Experimental and numerical investigations on failures due to plasticity in thin walled structures

A Jamil, A H Khan, S Ali

Department of Mechanical Engineering,
Z.H.C.E.T, Aligarh Muslim University, Aligarh-202002 (U.P.), India

azharjamil@zhcet.ac.in

Abstract. Plasticity is one of the prominent cause of failures in metallic materials and with the continuous efforts of mankind in exploring domains of high uncertainty, a thorough understanding of structural behaviour under extreme conditions is quite necessary. Particularly, in the case of thin walled structures, where geometrical and physical nonlinearities are quite significant and require sophisticated techniques for failsafe design. In this work, a numerical model was developed in the commercial finite element code of ABAQUS®, after experimentation of 6150 cold rolled steel used in thin walled pressure vessels. For this, proper experimentation quest was carried out based on the ASTM E8/E8M standard and dog-bone specimen were made with both planar and circular cross-sections. Ramberg-Osgood plasticity equation was employed for the smoothening of the test data along with the extrapolation of stresses at higher strain level. A tabular plasticity model was thereby made from the smoothened and extrapolated stress-strain experimental data, and implemented in the material definition module of ABAQUS®. Both plane stress CPS4R and plane strain CPE4R elements were employed in planar dog-bone specimen and C3D8R elements were used for three dimensional planar and circular dog-bone specimen. The results from the simulation were in excellent agreement with the experimental data with the plane stress analysis slightly underestimating, the plane strain analysis slightly overestimating the load and the three dimensional analysis lying in between the two.

Keywords Tensile test, plasticity, Ramburg Osgood plasticity model, FEA

1. Introduction

Cold rolled steel is widely used in industry for storage and transportation of liquids and gases and often these are susceptible to failure due to unavoidable peak loading in the lifetime of these structures. The test has performed to find out the point of failure in industrial components that occurs due to the internal forces such as internal fluid pressure into the thin walled pressure vessels. It occurs due to the hoops stress into components consequently plastic deformation initiated into the surfaces of components. Previous studies have performed tensile test for determining the ductility of materials and determine the plasticity criterion according to simulation analysis. Investigations on necking and post-necking behaviour of uniaxial stretched tensile specimen were performed by Lars *et.al.* [1], using an enhanced *2D* plane stress finite element model. Tensile tests were carried by Jain *et.al.* [2] on aluminium specimen for the determination of ductile fracture criterion. Weng *et.al.* [3], worked on ductile materials and presented standards, using finite element analysis. Experimental and numerical investigations were done by Eduardo *et.al.* [4] inorder to study the mechanical behaviour under tension of SAE 1045 steel sheet specimens. In the area of shear failure, Komori [5], implemented ductile failure criterion on finite element nodes. Fracture criterion of various materials along with failure analysis was done by Yang *et al.* [6]. *FEA* modelling of deformation behaviour of steel specimen was done by Banerjee *et al.* [7], under uniaxial tension test. Prediction of strains and deformation of model were compared with experiment data. The role of imperfection of necking behaviour in *FE* modelling results of uniaxial test was discussed. Chen [8], worked on tensile tests of 7075 aluminium obtaining true stress-strain values along with finite element simulation analysis. In the analysis various ductile fracture criteria were investigated revealing different levels of accuracy in the simulation. Edgar *et al.* [9], analysed the tensile test of mild steel using finite element method using a computer program called ABAQUS/CAE®. The results have been as the stress-strain graph. It is understood that the mild steel follows the Hooke's Law *i.e.*, stress is directly proportional to strain. After the linear region in the graph, there occurs necking on the sample and finally it breaks. Ahmed *et al.* [10], investigated the different stresses of thin walled pressure vessel. Equations of static equilibrium were used to determine the normal stresses and circumferential or hoop stresses.

2. Methodology

In this work, a tensile test of 6150 cold rolled steel was performed used in thin walled pressure vessels which is widely used in industry for storage and transportation of liquids and gases. For this proper experimentation quest was carried out based on the ASTM E8/E8M standard dog-bone specimen of both planar and circular cross-sections and determine the plasticity criterion according to a simulation analysis, as seen in Figure 1.

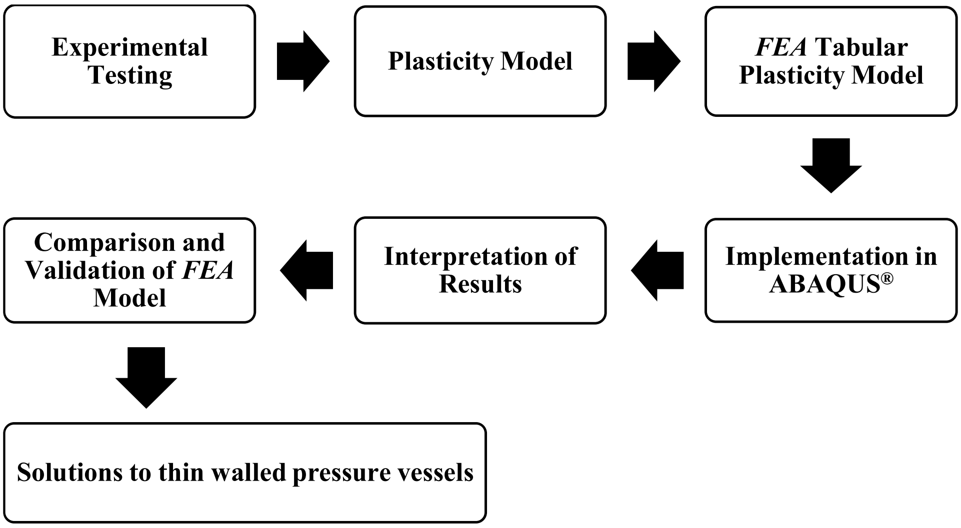
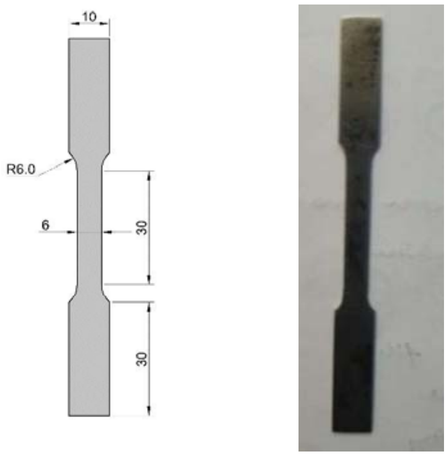


Figure 1. Systematic flow chart of methodology

3. Experimental Testing

Tensile tests were performed in accordance with ASTM E8/E8M for both rectangular and circular section specimen made up of 6150 cold rolled steel. The dog-bone specimen were designed and fabricated in the in-house workshop facility, the schematics of which is shown in Figure 2(a) and one of the fabricated specimen can be seen in Figure 2(b). Final dimensions of which are tabulated in Table 1. Experimental tests were finally performed on the in house universal testing machine (UTM) having a load capacity of 50kN as shown in Figure 3.



(a) Schematics of specimen *(b) Fabrication of test specimen*

Figure 2. Dog bone rectangular specimen based on ASTM/E8 standard

Table 1. Dimension of specimen

S. No.	Descriptions	Dimension (mm)
1	Gauge length	30
2	Gauge width	6
3	Grip length	30
4	Thickness	1

The mechanical properties of the materials so obtained are enlisted in Table 2. The specimen showed the typical stress-strain behaviour and post-ultimate strength it fractured as seen in Figure 4.

Table 2. Mechanical properties of 6150 cold rolled steel

Density	7.85 g/cm3
Young's Modulus	210 Gpa
Poison ratio	0.3

Figure 3. Universal testing machine

Figure 4. Fracture specimen

IOP Conf. Series: Materials Science and Engineering **396** (2018) 012065 doi:10.1088/1757-899X/396/1/012065

4. Development of the Plasticity Model

The data so obtained from the tests was averaged and was utilised for the plotting of load *vs.* extension. For the planar rectangular section the typical loads *vs.* deformation phenomenon is shown in Figure 5(a) whereas the loads *vs.* deformation for circular cross section is represented in Figure 5(b). From these the value of engineering stress-strain and true stress-strain was evaluated using the basic formulation of stress and strain represented in Equations 1 and 2, as seen in Figures 5(c) and 5(d) respectively for specimen with rectangular and circular cross-sections.

$$\varepsilon = \ln(1 + e) \tag{1}$$

$$\sigma' = \sigma(1 + e) \tag{2}$$

Where ε is known as true strain, 'e' is engineering strain, σ' is true stress and σ is engineering stress. This was further utilised for the calculation of plastic strain using the standard analytical relation represented in Equation 3.

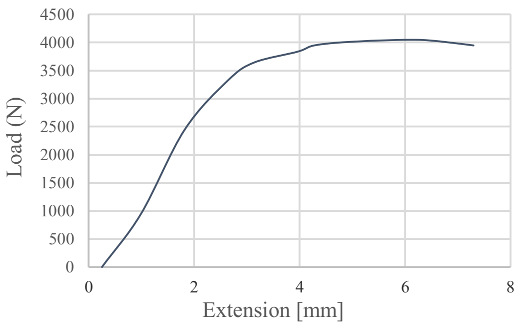
$$\varepsilon_p = \varepsilon - \frac{\sigma}{E} \tag{3}$$

4.1 Ramberg Osgood Plasticity Model

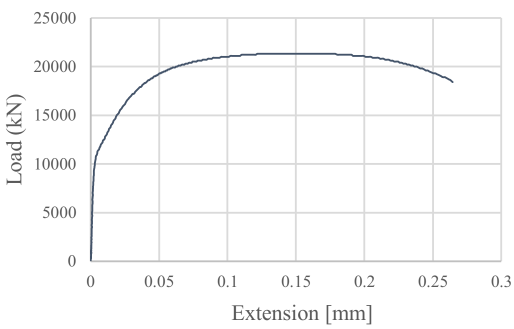
Ramberg Osgood plasticity model was implemented for the smoothening of the test data along with the extrapolation of stresses at higher strain level in accordance with Equation 4. This relationship is exponential and is used to describe the plastic strain in a material. It is especially useful for metals that harden with plastic deformation, showing a smooth elastic-plastic transition.

$$\frac{\sigma}{\sigma_0} = a\left(\frac{\varepsilon}{\varepsilon_0}\right)^n \tag{4}$$

σ_0 and ε_0 are true stress and logarithmic strain corresponding to 0.002 plastic strain and a and n are constants of the power law.

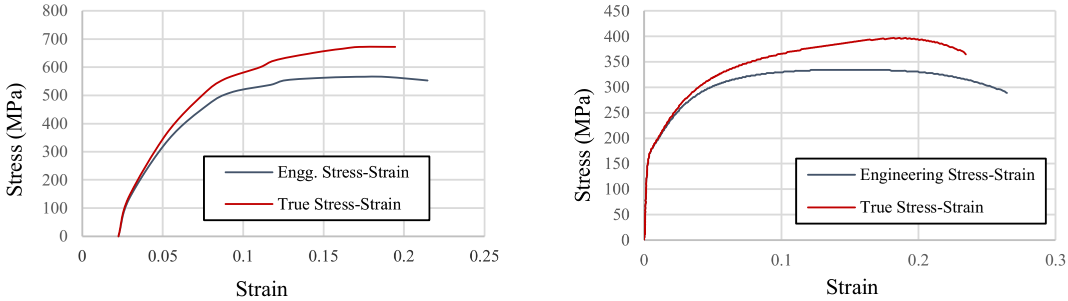


(a) Load vs. Extension curve for rectangular specimen



(b) Load vs. Extension curve for circular specimen

IOP Conf. Series: Materials Science and Engineering **396** (2018) 012065 doi:10.1088/1757-899X/396/1/012065



(c) Stress strain curve for rectangular specimen *(d) Stress strain curve for circular specimen*

Figure 5. Load-Extension and Stresses-Strains graphs

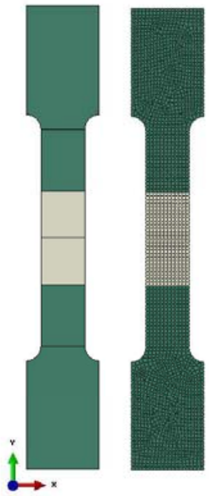
Finally the plastic model was developed after the extrapolations, represented in Table 3, and was finally implemented in the material definitions of ABAQUS®.

Table 3. Final plasticity model

Yield Stress [MPa]	Plastic Strain
1	0
136.2499876	0.0001
361.4609244	0.395023082
502.8731463	0.538420172
558.2141003	0.58376253
600.6772615	0.61560292
624.1550544	0.632254219
648.3509722	0.648771894
666.133737	0.660523157
672.8329214	0.664868961
671.8650519	0.664243779

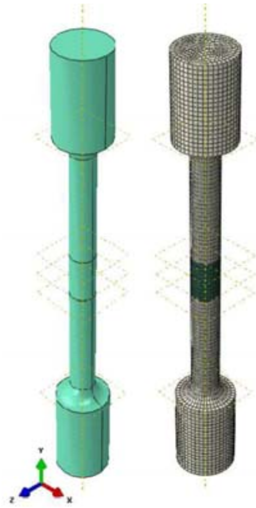
5. Modelling and Simulation

A numerical model of dog bone specimen of both rectangular and circular cross section was developed in the commercial finite element code of ABAQUS®. Rectangular cross-sectional model was modelled in both two dimensions (with 4 noded CPS4R and CPE4R elements corresponding to plane stress and plane strain conditions) and three dimensions using 8 noded C3D8R elements, as seen in Figure 6 (a) and 6(b). However, on the other hand, circular cross-sectional dog-bone specimen were modelled only in three dimensions using the 8 noded C3D8R elements, as evident from Figure 6(c). Material model so development was inserted in property module along with the value of modulus of elasticity and poison ratio. In interaction module, two coupling corresponding to two end condition of specimen. On one end was perfectly clamped with zero degree of freedom whereas the upper end is provided displacement load of 5mm.



(a) Two dimensional model of rectangular section specimen

(b) Three dimensional model of rectangular section specimen

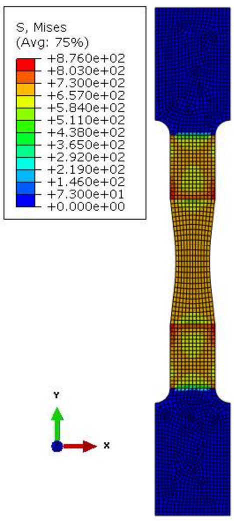


(c) Three dimensional model of circular section specimen

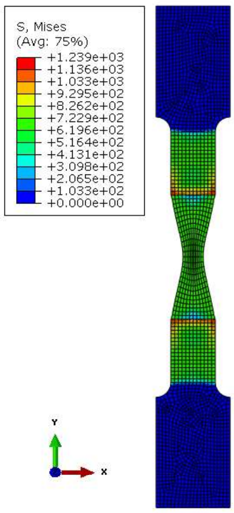
Figure 6. Modelling of rectangular and circular section tensile specimen.

6. Results and Discussion

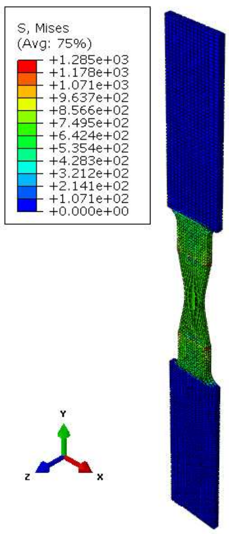
The simulation of rectangular dog bone reveal a maximum value of Von Mises stress equal to 876.0 MPa at the mid-section in plane stress conditions and 1239.0 MPa for plane strain condition at the end of 5mm displacement load, as seen in Figure 7(a) and (b) respectively. In three dimensional analysis the maximum attained level of Von-Mises stress was 1285.0 MPa corresponding to the same 5mm displacement loading, as seen in Figure 7(c). The corresponding strain levels were recorded to be 0.657 and 0.815 for plane stress and plane strain conditions as seen in Figures 8(a) and (b) respectively and 0.991 for the three dimensional analysis, depicted in Figure 8(c).



(a) Plane Stress conditions

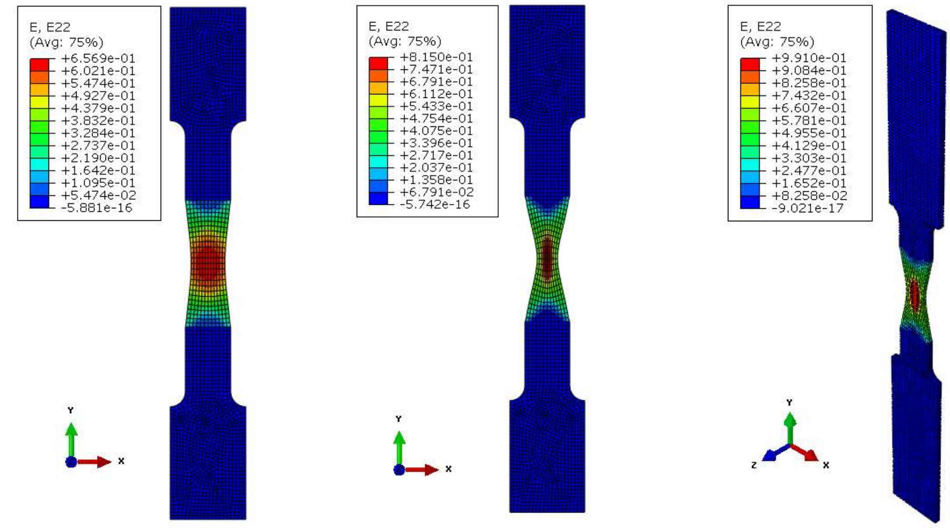


(b) Plane Strain conditions



(c) 3D Specimen

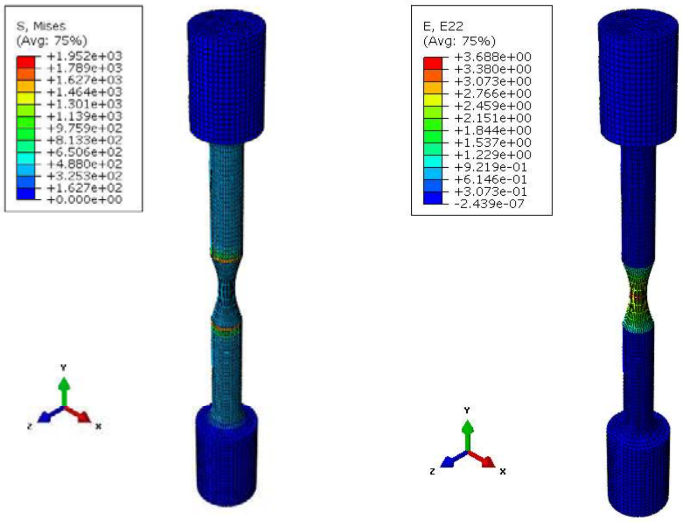
Figure 7. Von Mises Stress distribution of rectangular cross section tensile specimen.



(a) Plane Stress conditions *(b) Plane Strain conditions* *(c) 3D Specimen*

Figure 8. Strain distribution in principal direction of rectangular cross section tensile specimen.

For the circular cross section specimen, the maximum value of Von Misses stress was observed to be 1952.0 MPa and a corresponding strain of 3.66 in the principal direction corresponding to the same 5.0mm displacement load.



***(a)** Von Mises stress distribution* ***(b)** Strain in principal direction*

Figure 9. Stress-strain contours of circular cross section tensile specimen.

The results obtained from *FEA* model were compared with the experimental results and showed an excellent agreement with each other as can be seen in Figures 10 and 11 corresponding to rectangular and circular sections respectively. The results reveal close resemblance of the experimental data with plane stress and three dimensional analysis. This is quite expected for thin sections are more inclined towards plane stress conditions. Plane strain expresses a stiffer section owing to bulk material conditions of the interior

IOP Conf. Series: Materials Science and Engineering **396** (2018) 012065 doi:10.1088/1757-899X/396/1/012065

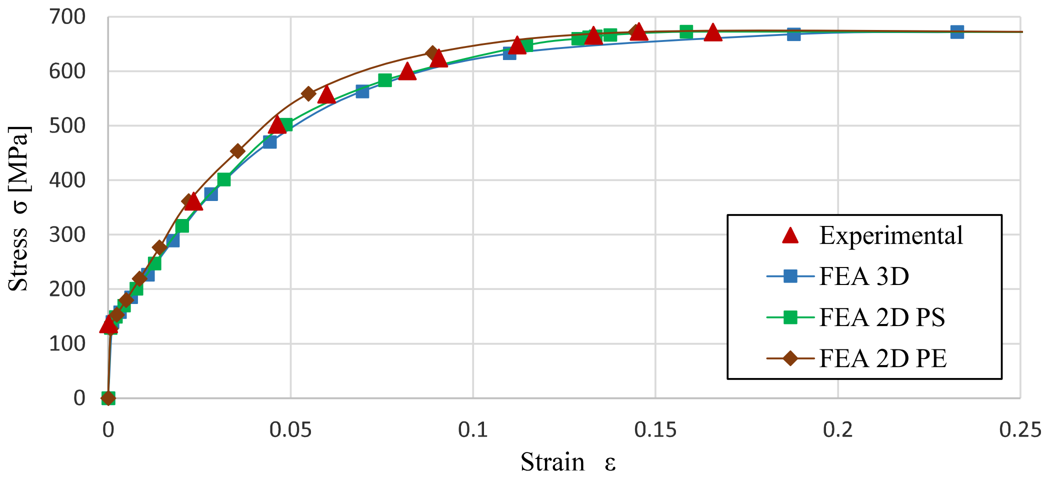


Figure 10. Stress vs. Strain graph for rectangular specimen

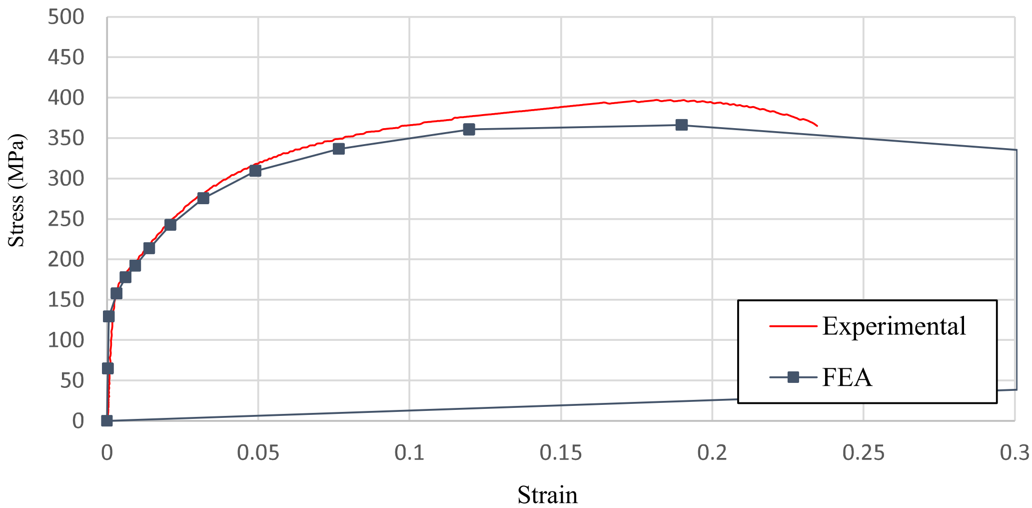


Figure 11. Stress vs. Strain graph for circular specimen

7. Conclusion

The material model was made with the help of the experimental test data. This model was implemented in commercial *FEA* code of ABAQUS® and simulating the experimental quest. Various conclusions can be drawn from work and with the validation of the approach, it can be extended to industrial components of high geometric complexities. The results from rectangular section reveal the closeness of the plane stress conditions to the experimental results which were also in accordance to the three dimensional analysis. Plane strain conditions in such a case slightly overestimates the stress distributions and thin sections are therefore best modelled in plane stress conditions when three dimensional analysis is quite challenging or not possible. The methodology is therefore proposed in the simulation of plasticity in industrial components like thin walled pressure vessels where hoop stress plays an important role in design considerations.

IOP Conf. Series: Materials Science and Engineering **396** (2018) 012065 doi:10.1088/1757-899X/396/1/012065

8. References

[1] Mikkelsen, Lars Pilgaard. "Necking in rectangular tensile bars approximated by a 2-D gradient dependent plasticity model." *European Journal of Mechanics-A/Solids* 18.5 (1999): 805-818.
[2] Jain, M., J. Allin, J., D. J. Lloyd, D.J., 1999. Fracture limit prediction using ductile fracture criteria for forming of an automotive aluminum sheet. International Journal of Mechanical Sciences 41, 1273-1288.
[3] Weng, T.L., Sun, C.T., 2000. A study of fracture criteria for ductile materials. Engineering Failure Analysis 7, 101-12
[4] Cabezas, Eduardo E., and Diego J. Celentano. "Experimental and numerical analysis of the tensile test using sheet specimens." *Finite Elements in Analysis and Design* 40.5-6 (2004): 555-575.
[5] Komori, K., 2005. Ductile fracture criteria for simulating shear by node separation method. Theoretical and Applied Fracture Mechanics 43, 101–114
[6] Banerjee, Dilip. "Finite Element Modeling of Deformation Behavior of Steel Specimens under Various Loading Scenarios Dilip Banerjee, Mark Iadicola, Adam Creuziger, and Timothy Foecke."
[7] Yang, X., Chen, C., Hu, Y., Wang, C., 2005. Damage analysis and fracture criteria for piezoelectric ceramics. International Journal of NonLinear Mechanics 40, 1204 -1213.
[8] Analysis and experiment of 7075 aluminum alloy tensile test.et al (2014).
[9] Jose, Edgar, and Tom Anto. "Analysis of Tensile Test of Mild Steel using Finite Element Method." *International journal of innovation in engineerinsg and technology* 5.4 (2015): 247-251.
[10] Ibrahim, Ahmed, Yeong Ryu, and Mir Saidpour. "Stress analysis of thin-walled pressure vessels." *Modern Mechanical Engineering* 5.1 (2015): 1-9.