Property Modelling

Finite-element analysis of quasi-static characterisation tests in thermoplastic materials: Experimental and numerical analysis results correlation with ANSYS

A. Arriaga\textsuperscript{a,\,*}, J.M. Lazkano\textsuperscript{a}, R. Pagaldai\textsuperscript{a}, A.M. Zaldua\textsuperscript{a}, R. Hernandez\textsuperscript{a}, R. Atxurra\textsuperscript{a}, A. Chrysostomou\textsuperscript{b}

\textsuperscript{a}Materials Department, Lea-Arटibai Ikastetxea, Xemein Avenue 19, 48270 Markina-Xemein (Bizkaia), Spain
\textsuperscript{b}London Metropolitan Polymer Centre, North Campus, London Metropolitan University, 166-220 Holloway Road, London, UK

Received 7 September 2006; accepted 25 October 2006

Abstract

The use of structural calculation software based on finite-element analysis is nowadays a common practice when designing new industrial products processed from thermoplastic materials. In order to make an adequate prediction of the service behaviour of plastic components, it is necessary to carry out appropriate analysis when working with the software. This requires both the correct mechanical characterisation of the materials used for inputting the required properties in the calculation code, and the specification of the different solution characteristics.

In the present work, both areas have been studied in order to find a good correlation level between experimental mechanical test results in thermoplastic materials (principally two material types have been evaluated, a Polypropylene PP BE677AI from BOREALIS and a polycarbonate/acrylonitrile–butadiene-styrene PC/ABS T 45 from BAYER) and simulation of the same tests in the finite-element code ANSYS.

Initially, a series of conditions that can affect the quality of the material data input and, therefore, the simulation results are defined. These are the testing conditions of the plastic samples, the methods of measuring different strain values in the uniaxial tensile tests or the conversion from engineering measured data to “true” values that can be analysed in the software.

Next, two quasi-static validation tests are defined for the comparison of the simulation and experimental results: these are the 3-point bending test and the plate penetration test using semi-spherical darts. Following the bending and penetration experimental tests, simulation of the tests under the same conditions in ANSYS was conducted, taking into account the different variables that can change the results obtained and, therefore, the correlation with the physical tests. The accountable variables include the use of different element types in the simulation (solid, shell and plane axis-symmetric elements), the use of different friction coefficients between the plastic and metallic parts, or the use of different values for the elastic Poisson’s ratio.

From the results obtained, it can be seen that the correlation level found in both materials and both testing modes, i.e. bending and penetration, is good both in the shape of the response curves and the quantitative values, even at high strain levels. A conclusion that can be extracted from the work is that the use of a correct friction coefficient is fundamental for

\textsuperscript{\*}Corresponding author. Fax: +34 94 616 91 60.
E-mail address: aarriaga@leartik.com (A. Arriaga).

042-9418/$ - see front matter \textcopyright 2006 Elsevier Ltd. All rights reserved.
doi:10.1016/j.polymertesting.2006.10.012

Please cite this article as: A. Arriaga, et al., Finite-element analysis of quasi-static characterisation tests in thermoplastic materials: Experimental and numerical analysis results..., Polymer Testing (2006), doi:10.1016/j.polymertesting.2006.10.012
good correlation between the experimental and simulation results. On the other hand, there are additional points to consider such as the use of different element types or the conversion of engineering values to “true” values where, for the tests described and the testing conditions used, no excessive differences are observed between one method and another.

Keywords: Thermoplastic; Finite-element; Characterisation; Stress; Strain

1. Introduction

The use of calculation codes based on the finite-element method in the design phase of new industrial products has resulted in cost reduction relative to the manufacture of physical prototypes and “trial and error” tests. The general use software such as NASTRAN, ABAQUS or ANSYS permit the modelling of a part or assembly, the input of material mechanical properties, the solving of loading conditions and the visualisation and evaluation of the obtained results.

Nowadays, polymeric materials, and more specifically thermoplastic materials, have passed from applications where aesthetics were the principal function to strict structural requirements in sectors like automotive (substitution of metallic parts, passenger and pedestrian protection components, instrument boards, etc.) or appliances (mobile phones, TV-audio devices, etc.).

One of the most challenging considerations when modelling thermoplastic materials is the feeding of the calculation code, with the appropriate test data, in order to represent the materials’ mechanical behaviour and determine suitable material models for plastic components. Usually, the former is met using data from the traditional uniaxial tensile test, even though the yielding behaviour in tensile, compression and shear differs for these materials. These deformation modes should also be tested in conjunction with the use of a material model having a pressure-dependent yielding criterion [1]. Moreover, the vast majority of material models developed for finite-element analysis (FEA) software are directed at metals, and it is well known that plastics present special characteristics, such as a linear viscoelastic behaviour at small strains leading to non-linearity in the stress–strain curve, differing yielding in tensile and compression modes (the Von Mises yielding is not satisfied), anisotropic hardening and no volume preservation in plasticity.

These considerations need to be borne in mind, but from the practical engineering or industrial viewpoint the approximation of metal plasticity should not be rejected. In the present work, it is intended to determine the correlation level reached when contrasting the 3-point bending test (where tensile and compression stress zones are generated when straining the sample) and the dart penetration test (where biaxial stresses are generated in the central region) adopting the classical elastic–plastic material models (isotropic hardening with Von Mises criterion).

When performing the simulations, certain variables have been determined and their influence on the results obtained have been checked in order to develop a validated working methodology for future calculation in these types of material. The factors studied are as follows:

- Material input data, conversion from engineering to “true” values.
- Effect of the elastic Poisson’s ratio.
- Effect of the friction coefficient between metal devices and plastic samples.
- Different element types for simulating the tests.

2. The tensile test for the generation of material input data in the simulation code

Most of the structural simulation software with finite elements allows the input of material mechanical behaviour in the form of stress–strain points from uniaxial tensile testing. This is due to the fact that the yielding criterion commonly used, in addition to the plastic material manufacturers’ data, is directed to this straight and simple method of testing and plastic flow verification.

The force measuring systems (load cells, extensometer gauges), the strain measurement devices (video, laser, mechanical extensometers) and the sample clamping systems determine the quality of the data obtained from the test. In this respect, the effectiveness of using a laser-measuring device in relation to a mechanical clip-on extensometer has been considered.
2.1. Tensile data acquisition and the conversion to suitable measures for ANSYS

The engineering stress–strain values, denoted as \( e \), must be converted to “true” values when high plastic strain applications are simulated. ANSYS permits the input of stresses in Cauchy form and strains in logarithmic or Hencky form:

\[
\sigma_t = \sigma_e (1 + \varepsilon_t)
\]

(1)

where \( \sigma_t \) is the “true” Cauchy stress.

\[
\varepsilon_t = \ln(1 + \varepsilon_e)
\]

(2)

and \( \varepsilon_e \) is the logarithmic or Hencky strain.

This conversion supposes volume preservation during plastic deformation of the material, that is, the Poisson’s ratio of the material, \( \nu \), remains 0.5 (incompressible material). Moreover, it supposes that the distribution of stresses along the specimen is uniform, which is true in the case of no necking in the central region of the sample.

This conversion has historically been successfully used with metallic materials; however, when dealing with thermoplastic materials some uncertainties arise in terms of its usefulness. It has been proved experimentally that, with thermoplastic materials particularly, the presence of elastomeric base ingredients, as in for example ABS, Poisson’s ratio can vary from 0.4 in the elastic range of tensile deformation to 0.1 in the plastic region [2].

When variation of the Poisson’s ratio is required in converting engineering data, another relation can be used

\[
\sigma_t = \frac{\sigma_e}{(1 - \nu \varepsilon_e)^2}
\]

(3)

where \( \nu \) is the Poisson’s ratio of the material.

The experimental determination of Poisson’s ratio from a tensile test requires noting of both the longitudinal and transversal strains using a double clip-on extensometer system or recording the test by the use of a video camera device and analysing the recorded pictures for strain calculations.

If no experimental data is available, approximation of the variation of Poisson’s ratio during the tensile test can assume that it varies from 0.3 to 0.5 for most materials between the elastic and plastic range (in the plastic region, volume preservation is also considered). Due to this, the variation of the Poisson’s ratio can be represented along the stress–strain curve, depending on the values for the tangent and secant moduli, as can be seen in Fig. 1.

In this way, the value for the Poisson’s ratio during the tensile test can be calculated using the following equation:

\[
v = 0.5 - 0.2 \left( \frac{E_S}{E_0} \right).
\]

(4)

\( E_S \) is the secant modulus at each point along the tensile stress–strain curve.

\( E_0 \) is the tangent modulus in the initial stage of the stress–strain curve.

This equation could be used in conjunction with Eq. (3) in order to take into account the variation of Poisson’s ratio.

2.1.1. Corrections for the plastic zone from the maximum point in stress

From the maximum point in stress, the conversion from engineering data to “true” values is not straightforward. For most plastic materials, as in metals, “necking” begins at maximum load, which can be considered a point of tensile instability; when the slope of the curve reaches a critical stress value, “necking” takes place. This point can be determined using the so-called Considier’s construction [3].

Beyond this point, the engineering data conversion should be completed iteratively, simulating the tensile test and correcting the plastic part of the curve until adequately representing the force–displacement curve obtained in the tensile test.

The engineering stress at the maximum point is:

\[
\sigma_e = \frac{F_{max}}{A_0},
\]

(5)

\( F_{max} \) is the maximum load along the tensile test.

\( A_0 \) is the initial cross-sectional area of the specimen.

Fig. 1. Poisson’s ratio values depending on the tangent and secant modulus.
and the true stress:

\[ \sigma_t = \frac{F_{\text{max}}}{A_t} \]  

substituting \( F_{\text{max}} \),

\[ \sigma_t = \sigma_e \frac{A_0}{A_t} \]  

or

\[ \sigma_t = \sigma_e \varepsilon_e \]  

where \( \varepsilon \) is the true strain at the maximum load point.

Starting from the maximum load point, a series of corrections can be made depending on the form of the experimental engineering stress–strain curve:

**Correction 1:** The case of constant engineering stress.

Supposing that the engineering stress–strain curve follows the form represented in Fig. 2, up to the point of maximum load the classical conversion from engineering to “true” values is used, and from this point the formula for stress conversion is as follows:

\[ \sigma_t = \sigma_{\text{max}} e^{(\varepsilon-\varepsilon_{\text{max}})} \]  

**Correction 2:** The case of a plateau zone.

Additional exponents can be used for correcting different zones within the engineering stress–strain curve when using correction 2; the case of the presence of a plateau region beyond the maximum load point. The resultant curve following such a correction can be seen in Fig. 3.

\[ \sigma_t = \sigma_{\text{max}} e^{(\varepsilon-\varepsilon_{\text{max}})^2} \]  

**Correction 3:** Case of a more pronounced plateau zone.

\[ \sigma_t = \sigma_{\text{max}} e^{(\varepsilon-\varepsilon_{\text{max}})^3} \]  

In this study, the corrections described have been applied to PP and PC/ABS. Different simulations have been carried out modelling the dart penetration test and comparing the results obtained with different data conversion methods.

2.2. Experimental work: quasi-static uniaxial tensile tests in different materials

This work specially concerns two materials, a polypropylene from BOREALIS (PP BE677AI) and a polycarbonate/acrylonitrile–butadiene-styrene from BAYER (BAYBLEND T45). Tests have also been performed for comparison purposes with an ABS SINKRAL L320 from POLIMERI and a polyamide 66 with a 30% of glass fibre (VYDINE R530 from SOLUTIA).

The tensile tests have been carried out on a universal testing machine, HOUNSFIELD H25KS. A 5 kN load cell has been used for force measuring purposes, and a mechanical clip-on extensometer and laser have been used for strain measurements.

2.2.1. Strain measurements using different extensometer types

Dog-bone multipurpose specimens (according to ISO 3167) were utilised. The samples have a total length of 158 mm, a cross section of 10 × 4 mm² and a straight central length of 80 mm.
Three different techniques were used to measure the strains generated in the tensile testing direction:

- measurement with a laser extensometer,
- measurement with a mechanical clip-on extensometer,
- measuring the distance between specimen clamps.

Fig. 4 illustrates the results when ABS SINKRAL L320 from BOREALIS was tested under various conditions. Comparing the curves for tests at the same speed, but utilising different strain measurement techniques, it can be seen that the stiffness calculated when measuring distance variation between the clamps is lower than when using the laser.

When using the laser system, an initial length of 50 mm (tensile standard) was allowed between the marks on the sample. For the second system, a clamping distance of 115 mm was used, again according to ISO 527. In both cases, a programmed testing speed of 50 mm/min was utilised.

The laser and mechanical clip-on extensometers have also been compared for PP and PC/ABS, as shown in Fig. 5.

In general, good agreement was reached between the measured values using these two methods. The results presented in the rest of the article have been generated with the laser system.

2.2.2. Test data extraction for implementation in the calculation code

Five tensile specimens were tested per test condition and an intermediate curve selected for data analysis. From the generated stress–strain data, approximately 20 points were entered in the finite-element software ANSYS.

When entering a modulus value, a modulus that defines the initial linear zone of the stress–strain curve and determines the yield point of the material should be used, rather than a modulus representative of very small strains.

This proves to be quite complicated to define in plastic materials, being viscoelastic in nature they do not have a clear linear portion, nor either a pronounced yielding point or transition from linearity to non-linearity.

These reduced points (see Fig. 6) were entered into ANSYS in the true classical conversion format for all the stress–strain data points set (no correction beyond the maximum load point). As can be seen in Fig. 7, different approximations can be made to select an adequate modulus that correctly represents the material’s stiffness and that defines the yield point.

The use of a tangent modulus gives the definition of a low yielding stress in comparison with real material’s behaviour, so the resulting plastic strains in the component will be too high. The use of a secant modulus gives a low stiffness value and, if it is a decisive factor relative to the behaviour of the part analysed, that property is underestimated.
For an adequate representation of the material’s stiffness, the MELAS (Multilinear Elastic) material model can be used, but if the interest is in evaluating the plasticity behaviour or the charge-discharge conduct, a MISO (Multilinear Isotropic hardening) or a MKIN (Multilinear Kinematic hardening)

Fig. 5. Comparison of the stress–strain curves recorded in PC/ABS and PP using two different strain-measuring systems.

Fig. 6. Treatment example with PC/ABS for data input in ANSYS.
Fig. 7. Different approximations for the determination of a modulus and hence, a yield point.

Fig. 8. Differing stress–strain curves of the analysed thermoplastic materials.

Please cite this article as: A. Arriaga, et al., Finite-element analysis of quasi-static characterisation tests in thermoplastic materials: Experimental and numerical analysis results..., Polymer Testing (2006), doi:10.1016/j.polymertesting.2006.10.012
material model should be chosen, taking into account the previously mentioned difficulties for adequately representing the modulus and yielding.

2.2.3. Stress–strain curves for different thermoplastic materials

Some of the stress–strain curves obtained for the analysed thermoplastics at a testing speed of 50 mm/min and using a laser-type extensometer are presented in Fig. 8.

2.2.4. Comparative engineering and “true” stress–strain curves

The conversion of data from engineering to “true” values was achieved in three different ways. (Note that conversion of strains was always by logarithms):

- classical conversion along the entire tensile test curve supposing volume preservation during plasticity,
- using a Poisson’s ratio that changes along the tensile stress–strain curve,
- using the classical conversion up to the maximum load point and then performing three different corrections (constant engineering stress case and two plateau corrections as described before).

The influence of inputting one or another conversion method within the simulation results and, hence, in the correlation with the experimental curves has been investigated for the plate perforation tests.

The graphs in Figs. 9 and 10 show the stress–strain curves obtained using different conversion methods.

When entering the strain points, total strains are normally entered and the software performs the calculations internally. However, there are some material models and codes that require the introduction of plastic strains as follows:

\[ \varepsilon_{\text{total}} = \varepsilon_{\text{elastic}} + \varepsilon_{\text{plastic}}, \quad (12) \]

\[ \varepsilon_{\text{plastic}} = \varepsilon_{\text{total}} - \sigma / E, \quad (13) \]

where \( \sigma \) is the true stress for every point of the curve and \( E \) is the material’s elastic modulus.

Fig. 11 shows the hardening curves for PP and PC/ABS. Classical stress conversion has been used along the entire curve.

Fig. 9. Resultant stress–strain curves for PC/ABS using different conversion methods from engineering to “true” data.
Fig. 10. Resultant stress–strain curves for PP using different conversion methods from engineering to “true” data.

Fig. 11. Hardening curves (plastic strains) for PC/ABS and PP.
3. The bending test for the correlation with the results obtained in ANSYS

Bending tests were performed to determine the correlation level obtained between the experimental tests and the simulation results in ANSYS modelling. Both utilise the same test samples and test conditions.

3.1. Testing procedure

The specimens used conform to ISO 3167 having a total length of 158 mm, a cross section of $10 \times 4 \text{mm}^2$ and a straight zone of 80 mm.

Different testing speeds have been used, all of them relatively slow, for the determination of the straining rate effect on the resultant curves. The testing method follows the standard ISO 178.

3.2. Calculation of the stresses and strains generated in the bending test

The stress–strain conversion formulae are based on consideration of material linearity, small strains and small specimen thickness. Their application in plastic materials is debatable but can be used for comparison purposes.

The strains generated on the external surfaces at the centre of the sample can be calculated using the following relation:

$$\varepsilon_f = \frac{6Dd}{L^2},$$

(14)

where $D$ is the displacement applied at the centre of the sample, $d$ is the specimen thickness and $L$ is the span distance.

The maximum stress on the external surfaces at the centre of the sample can be calculated as follows:

$$\sigma_f = \frac{3PL}{2bd^2},$$

(15)

where $P$ is the measured force, $L$ is the span distance, $b$ is the specimen width and $d$ is the specimen thickness.

3.3. Quasi-static bending tests in different materials

The force–displacement curves recorded in the universal testing machine are converted into stress–strain data using Eqs. (14) and (15). A series of bending stress–strain curves are represented in Figs. 12 and 13:

![Bending stress-strain curves in different thermoplastics](image_url)

Fig. 12. Bending stress–strain curves in different thermoplastics, using a testing speed of 50 mm/min.
3.3.1. Comparison between tensile and bending stresses

It can be seen that the stress–strain values calculated in bending are considerably higher than those represented in engineering values for the tensile test. It should also be noted that the formulas for bending do not take account of plasticity effects.

Concerning the simulation by finite elements, from the correlation viewpoint the compared curves are the force–displacement points obtained in the tests with the results obtained in ANSYS simulating the bending test.

4. The plate penetration test with semi-spherical darts for the correlation with the results obtained in ANSYS

The aim of the plate perforation test using semi-spherical steel darts is to determine the correlation level achieved between the experimental tests and the simulation of the same tests in the FEA software ANSYS.

---

Please cite this article as: A. Arriaga, et al., Finite-element analysis of quasi-static characterisation tests in thermoplastic materials: Experimental and numerical analysis results..., Polymer Testing (2006), doi:10.1016/j.polymertesting.2006.10.012
4.1. Testing procedure

The tested specimens were square plates of approximately $90 \times 90 \times 2$ mm. The test is similar to a dart drop test but in this case the testing speeds were very slow (up to a maximum of 1000 mm/min. available in the machine). The specimen clamping systems follows the standard ISO 6603-2; the plates were located on a steel annular support of an inner diameter of 40 mm and external diameter of 60 mm. The darts used have a diameter of 20 mm.

An illustration of the testing device can be seen in Fig. 14.

The stress–strain analysis in a plate penetration test is restricted to the calculation by means of a FEA system, even though there are some formulas that can be used for having some initial orientation [4].

In the elastic range of strains at the plate, the biaxial stresses at the central region (where the maximum stress will be located) can be estimated as shown in Eq. (16):

$$\sigma = \frac{3F}{2\pi t^2} \left(1 + v\right) \left(\ln \frac{R}{a} + \frac{a^2}{4R^2}\right)$$

where $F$ is the centrally applied force, $a$ is the contact radius of the dart with the plate, $t$ is the plate thickness and $R$ is the support radius of the plate on the metallic base.

A simple finite-element analysis in ANSYS shows large discrepancies between the stresses calculated using this formula and the FEA method.

From the correlation point of view, the curves compared are the force–displacement points from the tests and the results obtained in ANSYS simulating the penetration test.

5. Modelling with finite elements in ANSYS

When modelling the different experimental tests carried out in tensile, bending and plate penetration modes, a series of factors that can influence the behaviour of the model, and hence the results obtained in the simulations, should be taken into account. The most important factors are summarised below:

Modelling the material’s behaviour: This is one of the most important factors since the simulation response will mainly depend on how well the structural behaviour of the material is represented in the calculation code. In the work carried out, the behaviour of the material has been studied both in elastic and plastic ranges, evaluating the large deformation behaviour and material non-linearity. The MISO (multilinear isotropic) hardening elastic–plastic model available in ANSYS has been evaluated in order to define the stress–strain behaviour; a modulus and a yield stress define the elastic part, while the rest of the curve is modelled by different pairs of stress–strain points. The yield surface expands uniformly as yielding occurs in isotropic hardening. This model is recommended for large strain applications.

The model adopts the Von Mises yielding criterion. This is known not to be always precise for plastics (especially when hydrostatic stress components are generated in the part which, on the other hand, is not very usual in plastic components where small thickness values are present) but in the present work the intention is to determine the correlation level that is obtained with this classical “metals oriented model”.

The effect of using different Poisson’s ratio values has also been investigated, as has the use of different conversion methods from engineering stress–strain values to suitable “true” data. The Poisson’s ratio used is the elastic one, which is related to the low strain area of the entire stress–strain curve.

Modelling the test loading conditions: the different contact areas between the metallic testing devices and the plastic specimens have to be taken into account, as have the symmetries existing in the modelling geometries. The effect of using different friction coefficients between metallic and plastic parts has also been evaluated.

Use of different element types: The multipurpose specimens and square plates can be modelled using 3D solid or shell-type elements. In the case of plate penetration, a further simplification has been carried out: considering that the stress pattern in 3D follows a circumferential form, axis-symmetry has been considered for a plate radius where stresses are radial and low at the extremes.

Use of different solution characteristics: The calculations performed are static solutions. The presence of non-linearities—such as for instance the contacts with metallic parts, the large displacements and the material non-linearities, makes the solution convergence difficult in some cases. Therefore, selection of a correct number of solution substeps is fundamental for the consecution of the required results.
5.1. Modelling the uniaxial tensile test in ANSYS

The validation of a material model should be carried out in simple reproducible geometries that require a low computational cost in the FEA software. One material model validation method consists of modelling the same uniaxial tensile test from which the stress–strain data points have been recorded. If the force–displacement curve of the experimental test is adequately represented, the model can be considered valid for the uniaxial tensile mode, however it should also be calibrated for other deformation modes such as compression, biaxial or triaxial stress states or shear straining.

If the material suffers considerable “necking”, the reproduction of this effect is quite complicated in the simulation software. It is an instability from the simulation point of view since the stiffness of the structure changes abruptly to almost zero value at maximum load, then it suffers an engineering stress drop due to necking initiation, and again the stiffness increases, passing a plateau region. In this study, it has been seen that a fine mesh in conjunction with higher order solid elements (intermediate nodes) are recommended for a better matching of this singularity. A reproduction of the necking effect in ANSYS can be observed in Fig. 15.

Fig. 15. Reproduction of the “necking” effect simulating the tensile test specimen.

5.2. Modelling the 3 point bending test in ANSYS

The 3-point bending test has been used as a verification test of the elastic–plastic models implemented in ANSYS. The force–displacement curves obtained in the universal testing machine are compared with the results in the simulations.

The bending test has been modelled using two different element types: solid- and shell-type 3D elements. Shell-type elements are commonly used when analysing thermoplastic components.

Solid hexahedral with mid-side nodes (SOLID 95 in ANSYS), solid hexahedral without mid-side nodes (SOLID 45), solid tetrahedral with mid-side nodes (SOLID 92) and shell-type quadrilateral elements without mid-side nodes (SHELL 43) have been used in the simulations.

Figs. 16 and 17 show the used 3D mesh and the stress results obtained in ANSYS.

Fig. 18 shows the results obtained using different element types for multipurpose specimens subjected to bending.

The curves shown are those of PP from BOR-EALIS. It can be seen that the results are very similar even using different element types, though in the case of the shell element types the maximum forces are somewhat lower than using solid element types (approximately a 4% of difference in force values). This can be due to the fact that shell
elements are recommended when the thickness of the structure is very thin in comparison to the other dimensions. In this case, the thickness of 4 mm is considerable so the reproducibility of the stiffness of the sample along the thickness is poorer.

5.3. Modelling the plate penetration test in ANSYS

The plate perforation test using semi-spherical darts has been used as a verification test of the elastic–plastic models implemented in ANSYS. The
Fig. 18. Force–displacement response when simulating the bending test with different element types in ANSYS.

Fig. 19. 3D solid model of the plate perforation test.
Fig. 20. Von Mises stresses generated during the plate perforation test. Quarter symmetry is shown.

Fig. 21. Force–displacement response when simulating the perforation test with different element types in ANSYS in an ABS.
force–displacement curves obtained in the universal testing machine are compared with the results obtained in the simulations.

3D solid hexahedral elements without mid-side nodes (SOLID 45), 3D shell quadrilateral elements without mid-side nodes (SHELL 43) and 2D solid quadrilateral axis-symmetric elements without mid-side nodes (PLANE 42) have been used.

Figs. 19 and 20 show the used 3D mesh and the stress results obtained in ANSYS.

The force–displacement curves obtained for an ABS SINKRAL from POLIMERI using different element types are shown in Fig. 21.

It can be seen that the use of shell-type elements gives higher force values than using 3D solid or 2D solid axis-symmetric elements. There is a difference of approximately 7% in force for a displacement of 7 mm.

6. Study of the effect of different variables in the simulation results

A series of simulations have been performed using different values for:

- the elastic Poisson’s ratio of the material,
- the dynamic friction coefficient of the material.

A comparison of simulation results using different stress–strain conversion methods has also been carried out.

6.1. Use of different values for the Poisson’s ratio in the penetration test simulation

It has been observed (Fig. 22) that different values for the elastic Poisson’s ratio do not have an effect on the force–displacement behaviour of the dart penetration test. No measurements of the plastic Poisson’s ratio have been performed, which should in principle be more realistic for this type of test where high strains are generated. Moreover, the material models implemented in ANSYS only permit the input of an elastic Poisson’s ratio.

6.2. Use of different friction coefficients in the bending test simulation

For observing the effect of using different friction coefficients, three frontier values have been adopted for PC/ABS. It can be seen that the higher the friction coefficient the more pronounced the effect on the plastic regime of the response (beyond the maximum force point). At friction values of 0.60,

![Fig. 22. Effect of changing the elastic Poisson’s ratio. Results for PC/ABS.](image-url)
Fig. 23. Effect of changing the friction coefficient value in the bending simulation.

Fig. 24. Effect of using different conversion methods from engineering to true stress–strain data. Results for PP.
convergence problems arise and the force–displacement curve becomes quite noisy as can be seen in Fig. 23.

6.3. Use of different conversion methods of engineering stress–strain data

The PP results obtained in a plate perforation test are shown in Fig. 24. Solid-type elements have been used and the stress–strain conversion from engineering data to “true” values has been performed in three different ways:

- **True**: Classical conversion supposing volume preservation in the plastic region.
- **Variable Poisson**: Using the formula described in Section 2.1.
- **Corrections 1, 2 and 3**: The classical conversion is used up to the maximum load point and from that point different exponents are calculated.

The main differences between the various conversion methods can be particularly observed in the large deformation region of the force–displacement curve. The curve deviating most corresponds to the first correction method due to the stiffening behaviour of the stress–strain curve of this correction in the plastic region.

7. Comparison between experimental and simulation results in ANSYS

The different experimental and simulation curves compared in Figs. 25–28 reflect the correlation level that is obtained between the experimental bending and penetration tests and the FEA results for the PP and PC/ABS materials.

In the experimental tests, the curves shown are intermediate responses from different testing speeds when observing the effect of straining rate on the material’s behaviour. An attempt has been made to allocate the simulation response between the experimental curves tested at different speeds.

The simulation responses are the result of entering tensile stress–strain curves from test data at 50 mm/min. and using the true classical conversion.

In Fig. 25, it can be seen that the simulation response is between the experimental bending tests and close to the 5 mm/min curve. The higher the
friction coefficient, the flatter the slope (more horizontal) beyond the maximum load point. This does not correlate with the test behaviour. This means that relatively low friction values should be used to get realistic simulation results and in this case the zero friction approximation works properly.

In Fig. 26, the response obtained in the simulation with solid-type elements is close to the experimental curves. For vertical dart displacements beyond 7 mm, convergence problems appear due to the excessive deformation at the central region elements.

In this simulation, the use of a higher friction value will give responses over the experimental curves in the speed ranges tested.

As in the case of PC/ABS, the simulation results correctly reflect the experimental curves for PP in bending (Fig. 27). The influence of the friction coefficient on the simulation response must again be remarked upon. Here, the effect of the straining rate is more marked than in the case of the PC/ABS.

For the penetration test in PP (Fig. 28), the response obtained in the simulation with solid-type elements is again in concordance with the experimental curves. For values of 7 mm displacement upwards, convergence problems are present due to the high straining of central region elements.

In this simulation the use of a higher friction value will give responses over the experimental curves in the tested speed ranges.

A proper correlation would be expected for a rate-dependent plastic analysis with experimental data covering a range of strain rates. This is intended to be treated in further studies.

8. Conclusions

Overall, good correlation level has been reached with the PP and PC/ABS materials between the experimental 3 point bending and plate perforation tests and the simulation of the same tests in the FEA software ANSYS.

The use of an elastic–plastic material model in conjunction with isotropic hardening in plasticity...
Fig. 27. Comparison of experimental and simulated responses of the bending test in PP.

Fig. 28. Comparison of experimental and simulated responses of the penetration test in PP.
and Von Mises yielding criterion is an adequate choice for the materials studied and mechanical tests performed. The error level in comparison with the experimental responses is acceptable from an engineering point of view since, apart from the material model selection, there are other factors such as the friction coefficient specification that have a more relevant effect on the simulation results.

It has been seen that the results obtained are very similar in the plate perforation tests when using different conversion methods for converting engineering stress–strain values to “true” points. So, it can be said that the classical conversion method can result in a choice to take into account when working with these thermoplastic materials and these types of geometries.

When analysing the contact between the metallic supports and the plastic samples, the dynamic friction coefficient value is of fundamental importance to obtaining good correlation levels between the experimental tests and the simulations in ANSYS. In the present work, the limit values adopted have been extracted from material manufacturers’ catalogues for orientation purposes. It has been seen that the friction coefficient value in particular influences the force–displacement curve beyond the maximum load point, and that the use of high friction values deviates the response from the experimental curve (convergence problems and noisy response).

The elastic Poisson’s ratio does not affect the penetration test responses in the performed simulations; however, further studies should take into account the plastic Poisson’s ratio.

There are some differences between the results obtained with solid elements and with shell-type elements: in the bending test the thickness of the specimen is relatively high for shell-type specimens and the force results are somewhat lower than when using solid elements. In the plate penetration test, at high strain levels the response obtained using shell-type elements is stiffer than using 3D or 2D solid-type elements.

Acknowledgements

BAYER, BOREALIS, SOLUTIA and POLIMERI for the given thermoplastic materials.

London Metropolitan University for the supervision from Alicia Chrysostomou.

Lea-Artibai Ikastetxea, AZARO Fundazioa, FEDER and the Basque Country Government for giving founding for the development of this study.

References


Further reading

[8] F. Lutter, M. Münker, M. Wanders, Simulation Right through to Failure, Bayer, ATI.