IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

DESIGNING AND MODELING ANALYSIS OF DIFFERENT CRANKSHAFT BY USING ANSYS

[1]Giriteja Pitana, [2]M Rambabu, [3]I Satya, [4]G Swathi

[1]MTech Thermal Engineering Research Scholar, [2,3,4] Assistant Professor
[1,2,3,4] Department of Mechanical Engineering
[1,2,3,4,5] Visakha Institute of Engineering and Technology, Visakhapatnam, A.P, India.

Abstract: The reliability of any system, during which the linear displacement of a piston is regenerate into the rotation of a power transmission shaft, strongly depends on the reliability of the crankshaft. The crankshaft is that the vital element and any damage occurring to the crankshaft might put thesystem out of order. Crankshaft is massive volume production component with a complex geometry within the ICE. This converts the reciprocating displacement of the piston into a rotationof the crank. The crank shaft takes the power from piston that is generated because of combustionmethod within the combustion chamber of the cylinder. Throughout the power transmission method the load acts at a specific crank angle to the max and thus the connecting rod is analyzed for the stress developed, due to load conditions and the changes mentioned.

The existing works design first the 3D model of the engine parts are built in the software 'CATIA V5' and are then transferred to 'ANSYS'. The analysis of crank throw distortion and stress provides a conceptual support to enhance the design by weigh reduction. The proposed research works the 3D model of crankshaft system, obtained from CATIA V5 software is analyzed in ANSYS to assess the motionand loads acting on the crankshaft. Topology optimization is help to optimize the performance ofany machine. The present model is test in three different loading conditions 22624N, 32624N and42624N load is applied in crank shaft. After analysis ad comparison of all present models it is seenthat the lowest stress, deformation and weight are found in model 3 in all loading condition. The topology optimization process is help to reduce the material of workpiece without affecting the performance.

Index Terms: ANSYS, Crankshaft, CATIA V5, Forget steel, Topology optimization and Weight reduction

I.INTRODUCTION

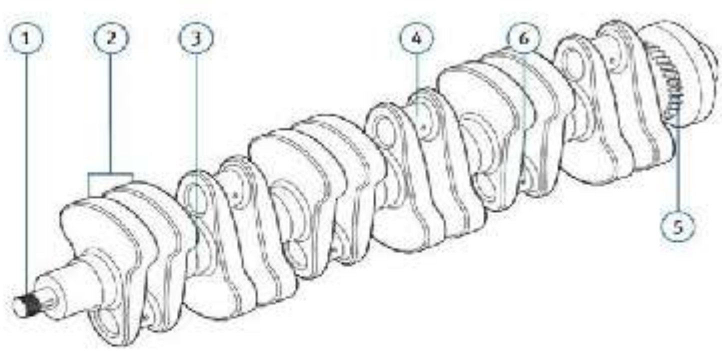
In various mechanical engineering applications the most widely used machine elements is Shaft. The crankshaft, impeller shaft, propellor shafts, camshafts etc. use shaft. Crankshaft is one among the most vital moving elements consisting of 2 web sections and one crankpin that convert the piston reciprocator displacement to a rotation with a four link mechanism. Crankshaft experiences massive forces from gas combustion. This force is applied to the highest of the piston and since the connecting rod connects the piston to the crank shaft, the forces are transmitted to the crankshaft. The magnitude of the forces depends on several factors that include crank radius, connecting rod dimensions, and weight of the rod, piston, piston rings, and pin. Since the crankshaft experiences an outsized range of load cycles throughout its service life, fatigue performance and durability of this element needs to be thought of within the style method. That the lifetime of internal combustion engine and reliability depend upon the strength of the crankshaft mostly and because the engine runs, the power impulses hit the crankshaft in one placeso another. Failures of shafts not only end in cost, however additionally in method period. The one among the most common causes of shaft failure is break. Fatigue failures are vital issues in mechanical styles. to determine the stress, geometry and dimensions Analysis, evaluations and engineering principles are utilized within the style. to confirm the lifetime of engine, strength calculation of crankshaft becomes a key issue. historically beam and area frame model were usedto calculate the stress of crankshaft however in these models the number of node is restricted. Withthe development of computer, additional and additional design of crankshaft has been used finite element methodology (FME) to calculate the stress of crankshaft.(Randhavan and Galhe, 2017).

The crankshaft is a moving part of the internal combustion engine (ICEIts main operate is to transform the linear motion of the piston into rotational motion. The pistons are connected to the crankshaft through the connecting rods. The crankshaft is mounted inside the engine block. The crankshaft is fitted into the engine block through its main journals. The connecting rods are fixed on the conrod journals of the crankshaft. On opposite sides of the conrod journals the crankshaft has counterweights that compensates outer moments, minimizes internal moments and thereforereduces vibration amplitudes and bearing stresses.

1.1 INTRODUCTION OF TOPOLOGY OPTIMIZATION IN ANSYS

Unless it's been topologically optimized, each part in an assembly most likely weighs over it has to. additional weight means that excess materials are being employed, loads on moving elements are more than necessary, energy efficiency is being compromised and shipping the part prices additional. Now, with Topology improvement technology, ANSYS Mechanical offers you the tools you need to style durable, light-weight elements for any application. You can outline
objectives simply and apply controls to confirm that producing necessities are met, minimummaterial thicknesses are set and exclusion areas are outlined.
Topology improvement in ANSYS Mechanical permits you to:

- Takings into account several static loads joined with optimizing for natural frequencies (modalanalysis)
- Fulfil necessities for minimum material thickness
- Observe rules around feature direction (for machining operations for example)
- Have scope for both cyclic and planar symmetry
- Easily validate results



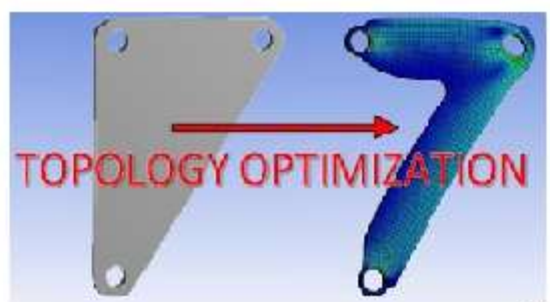


Fig.1.1 Crankshaft Nomenclature & Topology optimization

1.2 OBJECTIVE

- To model a crankshaft on software
- To perform static analysis of crankshaft
- To optimize the crankshaft design using topology optimization method.
- To reduce the overall weight of crankshaft implementing the new optimised design.
- To compare the existing crankshaft design with new optimised design in terms of obtainedvon-mises stress and deformation.
- To compare the weight of conventional crankshaft with new optimised design model.

1.3 PROBLEM STATEMENT

As crankshaft is generally made of ferrous material, it has high density the crankshaft become bulky. Because of bulky crankshaft weight of the engine is increases, hence selecting a new material to reduce the weight of the crankshaft without compromising on the properties of the material.

II.METHODOLOGY

2.1 STEP OF WORKING

Proposed methodology to be adopted while performing the design, optimization and analysis on crankshaft design:

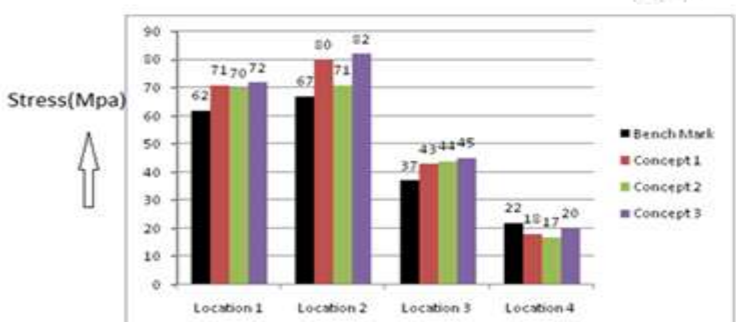
1. Design parameters in the present design were obtained by study of previous paper.
2. After obtaining the desired dimensions further the model in CATIA V5 will be created.
3. After the modelling part the design of crankshaft model will be further imported in Ansys static structural analysis workbench to perform the static structural analysis on it.
4. After the static analysis areas having higher and lower stress will be evaluated to perform the topology optimisation on it.
5. After the evaluation part further the model is imported in topology optimization work bench to have the areas which can be removed from the model having less stress values to have the weight reduction in overall crankshaft.
6. After the topology optimization again the model will be recreated in CATIA V5 and further static analysis will be performed.
7. After the analysis both the conventional and optimized model will be compared and evaluated to suggest the overall improvement in the crankshaft design.

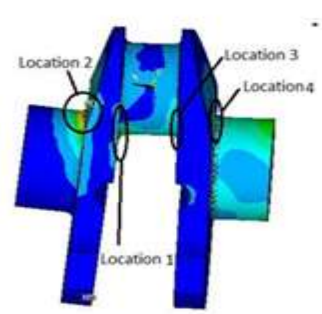
2.2 DESIGN ANALYSIS

Existing load condition and stress generation

The force acting on the crankpin for concept-1 due to gas loads at 4500 rpm. The maximum force acting on the crankpin is 22624 N. Similarly the maximum force acting on the crankpin due to gasloads at 4500 is 22066 N for concept-2 and 22303 N for concept-3.

Fig.2.1 Flow chart of methodology





It was found to be 22624 N, 22066 N and 22303 N respectively. The maximum stress for the bench mark model was found to be 67 MPa and whereas for the developed concepts 1, 2 and 3, it was found to be 80MPa, 71 MPa and 79 MPa respectively.

2.3 PROPOSED METHODOLOGY

Computer Aided Design gives point by point data on Computer Aided Design, Computer Aided Design Software, Computer Aided Design and Manufacturing, Computer Aided Design andEngineering and that's just the beginning. Computer Aided Design is associated with Cam and Computer Aided Design.

2.4 SOFTWARES IN USE

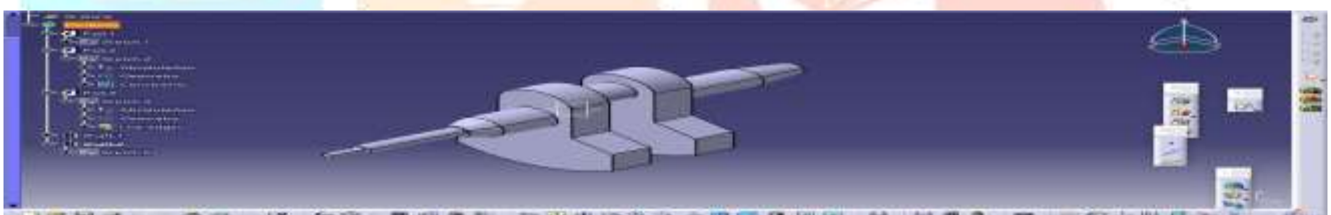
In light of the idea of CAD innovation, numerous CAD programming have been created by programming goliaths like Auto-work area Inc, Bentley, Dassult Systemes, Some of the main programming in the business are Auto-CAD, SOLIDWORKS, CATIA, Pro-Engineer, Uni-designs, Solid-Edge, STAAD Pro, Auto-Civil, Auto-work area Inventor and the rundown continues forever.

Because of CAD offices, the reiteration of work are limited, precise exactness can be accomplished; propagation isn't an issue now days. After change into computerized position it canlikewise be sent through electronic mail to any piece of the world as an editable document. Becauseof accessibility of a great deal of record designs, a similar document can be opened and utilized inan assortment of CAD programming.

Fig.2.2 Design View of Part 1 & 2

2.4.1 3D MODEL

In present research 1 basic model and 3 implement model is proposed. The crankshaft model 3D



geometry is created by using CATIA V5 and then it is import in ANSYS 19.0. And check all the result in three different load cases 22624N, 32624N and 42624N.

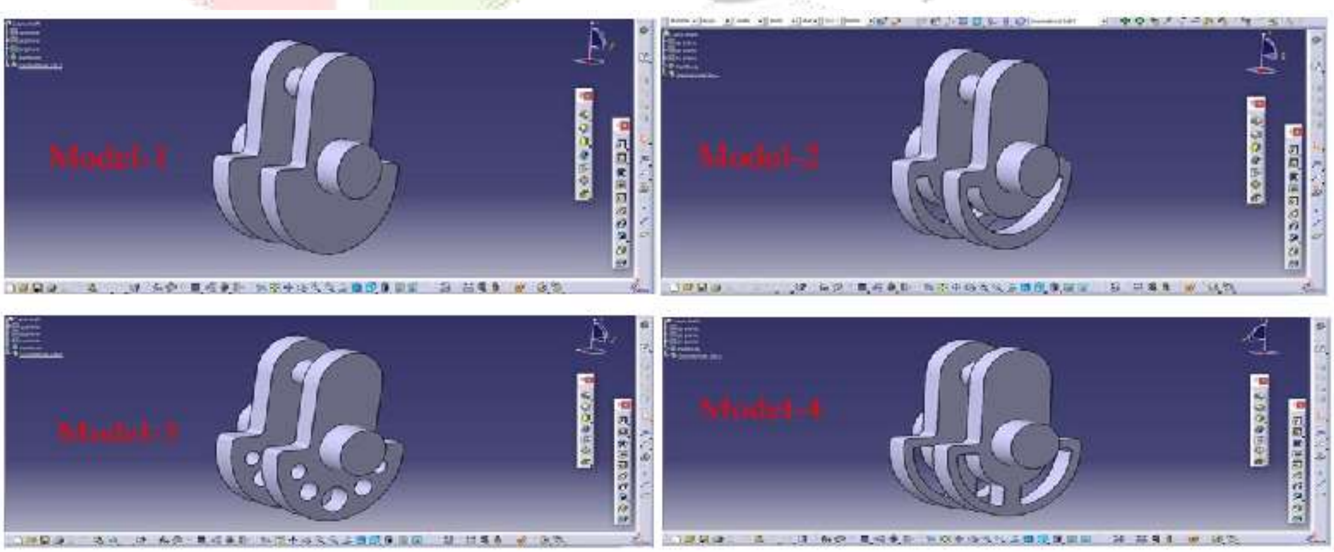


Fig.2.3Design View of Model 1,2,3&4

Table 3.1 Material Property

Material	Forget steel
Density (Kg/m3)	7833
Young's Modulus (MPa)	2.21E+05

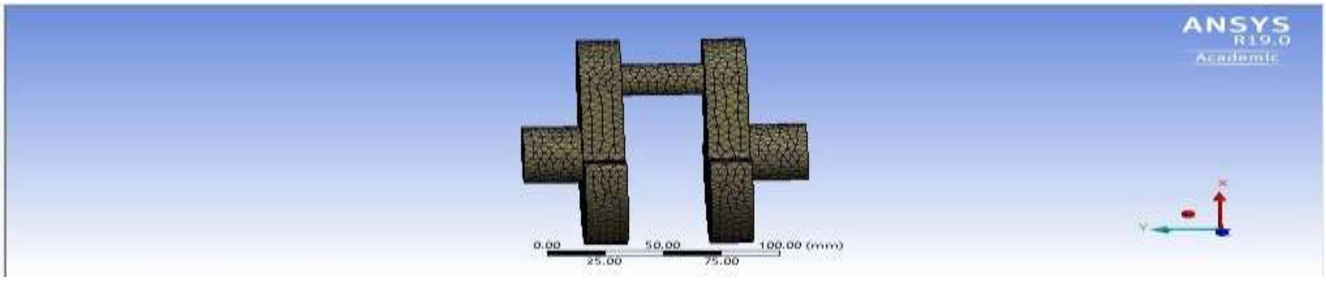


Fig.2.4 Meshing of crankshaft

2.5 Boundary condition

The FE model created was subjected to static structural analysis after assigning suitable material properties and boundary conditions.

The force acting on the crankpin for case-1 due to gas loads at 4500 rpm. The maximum force acting on the crankpin is 22624 N. The force acting on the crankpin for case-2 due to gas loads at 4500 rpm. The maximum forceacting on the crankpin is 32624 N.

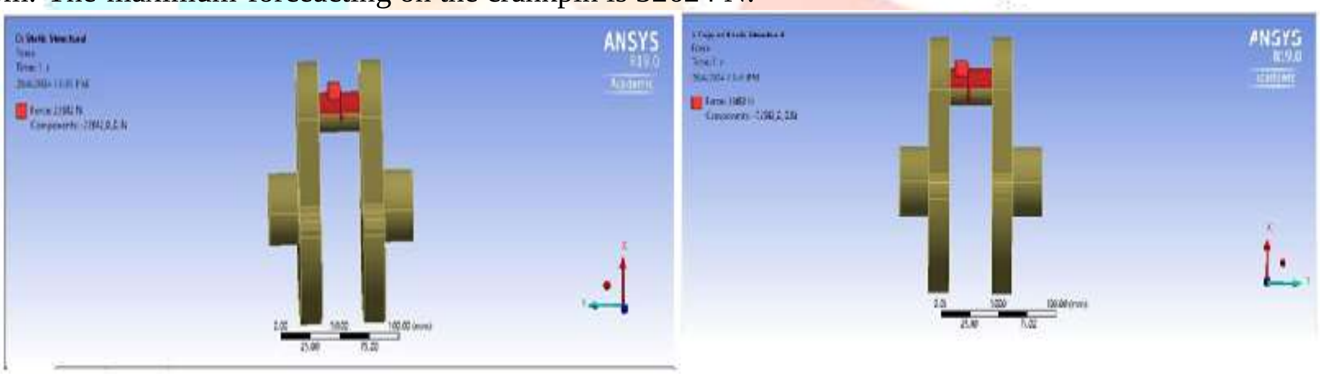


Fig.2.5 Applying force for Case-1 & Case-2

The force acting on the crankpin for case-3 due to gas loads at 4500 rpm. The maximum forceacting on the crankpin is 42624 N. Fixed support The crankshaft is fixed in both the end of shaft.

Fig.2.6 Applying force for Case-3 & Fixed support of Crankshaft

2.6 TOPOLOGY OPTIMIZATION

The ANSYS 19.0 provide the feature of topology optimization. After applying 22624N force the new model is proposed. The red region show that this material is does not affect the all over stressvalue. So all the new design is made only change in red region.

Fig.2.7 Topology optimization in ANSYS

III.SIMULATION RESULT

3.1 Case-1 In this case the maximum force acting on the crankpin is 22624 N.

(Model -1) Deformation: After applying 22624N force in crankshaft the maximum crankshaft is deformed in top of journal which is shown in figure 3.1(a) In this figure the following result is obtained, the blue legend shows the minimum deformation and red legend show maximum deformation in crank shaft. In this case the maximum deformation is 0.011156mm.

(Model-1) Equivalent Stress: After applying 22624N force in crankshaft the maximum stress ispresent in top of journal which is presented in figure 3.1(b). In this figure the following result is obtained, the blue legend shows the minimum stress value and red legend show maximum stress value in crank shaft. In this case 68.42 Mpa maximum stress is obtained.

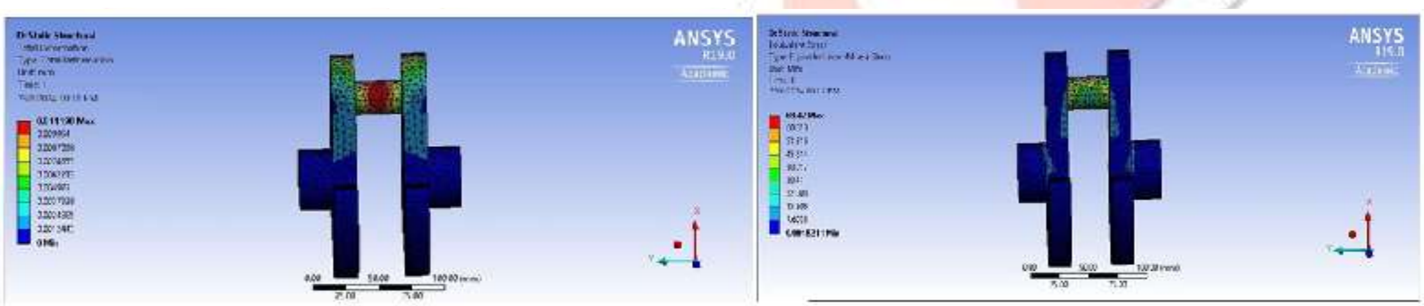


Fig.3.1 (a) Total deformation in Case-1 for Model-1 & (b) Equivalent stress in Case-1 for Model-1

(Model-4) Deformation: After applying 22624N force in crankshaft the maximum crankshaft is deformed in top of journal which is illustrated in figure 3.2(a). In this figure the following result is obtained, the blue legend shows the minimum deformation and red legend show maximum deformation in crank shaft. In this case 0.011207mm deformation is found.

(Model-4) Equivalent Stress: After applying 22624N force in crankshaft the maximum stress ispresent in top of journal which is displayed in figure 3.2 (b). In this figure the following result is obtained, the blue legend shows the minimum stress value and red legend show maximum stress value in crank shaft. In this case 4 74.156 Mpa maximum stress is found.

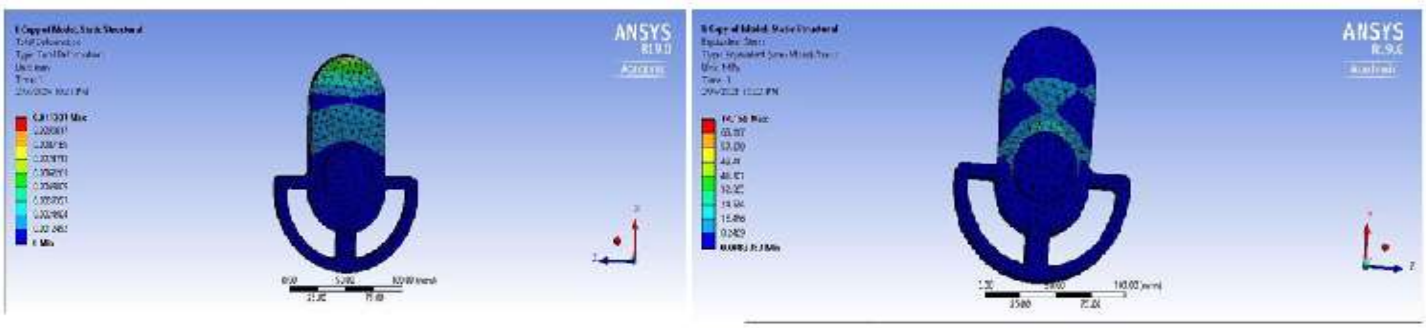


Fig.3.2 (a) Total deformation in Case-1 for Model-1 & (b) Equivalent stress in Case-1 for Model-4

IV.STRUCTURAL ANALYSIS OF CRANK SHAFT

The analysis parameter is shown in table 4.1.

Table 4.1: Properties of material

Material	Density(g/cm^3)	Youngs modulus(Gpa)	Poissions ratio
Aluminum alloy	2.6898	68.3	0.34
Titanium alloy	4.62	96	0.36
Magnesium alloy	1.8	45	0.35

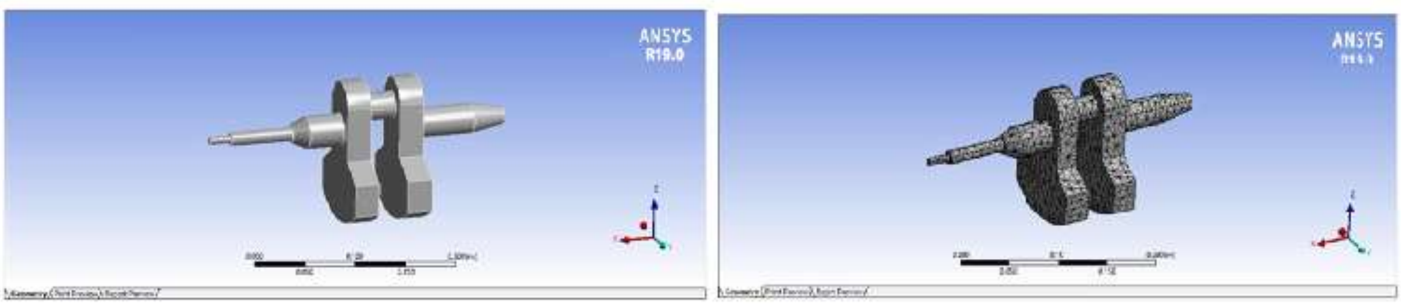


Fig.3.3 Model of Crank Shaft & Mesh of Crank Shaft

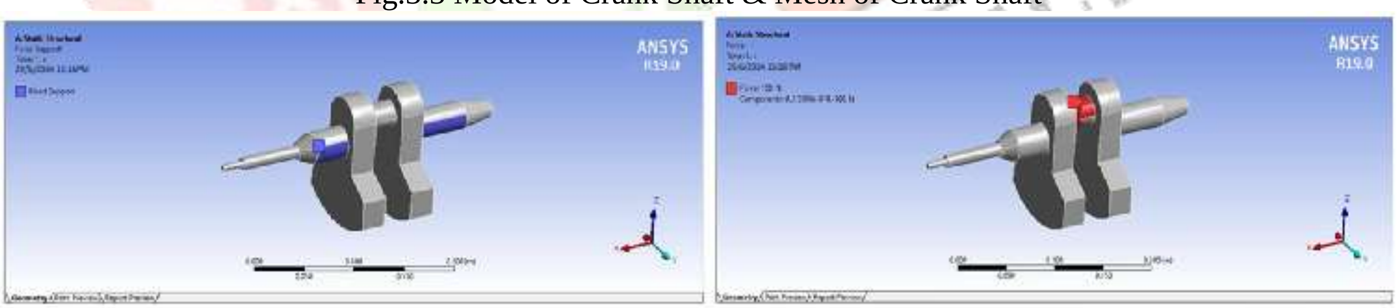


Fig.3.4 Fixed support of Crank Shaft & Load –force 100 N

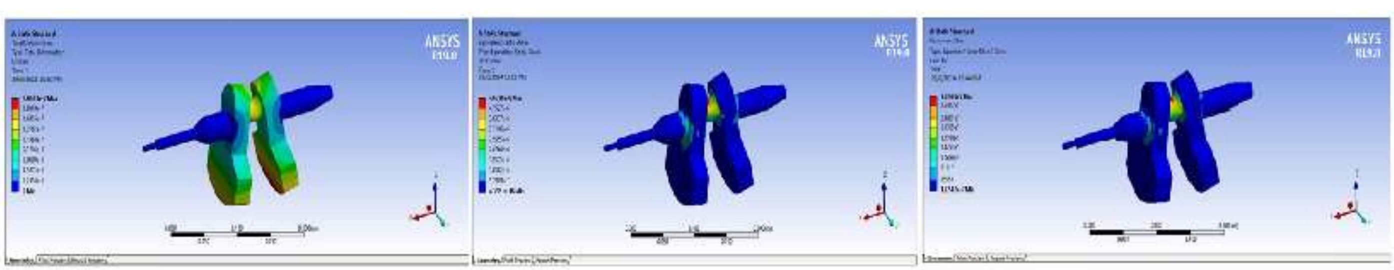


Fig.3.5 Total deformation of Crank Shaft & Strain of Crank Shaft & Stress of Crank Shaft for Aluminium Alloy

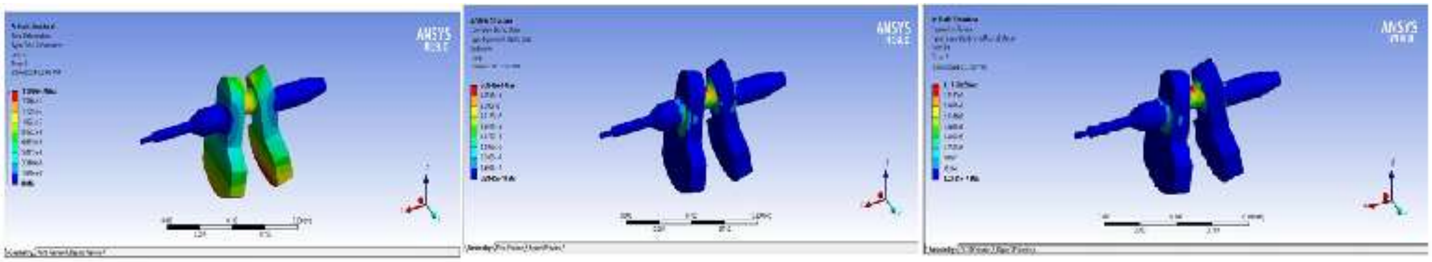


Fig.3.6 Total deformation of Crank Shaft & Strain of Crank Shaft & Stress of Crank Shaft for Titanium Alloy

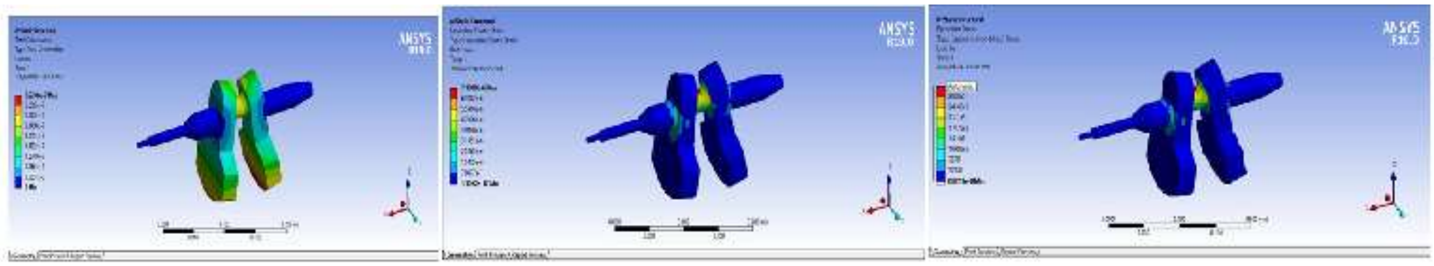


Fig.3.7 Total deformation of Crank Shaft & Strain of Crank Shaft & Stress of Crank Shaft for Magnesium Alloy

V. RESULTS & DISCUSSION

5.1 Deformation comparison

After performing analysis in Ansys following deformation are found as shown in table. The graph can be representing deformations in all 3 cases.

Table 5.1 Deformation comparison in all cases

Model	Case-1	Case-2	Case-3
1	0.011500	0.016144	0.021091
2	0.011182	0.016144	0.021079
3	0.011202	0.016149	0.021096
4	0.011207	0.057078	0.074564

5.2 Stresses Comparison:

After performing analysis in Ansys following Stresses are found as shown in table. The graph canbe representing stress in all 3 cases.

Table 5.2 Stresses comparison in all cases

Model	Case-1	Case-2	Case-3
1	68.42	98.68	128.86
2	68.70	108.57	148.83
3	63.49	91.533	119.57
4	74.156	176.84	231.01

5.3 Weight comparison:

After performing analysis in Ansys following Weight are found as shown in table. The graph canbe representing weight in all 3 cases.

Table 5.3 Weight comparison in all 3 cases

Case	Weight (Kg)
1	4.380
2	3.854
3	3.990
4	3.665

5.4 Validation of Work:

The maximum stress in previous is 67.14 Mpa and total deformation is 0.0433 mm which is varyclosed to over model at loading condition 22624N force on crankshaft. The maximum stress in proposed case 1 is 68.42 Mpa and total deformation is 0.0115 mm this value is very closed to previous work model.

Table 5.4: results for the analysis

Material	Total Deformation (M)	Strain	Stress (pascals)
Aluminum alloy	$2.0643e^{-7}$	$4.6718e^{-6}$	$3.2003e^{5}$
Titanium alloy	$1.5034e^{-7}$	$3.3248e^{-6}$	$3.1719e^{5}$
Magnesium alloy	$3.2248e^{-7}$	$7.1889e^{-6}$	$3.1815e^{5}$

5.5 CONCLUSION:

Crankshaft is a vital element of engine, failure even making engine useless conjointly needs expensive procurance and replacement. An intensive analysis within the past clearly indicates that the matter has not however been overcome fully and designers are facing lot of issues specially connected with multiaxial loading (Bending and Torsion), stress concentration and stress gradient and result of variable amplitude loading. Topology improvement was analyzed to the connecting rod. Based on these analysis results, concepts have been developed which reduce the weight of the crankshaft to a possible extent, without affecting the performance of the engine. The 3D model of crankshaft system, obtained from CATIA V5 software is analyzed in ANSYS to assess the motionand loads acting on the crankshaft. Topology optimization is help to optimize the performance of any machine. Some conclusions following are:

1. Topology optimization helps to optimize the performance of any machine. In these research three new models is proposed.
2. In previous research it is seen that The maximum stress for the bench mark model was found to be 67 MPa and whereas for the developed concepts 1, 2 and 3, it was found to be 80MPa, 71 MPa and 79 MPa respectively.
3. The present model is test in three different loading conditions 22624N, 32624N and 42624N load is applied in crank shaft.
4. After analysis ad comparison of all present models it is seen that the lowest stress, deformation and weight are found in model 3 in all loading condition.
5. The stress generate in three loading condition which are 22624, 32624, and 42624 N is 63.49, 91.53 and 119.57 Mpa respectively.
6. The weight of model 3 is 3.99Kg. The reduction of weight in present model is 9% less in overall weight of crank shaft.
7. It is conclude that the topology optimization process is help to reduce the material ofworkpiece without affecting the performance.
8. Modeling of crank shaft is done in catia v5 design programming by utilizing differentorders
9. The catia part record is changed over into IGS document and imported to ansys workbench.
10. First Static auxiliary investigation is done on spike gear at 100 N with three unique materials, for example, aluminum compound, titanium amalgam and magnesium composite in ansys workbench.
11. Maximum stress, twisting and most extreme strain are noted and arranged

REFERENCES:

[1] Rincle Garg, Sunil Baghla, "Finite element analysis and optimization of crankshaft", International Journal of Engineering and Management Reaserch, vol-2,Issue-6,ISSN: 2250- 0758, Pages:26-31, December 2012.

[2] C.M Balamurugan, R. Krishnaraj, Dr.M.sakhivel, K.kanthavel, Deepan Marudachalam M.G, R.Palani, "Computer Aided modelling and optimization of Crankshaft", International Journal of scientific and Engineering Reaserach, Vol-2, issue-8, ISSN:2229- 5518, August- 2011.

[3] Gu Yingkui, Zhou Zhibo, "Strength Analysis of Diesel Engine Crankshaft Based on PRO/E and ANSYS", Third International Conference on Measuring Technology and Mechatronics Automation, 2011.

[4] Abhishek choubey, Jamin Brahmbhatt, "Design and Analysis of Crankshaft for single cylinder 4- stroke engine", International Journal of Advanced Engineering Reaserch and studies, vol-1, issue-4, ISSN:2249-8974, pages: 88-90, July-sept 2012.

[5] R.J Deshbhratar, Y.R Suple, " Analysis and optimization of Crankshaft using FEM", International Journal of Modern Engineering Reasearch, vol-2, issue-5, ISSN:2249-6645, pages:3086-3088, Sept-Oct 2012.

[6] Farzin H. Montazersadgh and Ali Fatemi " Stress Analysis and Optimization of Crankshafts Subjected to Dynamic Loading", AISI, August 2007.

[7] Xiaorong Zhou., Ganwei Cai., Zhuan Zhang. Zhongqing Cheng., 2009, "Analysis on Dynamic Characteristics of Internal Combustion Engine INTERNATIONAL JOURNAL OF PROFESSIONAL ENGINEERING STUDIES Volume VIII /Issue 5 / AUG 2017 IJPRESCrankshaft System," International Conference on Measuring Technology and Mechatronics Automation.

[8] Farzin H. Montazersadgh and Ali Fatemi., 2007,"Dynamic Load and Stress Analysis of a Crankshaft," SAE Technical Paper No. 010258, Society of Automotive Engineers

[9] Jonathan Williams, Farzin Montazersadgh, and Alifatemi.,2007, "FatiguePerformance Comparison And Life Prediction Of Forged Steel And Ductile Cast Iron Crankshafts," Published in Proceeding of the 27th Forging Industry Technical Conference in Ft.Worth,Texas

[10] Shenoy, P. S. and Fatemi, A., 2006, "Dynamic analysis of loads and stresses in connecting rods,"IMechE, Journal of Mechanical Engineering Science, Vol. 220, No. 5, pp. 615- 624

[11] Zoroufi, M. and Fatemi, A.,2005, "A Literature Review on Durability Evaluation of Crankshafts Including Comparisons of Competing Manufacturing Processes and Cost Analysis", 26th Forging Industry Technical Conference, Chicago,

[12] Prakash, V., Aprameyan, K., and Shrinivasa, U.,1998, "An FEM Based Approach to Crankshaft Dynamics and Life Estimation," SAE Technical Paper No. 980565, Society of Automotive Engineers

[13] Payar, E., Kainz, A., and Fiedler, G. A., 1995,"Fatigue Analysis of Crankshafts Using Nonlinear Transient Simulation Techniques," SAE Technical Paper No. 950709, Society of Automotive Engineers

[14] Guagliano, M., Terranova, A., and Vergani, L., 1993,"Theoretical and Experimental Study of the Stress Concentration Factor in Diesel Engine Crankshafts, "Journal of Mechanical Design, Vol. 115, pp. 47-52

[15] Henry, J., Topolsky, J., and Abramczuk, M., 1992,"Crankshaft Durability Prediction – A New 3-D Approach," SAE Technical Paper No. 920087, Society of Automotive Engineers [16]. Stephens, R. I., Fatemi, A., Stephens, R. R., and Fuchs, H. O., 2001, "Metal Fatigue in Engineering,"2nd edition, John Wiley and Sons, New York, NY,USA co.