Journal of Multidisciplinary Approaches in Science (JMAS)

Journal homepage: https://jmas.biz/index.php

Improved Design of Compression Test Jig for Sheet Metal Using Finite Element Analysis

Open

Abdoulhdi A. Borhana*[,1], S.Dass[2], Ali Najah Ahmed[3] & M.A.Khattak[4]

[1,2] Department of Mechanical Engineering, College of Engineering, University Tenaga Nasional, Jalan IKRAM-UNITEN,43000 Kajang, Selangor, Malaysia
[3] Institute of Energy Infrastructure (IEI), Universiti Tenaga Nasional (UNITEN), Kajang, 43000, Selangor Darul Ehsan, Malaysia
[4] ARL Laboratory Services PTY Ltd. 13 61/55 Pine Rd, Yennora Sydney, NSW 216, Australia

ARTICLE INFO

Article history:
Received 9 January 2019
Received in revised form 1 March 2019
Accepted 8 March 2019
Available online 10 April 2019

Keywords:
Finite element (FE) method, True stress-plastic strain curve, Shear compression test, Shear compression metal sheet (SCMS) specimen

ABSTRACT

This paper discusses the improvisation of jig design of shear compression test for a metal part specimen from the previous jig design. In conjunction with that, this research leads to investigate the behavior which occurs on the metal part specimen during the shear compression impact at the gage section of 3.5mm by using the modified jig design. This paper also discusses the result of the metal specimen during shear compression impacts such as stress and strain and to examine the load-displacement through Finite Element Method. Finally, using the modified jig design, the result obtained from this research such as stress behavior, plastic strain deformation, and displacement are identified and studied.

Copyright © 2019JMAS - All rights reserved

1. Introduction

In the United States, the National Highway Traffic Safety Administration, the Federal Administration and the Department of Defense have been working on developing their own authoritative safety requirements and conducting research on crash safety. One of the researches is crashworthiness. Moreover, crashworthiness is more focused on automotive industry because crashworthiness indicates the proportion of the vehicle's basic capacity to plastically deform but then keep up an adequate survival space for its occupants in accidents including sensible deceleration loads. For example, the impact of the crash of the vehicles depends on the body materials of the vehicles [1].

The main purpose of crashworthiness design is to give importance to the passenger as the first priority and equipment as the second priority. In car industry, a lot of researchers keep on exploring and improving the design of the vehicle so that the vehicle structure can ingest the crash energy by

* *Corresponding author.*
E-mail address: amhmad@uniten.edu.my (Abdoulhdi A. Borhana)

controlled vehicle disfigurements while keeping up adequate space with the goal that the lingering crash energy can be overseen by the limitation systems to limit crash loads exchange to the vehicle occupants. Over the past seven decades, the material typically used in vehicle structures is sheet

E-mail address: amhmad@uniten.edu.my (Abdoulhdi A. Borhana)

metal which allows for the economic mass production of millions of units. Crashworthiness in the automotive industry is evaluated by a combination of testing and analysis.

For the testing part, a prototype is built to identify the energy absorption capacity. For the analysis part, numerical simulation technologies are being used. In this modern era, to achieve advanced performance design of car bodies, numerical simulation has been chosen as the most important method as it can express the characteristics of material accurately. In conjunction with that, it encourages more to facilitate the way toward overcoming the effect of the vehicle crashes on the sheet metal vehicles through the development of advanced technology vehicle [2]. The crash simulations that commonly used a process of analysis called the Finite Element Method. Based on the work by Borhana et al, [3] on the large strain of shear compression test on sheet metal specimens, when the experimental work is done, FE simulation is utilized to validate sheet material's hardening curve. Next, for the input data of the material hardening's behavior, the conditional true stress-plastic strain curve is used.

1.1 Sheet Metal Specimen

Sheet metal has variable significant thickness where the foil or leaf-type represents the thinner version of the metal sheet which is produced in the industry while plates represent the sheet metal pieces which are thicker than 6mm [4]. Using the torsion method, sheet metals can experiment where the metal sheet is welded into a tube which is thin-walled. Unfortunately, when the sheet metal is rolled and welded to make the tube, the sheet metal's structure can be changed and thus changes its visible properties. Torsion testing also causes the strain and stress to be not the same as the whole cross-section of the sample [5]. Based on the recent work by Rittel (2006) and Dorogoy (2006), a single specimen is used to find out both the quasi-static and the dynamic characteristic of a material. The specimen has an uncomplicated geometric shape which consists of a cylinder with a double slot machine of gage section at 45º on its faces [6].

1.1.1 Shear-compression specimen

Based on Choung et al. (2008), for both flat specimens and round specimen, the stress is uniaxial during uniform deformation [7]. Lee [8] did large strain testing using shear-compression specimen. The specimen comprises of a short chamber into which two spaces have been machined at 45^{o} to the longitudinal pivot. The principle thought is that upon longitudinal compression of the cylinder, the gage area will be exposed to the main condition of shear distortion.

1.2 Modified Design

Using shear-compression specimen (SCS), an experimental on the plastic deformation was done by Vural, Molinari, and Bhattacharyya [9]. The experiment was processed by the technique of digital image processing (DIC). Based on the analysis, the slot angle of shear-compression specimen which has an original value of 45° was modified to 35.26° so that to improve the consistency of stress-strain fields in gage area [9]. By analysis, it is proved that the improvement is fundamental because of the compatibility and boundary conditions that produce a homogenous deformation which is certainly fulfilled for the modified gage section of θ=35.26°. Actually, in 1953, Hill [10] already suggested differing the gage section angle between 0° to 35.26°. However, the slot should have a larger length compared to the width of the slot angles other than θ=35.26° [9].

1.3 Pure Shear Compression Test

1.3.1 General

Simple compression testing is basically simple, have simple sample geometry and can prevent necking [11]. Meyer et al. [12] directed straightforward shear tests in discovering the yield trial of sheet metal and to quantify the shear strains utilizing a digital image correlation strategy (ARAMIS). At the same time, Schikorra et al. [13] recommended that the planar shear tests moved toward becoming the most vital test for testing of sheet metals caused from the facility with which the pressure strain relations can be resolved and because of the nonappearance of frictional elements [13]

1.4 Finite Element Method

The most promising technique to observe the sheet metal for crash deformation is the finite-element method. Moreover, because of the CAE technology keep on improving, the deformation analysis of the shearing process is found out with the help of the finite element method [14]. To solve mechanical engineering problems which mostly deals with linear elastic problems which only consider the elastic modulus and Poisson's ratio as the material constant, finite element method (FEM) has become common which used as engineering tool [15]. Furthermore, FEM can produce stiffness and strength visualizations in a structural simulation and also to reduce the weight and cost of material. The area where the structures bend or twist can be visualized more detailed through FEM. Other than that, before a design is fabricated, FEM permits full design to be constructed, refined and optimized [16].

2. Methodology

2.1 Research Program Flow

The following Figure 1 shows the processor steps for the simulation analysis for metal specimen which is to study the behavior of metal specimen during compression testing.

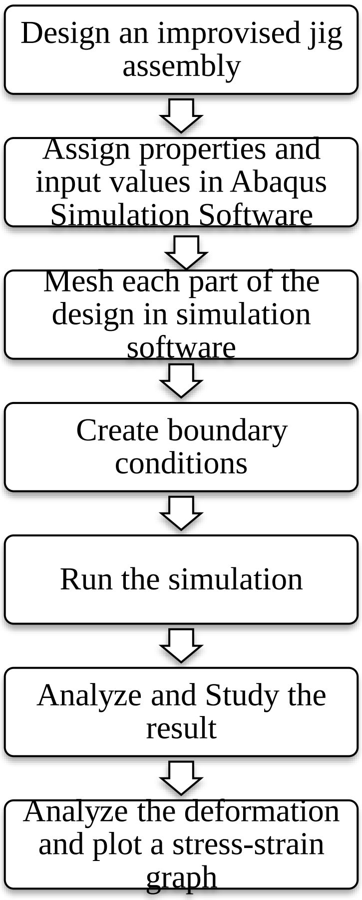


Fig. 1. Flowchart of the research

2.2 Modified Jig Design

2.2.1 Part A (Slider)

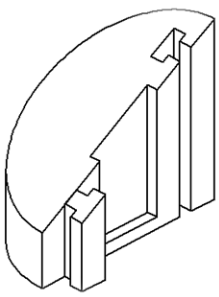


Fig. 2. The Isometric view of lower part A

2.2.2 Part B (Stopper)

This design is actually divided into two parts which **Fig. 2**shows the part A while**Fig. 3**shows the part B. Basically, part A act a slider where it has a slider gap so that the stopper at part B can slide in. But, improvisation has been made from earlier design which is the slider gap is not full until the end of the bottom surface but have a small gap down and the slider gap has a thickness of 1.6 mm. Next, the stopper at part B will have the dimension of 10 mm x 8 mm and a thickness of 1.6 mm and act as a lock. Next, a T-shaped stopper and T-shaped slider are placed at each side of part A and part B. The main reason of this stopper and slider to hold or lock part A and part B. This is very important because part A and part B should be held tightly during the experimental part. Because the jig design has 35°surface, so the T-shaped slider and stopper have three different heights each. For part A, the T-shaped stopper has heights of 28.3 mm, 27.6 mm and 25.5 mm while the T-shaped slider has a height of 16.4 mm, 12.2 mm and 11.5 mm. On the other hand, for part B, the dimension will be vice versa which means the T-shaped slider of part B will have the same dimension as T-shaped stopper of part B. All these dimensions are for lower part A and lower part B and it will be same for upper part A and upper part B. After part A and part B of the lower part and upper part are combined, the full assembly of the jig will have a diameter of 34 mm and a height of 40 mm.

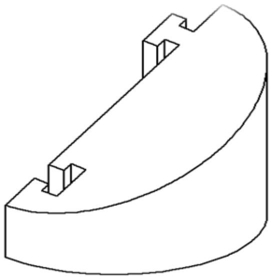


Fig. 3. The Isometric view of lower part B

3. Results

3.1 Simulation of the jig and specimen (Meshing)

All the parts of the jig are meshed before running the simulation. As shown in the figures above, the purple color indicates the seed edges which are planted at the edges at each part. The number of seeds planted at every edge is different and according to the different parts of the jig. For the meshing, a few mesh controls were set. Firstly, the element shape was quad-dominated and the technique used was structured for every part of the jig.

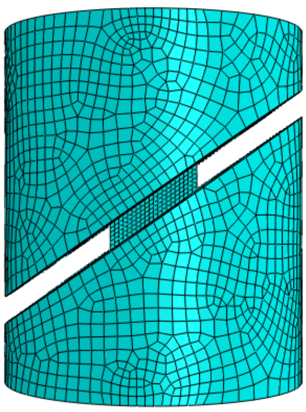


Fig. 4. The full assembly of the jig after meshing

3.2 Boundary Condition

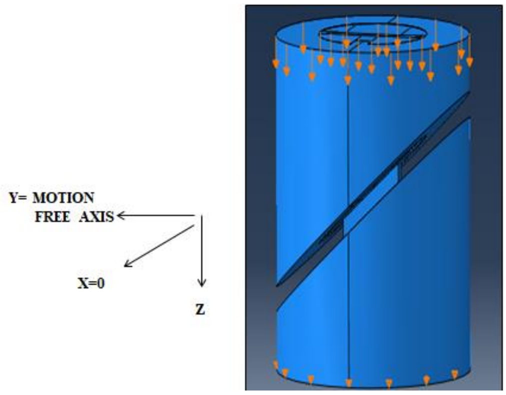


Fig. 5. The illustration of the boundary condition for the simulation

To obtain an expected result or output, a few boundary conditions have been set for the simulation of the jig. On the top of the jig, displacement has been applied which is 2mm (negative z-axis) while the jig's bottom part has been fixed. The motion in the x-axis is set to zero while the y-axis is set to free motion. Boundary conditions are very important and act as a necessary constraint.

3.2 Strain and Stress Value of Different Materials

Table 1. The strain and stress value of material 1

Material 1	
Strain	**Stress**
0	**70**
0.1	**87.5**
0.2	**105**
0.3	**122.5**
0.4	**140**
0.5	**157.5**
0.6	**175**
0.7	**192.5**
0.8	**210**
0.9	**227.5**
1	**245**
1.1	**262.5**
1.2	**280**
1.3	**297.5**

Table 2. The strain and stress value of material 2

Material 2	
Strain	**Stress**
0	**70**
0.1	**105**
0.2	**140**

0.3	175
0.4	210
0.5	245
0.6	280
0.7	315
0.8	350
0.9	385
1	420
1.1	455
1.2	490
1.3	525

Table 3. The strain and stress value of material 3

Material 3	
Strain	Stress
0	70
0.1	140
0.2	210
0.3	280
0.4	350
0.5	420
0.6	490
0.7	560
0.8	630
0.9	700
1	770
1.1	840
1.2	910
1.3	980

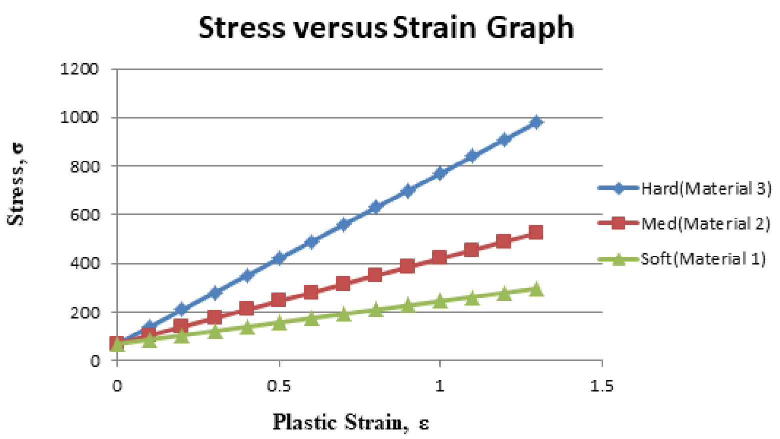


Fig. 6. Stress versus strain graph

3.3 Result of Simulation of Different Materials

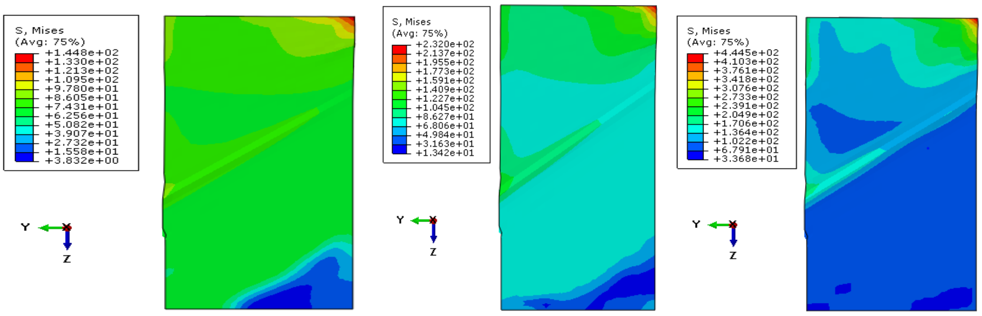


Fig. 7. The stress of specimen of Material 1, Material 2 and Material 3

Based on the simulation result analysis of the specimen of Material 1, it is found that the maximum stress value is 1.448e+02, for material 2 is 2.320e+02 while for material 3 is 4.445e+02.

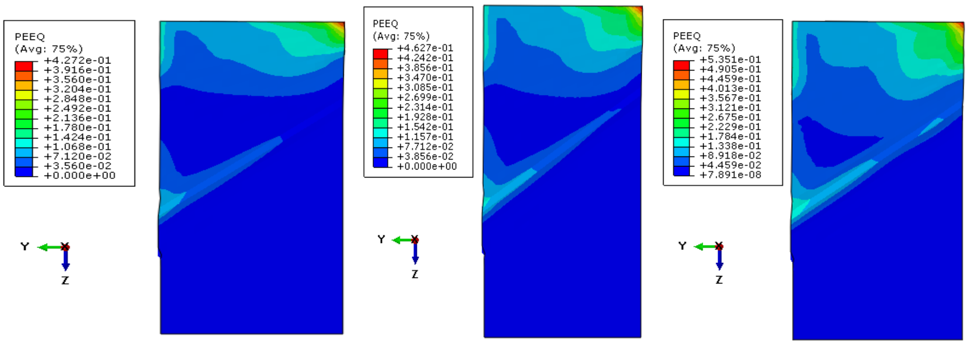


Fig. 8. The strain of specimen of Material 1, Material 2 and Material 3

Based on the simulation result analysis of the specimen of Material 1, it is found that the maximum strain value is 4.272e-01, for material 2 is 2.320e+02 while for material 3 is 5.351e-01.

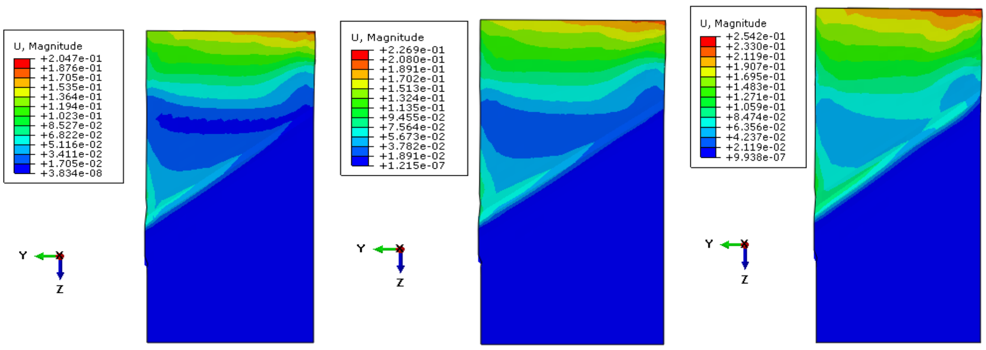


Fig. 9. The displacement of the specimen of Material 1, Material 2 and Material 3

Based on the simulation result analysis of the specimen of Material 1, it is found that the maximum displacement value is 2.047e-01 mm, for material 2 is 2.269e-01mm while for material 3 is 2.542e-01mm.

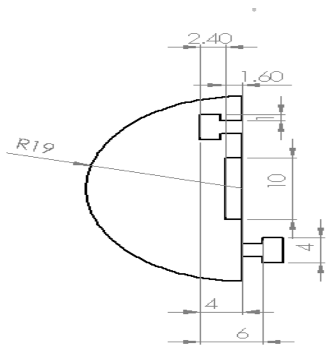
4. Conclusions

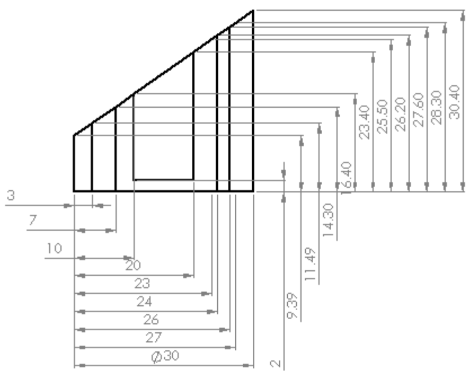
To conclude, a new design of jig assembly for metal compression testing has been designed to overcome and solve the problems found from the previous jig assembly design. This improvised jig design has a unique design and is simulated using Finite Element Method to study the behavior of metal part specimen during the shear compression impact. Based on the simulation results, their important values are being considered which are the stress values, plastic strain values and finally the displacement of the specimen. It is found that when the value of plastic strain increases, the value of the displacement value also increases. Basically, the strain is directly proportional to the displacement and this also tells that the deformation occurs at the specific area of the specimen also will be high. Finally, a graph on strain value and stress value between different materials has been plotted.

Acknowledgment
This research was funded by a grant from the Ministry of Higher Education of Malaysia (FRGS/1/2016/TK03/UNITEN /02/1).

Appendices

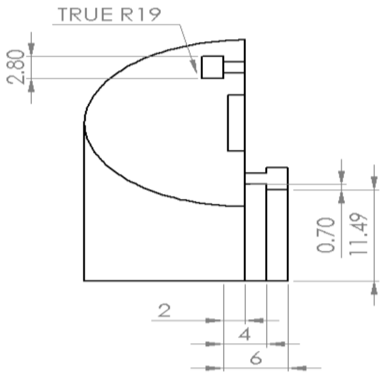
Details on geometry and dimensions of the shear compression jig assembly are illustrated in **Fig. 9** and **Fig. 10.**





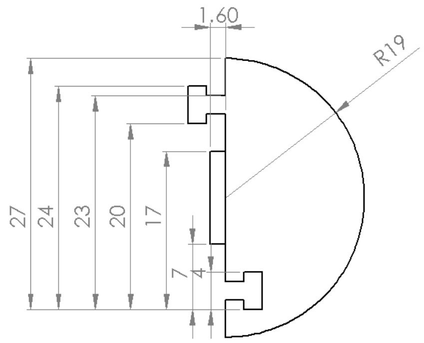
Top view of lower part A

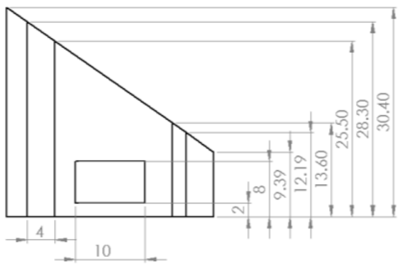
Right view of lower part A



Front view of lower part A

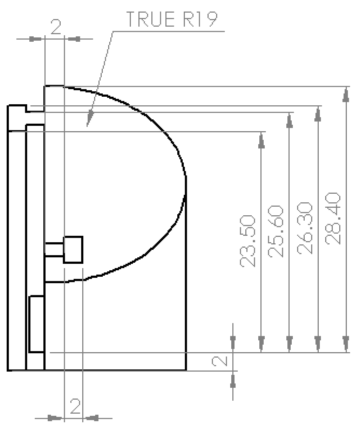
Fig. 9. Drawings of part A of the shear compression jig assembly illustrating geometrical parameters used in the study. Dimensions in mm.





Front view of lower part B

Left view of lower part B



Front view of lower part B

Fig. 10. Drawings of part B of the shear compression jig assembly illustrating geometrical parameters used in the study. Dimensions in mm.

References

[1] "Crashworthiness Data System Overview | National Highway Traffic Safety Administration (NHTSA)", *One.nhtsa.gov*, 2019. [Online]. Available: https://one.nhtsa.gov/Data/National-Automotive-Sampling-System-(NASS)/NASS-Crashworthiness-Data-System. [Accessed: 14- Jun- 2018].

[2] Severson, Kristine & Tyrell, David & Rancatore, Robert. (2005). Crashworthiness Requirements for Commuter Rail Passenger Seats.

[3] A. Borhana, H. Ali and M. Tamin, "Large Strain Shear Compression Test of Sheet Metal Specimens", Experimental Mechanics, vol. 53, no. 8, pp. 1449-1460, 2013.

[4] "Sheet metal", *En.wikipedia.org*, 2018. [Online]. Available: https://en.wikipedia.org/wiki/Sheet_metal. [Accessed: 07- Jun- 2018].

[5] R. Boger, R. Wagoner, F. Barlat, M. Lee and K. Chung, "Continuous, large strain, tension/compression testing of sheet material", International Journal of Plasticity, vol. 21, no. 12, pp. 2319-2343, 2005.

[6] M. Vural, G. Ravichandran, and D. Rittel, "Large strain mechanical behavior of 1018 cold-rolled steel over a wide range of strain rates", Metallurgical and Materials Transactions A, vol. 34, no. 12, pp. 2873-2885, 2003.

[7]J. Choung and S. Cho, "Study on true stress correction from tensile tests", *Journal of Mechanical Science and Technology*, vol. 22, no. 6, pp. 1039-1051, 2008.

[8] D. Rittel, S. Lee and G. Ravichandran, "A shear-compression specimen for large strain testing", Experimental Mechanics, vol. 42, no. 1, pp. 58-64, 2002.

[9] M. Vural, A. Molinari and N. Bhattacharyya, "Analysis of Slot Orientation in Shear-Compression Specimen (SCS)", Experimental Mechanics, vol. 51, no. 3, pp. 263-273, 2010.

[10] R. Hill, "A new method for determining the yield criterion and plastic potential of ductile metals", Journal of the Mechanics and Physics of Solids, vol. 1, no. 4, pp. 271-276, 1953.

[11]S. Kalidindi, A. Abusafieh and E. El-Danaf, "Accurate characterization of machine compliance for simple compression testing", *Experimental Mechanics*, vol. 37, no. 2, pp. 210-215, 1997.

[12] L. Meyer, S. Abdel-Malek and N. Herzig, "Experimental Methods for Characterizing of Sheet Metals at High Strain Rates", Key Engineering Materials, vol. 473, pp. 474-481, 2011.

[13] M. Schikorra, R. Govindarajan, A. Brosius, and M. Kleiner, "Springback Analysis of Sheet Metals Regarding Material Hardening", Advanced Materials Research, vol. 6-8, pp. 721-728, 2005.

[14]T. Ishiguro, Y. Yoshida, N. Yukawa, and T. Ishikawa, "Deformation Analysis of Shearing Process Using Results of Notched Round Bar Tension Test", *MATERIALS TRANSACTIONS*, vol. 50, no. 7, pp. 1671-1677, 2009.

[15] Ling Y (1996) Uniaxial true stress-strain after necking. AMP J Technology 5:37–48

[16] "Finite element method", En.wikipedia.org, 2018. [Online]. Available: https://en.wikipedia.org/wiki/Finite_element_method. [Accessed: 07- Jul- 2018].