Failure Analysis of Vise Jaw Holders

Essam A. Al-Bahkali[1], and Adel T. Abbas

Department of Mechanical Engineering, King Saud University

P.O. 800, Riyadh 11421, Saudi Arabia

[1]Email: ebahkali@ksu.edu.sa

Abstract: Sometime vise jaw holders of the hacksaw are broken due to accidental heavy loads or machine misusing. These broken part are the stationary and movable vise jaw holders and the connecter power screw between them. With the aim of investigating the failure of these components, a three dimensional finite element model for stress analysis has been performed. It starts by identifying the broken parts of the hacksaw machine. In addition, the material of the broken parts is identified, the CAD model is built, and the hacksaw mechanism is analyzed to get the accurate applied loads on the broken parts. After analyzing the model using Abaqus CAE software, the results show that the location of the high stresses are identical with the original broken parts. Furthermore, the power screw was subjected to a high load which let the power screw to be deformed. Also, the stationary vise jaw holder was broken by impact. This is because, it is not touch by the power screw until the movable vise jaw holder is broken. A conclusion of the failure analysis is drawing and a suggesting to improving the design of the broken parts are suggested.

1. Introduction

Dowling (1999) said that the designers of machines or structures must achieve acceptable levels of performance, while at the same time assure the part is safe and durable. Therefore, it is necessary to avoid excess deformation, such as bending, twisting, or stretching, of the machine's components. In addition, cracking in components must be avoided entirely to prevent the crack from progressing to the point of complete fracture (Leonel, 2011).

To avoid structural failure, the stress in the component must not exceed the strength of the material, where the strength is simply the stress that causes a deformation or fracture failure.

Failures occurs in mechanical structure due to various reasons. Boyer et. al. (1975) showed that failures could occur due to mechanisms and environmental factors. He also suggested that failure analysis of a metal structure requires identifying of the type of failure mode. The failure mode is classified as either deformation or fracture. The process of identifying a failure mode is complicated because different techniques can be used to determine the actual cause of failure (Layer, 2002, Irisarri, 2010).

Hacksaw machine is a machine tool designed to cut material to a desired length or a certain shape. It draws a blade with cutting teeth through a workpiece and is faster and easier than a hand hacksaw. The hacksaw is used primarily to produce an accurate cut on the workpiece.

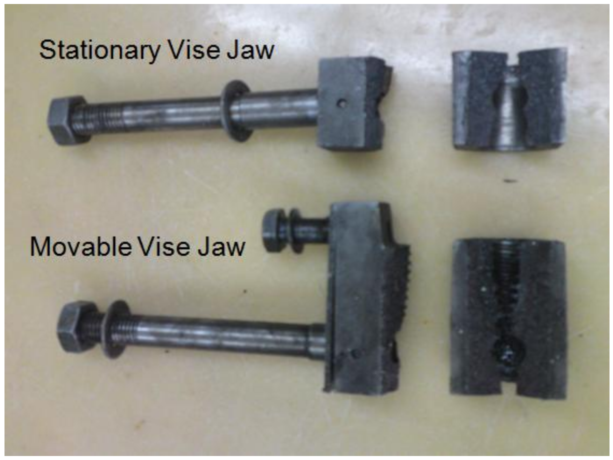
Reciprocated hacksaw machine is a repetitive up-and-down or back-and-forth motion. This motion is found in a wide range of mechanisms, including reciprocating engines and pumps. It is used for square or angle cutting of workpiece. Figure 1 shows a typical type of reciprocated hacksaw machine.

Figure 1. A reciprocated hacksaw machine.

The main parts of the hacksaw machine are the base, vise, and frame. The size of a power hacksaw is usually determined by the largest piece of metal that can be held in the vise and sawed.

Occasionally, the vise jaw holders of the hacksaw break due to heavy loads as shown in Figure 2. These broken parts are the stationary and movable vise jaw holders and the connecter power screw between them. During the investigation, all available data must collected, examined, and evaluated to determine the sequence of incidents that led to the failure. The immediate objective is to find the root causes of the failure to determine the compensation for the induced damage, even if the long-term objective is to prevent similar failures in new recommendations. Typically, these failure analyses involve obtaining samples for material analysis and mechanical tests.

In this paper, a failure analysis is carried out on vise jaw holders and power screw to identify the failure mode and to provide recommendations to resolve the problem. A three-dimensional finite element model for stress analysis was performed to provide a more realistic and clear explanation of the failure mechanisms.



(a) Broken stationary and movable vise jaw holders

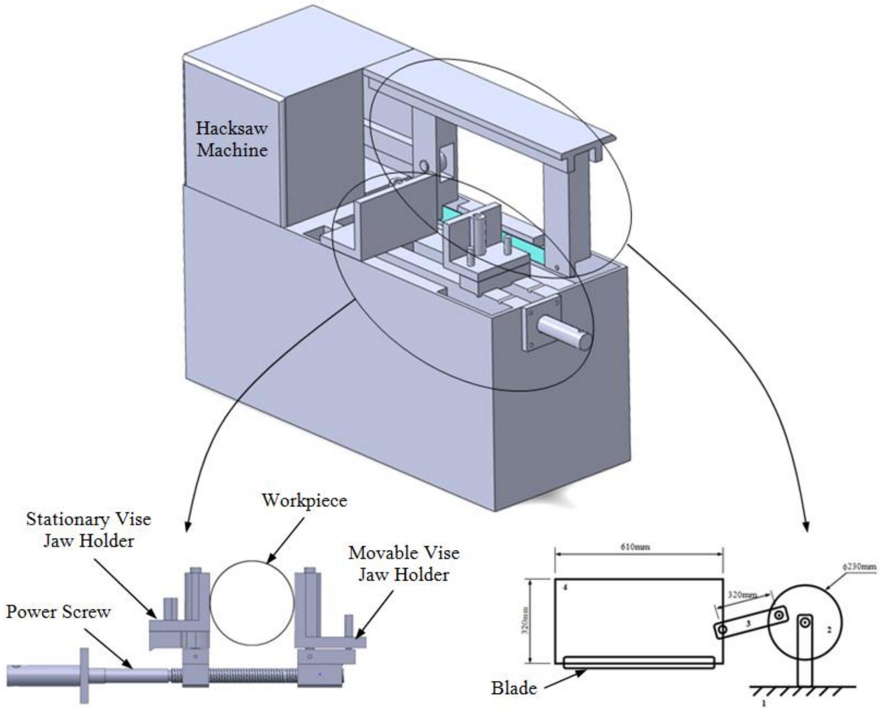
(b) Deformed power screw

Figure 2. Broken parts of hacksaw machine.

2. Analysis of Hacksaw Machine

2.1 Force Analysis

To be able to carry out the stress analysis for the hacksaw parts, it is extremely important to determine the maximum forces. The forces are from tightened threaded screw of the vise and from the maximum acceleration of the slider link of the hacksaw mechanism, as shown in Figure 3.



(a) Force coming from the threaded screw (b) Force coming from slider link

Figure 3: Hacksaw Machine showing the type of loads effect the vise jaw holders

2.1.1 Force coming from the threaded screw

The power screw of the hacksaw machine is single square-thread screw. The major diameter of the screw is 26mm, and the pitch is 5mm. To tighten the power screw, it is important to understand the concept of the preload and maximum load that power screw can withstand.

The preload force, F_i for nonpermanent connections can be calculated by

$$F_i = 0.75F_p \quad (1)$$

where F_p is the proof load and is obtained from

$$F_p = A_t\, 0.85S_y \quad (2)$$

where A_t is the tensile stress area.

For the case of our hacksaw machine, its geometry is the following: major diameter D=26mm, miner diameter d =20.5mm, and mean diameter d_m= 23.25mm. The tensile stress area can be calculated using the pitch diameter d_p= D – 0.649519 = 22.75mm. Thus, The preload force F_i is 54.24kN for low-carbon steel (S_y = 225MPa). This preload force is the maximum load that the power screw can withstand. If the load exceeds this limit, it could lead to a plastic deformation of the power screw. Hence, the wrench torque, which has a built-in dial that indicates the proper torque, is used to apply a load below the specified preload.

2.1.2 Force coming from slider link

The force that transmits to the Vise Jaw holders from the slider link is a result of the motion of the mechanism on the workpiece. Based on the analysis of a hacksaw machine with a horse power of 2.2kW and crank shaft angular velocity of 180rpm, the maximum acceleration of the slider link is 50m/s^2 (see Figure 3).

Thus, to calculate the force of the link (Norton 2004),

$$F_s = m(\dot{d} + \mu g) \quad (3)$$

where

m = the mass of the slider link (25kg based on our lab calculation)

μ = the friction between the saw and the workpiece

$\dot{d}$ = acceleration of the slider link (m/s^2)

g = gravity acceleration (m/s^2)

Thus, the maximum slider force is F_s= 1200N, and the minimum force is zero.

2.2 Material Analysis

The material analysis was conducted using a scanning electron microscope (SEM) to identify the materials used to manufacture the vise jaw holders. Table 1 shows the chemicals composition of the parts. Figure 4 shows the amount of graphite present as well as the shape, size, and distribution of the graphite flakes. It is clear from the material analysis that the material is gray cast iron.

Another material analysis is conducted using Brinell hardness test, to verify the type of martial used. The analysis showed that the hardness of the material was 174HB, which confirmed the results that the martial is gray cast iron. Based on the analysis performed above, the material properties according to the American Society for Testing and Materials (ASTM) and (Krause 1969), are shown in Table 2.

Table 1: Chemical composition of vise jaw holders

Element	Weight %
C	4.3
Si	2.13
Mg, So, P	0.625
Fe	92.945
Totals	100.00

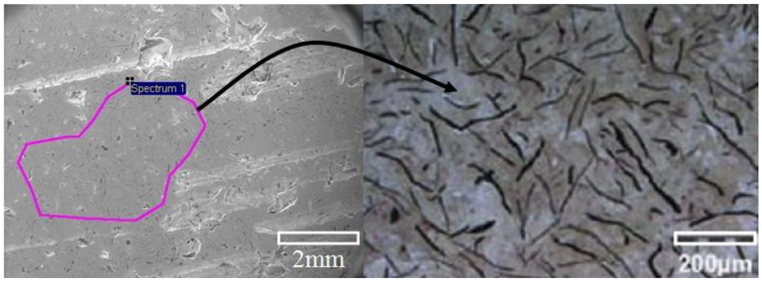


Figure 4: A microscopic picture shows the amount of graphite flakes

Table 2: Mechanical Properties of Gray Cast Iron ASTM 25

Tensile Strength (MPa)	Compressive Strength (MPa)	Modulus of Elasticity (GPa)	Endurance Limit (MPa)	Brinell Hardness	Fatigue Stress concentration Factor K_f
179	668	100	79	174	1.05

2.3 Visual Inspection

Parts fail because their stresses exceed their strength. The type of stresses that cause the failure could be tensile, compressive, or shear. Failure depends on the material and its relative strengths in tension, compression, and shear. It also depends on the type of the loading (static or dynamic) and on the presence or absence of cracks in the material. Static loads are slowly applied and remain essentially constant with time. Dynamics loads are either suddenly (impact) applied loads or repeatedly varied with time (fatigue loads), or both. Fatigue failures always begin at a crack. The crack may have been present in the material since its manufacture, or the crack may have developed over time due to cyclic straining around stress concentrations. Fatigue cracks generally begin at a notch or other stress concentration. Thus it is critical that dynamically loaded parts be designed to minimize stress concentrations.

A close inspection of the failed components reveals the following modes of failure:

- A crack occurred in both the stationary and movable vise jaw holders.
- The power screw was subjected to a large deformation.
- In the movable vise jaw, the crack initially began near the inside hole, as shown in Figure 5 shows.
- In the stationary vise jaw, the crack initially began from the side, as shown in Figure 6.

These observation suggests that the parts were subjected to a large load. To investigate these failures further, a finite element model was developed to understand the mechanism of failure that causes the vise jaw holders to fail and is discussed in the next section.

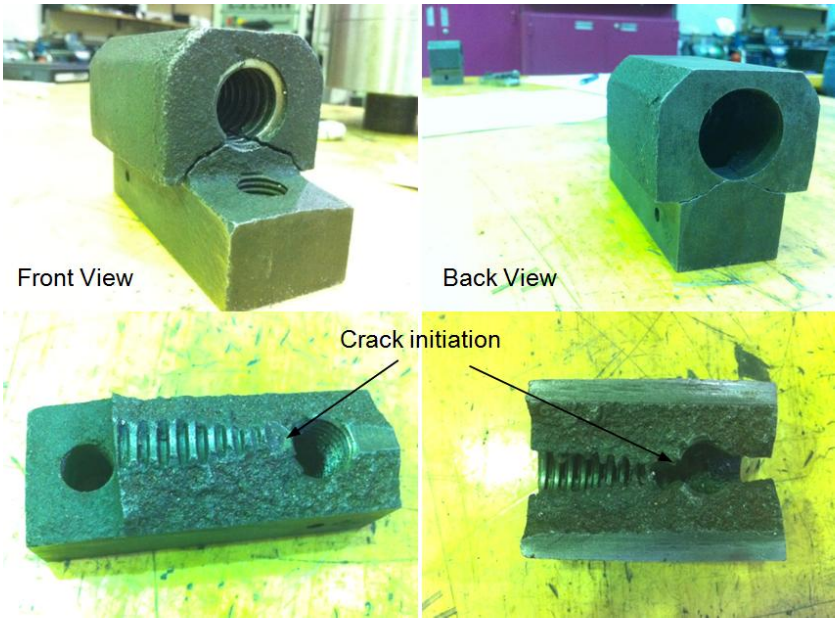


Figure 5: Movable vise jaw shown the location of crack initiation

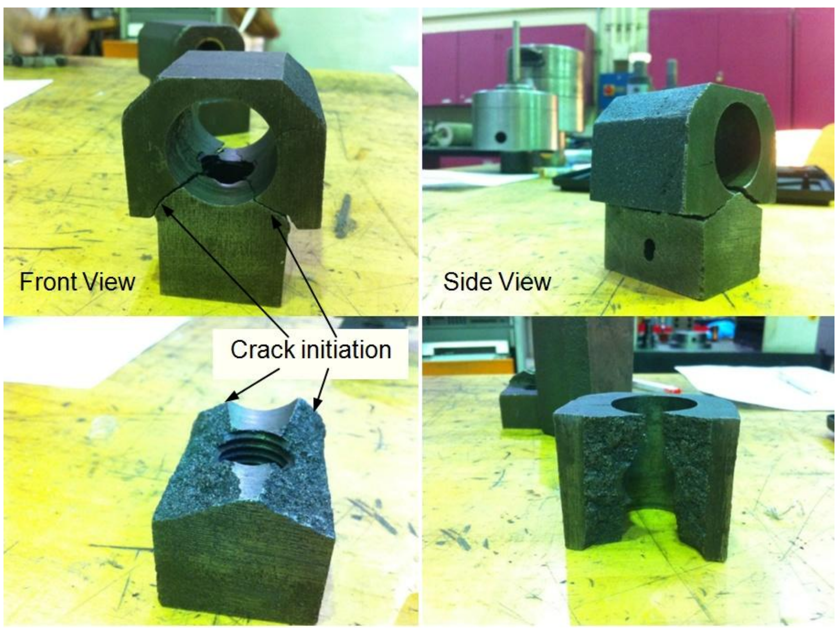
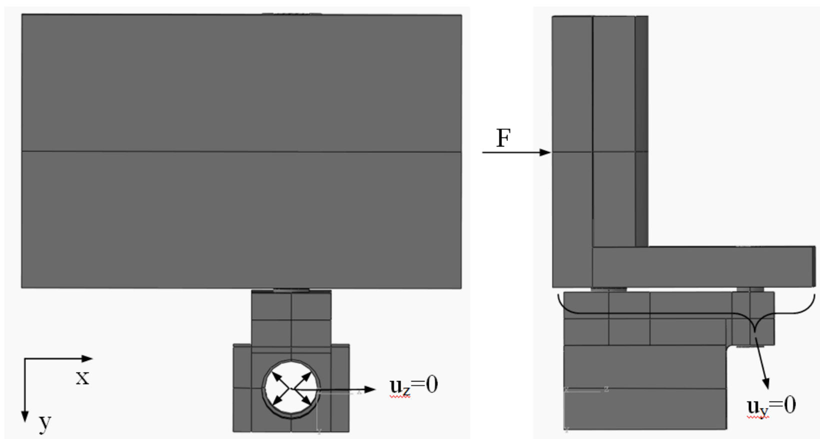


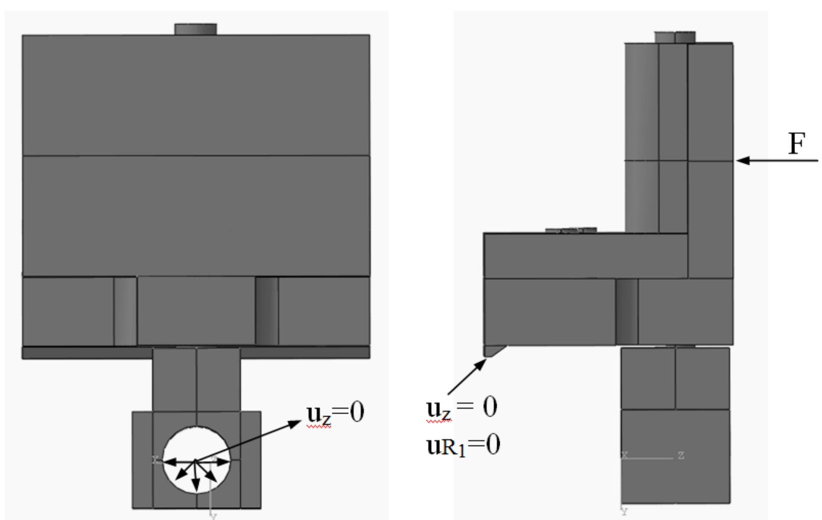
Figure 6: Stationary vise jaw shown the location of crack initiation

3. Finite Element Modeling

The art of the finite element analysis lies in the representation of a real mechanical structure and its loading by a mathematical model, which then can be analyzed. An accurate and efficient idealization was attempted to create a model as similar as possible to the real structure from the geometric and loading point of view. Two finite elements models were considered in the present work. The considered models were the stationary vise jaw holder and the movable vise jaw holder. Figure 7 shows the configuration, constraints and loading conditions for both models.



(a) Movable Visa Jaw Holder



(b) Stationary Visa Jaw Holder

Figure 7: Movable and Stationary visa jaw holder models

The mechanical boundary conditions associated with finite element model can be summarized as the following:

Movable Visa Jaw Holder

1. On the base of the Vise Jaw u_y=0, is imposed
2. Inside the hole of the movable vise jaw holder, the direction of the u_z is fixed.
3. The model is subjected to a maximum constant tensile force of F_i+ F_S=55.44kN at the face of the vise jaw.

Stationary Visa Jaw Holder

1. On the side edge of the vise jaw bed u_z=0, and the rotation uR_1 about x-axis is zero.
2. Inside the hole of the stationary vise jaw holder, the direction of the u_z is fixed in only half of the hole.
3. The model is subjected to a maximum constant tensile force of F_i+ F_S=55.44kN at the face of the vise jaw.

3.1 Finite Element Mesh

The finite element computation was performed using ABAQUS/standard (2012). The finite-element meshes of these models were generated using eight-node-linear brick reduced integration elements (C3D8R). Figure 8 shows the FE meshes for both models respectively. The numbers of elements for the different models that were used in current study, after several refined meshes to insure the conversion of FE results, are given in Table 3.

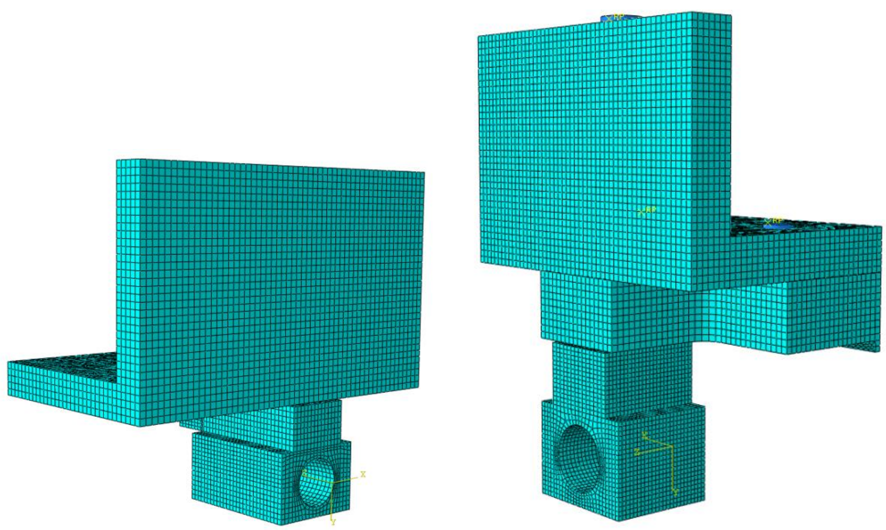


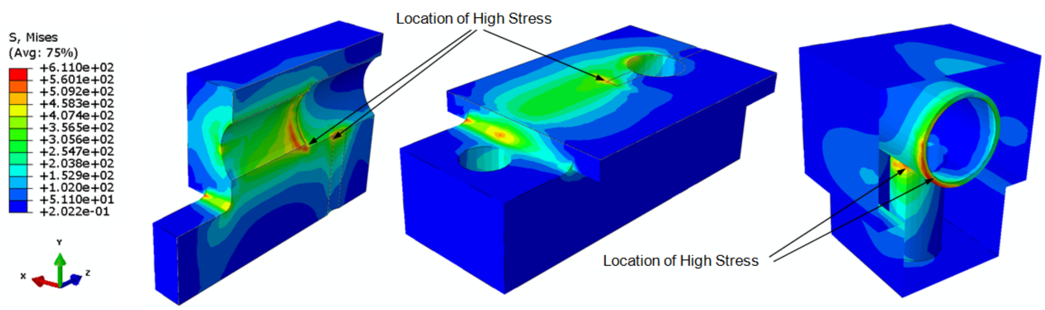
Figure 8: The finite element mesh for movable and stationary vise jaw holders

Table 3: Number of elements used in different models

Model	movable vise jaw holder model	stationary vise jaw holder model
Part	12316	13934
Visa Jaw	21122	28474
Visa Jaw Bed	N/A	14690

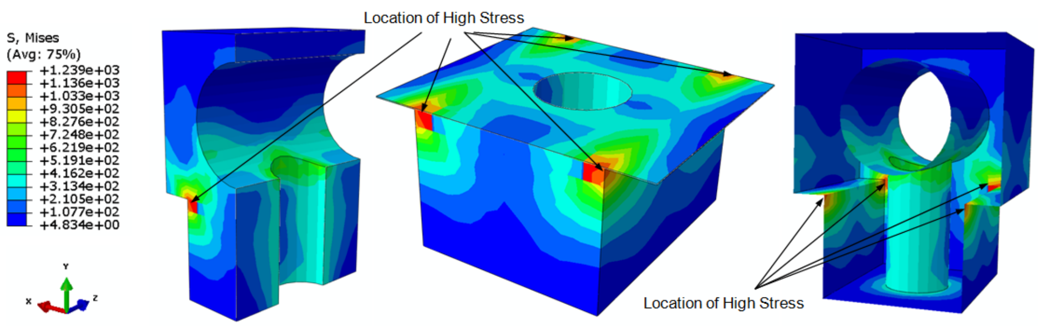
3.2 Finite Element Result

A contour plot for both vise jaw holders was obtained. Figure 9 shows the Von Mises stress contour for the movable vise jaw holder model. Different sections were created to identify the location of the high stress. Figure 10 shows the Von Mises stress contour for the stationary vise jaw holder model. Again, different sections were made to identify the location of the high stress.



(a) X-Plane cut (b) Y-Plane cut (c) Z-Plane cut

Figure 9: Von Mises stress contour for movable vise jaw holder model



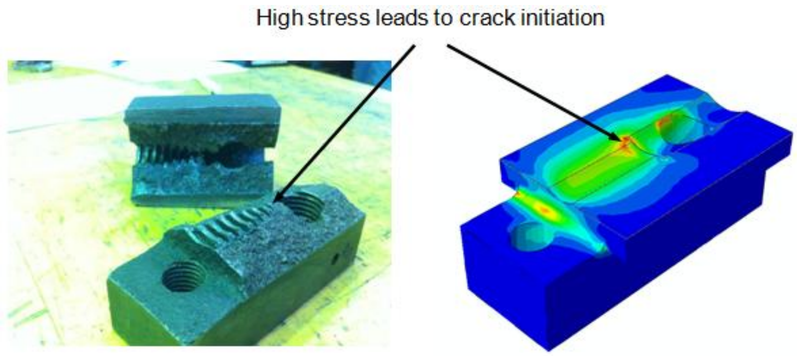
(a) X-Plane cut (b) Y-Plane cut (c) Z-Plane cut

Figure 10: Von Mises stresses for stationary vise jaw holder model

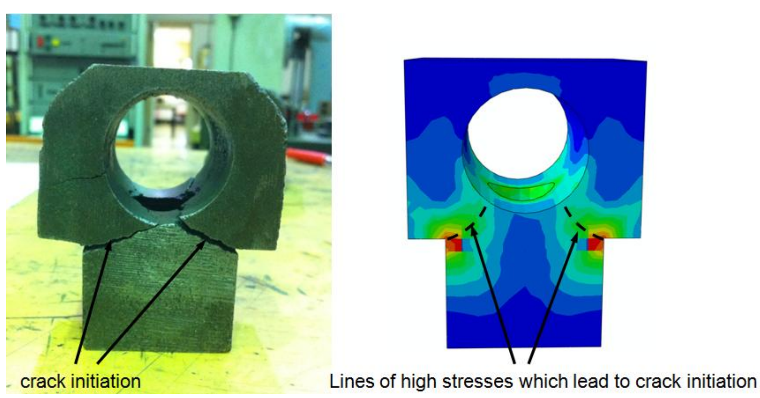
3.3 Discussion

The broken parts of the hacksaw machine were first analyzed to determine its material, dimension, and the maximum load that it could be subjected to. By analyzing the broken parts, we found that the material used to manufacturing these parts was gray cast iron. In addition, the power screw was subjected to a high load, which let the power screw to be deformed and exceeded its preload limit.

After analyzing the hacksaw machine using Abaqus/standard, the results showed that the stresses were exceeded the elastic limit of the material. It also showed that the location of the maximum stresses that lead to fracture were identical to the location of the fracture in the broken parts, as shown in Figure 11. Moreover, the stationary vise jaw holder was not directly broken because the stationary part was not touched by the power screw until the movable part was broken.



(a) Location of crack initiation for Movable vise jaw



(b) Location of crack initiation for stationary vise jaw

Figure 11: Comparison between the Finite element models and the physical parts

4. Conclusion

Vise jaw holders of the hacksaw machine break due to accidental heavy loads. With the objective of investigating the failure of these components, a three-dimensional finite element model was created. The material of the broken parts was identified, the CAE model was built, and the hacksaw mechanism was analyzed to determine the actual applied loads on the broken parts. After analyzing the model using Abaqus/Standard, the results showed that the location of the high stresses was identical with that of the original broken parts. Furthermore, the power screw was subjected to a high load, which deformed the power screw. Also, the stationary vise jaw holder was broken by impact because it was not touched by the power screw until the movable vise jaw holder was broken. To avoid this human error, a wrench torque, which has a built-in dial that indicates the proper torque, could be used to develop the necessary load required to hold the workpiece. We could also improve the design of the vise jaw holders by adding a fillet at sharp edges and using a more ductile and higher strength material.

5. References

1. ABAQUS, User's Manual, Version 6.11, Dassault Systémes Simulia Corp., Providence, RI.
2. Boyer H. E. et al., "Metals handbook 10-failure analysis and prevention." American Society for Metals, eight edition, pp.1-10,1975.
3. Irisarri A.M., and Silveria E., "Study of failure of one machining tool." Engineering Failure Analysis, Vol. 17, pp.380-386, 2010.
4. Krause D. E., "Gray Iron-A Unique Engineering Material Gray, Ductile, and Malleable Iron Castings-Current Capabilities," ASTM STP 455, American Society for Testing and Materials, Philadelphia, pp. 3-28, 1969.
5. Layer J., Adler T., et al.,"ASM handbook-failure analysis and prevention." ASM International, 11ed, pp.14-65, 2002.
6. Leonel E. D., Venturini W. S., and Chateauneuf A., "A BEM model applied to failure analysis of multi-fractured structures." Engineering Failure Analysis, Vol. 18, 1538-1549, 2011.
7. Norton R. L., "Design of Machinery," McGraw Hill, Third Edition, 2004.