



TENSILE TEST AND FEA CORRELATION OF ABS PLASTIC

1Tusharkumar B. Jawalkar, 2Prashant A. Shirke, 3Sanjay T. Satpute

1S. Y. M. Tech. Student, 3 Professor,

Automobile Engineering Department,

Rajarambapu Institute of Technology, Sakharale, Dist. Sangli, Maharashtra, India

2CAE analyst, Johnson Controls Automotive Ltd

Email: 1tushar.jawalkar17@gmail.com, 2prashant.shirke@jci.com, 3sanjay.satpute@ritindia.edu

Abstract

This paper presents tensile behaviour of ABS polymer. Tensile deformation behaviour of Acrylonitrile Butadiene Styrene (ABS) was experimentally investigated, following a practice guideline by ASTM. Tests were performed for various strain rates. Test specimen was preprocessed using CAE software Hypermesh. Results obtained by experimentation were validated using analysis software Abaqus and Ls-Dyna. After ultimate stress, tested Stress – Strain curve had negative slope. During preparation of material card it was a problem. As solver doesn't allow negative slope. Curve was smoothening by establishing a procedure. There was problem with Young's modulus from Stress vs. Strain curve of test results. Some assumptions were made for Young's modulus from test results. It was seen that, specimen thickness contract at higher rate than its width. As strain rate for tensile test increases, its yield stress also increases. Tensile strain rate was maximum in necking region.

Keywords: Tensile test, CAE correlation, Hypermesh, Abaqus, Ls-Dyna.

1. Introduction

Engineering plastics are used in instrument panels, interior trim, fuel tanks and other vehicle applications. Due to their viscoelastic nature, plastics exhibit important rate dependence in their stress–strain responses. The strain rate dependent stress–strain curves of these materials are mandatory input in dynamic finite element (FE) analysis for crashworthiness prediction [9].

The experimental technique to create data at these strain rates is a research topic of practical importance [9]. Modelling and predicting the behaviour of these structures made of polymers require the knowledge of mechanical response of the materials [1].

To obtain valid stress–strain data in a material test, the specimen should be in a state of stress equilibrium, and undergo homogeneous deformation in the gage section. Usually, necking in the polymeric specimen arises at relatively small strain, which results in the inhomogeneous deformation of polymers [1].

2. Material and Specimen

Polymeric material used in this study was Acrylonitrile Butadiene Styrene (ABS). Material was specified according to ASTM D638, Type I [14]. It has variation in thickness from 3.2 mm to 3.8 mm across the length. The

specimen has dumbbell shape. The straight gauge section has length of 50mm and width of 13mm [14]. The test dumbbells were injection moulded by the material supplier. The nominal thickness of the specimens was 3.2 mm. Figure 1 shows the geometric dimensions of the test specimen as per ASTM D638 standards [14].

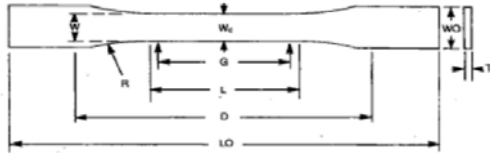


Fig. 1 Geometry and Dimensions of specimen[14].

Table 1. Dimensions for specimen as per ASTM D638 [14].

| Dimensions | Type I Specimen | Tolerances |
|------------------------------|-----------------|---------------|
| W - Width of narrow section | 13 | +0.5(-0.02) |
| L - Length of narrow section | 57 | +0.5(-0.02) |
| W.O - width overall, min | 19 | +1.18(-0.125) |
| L.O - Length overall | 165 | No max |
| G - Gauge length | 50 | +0.25(+0.010) |
| D - Distance between grips | 115 | +5(+0.2) |
| R - Radius of fillet | 76 | +1(-0.01) |
| T - Thickness | 3.2 | +0.4 |

3. Experimentation

ABS dumbbells were tested with UTM machine. Test was conducted according to ASTM D638 test guidelines, which is standard for tensile test of Plastics [14]. Dumbbells were tested with variable strain rate of 10mm/min, and 100mm/min. For each strain rate three specimens were tested. Results obtained are Load (N) and Deflection (mm) for each specimen. From results found calculation performed for Engineering Stress, Engineering Strain, True Stress, and True Strain. Experimentation gives load (N) and Deflection (mm) only. This curve is then converted in to Engg. Stress vs. Engg. Strain and True Stress vs. True Strain curve. Once Young’s modulus was finalized, True stress vs. Plastic strain curve was obtained.

A. Calculations

$$Engg.Stress(s) = \frac{Load(N)}{Cross\ Sectional\ Area(Sq.mm)} \quad \text{Equation 1}$$

$$Engg.Strain(e) = \frac{Change\ in\ length}{Original\ Length} \quad \text{Equation 2}$$

$$True\ Stress(\sigma) = Engg.Stress(s)[1 + Engg.Strain(e)] \quad \text{Equation 3}$$

$$True\ Strain(\epsilon) = \ln[1 + Engg.Strain(e)] \quad \text{Equation 4}$$

$$Plastic\ Strain = True\ Strain - \frac{True\ Stress}{Young's\ Mod.} \quad \text{Equation 5}$$

For calculation of engineering stress cross sectional area of each specimen after break was used. Then calculated Stress for each specimen and averaged it. Curves were plotted with average engineering stress values obtained by calculation.

B. Material Card Preparation

In material card of Hypermesh with profile Abaqus, material type is MATERIAL and card image is ABAQUS_MATERIAL [13].

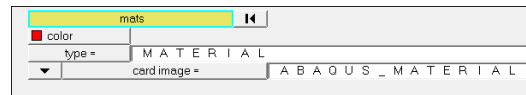


Fig. 2. Material card used in Hypermesh (Abaqus).

In properties of Hypermesh with profile Abaqus, card image used is SOLIDSECTION. Property type is SOLID SECTION [13].

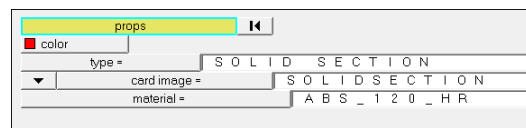


Fig. 3. Property card used in Hypermesh (Abaqus).

In material card of Hypermesh with profile Ls-Dyna Keyword 971, material type is ELASTIC-PLASTIC and card image is MATL24 [12].

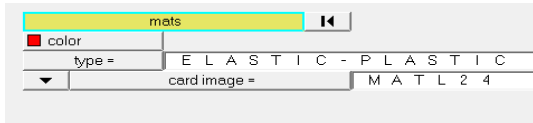


Fig. 4. Material card used in Hypermesh (Ls-Dyna).

In material card of Hypermesh with profile Ls-Dyna Keyword 971, material type is VOLUME and card image is SectSld [12].

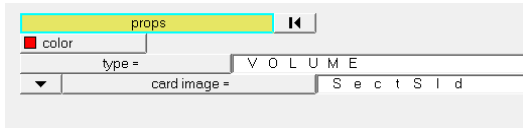


Fig. 5. Property card used in Hypermesh (Ls-Dyna).

For simulation result calculations solver demands plastic strain and True Stress values as input [13, 12]. If given curve have negative slope, its error during performing calculation for Abaqus and Ls-Dyna[12,13]. To exclude this error procedure was established. After ultimate stress where curve starts negative slope those stresses were eliminated performing trial and error to establish smooth curve with positive slope.

Density of ABS is 1.05×10^{-09} tonns/mm³, Poisson's ratio (Nu) is 0.35 for ABS plastic. There was problem with Young's modulus from experimental data. Initially tried young's modules from test results, simulation results obtained with this young's modulus were not as accurate as required. So assumption was made to use value of young's modules for simulation, by performing procedure and made trial and errors on it. A line was drawn at 0.2% strain which is parallel to elastic limit of Stress vs. Strain curve. This procedure gives value of Young's Modulus as 3777.456 MPa.

4. Results

Test results obtained are Load(N) and Deflection(mm). Curves for test results are as:

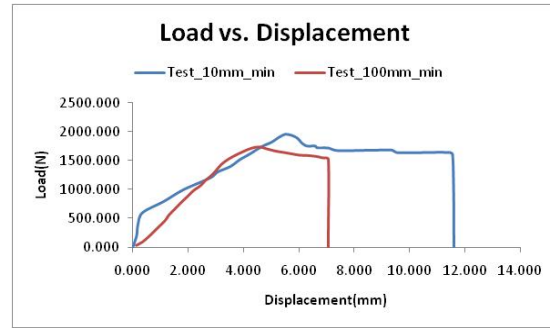


Fig. 6: Load vs. Deflection (Test Results).

From test data of Plastic material, as strain rate is increased ten times Yield value and its elastic limit increases. But total displacement of plastic at break is about half than earlier strain rate.

Data obtained from test were then calculated and converted into Engg. Stress, Engg. Strain, True stress, True strain, and Plastic strain. Curve plotted are with average of all specimen tested for each strain rate.

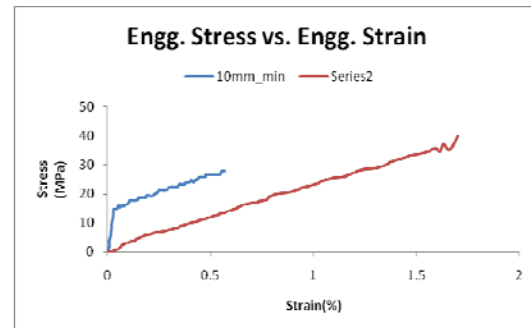


Fig. 7: Engg. Stress vs. Engg. Strain.

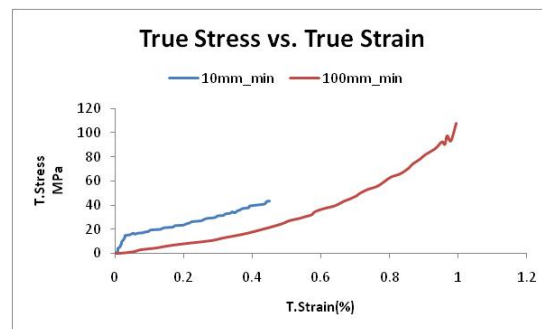


Fig. 8: True stress vs. True strain.

In the True stress vs. True strain curve, curve shows negative slope after ultimate stress value. As negative slope values are not required by solver for calculations, values of Engg. Stress vs. Engg. Strain and True Stress vs. True Strain curves are not considered and

also not plotted in above graphs (Fig.7 and Fig.8). Abaqus solver needs True stress and Plastic strain values as input. True stress vs. Plastic strain curve also shows negative slope. This was crucial problem during simulation. To resolve this issue, technique was developed. From the observation of curve, it is clear that curve shows negative slope after ultimate stress value. In actual testing, its region where necking formation starts in the test specimen. For the analysis purpose this was very important region of True stress vs. Plastic strain curve.

From the observation of curve, it is clear that curve starts bending after yield stress. Specimen enters in plastic phase from elastic phase. This bending of curve is still considerable, because it shows still positive curve.

Based on trial and error technique, trials were made with region of curve from ultimate stress to breaking point. Trials were made such that curve shows constant slope/positive slope and it will not affect formation of necking region in simulation.

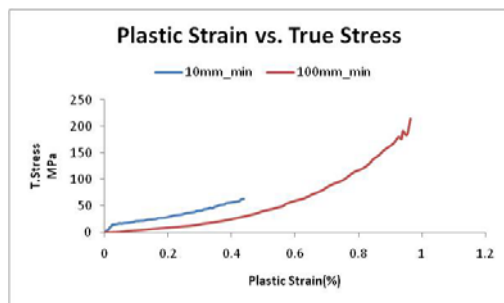


Fig. 9: True Stress vs. Plastic Strain.

Engg. Stress vs. Engg. Strain, True stress vs. True strain and True Stress vs. Plastic Strain curves shows that as strain rate is increasing, its failure value i.e. yield value is also increasing.

A. Correlation of Test and CAE results

Test specimen was first meshed in preprocessor Hypermesh, with profile Abaqus (Standard3D) and for Ls - dyna with profile keyword971. Specimen was meshed with solid mesh (Hexa elements) with element size 0.5, 1 and 1.5mm. After it, counted number of nodes and elements for each type of mesh. From the

simulation results noted values of max. stress, strain and displacement for each type of mesh. Results showed that, it is optimum to use 1mm mesh size. Mesh size of 0.5 mm gives better results but it is time consuming for calculation.

One side of dumbbell was constrained with zero degree of freedom and at another side tensile load was applied with only one degree of freedom, allowing specimen to move in only X - direction. Constraints were applied up to 25 mm from both ends as per ASTM D638 standards.

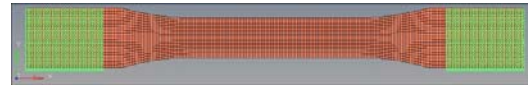


Fig. 10: 1mm element size mesh of dumbbell specimen.

Specimen was simulated in Abaqus and Ls-dyna. CAE results in Abaqus shows failure of specimen as actual test specimens.

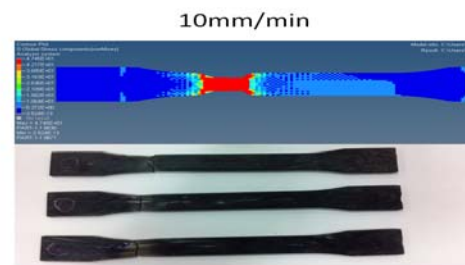


Fig. 11: CAE and Test failure of specimen at 1mm/min strain rate.

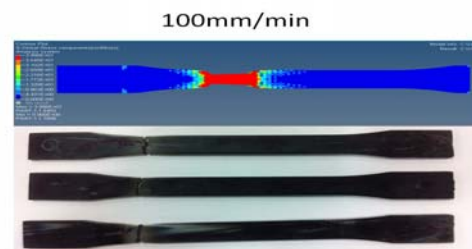


Fig. 12: CAE and Test failure of specimen at 10mm/min strain rate.

From above (Fig.11 and Fig.12) failure figures, it's clearly seen that as test specimen fails, CAE failure is also in the same region of specimen. CAE failure at each strain rate shows that tensile strain rate is maximum in necking region. As in above (Fig.11 and

Fig.12) failure correlation is matched, specimens were simulated in Abaqus and Ls-dyna. In both solver failure is analysed and plotted results were also matched.

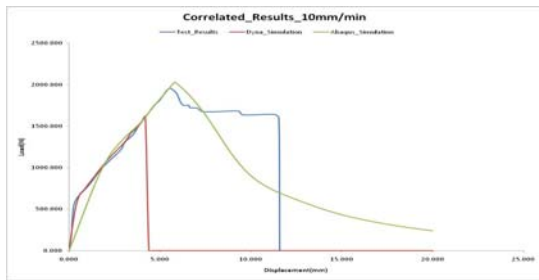


Fig. 13 Correlation of CAE and Test results of specimen at 10mm/min strain rate.

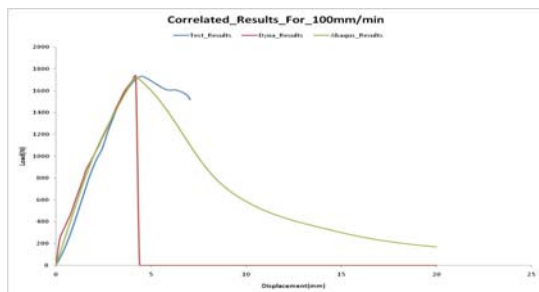


Fig. 14 Correlation of CAE and Test results of specimen at 100mm/min strain rate.

Above graphs (Fig. 13 and Fig.14) shows simulation results obtained in Abaqus and Ls-dyna and Test results obtained are correlated up to Yield point in both cases. From CAE simulation and experimental test it was observed that specimen thickness contract at higher rate than its width.

Acknowledgements

This research work was sponsored by Johnson Controls Automotive Ltd. Pune. I give my sincere thanks to Mr. Prashant Shirke, Mr. Pankaj Wankhade, Mr. Saurabh Joshi, Mr. Amit kumar Singh, Mr. Pavan Raj, Mr. Ranganath L, Prof. S.T. Satpute and entire team of CAE at Johnson Controls Automotive Ltd. Pune without their support analysis and correlations were not possible.

References

- [1] Qin-Zhi Fang, T.J. Wang, Hui-Min Li, Large tensile deformation behavior of PC/ABS alloy, *Polymer* 47, pp.5174–5181, (2006).
- [2] Z.N. Yin, T.J. Wang, Hui-Min Li, Deformation response and constitutive modeling of PC, ABS and PC/ABS alloys under impact tensile loading, *Materials Science and Engineering A* 527), pp.1461–1468, (2010).
- [3] ManSoo Joun, Jea Gun Eomc, Min Cheol Lee, A new method for acquiring true stress–strain curves over a large range of strains using a tensile test and finite element method, *Mechanics of Materials* 40, pp.586–593, (2008).
- [4] Qin-Zhi Fang, T.J. Wang, H.G. Beo, H.M. Li, Effect of cyclic loading on tensile properties of PC and PC/ABS, *Polymer Degradation and Stability* 93, pp.1422–1432, (2008) .
- [5] T.S. Srivatsan, Meslet Al-Hajri, J.D. Troxell, The tensile deformation, cyclic fatigue and final fracture behavior of dispersion strengthened copper, *Mechanics of Materials* 36, pp.99–116, (2004).
- [6] Matthias Weiss, Bernard Rolfe, Peter D. Hodgsona, Chunhui Yang, Effect of residual stress on bending of aluminum, *Journal of Materials Processing Technology* 212, pp.877–883, (2012).
- [7] Isabel Duarte, Matej Vesenjok, Lovre Krstulovic´-Opara, Dynamic and quasi-static bending behavior of thin-walled aluminum tubes filled with aluminum foam, *Composite Structures* 109, pp. 48–56, (2014).
- [8] J.A. Paquette, S. Kyriakides, (2006), Plastic buckling of tubes under axial compression and internal pressure, *International Journal of Mechanical Sciences* 48 ,855–867.
- [9] Xinran Xiao, Dynamic tensile testing of plastic materials, *Polymer Testing* 27, pp.164–178, 2008.
- [10] Yang Hao, Wu Xiao-Xiang, San Xing-Yuan, Shen Li, and Zhu Xin-Kun ,Influence of injection parameters and mold materials on mechanical properties of ABS in plastic injection molding, *Procedia Engineering* 36, pp.307 – 315, (2012)
- [11] Egor P. Popov, “Engineering mechanics of solids”, Pearson Education, Inc.
- [12] Ls – Dyna manual version 971, Livemore software Technology Corporation.
- [13] Abaqus6.14 online documentation © Dassault systems 2014.
- [14] Standard test method for tensile properties of plastics, ASTM International.