FEM ANALYSIS AND OPTIMIZATION OF CRANKSHAFT USING ANSYS

Punit Nagra[1], Dr. Mahesh Sawhney[2]

M.tech Scholar in Shree Sidhivinayak educational trust of institutions[1],

Professor in Shree Sidhivinayak educational trust of institutions[2]

ABSTRACT: In this study, a single cylinder 4 –stroke diesel engine crankshaft was taken and a static investigation will be conducted to get stress magnitude's variation at crankshaft's critical locations. A model will be created in SolidWorks 11.0 of the crankshaft and imported into Ansys 13.0 to carry out static analysis. The meshing of the crankshaft will be done; boundary and load conditions will be used according to the crankshaft's mounting conditions on Finite element model of the crankshaft. Furthermore, the crankshaft is optimized with the help of the analysis results. The crankshaft is analyzed using a static analysis method. The materials considered for crankshaft are EN308. By comparing; Gopal et al. 2017 and examining the results of static analysis, three was an increase in the stress as well as deformation is less when EN308 Crankshaft is taken. Weight Optimization will be achieved by varying the crank pin diameter. Also, it is required that stress range in FE analysis does not go beyond the stress range magnitude presented in the original crankshaft.

Keywords: Crankshaft, EN308, SolidWorks, ANSYS.

1. INTRODUCTION

In an I.C engine, one of the large components is the crankshaft through which piston's reciprocating motion is converted into rotatory motion with the help of a 4 bar link mechanism. The crankshaft is a large and very complex dynamic structure. The main components of a crankshaft are one crankpin bearing, two journal bearings and shaft part. Within the central bearing the shaft parts revolve, the large area of the connecting rods is connected to the crank pins, and the shaft parts and crank pins are connected through the crank webs or arms (also called cheeks). At the times of power stroke, the crankshaft is supposed to bear downward force without bending too much. Hence the life, as well as service, are viewed from the strength perspective largely.

1.1 Function of Crankshaft in IC Engines

A four-bar slider-crank mechanism is constituted by piston, connecting rods and crankshaft. This mechanism helps in converting the piston's sliding motion into rotary motion. The engines are being designed in such a way that it will provide the rotatory output, the reason behind this is that the when input is provided to other devices which is in rotatory form that is more applicable and practical. Additionally, it was observed that an engine's linear displacement is also not smooth which results in gas combustion in combustion chamber. Hence, this displacement resulted in unexpected shocks and when this output is used by another devices as input that may damage it.

Figure 1 shows the crankshaft's mounting in an engine and for a 4-stroke cycle engine during an engine cycle is represented through a P-V diagram in figure 2. Cylinder's volume when the piston is at BDC (Bottom Dead Center) is represented by V_{bdc} and V_d represents the volume swept by the piston.

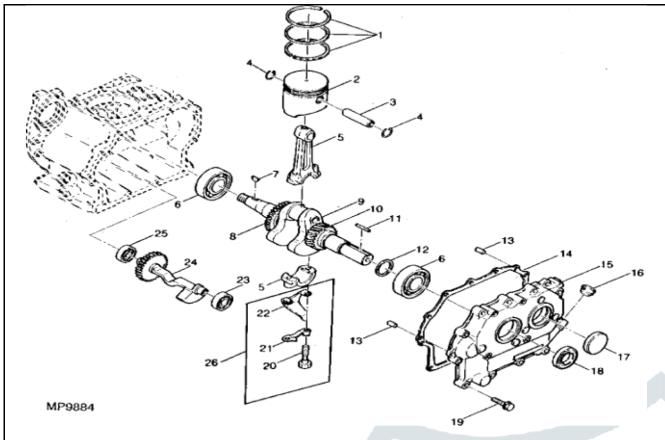

Fig 1: Crankshaft mounting in Single cylinder engine (http://www.deere.com/, 2005).

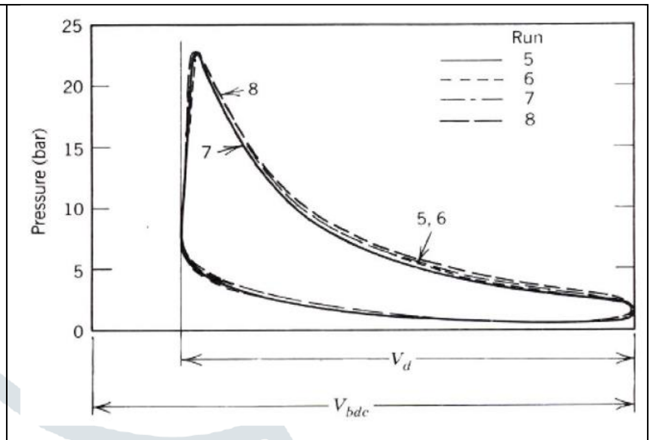

Fig 2: P-V diagram

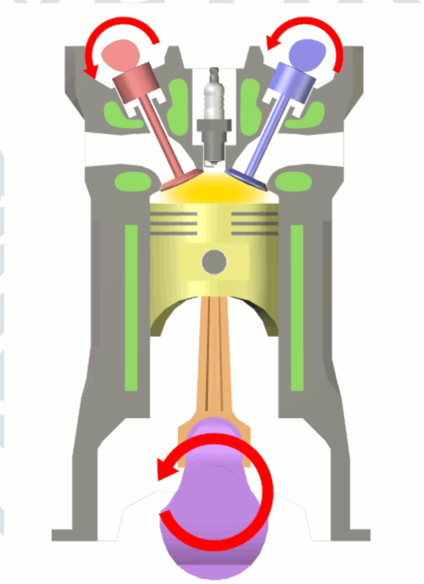

Fig 3: Side view (Purple in color) of the engine block at the time of combustion

1.2 Materials and Manufacturing Processes

The main materials for crankshaft that is used by the industry are cast iron and forged steel. For the automotive industry, the performance comparison on the basis of impact, cyclic and static loading are of main concern. Furthermore, a comprehensive comparison is conducted between processes of manufacturing w.r.t. crankshafts finished cost, manufacturing aspects, and mechanical properties.

1.3 Finite Element Method

The FEM is basically a result of an era of electronic digital computer. The numerical approximations have many same features with the other approaches. The high-speed computer shows some more features and has many advantages. For a difficult or complex program like nonlinear stress-strain behavior and non-homogeneous materials as well as complex boundary conditions, these methods are easily programmed.

For the engineering problem,FEMis a good technique because with thisestimated solutions are obtained. FEM is an analysis tool and due to its flexibility as well as diversity, it is very popular in industries as well as in schools and colleges. It is easy to obtain the approximation value of an experiment rather than an exact solution by the help of these tools. But for engineers, it is not used to find the analytical mathematical solution.

1.4 ANSYSIntroduction

In 1970, Dr. John A. Swanson established this company with the name “Swanson Analysis Systems, Inc. SASI”. Their main objective was to manufacture and promote the FEA software for structural physics through which the heat transfer (thermal), dynamic (moving) and static (stationary) problems can be simulated. There was a parallel growth in SASI business along with computer technology development as well as engineering requirements.Every year there was 10%-20% growth in the company and it was sold in 1994. The SASI’s leading software was considered as the main product by the new owners and named it ANSYS. An extensive range of mechanical problems can be solved numerically with the help of ANSYS, being a general purpose finite element modeling package. These mechanical issues are electromagnetic and acoustic problems, heat transfer and fluid problems, as well as static/dynamic structural analysis (both linear and non-linear).

Various ANSYS software are ANSYS (FEA), ANSYS/LS-Dyna, ANSYS Structural, ANSYS Fluent & CFD, ANSYS Transient Dynamic Analysis, ANSYS Buckling Analysis, ANSYS Thermal, ANSYS Coupled Fields, ANSYS Modal Analysis, ANSYS Harmonic Analysis, Robust Meshing, and Superior CAD Interface.

2. LITERATURE REVIEW

Gopal et al. (2017) describe thecrankshaft, piston, as well as connecting rods of a four-wheeler petrol engine in his study. Assembly’s components should have been rigid as well as moved like a machine. Connecting rod, engine piston, and crankshaft are the major parts of the assembly and as per the given data or design, they are modelled and manufactured. In ANSYS the FEM is done as well as Hyper Mesh was done by meshing.

Sandhya et al. (2016) analyzed the crank’s crankshaft with the use of Finite element software ANSYS. With using 3 materials crankshaft’s static analysis is done in various orientations. Two crank position’s results are proved with theoretical calculations for every material.

HailemariamNigus (2015) said in his paper about the kinematics formulation internal combustion engine crank mechanism. The vector loop method is used for kinematics formulation of the crank mechanism as well as piston's position is to explain by using cosine rules. With the help of MATLAB software, the mathematical algorithm and visualization of the 2D model system are obtained with the use of 2D Auto CAD software.

K. Prasad and A. V. S. S. Somasundar (2014) carried out static analysis. The meshing of the crankshaft was done; loads and boundary conditions which were applied like the mounting conditions of the crankshaft on FEM of the crankshaft. The cast iron crankshaft optimization is carried out by results that were achieved from the analysis.

K. Thriveni and Dr. B. Jaya Chandraiah (2013) studies about that, from a single cylinder 4-stroke I.C Engine, how the static analysis is carried out on crankshaft. CATIA-V5 Software is used to create crankshaft's modeling. At critical areas of the crankshaft (FEA), FEM is used to get the variances of stress with using the boundary conditions of ANSYS software.

C. M. Balamurugan et al (2011) study the Computer-aided modeling and optimization analysis of the crankshaft. This paper was based on ductile cast iron and forged steel, the manufacturing techniques and the result of these two were compare and evaluate to calculate the performance of each other. On forged steel crankshafts as well as on cast iron crankshaft, the dynamic simulation was conducted in the paper which is generated from single cylinder 4-stroke diesel engine.

3. RESEARCH METHODOLOGY

3.1 Methodology

In this work, Solidworksused for creating the three-dimensional CAD model and ANSYS has been used for the FEA. The static structural analysis is done by ANSYS 13.0 Simulation Software to Engine configuration where crankshaft fits is same as of Gopal et al. (2017).

3.2 Objective and Proposed Work

For modeling and optimization, we have selected a base paper of Gopal et al. (2017), they describe thecrankshaft, pistonas well as connecting rodof a 4-wheeler petrol engine in their study. In their work, the assembly's main components that are crankshaft, connecting rod, as well as engine piston, are modeled and then connected according to a particular design. Furthermore, FEA is performed in ANSYS. The crankshaft is manufactured generally using cast iron or alloy steel.However, in our work, the material of crankshaft is same as of Gopal et al. (2017).A crankshaft model will be created in CATIA and imported into ANSYS to carryout an analysis.

3.3 Finite Element Method for CAD Modal of Multi-Leaf Spring

The Crankshaft with material properties and boundary conditions are introduced in ANSYS and then analyze the results. The EN308 and High Alloy Steel are used as crankshaft analysis materials. FEA (Finite Element Analysis) is used in performing stress-deflection analysis. The ANSYS-13.0 is used to perform the complete analysis procedure. The FEA process is categorized into three stages: Pre-processor, Solution, and Postprocessor.

After that a validation stage is performed which mainly focus on to perform a comparison between FEM solutions with experimental tests or analytical solutions, or known published results so that the FEM model's correctness is validated.

4. DATA COLLECTION AND ANALYSIS

4.1 Material used

The piston's reciprocating (up and down) motion is converted into rotatory (turning) motion by the crankshaft. The wheel's turning motion is actually provided by this. Connecting rods are used to connect the piston and the crankshaft. Mainly the cast iron or alloy steel is used as crankshaft materials.

4.1.1Material properties

Table 1: Material Properties of EN308

PARAMETERS	VALUES
The material used- Steel	**EN308**
Young's Modulus E (N/mm^2)	205000
Poisson's Ratio	0.29
Density (kg mm^-3)	0.00000785

4.2 Computational study

The 3D-modeling is performed by a specialized software tool that is known as CAD modelling. Several complicated parts are included in the multi-leaf spring structures model that cannot be developed by other Finite Element and CAD modelling software. The crankshaft's CAD modellingis performed by using SolidWorks-11 software.

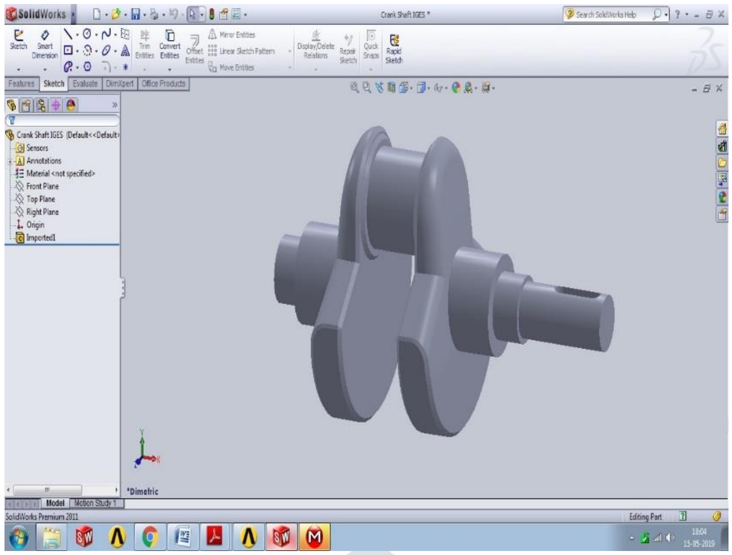

Fig 4: Crankshaft in Solid Works 11.0

4.3 Static structural analysis using ANSYS

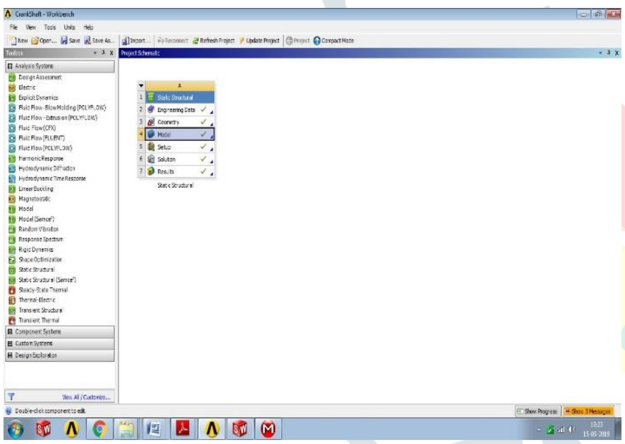

Fig 5: ANSYS geometry import

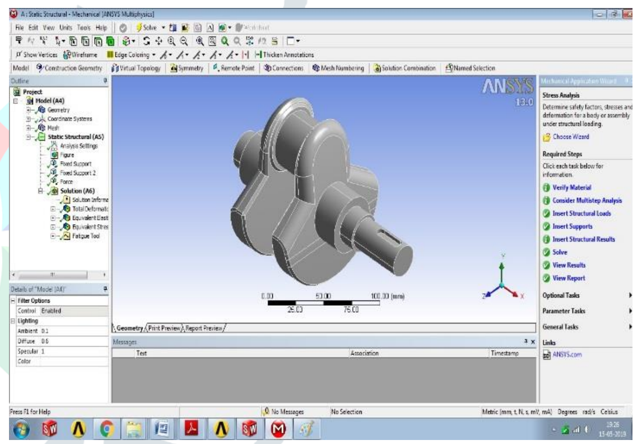

Fig 6: Crankshaft ANSYS CAD model for CAE analysis

4.4 Analysis environment setting

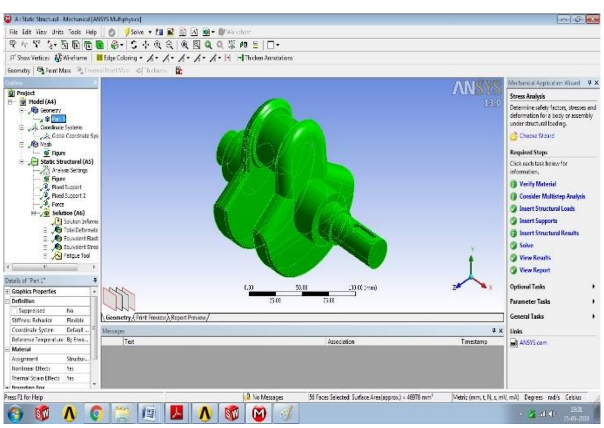

Fig 7: Assigningcrankshaft material properties

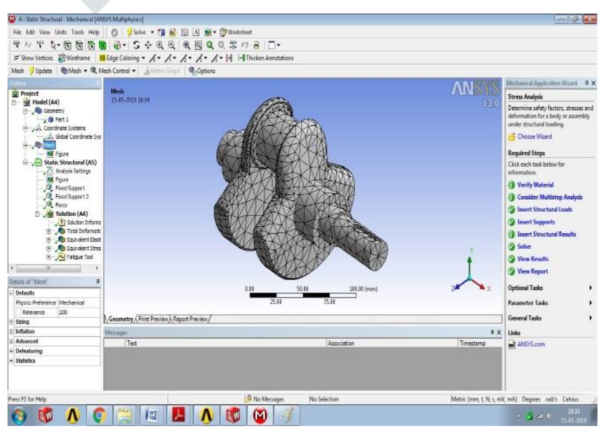

Fig 8: Crankshaft’s meshed model

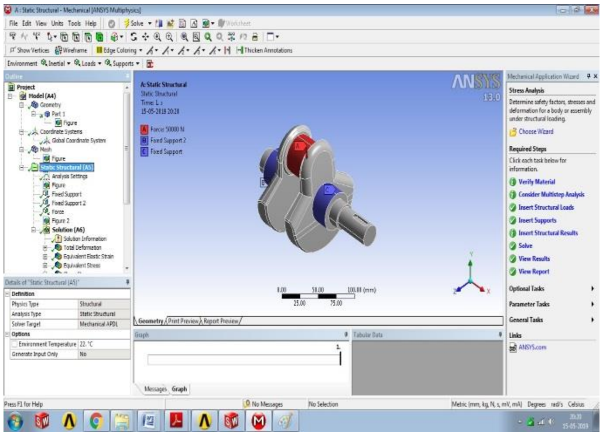

Fig 9: Boundary condition of crankshaft model

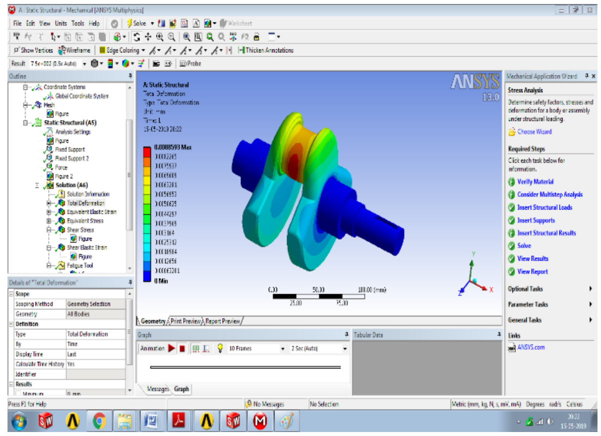

Fig 10: Total deformation at 50000 N load

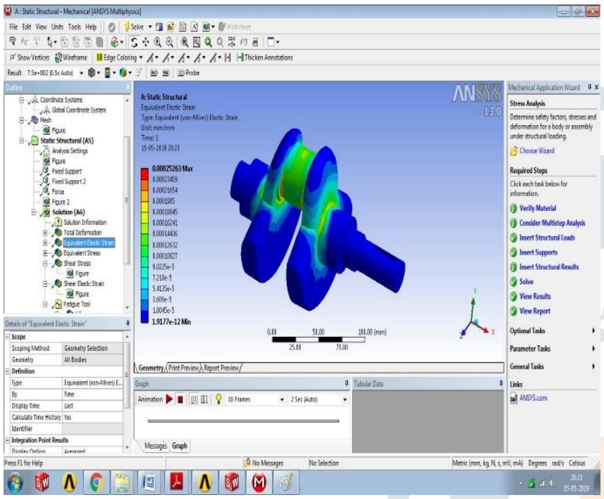

Fig 11:Crankshaft Equivalent Elastic Strain at 50000 N load

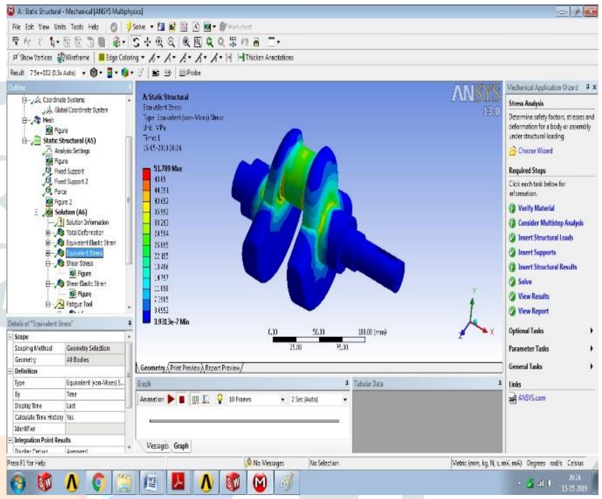

Fig 12: Crankshaft Equivalent Elastic Stress at 50000 N load

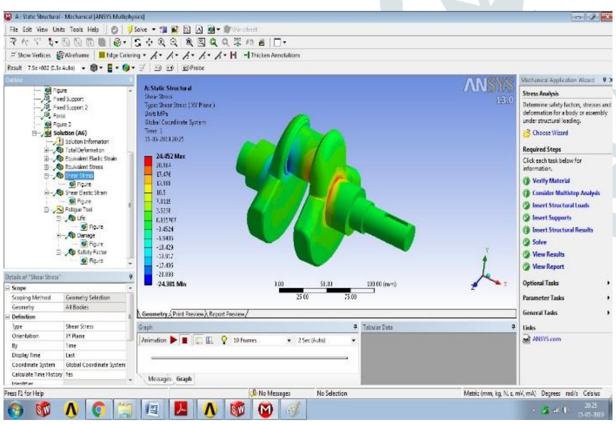

Fig13: Crankshaft share stress at 50000 N load

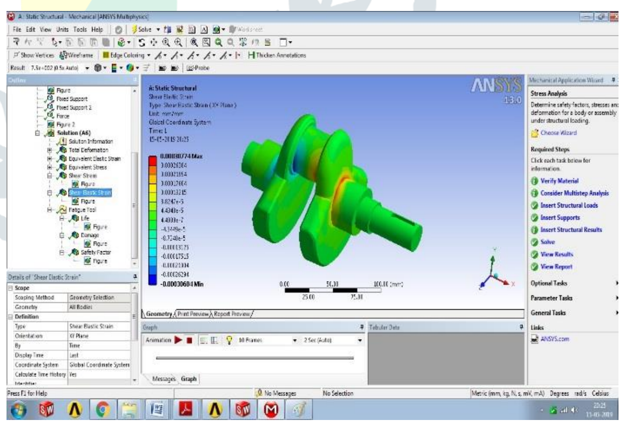

Fig 14: Crankshaft shear strain at 50000 N load

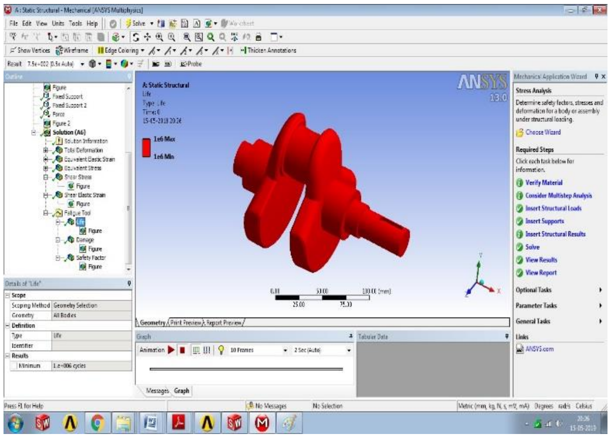

Fig 15: Crankshaft life at 50000 N load

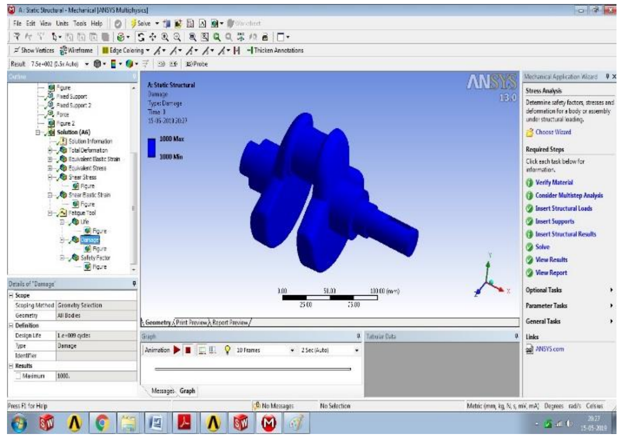

Fig 16: Crankshaft damage at 50000 N load

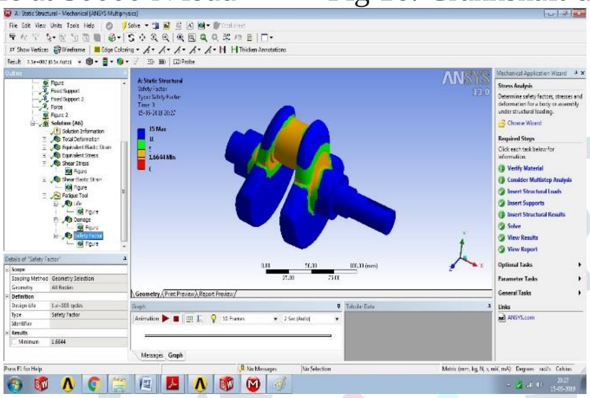

Fig 17: Showing Crankshaft Safety Factor at 50000 N load

5. RESULTS AND DISCUSSION

On the basis of CAE analysis the following results are obtained as represented byTable 2:

Type	Load (N)	Total deformation (mm)	Equivalent Elastic Strain (mm/mm)	Equivalent Elastic Stress (Mpa)	Shear Stress (Mpa)	Share Strain (mm/mm)
Max	50K	0.0088593	0.0002563	51.789	24.452	0.00030774
Min	0K	0.0	$1.9177*10^{-12}$	$3.9313*10^{-7}$	-24.381	-0.00030684

Static analysis was done on the crankshaft. The materials considered for crankshaft are EN308.By comparing [Gopal et al. 2017] and observing the static analysis results:

- It is cleared from our results that FEA Results Conformal is equivalent to that of Gopal et al. Therefore, it is concluded that for reducing the time consumption of theoretical work the FEA can be used as a great tool. Also, it was observed that the center of the neck surface of crankpin has the greatest deformation. The areas like near the central point Journal and fillet between crank cheeks and crankshaft journal experiences the maximum stress. Although, the high-stress area is observed at edges of the main journal.
- The stresses are decreased and displacements are less when EN308 Crankshaft is taken.

- As compared to material yield stress, the Von-Misses stresses value is very small which concludes that our design is safe and sound, therefore, optimization must be used for reducing the cost of the material.
- Crankshaft's dynamic analysis is performed after performing the static analysis, and results of dynamic analysis were realistic and overestimate results were shown by static analysis. Deformation and accurate stresses are used as a critical input for crankshaft's optimization and fatigue analysis.
- **Analysis Results:** Finally, we conclude that the cost of expensive experimental work can be reduced by using dynamic FEA.

6. CONCLUSION AND FUTURE SCOPE

Table 2 represents the various analysis results that are obtained after crankshaft is tested under static load containing fatigue life, deflection, andstresses. As the EN308 crankshaft easily survived through the static load, we determined that there exists no concern in terms of strength, when the EN308 crankshaft is replaced by the cast iron crankshaft. Additionally, by mass production, the cost of EN308 crankshaft is also reduced. Lastly, it is concluded that this project will help in making an impressive mark in the area of automobile industries.

7. REFERENCES

[1] A text book of Machine Design by R K Jain.

[2] Text book on Machine Design by R.S. Khurmi.

[3] Bhandari, V. B., 1994, "Design of Machine Elements," Tata McGraw-Hill.

[4] HAUG, E.J., WEHAGE, R.A., and BARMAN, N.C., 1982, "Dynamic Analysis and Design of Constrained Mechanical Systems", Journal of Mechanical Design.

[5] NIKRAVESH P.E., 1988, Computer-Aided Analysis of Mechanical Systems, Prentice Hall, Englewood Cliffs, New Jersey.

[6] MbonoSamba YC, Pascal M (2001) Dynamic analysis and numerical simulation of flexible multibody systems. MechStruct Mach 29(3).

[7] G Gopal, Dr. L Suresh Kumar, K VijayaBahskar Reddy, M Uma Maheshwara Rao, G Srinivasulu (2017), Analysis of Piston, Connecting rod and Crankshaft assembly, Materials Today: Proceedings, 4 (2017), 7810–7819.

[8] K.Sandya, M Keerthi, K.Srinivas (2016), Modelling and Stress Analysis of Crankshaft Using FEM Package Ansys, 3(1), 687-693.

[9] Surekha S. Shelke, Dr. C. L. Dhamejani, A. S. Gadhave (2016), Modeling and Analysis of the Crankshaft Using ANSYS Software, International Journal of Engineering Science and Computing, 6(6), 6796-6800.

[10] K. Prasad, A.V.S.S. Somasundar (2014), Modeling and Optimization of Crankshaft Design using ANSYS, International Journal of Engineering and Management Research, 4(4), 285-289.

[11] Sanjay B Chikalthankar, V M Nandedkar, Surender Kumar Kaundal (2013), Finite Element Analysis Approach for StressAnalysis of Crankshaft under Dynamic Loading, International Journal of Scientific & Engineering Research, 4(2), 1-6.

[12] Amit Solanki, JaydeepsinhDodiya (2014), Design and Stress Analysis of Crankshaft for SingleCylinder 4-Stroke Diesel Engine, International Journal for Research in Applied Science and Engineering Technology, 2(5), 320-324.

[13] K. Thriveni,Dr. B. Jaya Chandraiah (2013), Modeling and Analysis of the Crankshaft Using Ansys Software, International Journal of Computational Engineering Research, 3(5), 84-89.

[14] R. J. Deshbhratar, Y. R. Suple (2012), Analysis & Optimization of Crankshaft Using Fem, International Journal of Modern Engineering Research, 2(5), 3086-3088.

[15] Farzin H. Montazersadgh and Ali Fatemi – "Stress Analysis and Optimization of Crankshafts Subject to Dynam-ic Loading" - FIERF, AISI

[16] MuhammedNasiruddin Bin Anidin –" Finite element analysis of upper crank shaft six stroke engine using CAE Software" – University Malaysia Pahang.

[17] Steve Smith – " Utilizing crankcase deflection analysis to improve crankshaft design and engine performance" – Oxford, U.K

[18] Jonathan Williams, FarzinMontazersadgh and Ali Fatemi –"Fatigue performance comparison and life prediction of forged steel and ductile cast-iron crankshafts" –University of Toledo, Ohio.

[19] M.Zorouf and A.Fatemi- "The durability evaluation of crankshafts including comparisons of competing manu-facturing processes and cost analysis.

[20] ZissionsP.Mourelatos –"A crank shaft system model for structural dynamic analysis of internal combustion en-gines".