SSRG International Journal of Recent Engineering Science
ISSN: 2349–7157 / https://doi.org/10.14445/23497157/IJRES-V8I2P101
Volume 8 Issue 2, 1-9, Mar-Apr, 2021

Original Article

Modeling and Analysis of Crankshaft (Using ANSYS)

Himanshu Bist[1], Himanshu Bhatt[2]

[1,2]*Department of Mechanical Engineering, Delhi Technological University, Shahbad Daulatpur, Delhi, India*.

Received: 01 March 2021 Revised: 01 April 2021 Accepted: 11 April 2021 Published: 27 April 2021

Abstract *- Crankshaft is an important engine component used for converting the reciprocating motion of the piston to rotary motion. It is necessary to ensure that the crankshaft does not fail under load for the smooth functioning of the engine. In this paper, we performed the structural analysis of a single-cylinder crankshaft using ANSYS for 4 different materials to see which is the most suitable for it. We used Solidworks for creating the 3-d model of the crankshaft, which was then imported to ANSYS, where mesh was created and forces were applied. Then the deformation and stresses were analyzed and compared to find the regions of high-stress concentration and to see the most compatible material.*

Keywords *- Crankshaft, ANSYS, Solidworks, Modelling, Cylinder.*

1. Introduction

The crankshaft is a big feature with a complicated geometry in the I.C engine that uses a four-bar relation system to transform the reciprocating displacement of the piston to a rotary motion. Shaft sections, two journal bearings, and one crankpin bearing make up the crankshaft. Shaft parts that rotate in main bearings, crank pins to which the connecting rod's big end is attached and crank arms or webs that link the crank pins and shaft parts.

Furthermore, an engine's linear displacement is not smooth; since the combustion chamber induces the displacement, it has abrupt shocks. Crankshafts are used to convert abrupt displacements into a smooth rotary output and are the input to several instruments, including engines, pumps, and compressors. It is also worth noting that using a flywheel helps to balance out the shocks.

The crankshaft is subjected to significant forces as a result of gas combustion. This force is applied to the top of the piston, and since the piston is attached to the crankshaft by the connecting rod, the force is transferred to the crankshaft. The magnitude of the forces is determined by a number of variables, including crank radius, connecting rod dimensions, connecting rod weight, piston, piston rings, and pin.

Torsional and bending loads are created by combustion and inertia forces acting on the crankshaft. The crankshaft must be powerful enough to withstand the downward force of the power stroke without undue bending, so the internal combustion engine's stability and life are highly dependent on its strength.

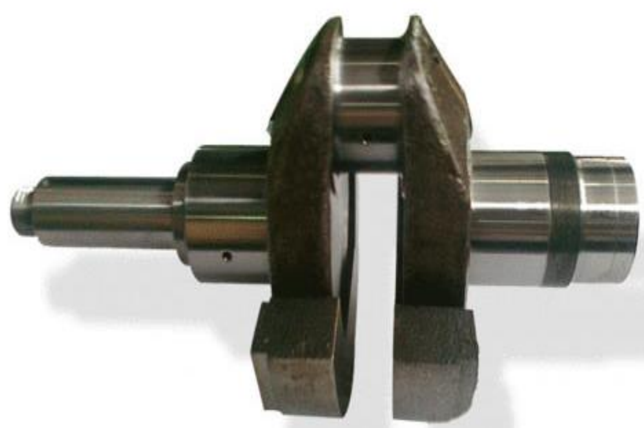

Fig. 1 A single cylinder crankshaft [1]

The crankpin functions as a built-in beam with a spread load that varies with the crank direction. Each web resembles a cantilever beam that is bent and twisted. The bending moment, which causes tensile and compressive pressures, and the twisting moment, which causes shear stress, are the two main causes of crankshaft failure.

There are various causes of engine failure; one of the most frequent is fatigue in the fillet areas caused by the bending load caused by combustion. The load from the piston is transferred to the crankpin at the moment of combustion, creating a significant bending moment on the whole geometry of the crankshaft. There are stress concentrations at the root of the fillet regions, and these high-stress range positions are where cyclic loads could cause fatigue crack initiation and fracture.

2. Literature Review

Prasad et al. [2] used CATIA V5 software to create a three-dimensional body of a single-cylinder crankshaft engine and later analyzed it using HyperMesh software to compare the result with their theoretical calculations. Their results matched their theoretical calculations to conclude that their design was safe.

Reddy et al. [3] created the model of a crankshaft using Pro-E software and then performed its Finite Element Analysis (FEA) using ANSYS for the composite material, and forged steel and results showed that the composite material had better performance than forged steel

Rinkle Garg and Sunil Baghl created a crankshaft model. [4] using Pro-E Software, and then it was imported to ANSYS software. They made different designs by changing web thickness and reducing material from the journal. The result shows an improvement in the strength of the crankshaft as the maximum limits of stresses, total deformation, and strain are reduced.

A 3-d model of the crankshaft of a diesel engine was created by Jonafark and Reddy [5] using Unigraphics software and was then performed Dynamic Analysis of it in ANSYS for Cast iron and forged steel body to compare their results and see high-stress concentration areas which came along the edge of the main journal.

Design specifications were calculated by Harshada et al. [6] for the crankshaft. Then the model was created using CATIA, which was then later analyzed using HYPER MESH for four different grades of steel to see which was the most suitable out of them.

Deviyaprasanth [7] performed the structural analysis and harmonic analysis of a single cylinder crankshaft using ANSYS by creating a tetrahedron mesh model, whose 3-d model was created in SolidWorks and calculated the von-mises stresses and shear stresses. They were under yield stress values, thus making his design safe.

Shenkar and Biradar[8] calculated the design specifications for a 3-d model of a single-cylinder crankshaft and created its 3-d model using Pro/E software, and then performed its structural analysis using ANSYS to validate their results. They observed high-stress concentrations in fillet areas.

3. Modelling

The first step in this project was to construct a 3-D model of the crankshaft, which was then analyzed with ANSYS. The 3-D model was made in SolidWorks according to the requirements specified in TABLE 1.

Table 1. Specifications of crankshaft [9]

Parameter	Value (mm)
Diameter of Crank Pin	44
Length of pin	33
Web Thickness (Left and Right)	24
Web Width (Left and Right)	64
Bore Diameter	86
Shaft Diameter	42
Max. Cylinder pressure	25 bar

4. Calculations [10]

D = Bore Diameter, P_{max} = Maximum cylinder pressure

Force on piston = $F_P = [\pi D^2 P_{max.}]/4 = 14.514$ KN

ϕ = inclination of connecting rod with the line of stroke;

θ = crank angle

$\text{Sin}\phi = \frac{\sin\theta}{L/R} \approx \frac{\sin 35}{4} \rightarrow \phi = 8.24^o$

Now, Thrust force on connecting rod = $F_{.Q.}$

$F_{.Q.} = F_{.P.} / \cos\phi$

Therefore, $F_Q = 14.675$ KN

Now, force on the crankshaft will have tangential and radial components

a) Tangential force on crankshaft:

$F_T = F_Q \sin(\theta + \phi) = 10.05$ KN

b) Radial force on crankshaft :

$F_R = F_Q \cos(\theta + \phi) = 10.68$ KN

5. Meshing and Analysis

The crankshaft was subjected to Finite Element Analysis (FEA) in this situation. Finite Element Analysis (FEA) is a computational tool for breaking down a complex structure into tiny parts known as components.

The program creates a concise understanding of how the system works as a whole by implementing equations that control the behavior of these components and solving them all.

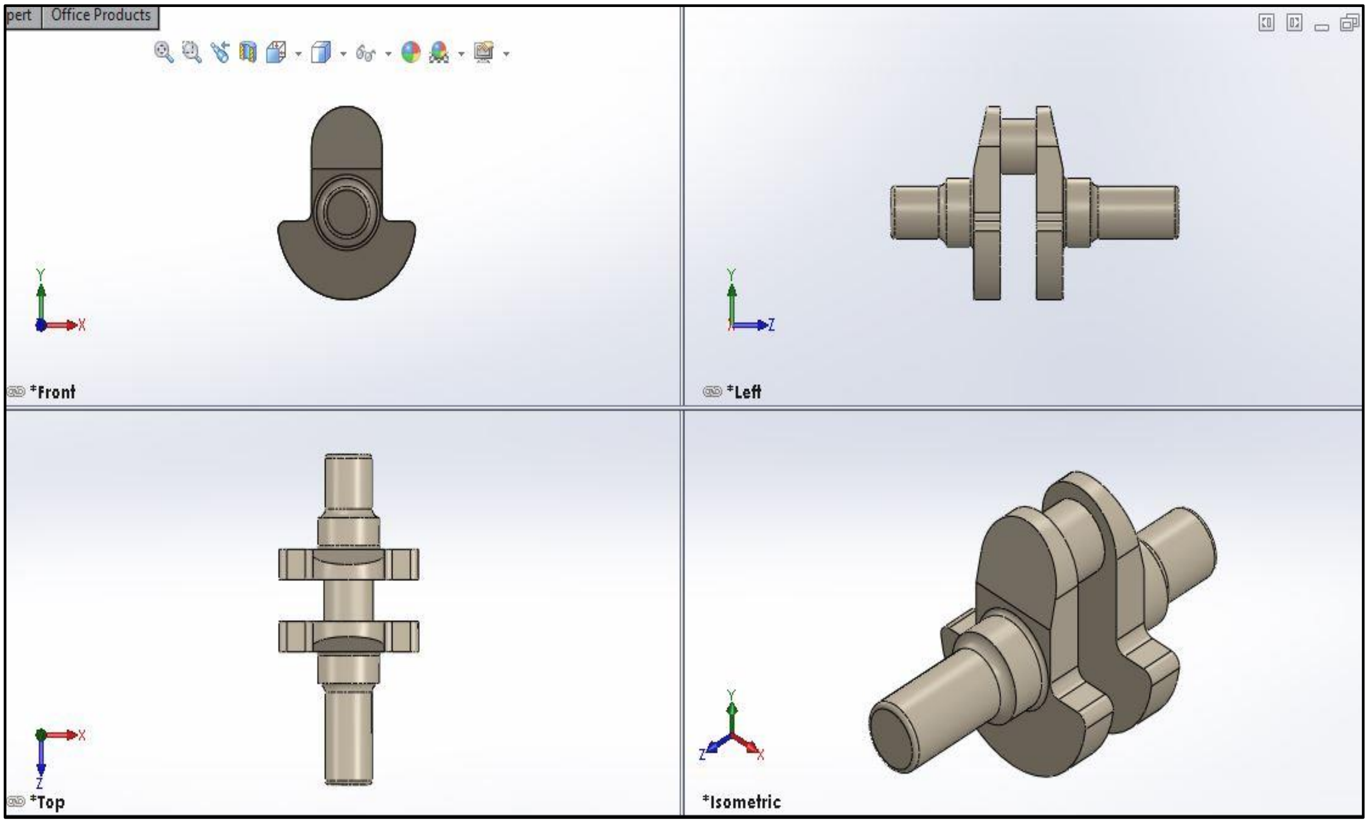

Fig. 2 Various views of the 3-D model of the crankshaft created in solidworks

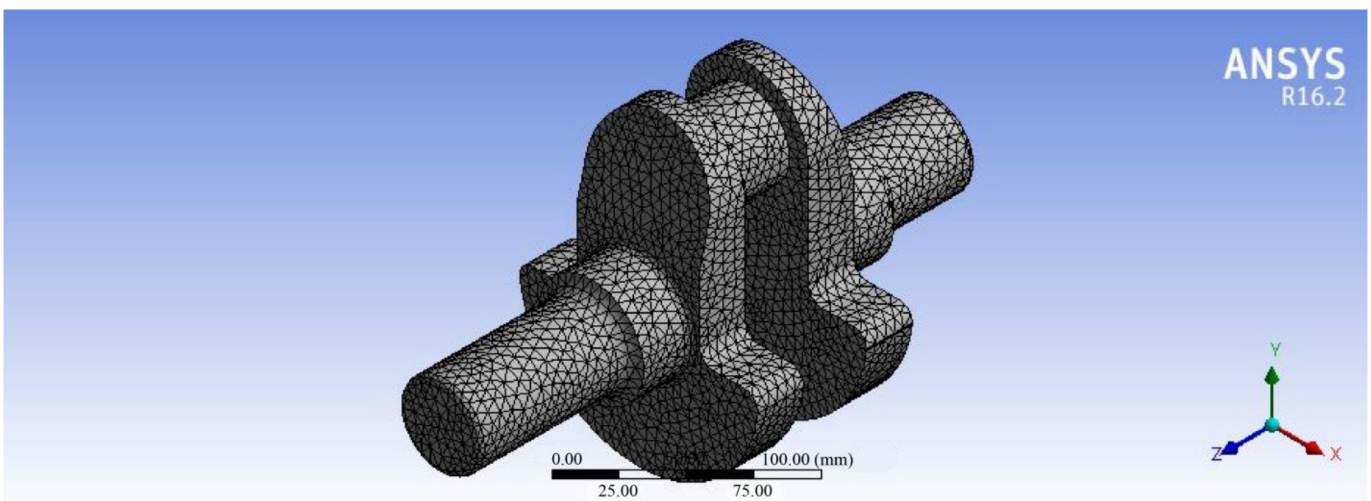

Fig. 3 Mesh model of crankshaft

Table 2. Materials used and their properties

	Young's Modulus	Poisson's Ratio	Density	Ultimate Tensile Strength
Gray Cast Iron	1.1E11 Pa	0.28	7200 kg/m^3	240 MPa
Structural Steel	2E11 Pa	0.3	7850 kg/m^3	460 MPa
Aluminium Alloy	7.1E10 Pa	0.33	2770 kg/m^3	310 MPa
Titanium Alloy	9.6E10 Pa	0.36	4620 kg/m^3	1070 MPa

For conducting FEA, a number of techniques and research tools are used. We are evaluating Ansys 16.2 program in this design. Ansys Inc. is a major manufacturer of finite element analysis applications. It has a user-friendly graphical user interface. The 3-D model developed in Solidworks was subjected to static structural analysis.

To start, the 3-D geometry was generated and imported into ANSYS. A mesh was built on the model after it was imported into ANSYS. Meshing is a step in the engineering simulation process that involves breaking down complex geometries into basic elements that can be used as isolated local approximations of the wider domain[11].

After creating the mesh model, the boundary conditions were applied to the model where the fixed support and the forces calculated before were applied, and then the mesh was solved.

6. Result

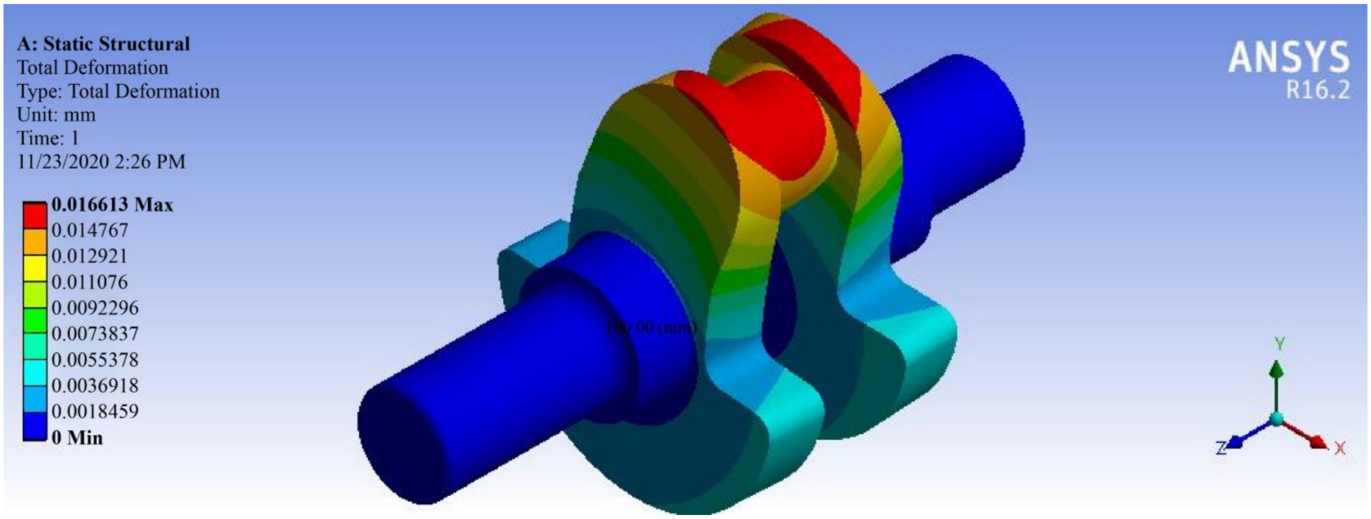

Fig. 4 Total deformation for gray cast iron

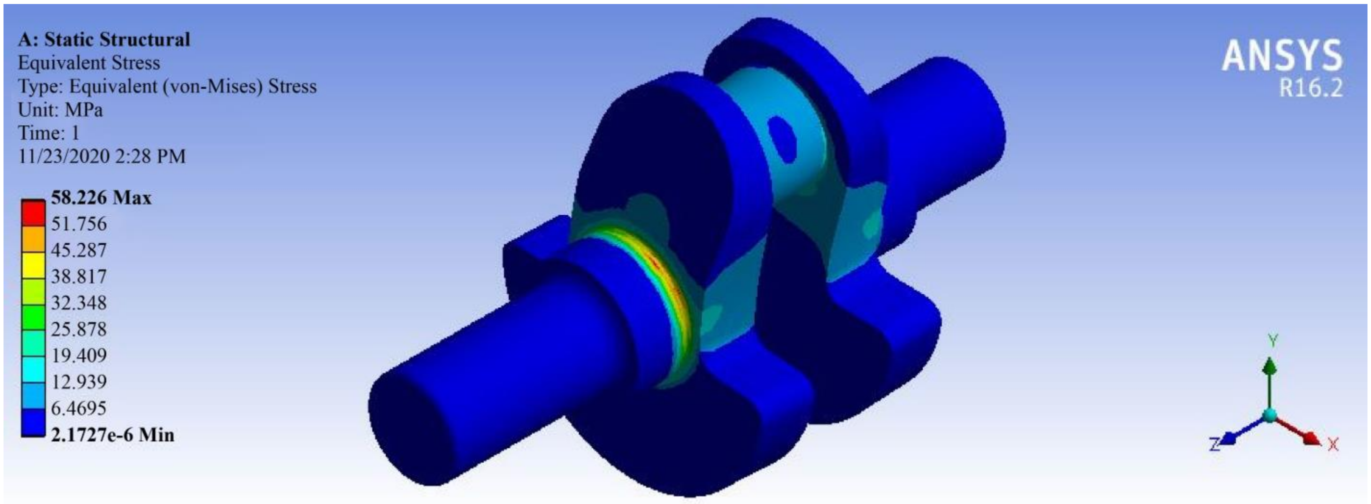

Fig. 5 Equivalent stress for gray cast iron

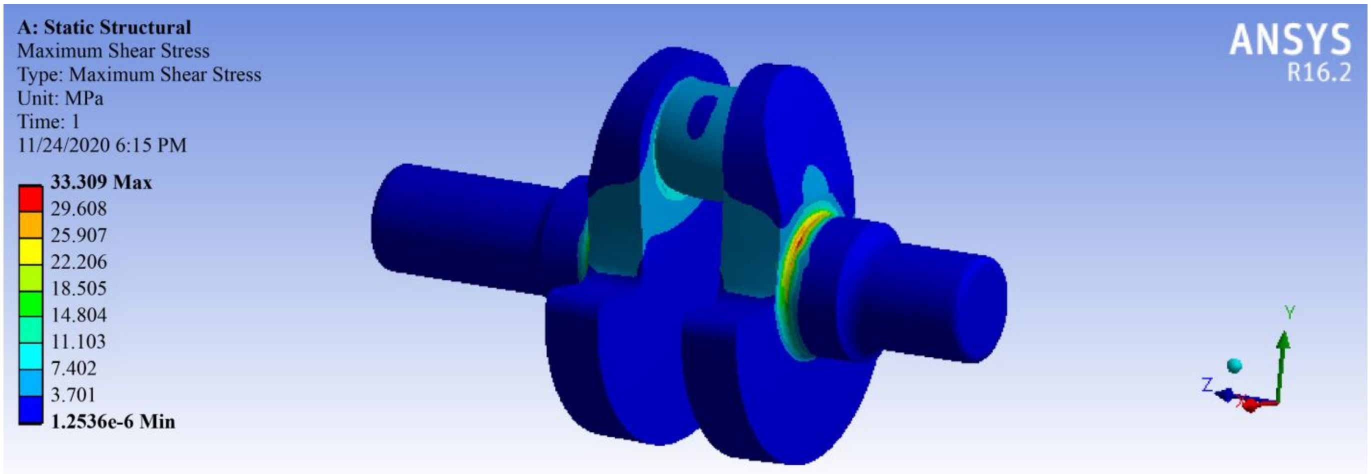

Fig. 6 Maximum shear stress for gray cast iron

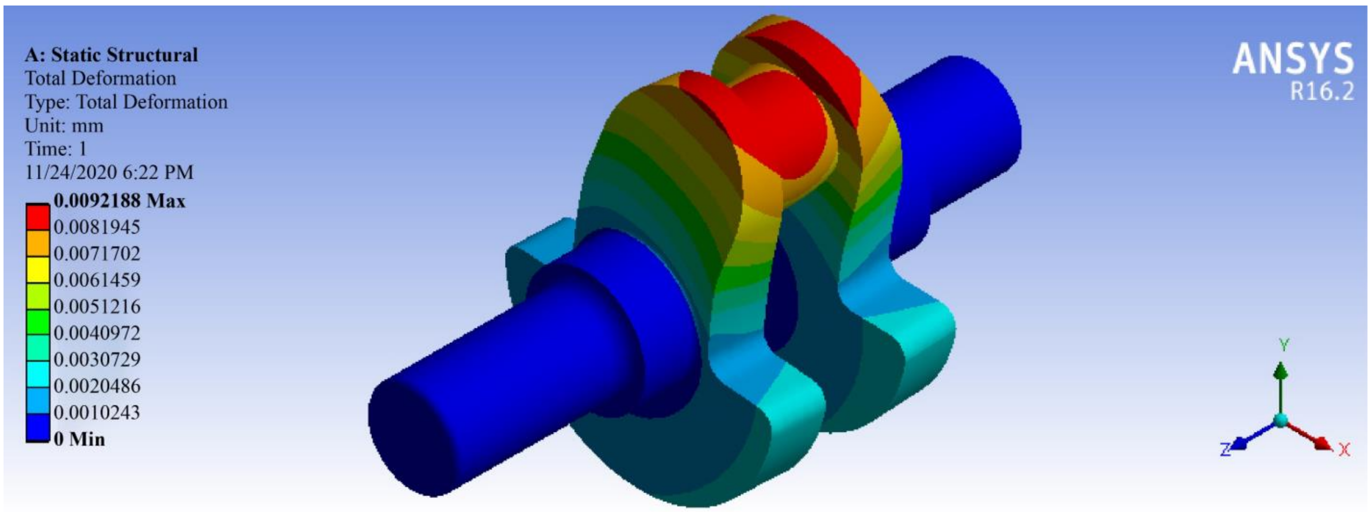

Fig. 7 Total deformation for structural steel

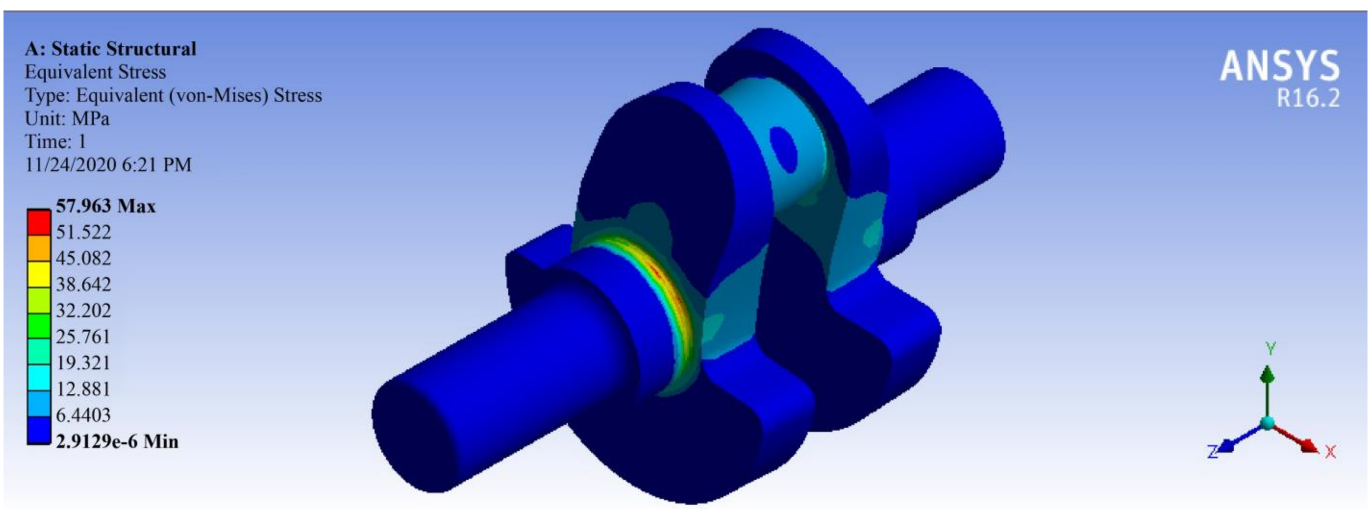

Fig. 8 Equivalent stress for structural steel

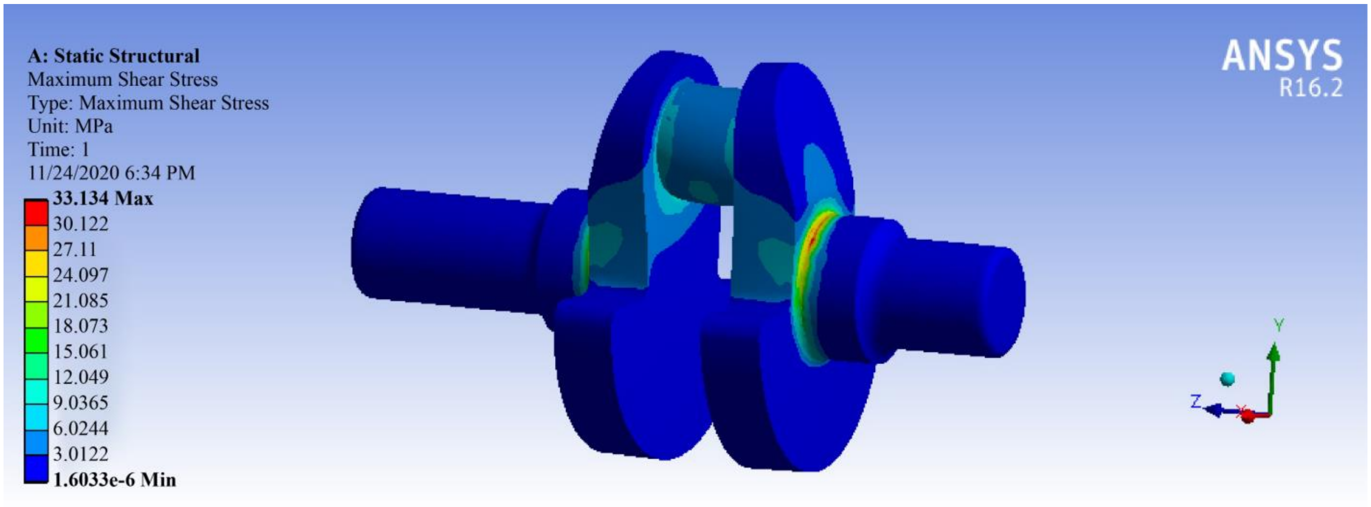

Fig. 9 Maximum shear stress for structural steel

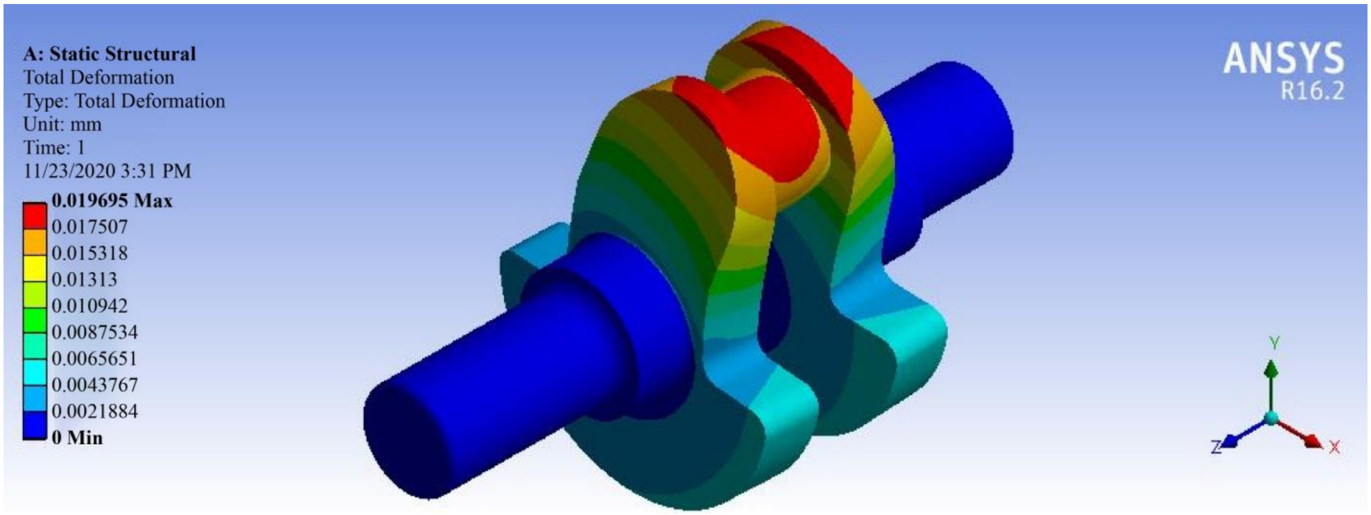

Fig. 10 Total deformation for titanium alloy

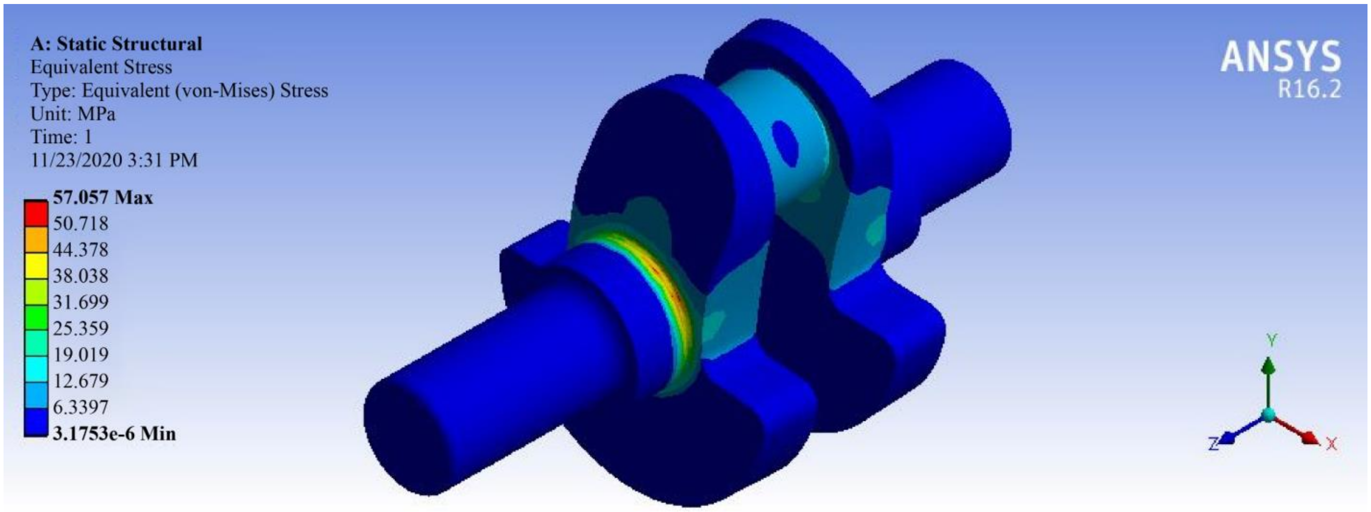

Fig. 11 Equivalent stress for titanium alloy

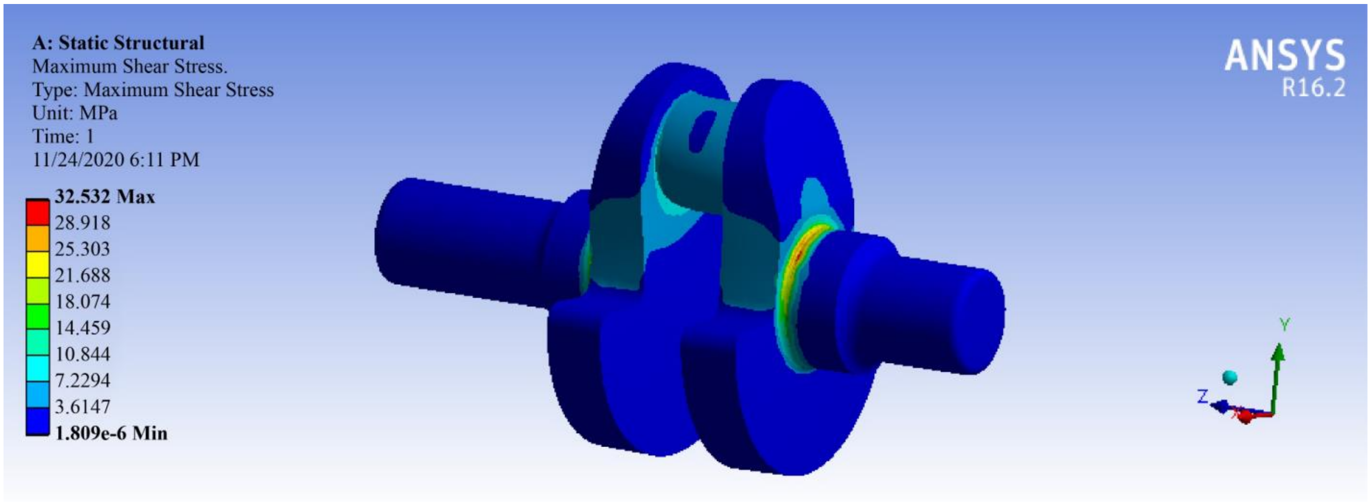

Fig. 12 Maximum shear stress for titanium alloy

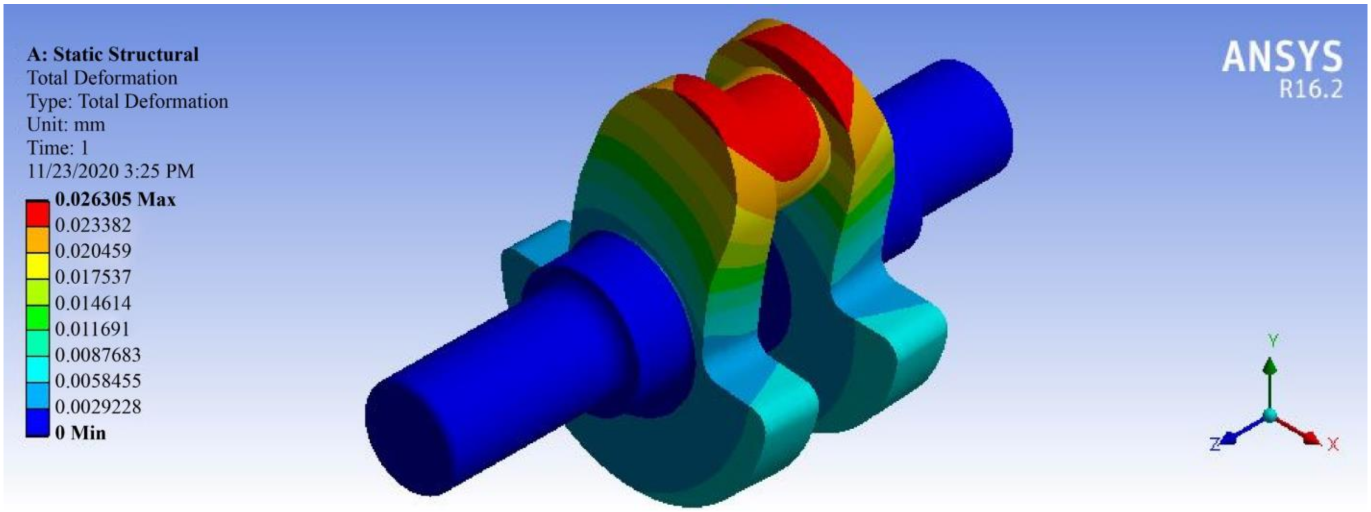

Fig. 13 Total deformation for aluminium alloy

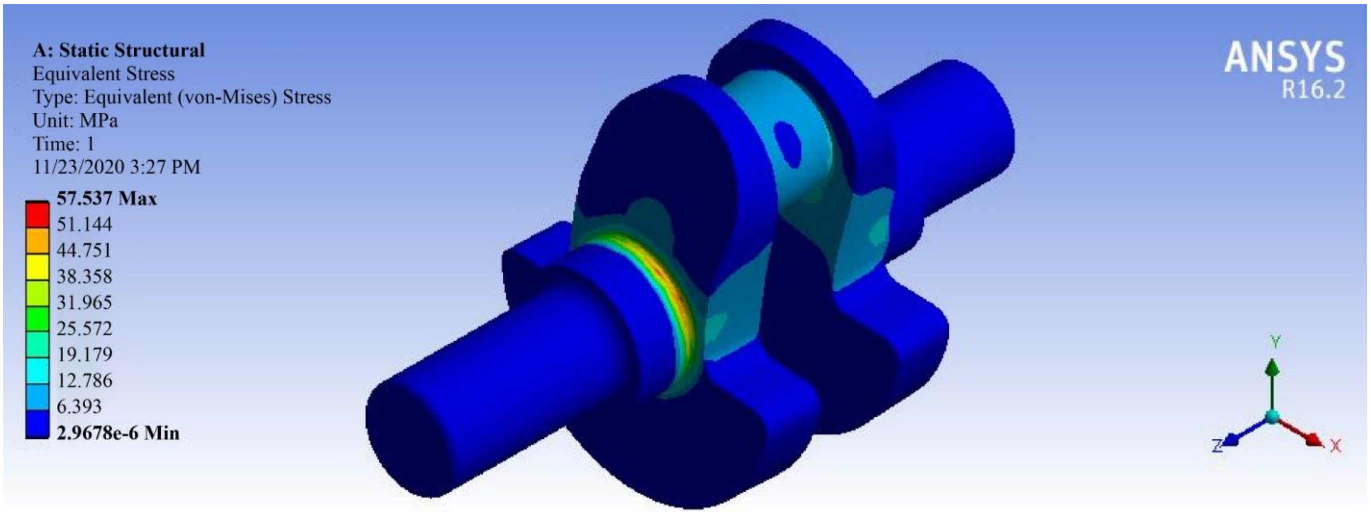

Fig. 14 Equivalent stress for aluminium alloy

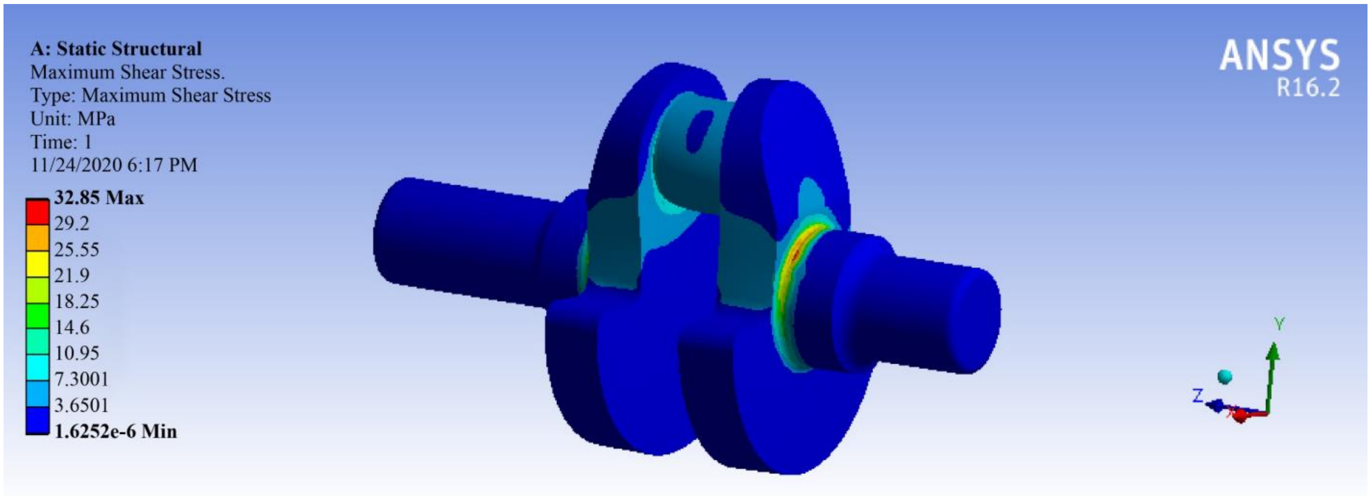

Fig. 15 Maximum shear stress for aluminium alloy

Table 3. Stresses and deformations values from ansys

	Deformation (mm)	Equivalent Stress (MPa)	Max. Shear Stress (MPa)
Gray Cast Iron	0.016613	58.226	33.309
Structural Steel	0.0092188	57.963	33.134
Aluminium Alloy	0.026305	57.537	32.85
Titanium Alloy	0.019695	57.057	32.532

7. Conclusion

In Solidworks, a three-dimensional model of an actual crankshaft design was developed. The three-dimensional model was then imported into ANSYS, where a mesh was generated for the study. The model's material was then chosen, and boundary constraints were added to the mesh model to solve the mesh. The 4 different materials, Grey cast iron, Structural Steel, Titanium Alloy, and Aluminium Alloy, were used for the body of the crankshaft.

For all 4 materials used, it was observed that the maximum deformation is taking place at the center of the crankpin in the crankshaft. In addition, for all four materials tested, the maximum equal stress and maximum shear stress were found in the fillet region of the crankshaft journal and crank cheeks. The minimum deformation value was observed for the Structural steel, whereas the minimum value of equivalent and shear stress was observed for Titanium alloy.

As a result of the findings, it can be inferred that, of the four, structural steel is the best material for the crankpin region, where the most deformation occurs, and titanium alloy is best for crankshaft journals and crank cheeks.

References

[1] Single Cylinder Crankshaft - [Online]. Available: https://www.indiamart.com/prodde tail/single-cylinder-c rankshaft-5868264055.html

[2] K.Durga Prasad, K.V.J.P.Narayana, and N.Kiranmayee, “Design and Stress Analysis of Crankshaft for Single Cylinder 4 Stroke Diesel Engine,” *International Journal of Engineering Research & Technology (IJERT),* vol. 7, no. 11, pp. 154-159, 2018. [CrossRef] [Google Scholar] [Publisher Link]

[3] M.Guru Brahmananda Reddy et al., “Design & Analysis of Crankshaft by Forged Steel & Composite Material,” *International Journal of Engineering Trends and Technology (IJETT),* vol. 46, no. 1, pp. 29-35, 2017. [CrossRef] [Google Scholar] [Publisher Link]

[4] Rincle Garg, and Sunil Baghla, “Finite Element Analysis and Optimization of the Crankshaft,” *International Journal of Engineering and Management Research,* vol. 2, no. 6, pp. 26-31, 2012. [Google Scholar] [Publisher Link]

[5] N.Tracy Jonafark, and Y.V.Mohan Reddy, “Computer Aided Design and Analysis of Crankshaft for Diesel Engine,” *International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)*, vol. 3, no. 4, pp. 39-46, 2013. [Google Scholar] [Publisher Link]

[6] Dhekale Harshada et al., "Design, Analysis & Optimization of Crankshaft Using CAE," *IPASJ International Journal of Mechanical Engineering (IIJME)*, vol. 4, no. 4, pp. 33-38, 2016.[Google Scholar] [Publisher Link]

[7] S.R.Deviyaprasanth, "Evaluation of Fatigue Life of the Crankshaft," *International Journal of Scientific & Engineering Research*, vol. 7, no. 4, 2016. [Publisher Link]

[8] Sujata Satish Shenkar, and Nagraj Biradar, "Design and Static Structural Analysis of Crankshaft," *International Journal of Scientific Engineering and Technology Research (IJSETR),* vol. 4, no. 7, pp. 1393-1398, 2015. [Google Scholar] [Publisher Link]

[9] Amit Solanki, and Jaydeepsinh Dodiya, "Design and Stress Analysis of Crankshaft for Single Cylinder 4-Stroke Diesel Engine," *International Journal For Research in Applied Science and Engineering Technology (IJRASET*), vol. 2, no. 5, pp. 320-324, 2014. [Google Scholar] [Publisher Link]

[10] Jaimin Brahmbhatt, and Abhishek choubey, "Design and Analysis of Crankshaft for Single Cylinder 4 Stroke Diesel Engine," *International Journal of Advanced Engineering Research and Studies,* vol. 1, no. 4, pp. 88-90, 2012. [Google Scholar] [Publisher Link]

[11] Ansys Meshing, [Online]. Available: https://www.ansys.com/products/platform/ansys-meshing

[12] K. Thriveni, and B.JayaChandraiah, "Modeling and Analysis of the Crankshaft Using Ansys Software," *International Journal of Computational Engineering Research*, vol. 3, no. 5, pp. 84-89, 2013. [Google Scholar] [Publisher Link]