Fatigue analysis of link of punch machine using SolidWorks

Shubham Ambastha
shubhams.saurav@gmail.com
Bhilai Institute of Technology, Raipur, Chhattisgarh

Devnarayan Dhiwar
8988devnarayan@gmail.com
Bhilai Institute of Technology, Raipur, Chhattisgarh

ABSTRACT

A connecting link of a punching machine work as a medium for transferring the force between the motor and the cutting tool of the punching machine. This work deals with fatigue analysis of connecting link of punching machine using the Solidworks software for modelling and simulation of problem. This work is based on the principle of analysis of connecting link of engines i.e. configuration and properties of slider crank mechanism. This work deals with problem-based due to the self-weight of the link, the reactive force and fluctuating repeated loads offered during the working, probable chances of failure of the link are there. The given punching machine is also known as servo or electro punching press machine that works by the conversion of produced torque into a linear force via a ball screw. The failure of a link is also responsible for damage of motor and flywheel. In this work, Solidworks 2016 software is used for finite element analysis of connecting link and other components of the machine. The analysis gives the result in terms of properties that are stress, working life cycle, strain and displacement. The magnitude of the above mentions properties result are respectively- Minimum 9.5316e+006 N/m^2 and Maximum 2.70489e+008 N/m^2, Minimum 17252.7 cycle and Maximum 1e+006 cycle, Minimum 3.98234e-005 and Maximum 0.00215751, Minimum 0.00475148 mm and Maximum 0.314305 mm.

***Keywords*—** *Fatigue failure, Punching press, Fluctuating load, Connecting link, Motor*

1. INTRODUCTION

1.1 Punching press machine

In punching machine punch used to create the holes in given sheet and press is a mechanism which is used for forcing die to penetrate into the material. The punching of holes is frequently established with a die that used in bending and forming. The punching press is designed according to the requirement for the making of various types of shaped holes and cutouts on sheet metal and plate material. The punching press is fitted with different types of punches and dies of the size and shape of the hole according to our requirement. If irregular nonstandard holes create, the modern punch press is capable of nibbling action. This refers to the action of a series of a successive number of hits followed by a predetermined desired pattern that creates the cut out from the plate. This punching action is accomplished by a vertical moving ram that forces the punch by an input power delivered by the motor, the material and into a die through which the resulting slug is ejected. Here additionally, a component to hold the material in place of the punch is withdrawn, call a stripper, is a fully integral part of the punch tool of this machine. This press ram may be activated manually, mechanically, or hydraulically. This punching press, usually, is capable of generating about four tons of force on the working plate. Given punching, presses use an integrated mechanism of flywheels, gears, and eccentrics to stroke the ram to generating the holes on the sheet. Here hydraulic presses use oil pressure to perform the punching action. This given last two types can generate from 9 to 70 tons of force with some larger components creating over 160 tons. Mechanical punching presses can generally operate faster than the hydraulic models but the latter can exert more punching pressure more uniformly on the given working thicker workpieces. The capacity of the punching press is determined by not only the available tonnage, but also by the effective throat depth on the given workpieces.

1.2 What is fatigue?

If any component working under the repeated loading and unloading that causes weaken objects over time even when the induced stresses are considerably less than the allowable stress limits. This phenomenon is known as fatigue. Each working cycle of stress fluctuation weakens the object to some extent. After the number of cycles, the object becomes so weak that it fails. Fatigue is the primary cause of the failure of many objects, which works under the fluctuating loading especially those made of metals. Examples of failure influence on fatigue included, rotary machinery, aeroplane wings, bolt, consumer products, offshore platforms, ships, vehicle axles, bridges, and bones.

1.3 Importance of connecting link

A connecting link of the punching machine is influenced by the complex state of loading. It carries high cyclic loads of the order of repeated loading, which contains a range from the high compressive load's due reaction from the plate and motor input power to

high tensile loads due to inertia. Therefore, the durability of this component is of critical importance. According to these factors, the connecting link has been the topic of research for a different topic of research for different aspects such as production technology, fatigue analysis, performance simulation, and material. The connecting link fatigue analysis is done by the Solidworks software using the principle of finite element techniques and, optimization techniques, improvement in the technique of production, and optimize the manufacturing cost by this analysis.

1.4 Analysis using SolidWorks

SolidWorks is a solid modelling Computer-Aided Design (CAD) and Computer-Aided Engineering (CAE) computer program that runs on Microsoft Windows. SolidWorks is published by Dassault Systems. SolidWorks® Simulation is a design analysis application fully integrated with SolidWorks. It provides a one-screen solution for stress analysis and also enables to solve large problems quickly using our personal computer. SolidWorks Simulation Professional allows for the setup/evaluation of a virtual fatigue analysis within the SolidWorks product suite. This will allow the user to answer the question as to whether a product will withstand usage requirements over a period of time and not just if the product will hold up to a single specified load environment. Finite element analysis is a computer simulation techniques used in engineering analysis. It uses a numerical technique called finite element method. The finite element method is one of the most used methods in engineering. This method having various sub-steps involves and that through solve geometry.

2. METHODOLOGY

Solidworks Motion simulates the mechanisms of assemblies with moving components and calculates forces that develop on components during motion. After running a motion study, we can export it to Simulation the forces developed on individual parts to calculate their stresses and displacements. Also export motion loads for one or multiple time frames. The simulation creates a design scenario for each selected time frame populated with motion loads developed at a particular time frame.

2.1 Create part models and assembly

The components are as follows:
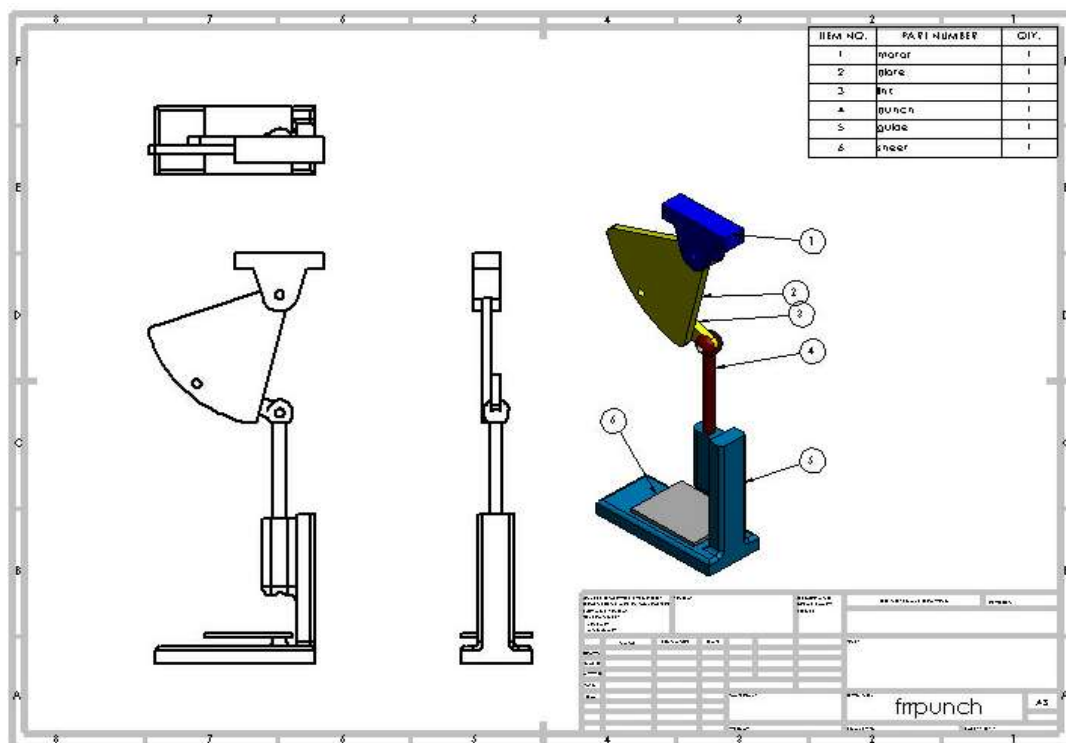- Motor
- Plate
- Link
- Punch
- Sheet

Fig. 1: Punch Machine

Table 1: Model information

Model Name	Volumetric Properties
	Mass:0.15531 kg Volume:2.15708e-005 m^3 Density:7200 kg/m^3 Weight:1.52204 N

2.2 Importing Motion Loads
SOLIDWORKS Motion simulates the mechanisms of assemblies with moving components and calculates forces that develop on components during motion. After running a motion study, we can export it to Simulation the forces developed on individual parts to calculate their stresses and displacements. Also, export motion loads for one or multiple time frames. The simulation creates a design scenario for each selected time frame populated with motion loads developed at a particular time frame. Different steps involved in this process are:
(a) Define load bearing faces and run a motion analysis study.
(b) Export loads from SOLIDWORKS Motion to SOLIDWORKS Simulation for multiple time frames.
(c) Run design scenarios created from motion frames.
(d) Graph design scenario results and identifies critical time instants.

The motion study, defined in the assembly document includes:
- Rotary motor applied at the Plate part (Oscillating, 1 Hz frequency, 20 degrees amplitude).
- Gravity load.
- Contact between parts Punch and Sheet with selected material Steel (Dry).
- Study properties set to 25 frames per second for motion analysis. Analysis duration set to 5 seconds.
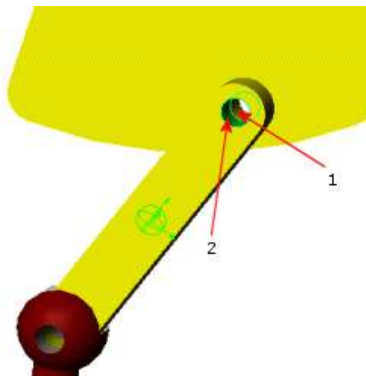

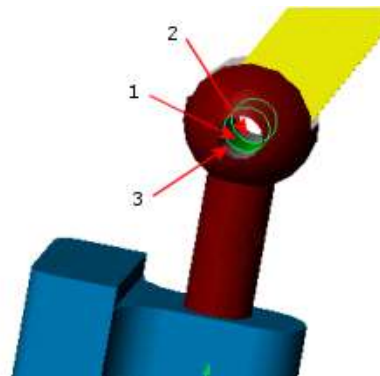Fig. 2: 1st Load bearing faces   Fig. 3: 2nd Load bearing faces

Now importing motion loads and defining design scenarios. Simulation automatically creates a new design study in the link part document with predefined design scenarios. Each design scenario has the loads developed during motion at the particular time instant of the associated frame. By analyzing the link for all design scenarios, we can locate critical time instants when maximum stresses are likely to develop in the part.

2.3 Fatigue Analysis of a Link Part
The Link part of the Punch assembly is analyzed using fatigue analysis. The loads for the reference static study are imported from SolidWorks Motion as illustrated by the Importing Loads from SolidWorks Motion Different steps involved in this process are:
(a) Defining a fatigue study
(b) Setting properties of the fatigue study
(c) Defining constant-amplitude fatigue events
(d) Viewing an SN curve for the part material
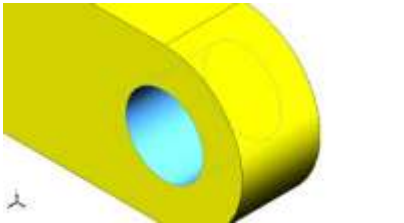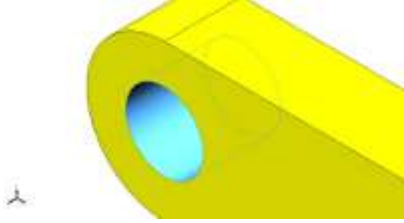(e) Viewing fatigue results

Analysis type: Fatigue (Constant Amplitude)
Event Interaction: Random
Computing alternating stress using: Equivalent stress (von Mises)

Table 2: Loads and Fixtures

Load name	Load Image	Load Details	
ALT-Gravity		Reference: Values:	Plane 1 -9.65976 1.22505 -4.49749e-017
ALT-CentriFugal		Centrifugal, Ref: Angular velocity: Angular acceleration::	Plane1 0 rad/s 7.73172 rad/s^2

ALT-RemoteLoads-1		Entities: Type: Coordinate System: Force Values: Moment Values: Reference coordinates: Components transferred:	1 face(s) Load (Direct transfer) Global Cartesian coordinates -6378.24, -0.00332865, -3.33089 N -2.74755, -0.095674, -5.87433e-013 N.m 0.05 -7.25114e-016 0.005 m Force and Moment
ALT-RemoteLoads-2		Entities: Type: Coordinate System: Force Values: Moment Values: Reference coordinates: Components transferred:	1 face(s) Load (Direct transfer) Global Cartesian coordinates 6378.45, -0.00684366, 3.33089 N 2.74749, -0.238481, 1.45961e-017 N.m -0.05 -1.73472e-018 0.005 m Force and Moment

After applying different loads in faces of the link part we do motion study and in that, we apply design scenarios which provide us about different stress induced in the Link part.

Here we can see that the maximum von Mises stress is likely to occur in the Link part at the time frames that correspond to design sets 4, 5, and 6. The time instants for these frames are 2.44 sec (frame 62), 2.48 sec (frame 63), and 2.52 sec (frame 64), respectively.
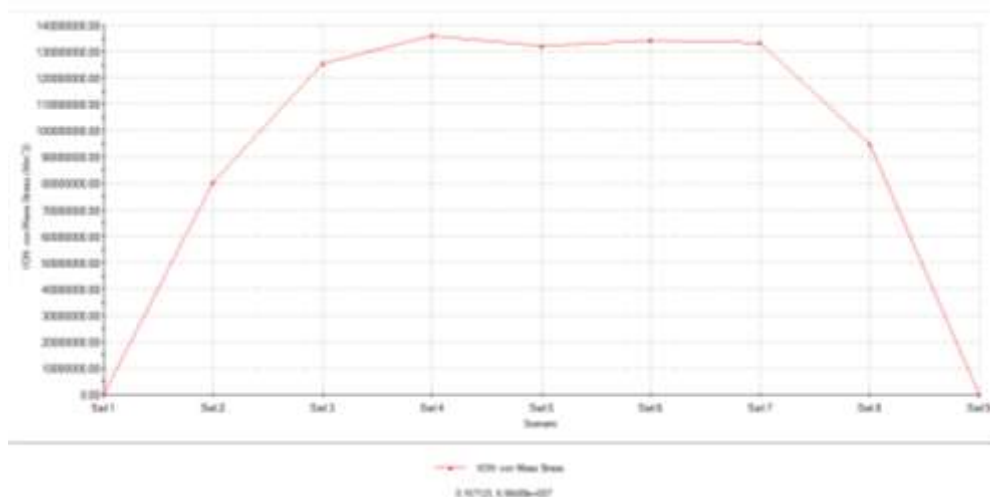

Fig. 4: History graph for Von Mises Stress

3. RESULT
- We are seeing that the results for the damage factor indicate that the specified event consumes about 5.7 % of the life of the model.
- The life plot shows that failure due to fatigue is likely to occur at the two holes after approximately 17,580 cycles.
- Von Mises Stress: The minimum stress induced in this link is 9.5316e+006 N/m^2 while maximum stress is 2.70489e+008 N/m^2.
- Resultant Displacement: The Displacement in this link is 0.00475148 mm at node 198 while the maximum is 0.314305 mm at node 31.
- Equivalent strain: Minimum strain is 3.98234e-005 at Element 163 and maximum is 0.00215751 at Element 256.
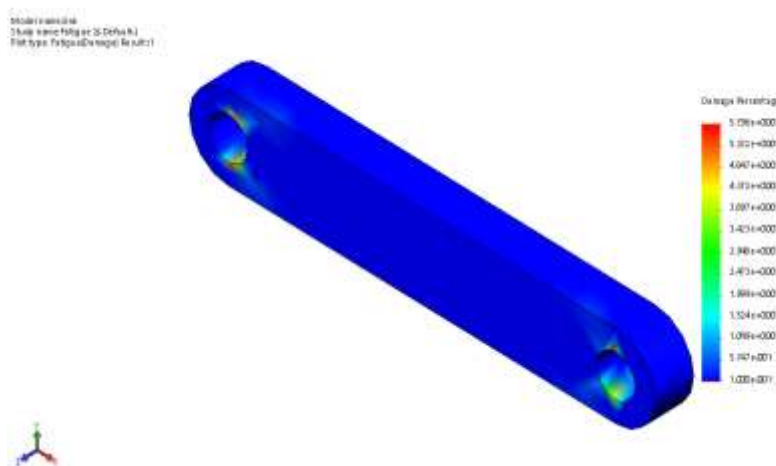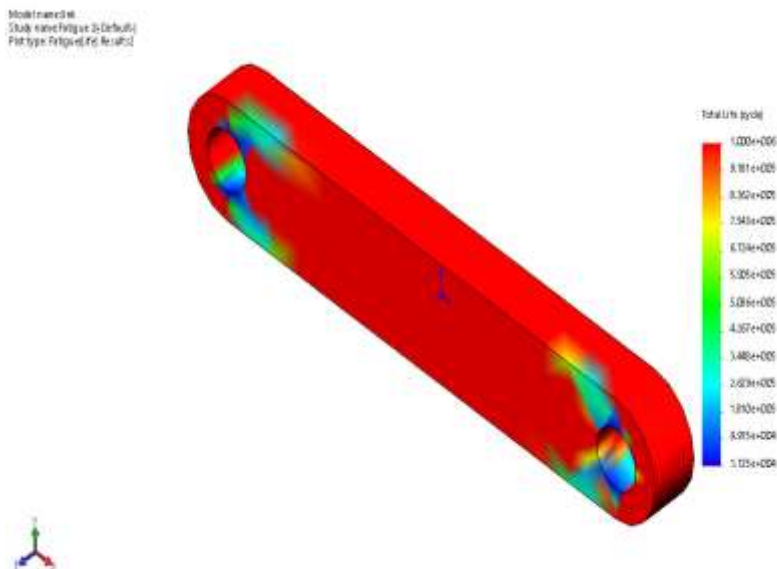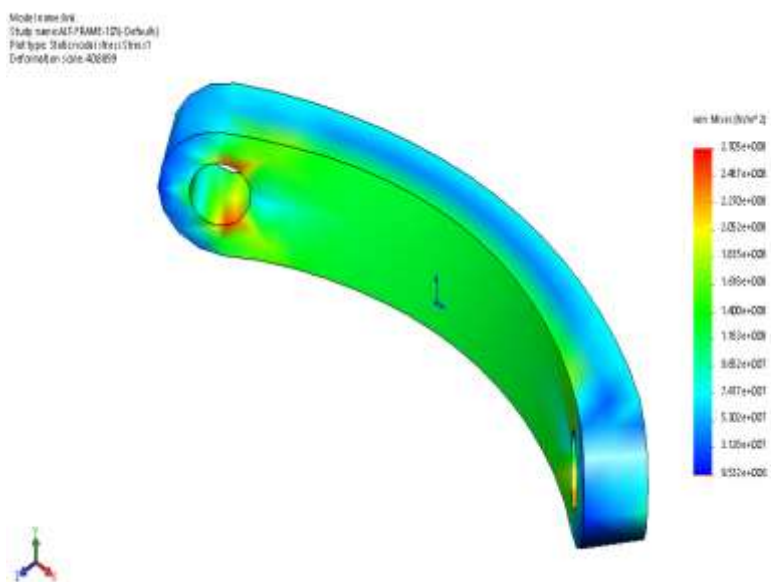

Fig. 5: Damage plot result

Fig. 6: Life plot result
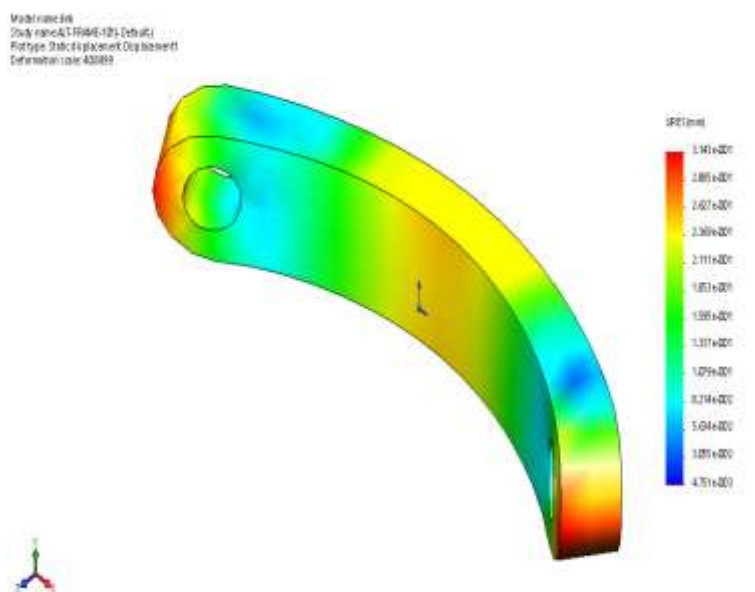

Fig. 7: Von Mises stress
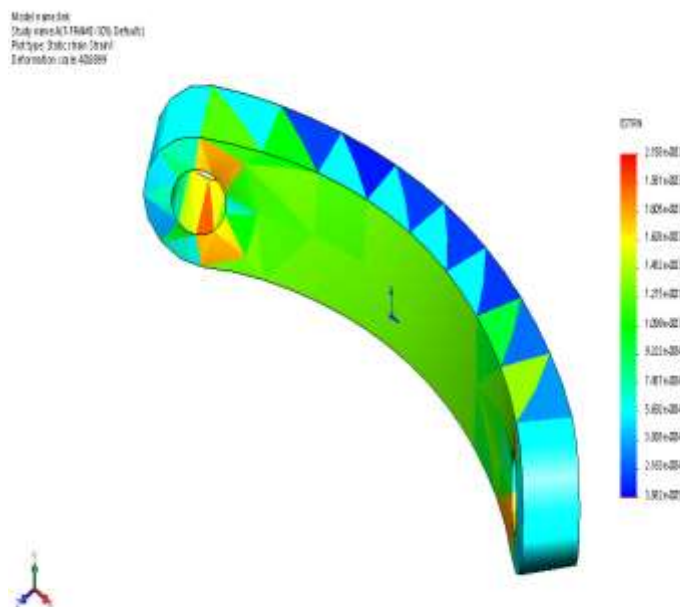

Fig. 8: Resultant Displacement

Fig. 9: Equivalent strain

4. CONCLUSION
The conclusion of this study is that the connecting rod can be designed and improved under a load range comprising tensile and compressive load corresponding to the 360-degree crank angle at the maximum speed and peak pressure at the extreme load respectively. Furthermore, in this way the existing connecting rod can be replaced with a new connecting rod made of Gray Cast Iron that is 10% lighter and 25% less expensive, in spite of lower strength Gray Cast Iron compared to the existing forged steel. Connecting rods can be replaced by steel forged connecting rods with a probable cost reduction of approximately 15% or higher, with similar or better fatigue behaviour. Fatigue strength was the most significant factor in the optimization of this connecting rod.

5. REFERENCES
[1] Nagesh H R & Sannamani L G, "Static Analysis of Punch and Die Used For LSV Link Bracket Using Solidworks", Dept. Of Tool Engineering, Govt. Tool Room and Training Centre, Mysore, India. (IJIMIINDS).
[2] Bhavesh N Khichadia., Dipesh Kumar Chauhan., "A Review On Design And Analysis Of Mechanical Press Frame", Mechanical Engineering Department, RK University, Rajkot, Gujarat, India.
[3] Hyounsoo Park., Young Sang Ko., Seung Cheal Jung., Byung Tag Song., Yong Ha Jun., Byung Cheol Lee., Jong Dae Lim., "Development of Fracture Split Steel Connecting Rods" SAE Technical Paper Series, Paper No. 2003-01-1309.
[4] Rabb R., 1996, "Fatigue failure of a connecting rod" Engineering Failure Analysis, Vol. 3, No. 1, pp. 13-28.
[5] Afzal, A., 2004, "Fatigue Behavior and Life prediction of Forged Steel and PM Connecting Rods", Master's Thesis, University of Toledo.
[6] Dorian Nedelcu., Marian-Dumitru Nedeloni., Daniel Daia., "The Kinematic and Dynamic Analysis of the Crank Mechanism with SolidWorks Motion", University of Resita P-ta Traian Vuia 1-4, 320085 Resita Romania.
[7] Serag, S., Seviean, L, Sheha, G. and I. El-Bestway., 1989, "Optimal design of the connecting-rod", Modeling Simulation and Control, B, ASME Press, Vol.24, No.3, pp 49-63.