Kijang LGX camshaft’s failure analysis using the finite element method approach

Riduwan Prasetya *, Andoko
Mechanical Engineering Department, State University of Malang
Semarang 5, Lowokwaru, Malang 65145, Indonesia
*Email: ridwanprasetya1@gmail.com

Abstract. The camshaft on the Kijang LGX was broken after 14 years of use. This study aims to determine causes of camshaft fracture using finite element approach. The software used is Solidworks and ANSYS 17.0 for FEM simulation. The results to be obtained from the simulation results are stress, strain, and deformation which are then analyzed. The results showed that the greatest stress occurred in the fillet section where the area with highest stress concentration was 168.93 MPa. This value is smaller than the compressive strength so that the applied stress is within the limits of the strength of the material. Maximum stress location is 7 mm away from the fracture site (53.17 MPa), therefore the stress is not the main cause of the camshaft fracture and it can be assumed that there is a defect. The maximum strain is located at the same location as the maximum stress which is 0.0015361 mm/mm. The maximum deformation value is 0.016741 which is far from the fracture (located at the end of the cam). This value is included in the elastic deformation of the material so that the load applied to the camshaft does not change shape / does not lead to plastic deformation that triggers the appearance of cracks.

Keywords: camshaft, failure analysis, finite element method

1. Introduction
The camshaft is a very important part of an engine. Its function is to repeatedly open and close the valve so that the suction, compression, power, and exhaust steps in the engine combustion cycle can occur [1]. The high work demands at the same time play an important role in that the camshaft must be made with minimal or no defects [2].

However, minimized defects do not prevent objects from failure [3]. It is these failures that have occurred which are then identified as material for evaluation so that similar failures do not occur again [4]. The process of identifying a failure can be approached using a failure analysis consisting of several methods, one of which is the finite element method [5,6]. The finite element method was developed in 1950 which gave birth to various finite element method software [7,8]. FEM has the advantage that it is cheaper both in terms of time and cost when compared to experimental testing (actual testing) but can describe actual events [9,10].

This research will discuss the camshaft failure that occurs in a minibus 7 passengers using the finite element method. The goal is to see the distribution of stress, strain, and deformation, which
is then analyzed whether these parameters are the cause of the camshaft failure. The camshaft of this car failed in the form of a fracture in the journal bearing between cylinder 1 and cylinder 2 (Figure 1). The camshaft broke occurred after the car was used for 14 years and there was no damage to other parts.

Previous research regarding camshaft failure was conducted by L. Ping et. al. in 2009 on the Fukang car [11]. The cause of camshaft fracture is that it tends to be too much chilled in the transition area which has the potential to affect the formation of cracks, propagate cracks, and reduce the loading capacity of the camshaft [11]. Davale and Muttagi [12] examined the failure of a camshaft made of nodular cast iron. The camshaft fracture occurs suddenly at a location very close to the journal bearing. The main cause of camshaft fracture is casting defects. Wanjari and Parshiwankar [13] researched camshafts made from Chilled Iron casting and found that factors that influence camshaft failure are material properties, engine speed, engine load, and lubricant properties.

![Figure 1. Camshaft with fracture](image)

### 2. Method and material

The camshaft model was drafted using Solidworks 2012 software which is shown in Figure 2. The camshaft dimensions were obtained from direct measurements of the broken camshaft which was confirmed by manual. The FEM software used in this study is ANSYS 17.0. The mesh size is 2 mm with 1044856 nodes and 661127 elements (Figure 3). The loads acting on the camshaft include the force from the spring, moment and friction. The spring force is the greatest force received by the camshaft at 7149 N, while the friction is neglected. The results to be obtained from the simulation are stress, strain, and deformation which are then analyzed whether these parameters are the cause of camshaft failure. The material used on the camshaft is grey cast iron grade G2500 (SAE Grade) / ASTM 48 Class 25 which has mechanical properties shown in Table 1 [14].

![Figure 2. Camshaft modelling](image)
3. Result and discussion

The stress distribution on camshaft is shown in Figure 4, where the smallest stress is $7.49 \times 10^9$ MPa and the maximum stress is 168.93 MPa. Figure 4 shows the location of the maximum stress on the camshaft, namely on the fillet camshaft followed by the stress in the fracture area.

Stress is defined as the load borne by a material. The location of the maximum stress on the fillet camshaft is an area with a high stress concentration due to changes in geometry so it is natural that this area is the location of the maximum stress [15]. The stress concentration will increase the local stress to be greater than the applied stress. If the local stress exceeds the yield stress, local plate deformation will appear. Plastic deformation is a parameter that controls crack initiation throughout the fatigue process [16].

![Figure 3. Meshing on the camshaft](image3)

![Figure 4. Stress distribution on the camshaft](image4)
In the simulation results, the highest stress in the stress concentration area is 168.93 MPa, which is smaller than the compressive strength value. Whereas in the fracture section it is 53.17 MPa. So that plastic deformation will not form in this area. This shows that when a load is applied to the camshaft, the stress occurs within safe limits of material strength. However, plastic deformation at the micro scale may occur due to stress concentrations at the micro scale [17]. So it is necessary to observe further in knowing the local plastic deformation of the broken part in the microstructure.

The thing that needs to be considered further is the correlation between the fracture location and the maximum stress that occurs. The maximum stress (fillet section) is 7 mm from the fracture location. This indicates that the stress is not the main cause of the camshaft fracture and it can be assumed that there is another defect causing the camshaft to break (which needs further investigation).
The strain distribution on the simulated camshaft is shown in Figure 6. The maximum strain value is shown further in Figure 7 at 0.0015361 mm/mm along with the strain value in the fracture area of 0.00049073 mm/mm. The location of the highest strain is at the fillet camshaft which has the highest stress concentration.

Strain is one of the parameters that contributes to the development of a crack [18]. At the macro scale the magnitude of the strain rate will affect the development of fatigue cracks, the fatigue crack will not increase if the stress value decreases [19]. Whereas at the micro scale, what affects the crack growth is the strain energy which drives the force to propagate the crack [20].
The deformation distribution when a load is applied to the camshaft is shown in Figure 8. The deformation range is from 0 mm to 0.016741 mm. In the fracture section, the total deformation is 0.0030866 mm shown in Figure 9. Deformation is a value related to the change in shape that occurs in a material due to the load being applied. The highest deformation occurs in the area subjected to the maximum force, namely at the end of the cam (away from the fracture). In the area with the highest stress the deformation magnitude is 18% lower which enters the elastic deformation region. The elastic deformation will return the material to its original shape when the load is removed and is inversely proportional to the plastic deformation [21]. So that the load applied to the camshaft does not lead to plastic deformation that causes cracks to appear.

4. Conclusion
The maximum stress on the camshaft lies in the fillet section (highest stress concentration) of 168.93 MPa. This value is smaller than the compressive strength value so that the applied stress is within the safe limits of the material strength. However, plastic deformation at the micro scale may occur due to stress concentrations at the micro scale. Therefore it is necessary to observe further in knowing local plastic deformation in the broken part in the micro structure. The location of the highest stress is 7 mm from the fracture site (53.17 MPa), thus stress is not the main cause of the camshaft fracture and it can be assumed that there is a defect. The maximum tensile value of 0.0015361 mm/mm lies in the fillet section. Strain contributes to the development of a crack. The maximum deformation value lies on the part of the camshaft which is subjected to a force of 0.016741 mm (far from the fault). This value is included in the elastic deformation of the material, so that the load applied to the camshaft does not lead to plastic deformation that causes cracks to appear.

5. References
[1] X. lei Xu, Z. wei Yu, and S. mei Mao, “Effect of extra-large compound calcium-aluminoosilicate inclusions on cracking of camshafts,” Eng. Fail. Anal., vol. 110, no. June 2019, p. 104408, 2020.
[2] Z. Yu and X. Xu, “Failure investigation of a truck diesel engine gear train consisting of crankshaft and camshaft gears,” Eng. Fail. Anal., vol. 17, no. 2, pp. 537–545, 2010.
[3] A. Kurniawan and Andoko, “Stress and Crack Simulation on Axle Housing Mitsubishi L300 Pickup Car using Finite Element Method,” IOP Conf. Ser. Mater. Sci. Eng., vol. 494, no. 1, pp.
1–7, 2019.

[4] A. Andoko, P. Paryono, R. Prasetya, R. P. Jeadi, P. Kurniawan, and D. R. Pradica, “Simulation of the effect of energy absorption on crashbox with full crash initiator and without crash initiator,” in AIP Conference Proceedings, 2020, vol. 2262.

[5] A. Andoko et al., “Simulation of winglet with bend angles of 45, 60 and 90 degree,” in AIP Conference Proceedings, 2020, p. 040014.

[6] A. Andoko et al., “Failure analysis of connecting rod,” in AIP Conference Proceedings, 2020, p. 050013.

[7] A. Andoko et al., “The influence of crash initiator placement towards the application of energy crash box due to impact load,” in AIP Conference Proceedings, 2020, p. 040016.

[8] Andoko and P. Puspitasari, “Characteristics of leaf spring strength of material 65si7 and material C17000 using finite element method,” AIP Conf. Proc., vol. 1778, no. October, 2016.

[9] A. Andoko et al., “Simulation on CNC 5 axis milling spindle bolt using finite element method,” in AIP Conference Proceedings, 2020, vol. 2262.

[10] Andoko and W. R. Amin, “Investigation of Stress, Deformation, and Cracks in the Brakes of Car Using Finite Element Method,” IOP Conf. Ser. Mater. Sci. Eng., vol. 515, no. 1, 2019.

[11] L. Ping, L. Fengjun, C. Anke, and W. Bokang, “Fracture analysis of chilled cast iron camshaft,” vol. 6, no. 2, pp. 104–108, 2009.

[12] A. S. Davale and V. R. Muttagi, “Study of Modeling and Fracture Analysis of Camshaft ’ A Review .,” vol. 2, no. 6, pp. 835–842, 2012.

[13] R. V Wanjari and T. C. Parshiwankar, “Failure of Camshaft,” no. 6, pp. 248–250, 2013.

[14] ASTM, “Standard Specification for Automotive Gray Iron Castings,” vol. 03, no. Reapproved, pp. 1–5, 2000.

[15] A. Andoko and P. Puspitasari, “The Fatigue Crack Growth Rate Due to Single-Step Austempered Heat Treatment in Nodular Cast Iron,” MATEC Web Conf., vol. 97, p. 01028, Feb. 2017.

[16] S.-T. Tu and X.-C. Zhang, “Fatigue Crack Initiation Mechanisms,” Ref. Modul. Mater. Sci. Mater. Eng., no. December, pp. 0–23, 2016.

[17] A. Andoko et al., “Failure analysis on the connecting rod by finite element method,” in AIP Conference Proceedings, 2020, vol. 2262.

[18] A. Andoko, “The effects of strain amplitude and low cycle fatigue (LCF) behavior on nodular cast iron with two-step austempering (TSA) process,” Mater. Res. Express, vol. 6, no. 9, p. 095705, Aug. 2019.

[19] S. A. Brauer et al., “Strain Rate and Stress-State Dependence of Gray Cast Iron,” J. Eng. Mater. Technol. Trans. ASME, vol. 139, no. 2, pp. 1–11, 2017.

[20] M. Taniguchi and Y. Kariya, “Effect of strain rate and temperature on micro fatigue crack propagation of Bi-Sn eutectic alloy,” Mater. Trans., vol. 57, no. 6, pp. 853–859, 2016.

[21] M. A. Al-nawafleh, “Fatigue Life Improvement of Spheroidal and Compacted Graphite Cast Iron,” no. January, 2018.