Static Analysis of Crankcase for Single Cylinder High Speed Diesel Engine

G.A.Bhosale
Department of Mechanical Engineering
Dean Academic, Yashwantrao Bhonsale polytechnic, sawantwadi

Dr. V.V. Kulkarni
Department of Mechanical Engineering
Dean Academic, Sanjay Ghodawat Institute of Technology, Atigre

Abstract -The single cylinder engines are extensively used in agricultural areas for several purposes such as water pumping, harvesting machines, spraying of pesticides etc. The use of diesel as fuel and having only one cylinder makes them an economic alternative due to their low investment cost and limited fuel consumption. Therefore these engines are important auxiliary agricultural tool for rural areas. Because of excessive usage and overloading in field applications more failures are reported on different engine components. Crankcase is located at the bottom of the cylinder block. It can be molded as a part of a block or bolted on separately. . The failure investigation includes visual inspection, chemical composition analysis, microstructure examination, hardness determination, static analysis as well as dynamic analysis (modal and harmonic analysis). In this paper, failure investigation has been conducted on crankcase of single cylinder high speed diesel engine by static analysis. A static analysis is used to determine the displacements, stresses, strains and forces in structures and components caused by loads that do not induce significant inertia and damping effects. In this static analysis, the crankcase model is created by CATIA 14.5 software. Then, the model created by CATIA is imported to ANSYS software

Keyword - Crankcase, Static Analysis, Ansys Displacement.

I. INTRODUCTION

Finite Element Analysis (FEA) is a computer based numerical technique for calculating the strength and behavior of engineering structures. It can be used to calculate deflection, stress, vibration, buckling behavior and many other phenomena. It also can be used to analyze either small or large scale deflection under loading and applied displacement. It uses numerical technique called the Finite Element Analysis. In finite element method, the actual continuum is represented by the finite elements. These elements are considered to be joined at specific points called nodes or nodal points. As the actual variation of the field variables (displacement, temperature, pressure or velocity) inside the continuum is not known, the variation of the field variable inside the finite is approximated by a simple function.

The approximating functions are also called as interpolation models and are defined in terms of field variables at the nodes. When the equilibrium equations for the whole continuum are known, the unknowns will be the nodal values of field variables.

In this paper finite element analysis is carried out using FEA software ANSYS. CAD model of crankcase of a single cylinder diesel engine as shown in fig 5.11. The primary unknowns in this structural analysis are displacement and other quantities such as strains, stresses, reaction forces are then derived from the nodal displacements.

Flow chart for finite element analysis

Finite element analysis should have the steps as follows:

a) Create 3D CAD model: - Use any of the 3D CAD modeling tools like ProE, Catia, and solid Edge etc. for creating the 3D geometry of the part/assembly of which you want to perform FEA.

b) Clean up the 3D CAD model: - Some features of the 3D CAD geometry may not be that important for the FEA but increase the complexity of meshing drastically; you need to remove those features of the CAD model.

c) Save the 3D CAD geometry in neutral format:- Save the 3D CAD geometry in neutral format like IGES, STEP etc. though some of the FEA packages allow importing the CAD geometry directly from some of the 3D CAD packages.

d) Importing 3D CAD geometry to FEA package: - Start the FEA package and import the CAD geometry into the FEA package.
e) Defining material properties: - You need to tell the FEA package which material you are using for the part. By this process you have to tell modulus of elasticity, poisons ratio and all other necessary properties require for the FEA.
f) Meshing: - Meshing is a critical operation in FEA. In this operation, the CAD geometry is divided into large numbers of small pieces. The small pieces are called mesh.The analysis accuracy and duration depends on the mesh size and orientations. With the increase in mesh size, the finite element analysis speed increase but the accuracy decrease.
g) Defining boundary condition:- You have to tell the FEA package where you want to apply loads and where you want to rest the part/assemble (constraints).
h) Solve: - In this step you tell the FEA package to solve the problem for the defined material properties, boundary conditions and mesh size.
i) Post processing: - You view the results of the solution in this step. The result can be viewed in various formats: graph, value, animation etc.

Design calculation for crankcase.

The configuration of a single cylinder four stroke diesel engine for this crankcase is tabulated in Table 5.1.

Table 5.1- Specification of a single cylinder diesel engine. (Rocket Engine)

MODEL	PPD8D
CAPACITY	345 CC
BORE	87.5 MM
STROKE	80 MM
COMPRESSION RATION	16.7 : 1
RATED POWER	8 HP @2600 RPM
MAX. GAS PRESSURE	80 Kg/cm^2 (78.43 BAR)
MEAN EFFECTIVE PRESSURE	8 Kg/cm^2.

Reaction at bearing 1 and 2 in crankcase hole where crankshaft support.

Design of crankshaft when the crank is at an angle of maximum twisting Moment.

Force on the Piston F p = Area of the bore x Max. Combustion pressure.

$F_p = \pi/4 \times (D)2 \times P_{max}.$

$F_p = \pi/4 \times (87.5)2 \times 7.843$ N.

$F_p = 47.14$ KN.

In order to find the thrust in the connecting rod (FQ), we should first find out the angle of inclination of the connecting rod with the line of stroke (i.e. angle Ø).

We know that,

$\sin Ø = \sin \theta / L/R.$

$\sin Ø = \sin 350 / 4.$

$Ø = 8.24°.$

Which implies Ø = 8.24°

We know that thrust in the connecting rod

$FQ = F_p / \cos Ø$

From this we have, Thrust on the connecting rod

$FQ = 47.14 / \cos (8.24)0.$

$= 47.14 / 0.989.$

FQ = 47.66 KN

Thrust on the crankshaft can be split into Tangential component and the Radial component. Tangential force on the crank shaft,

FT = FQ sin (θ + Ø).
FT = 47.66 sin (43.23)0.
FT = 32.64 KN.

Radial force on the crank shaft,

FR =FQ cos (θ + Ø).
FR = 47.66 cos (43.24)0
FR =34.71 KN.

Reactions at bearings (1 & 2) due to tangential force is given by,

HT1=HT2 = FT × b1 / b ………………… (Since b1=b2=b/2).
= FT × b / 2b.
= FT / 2.
= 17.32 KN.

Similarly, Reactions at bearings (1 & 2) due to radial force is given by,

HR1 = HR2 = FR × b1 / b ………………… (Since b1=b2=b/2)
= FR × b / 2b.
= FR / 2.
= 17.35 KN.

For this analysis , factor of safety is taken as 1.5,and apply radial force 26 KN.

HR1 = HR2= 17.35 × 1.5 = 26.02 KN.

Procedure of static Analysis

Static analysis of existing model of single cylinder crankcase

a) First, I have Prepared Assembly in CATIA for crankcase of single cylinder diesel engine and Save as this part as IGES for Exporting into ANSYS Workbench Environment as shown in figure 5.11. Import IGES Model in ANSYS Workbench Simulation Module.

b) Apply Material properties for Crankcase (FG260 Gray cast iron).
Material Details
Material Type: - Gray cast iron
Designation: - FG260
Modulus of Elasticity:-128520N/mm^2 (1.28 x 10^5 N/mm^2).
Ultimate tensile strength (MPa):- 850
Elongation (%):-13
Poisson ratio:-0.26

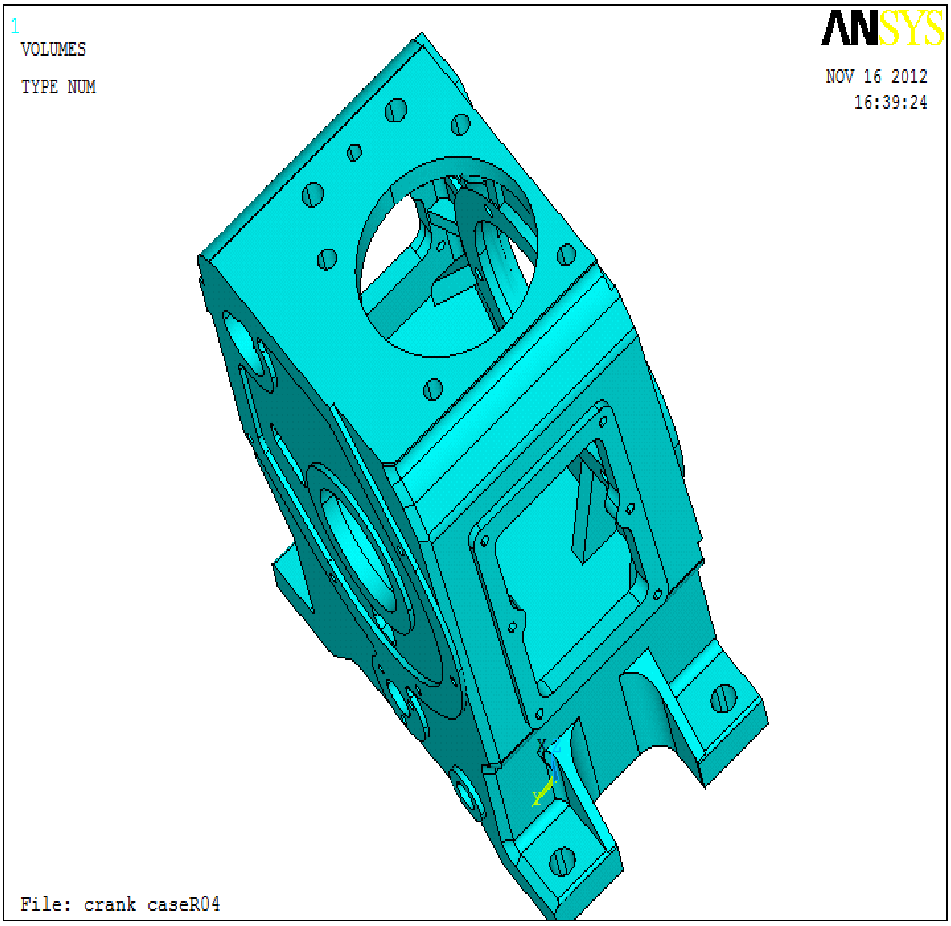

Figure.5.11. Crankcase of single cylinder diesel engine in ANSYS.

c) Mesh the existing model of Crankcase of single cylinder four stroke diesel engine as shown in figure 5.12.

Mesh Statics:

Type of Element: Tetrahedrons (10 Noded)

Number of Nodes: 469434.

Number of Elements: 270070.

Statistics	
Nodes	469434
Elements	270070

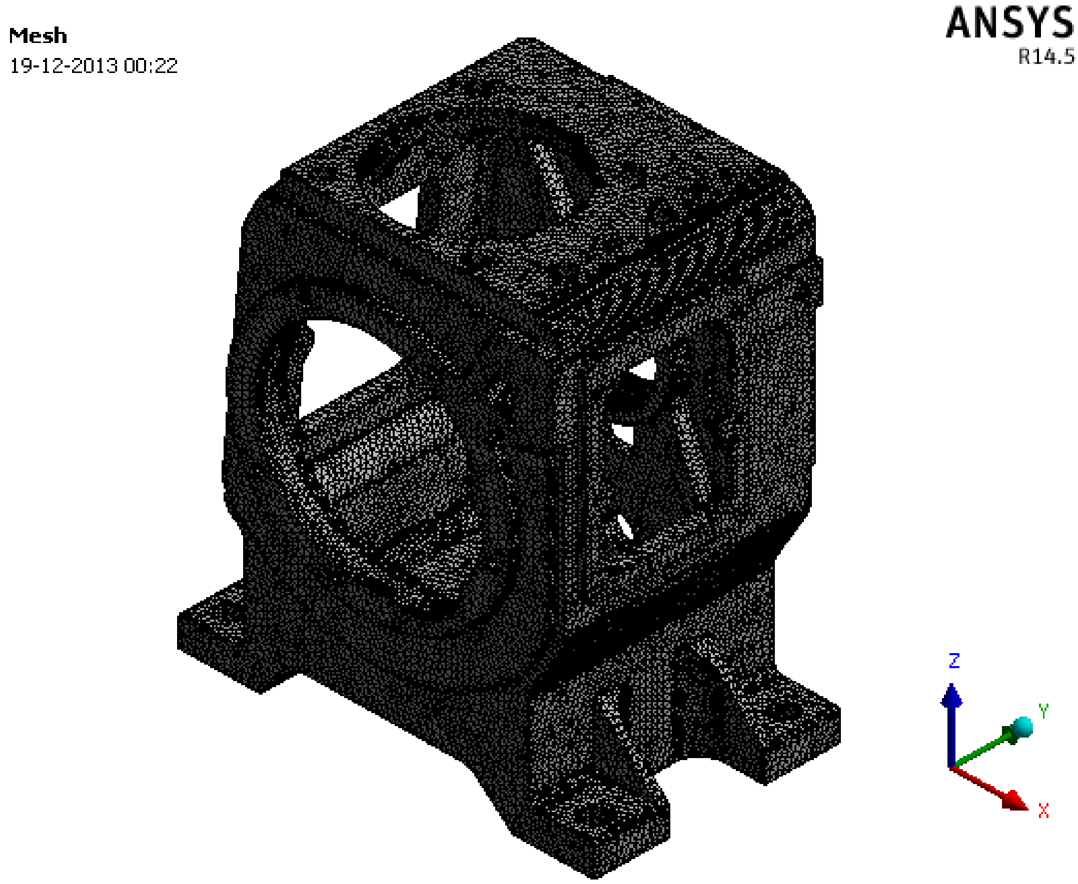

Figure.5.12: Meshed existing Model of Crankcase of a single cylinder diesel engine.

d) Define boundary condition for Analysis.

Boundary conditions play an important role in finite element analysis calculation here; I have taken radial remote displacements for bearing supports are fixed.

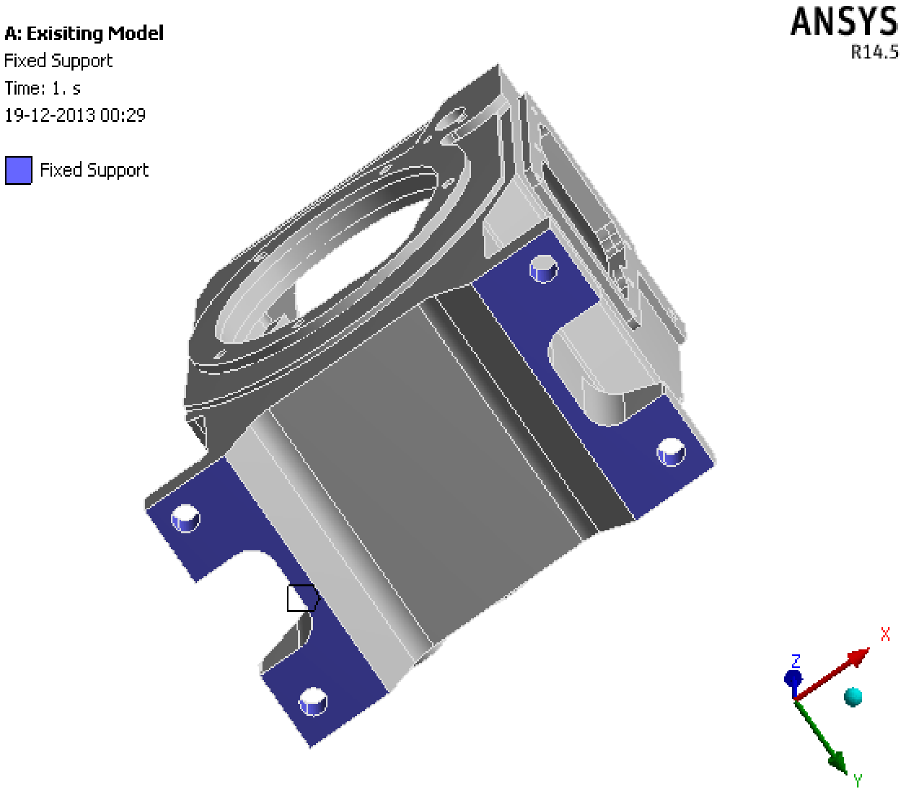

Figure.5.13: Fixed support on crankcase of single cylinder diesel engine.

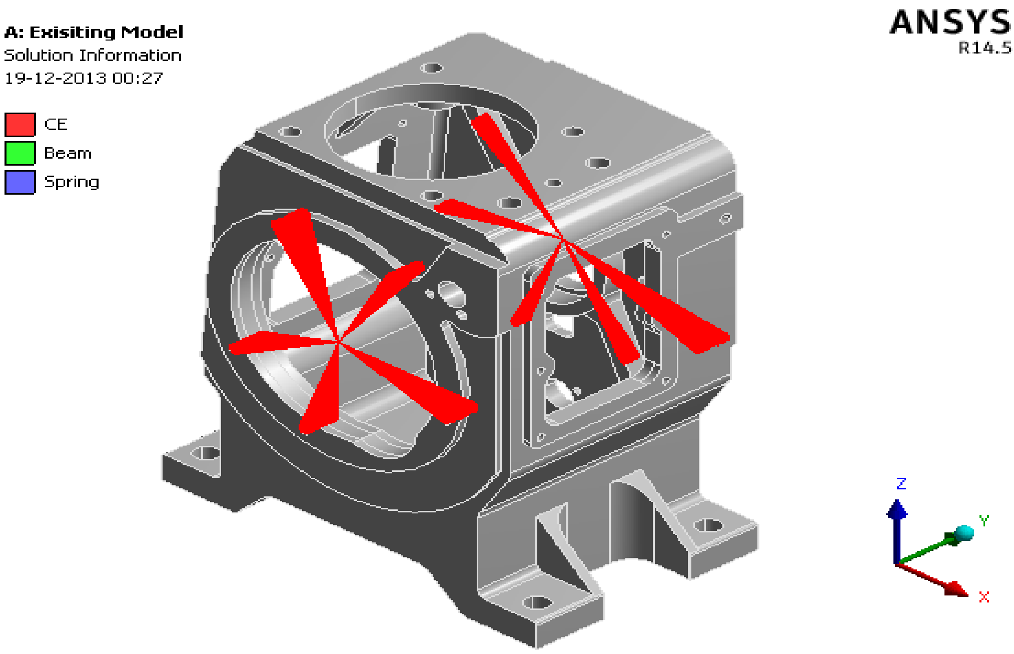

Figure.5.14: Apply radial force on crankcase of single cylinder diesel engine.

d) Define type of Analysis.

Type of Analysis:-Static Structural of crankcase of single cylinder diesel engine as shown in figure 5.13

e) Run the Analysis.

f) Get the Results.

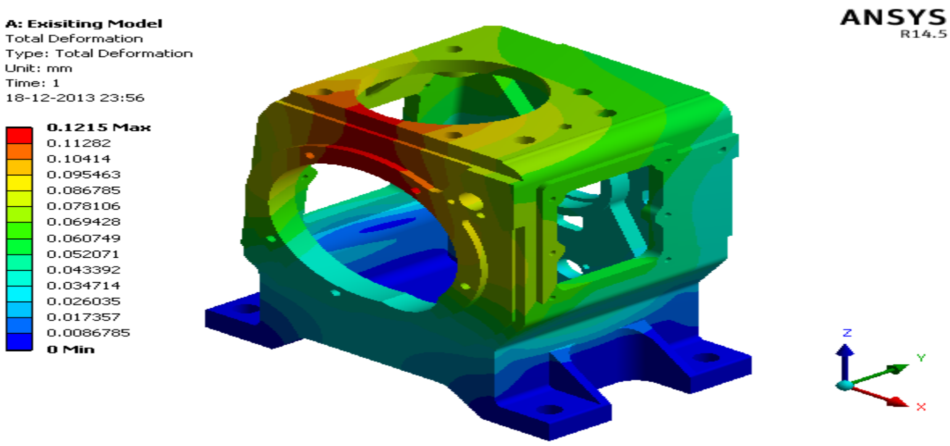

Fig. 5.15: Total deformation of crankcase of single cylinder diesel engine.

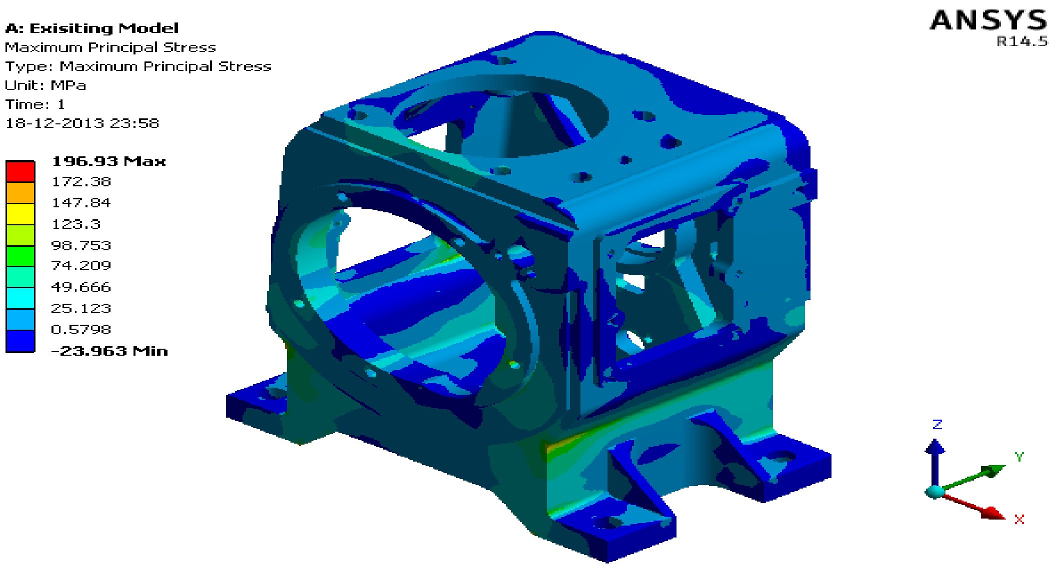

Figure.5.16: Maximum principal stress of crankcase of a single cylinder diesel engine.

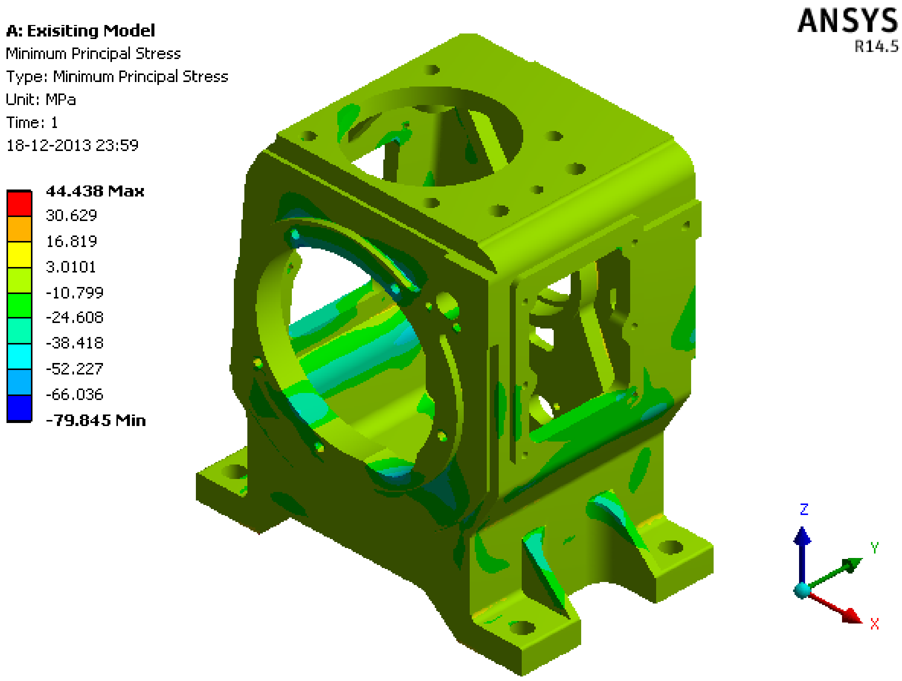

Fig.5.17: Minimum principal stress of crankcase of a single cylinder diesel engine.

II. CONCLUSION

In this static analysis, the crankcase model is created by CATIA 14.5 software. Then, the model created by CATIA is imported to ANSYS software.

Result Table 5.2:-Result Table of static analysis.

SR.NO	PARAMETERS	STATIC ANALYSIS
1	Total Deformation (In mm).	0.1215
2	Max.principal Stress (Mpa).	196.93

Maximum principal stress theory is useful for brittle material. Max principal stress theory state that failure will occurs when maximum principal stress developed in the body exceeds uniaxial ultimate tensile/compressive strength(or yield strength) of the material.

From static analysis, it can be clearly seen that existing model not safe considering max. Principal stress is more than the maximum allowable yield stress of cast iron. (154 MPa). It is clear that existing model has sharp corners at ribs and pockets thatswhy stress concentresstion is high. So it is required to eliminate sharp corners by appropriate fillets.

REFERENCES

[1] Element Method and Application in Engineering Using ANSYS by Erdogan Madenci, Ibrahim Guven.

[2] R. K. Pandey, "Failure of diesel-engine crankshafts", Engineering Failure Analysis, 10(2003) 165-175.

[3] J. L. A. Ferreira, A. B. de S. oliveria, A. B. Souza, V.G. Carloni, D. A. Leite, " Structural Practical engineering failure analysis by Hani M. Tawancy, Anwar UL-Hamid.

[4] The Finite integrity analysis of the main bearing cap screws of the turbo-diesel engine crankshaft", Engineering Failure Analysis,12(2005) 273-286.

[5] Ryuichiro Ebara, katsuaki Kubota, "Failure analysis of hot forging dies for automotive components", Engineering Failure Analysis, 14(2007) 725-730.

[6] S. K. Bhaumik, M. Sujata, M.A. Venkataswamy, "Fatigue failure of aircraft components", Engineering Failure Analysis, 15(2008) 675-694.

[7] F. Jimenez Espadafor, J. Becerra Villanueva, M. Torres Garcia, "Analysis of a diesel generator crankshaft failure", Engineering Failure Analysis, 16(2009) 2333-2341.

[8] M. Fonte, M. de Freitas, "Marine main engine crankshaft failure analysis: A case study", Engineering Failure Analysis, 16(2009) 1940-1947.