Experimental testing and numerical modeling of AM50A magnesium alloy for structures subjected to large deformation

W. Altenhof, M. Laframboise, J. Losher, A. Raczy, A. Alpas
Department of Mechanical, Automotive, and Materials Engineering, University of Windsor, Canada

Abstract

Automotive companies place a significant amount of effort on lowering the overall weight of a vehicle. Lightweight aluminum and magnesium alloys are replacing automotive structures which were traditionally developed from steels. With the increase in safety regulations automotive manufacturers have implemented finite element methods for the crashworthiness analysis of vehicular components subjected to large deformation. Material modeling of magnesium alloys and the use of constitutive equations to describe material behaviour is not as extensive as that of carbon steels. This research concentrates on the development of a material model for the AM50A magnesium alloy. Experimental and numerical simulation of standardized tensile and charpy testing procedures have been used to develop and validate the computer model. Further model verification has been conducted by correlating experimental and numerical impact testing of AM50A magnesium alloy steering wheel armatures.

1 Introduction

The need for fuel efficient automobiles has caused vehicle manufacturers to utilize aluminum and magnesium alloys for a variety of automotive structures in both load bearing and non-load bearing situations. An understanding of how these alloys behave under large deformation and at elevated strain rates is
important in the development of vehicles, which maintain a good level of safety under crash or impact conditions.

This research concentrates on the AM50A magnesium alloy and its application into the automotive industry, as a material for a steering wheel armature. An armature is the backbone of the steering wheel. It supports the electronic devices and other aesthetic materials on the steering wheel. The structural capabilities of the steering wheel are largely dependent upon the armature design [1]. The two most significant loading situations that the armature is subjected to are fatigue and impact loading conditions. Initial design for crashworthiness of the armature requires the use of finite element (FE) software to analyze the loads and deformations that the armature experiences when impacted with another entity. Although the steering wheel engineer would prefer the FE simulations to predict the actual response of the armature (and a good correlation between experimental and numerical tests), often numerical simulations can only provide trends in design modifications.

A number of researchers have studied the deformation of magnesium alloys. Aune et al. [2] have studied the behavior of three different die-cast magnesium alloys (AZ91D, AM60B, and AM50A) subjected to strain rates ranging from 15·s^{-1} to 130·s^{-1}. The findings from this investigation have shown that while a rate dependency for material stress behavior existed, the rate of deformation did not affect the elongation. Carlson [3] has also shown similar findings for the magnesium alloy AM60B; flow stress for this material is a function of strain rate over the range of 10^{-3}·s^{-1} to 10^{3}·s^{-1}. Other researchers have also observed strain rate dependencies for various magnesium alloys, however, it appears that the sensitivity is dependent upon the alloy [4].

Aune et al. [2] have determined the Johnson-Cook parameters for the AM50A magnesium alloy neglecting any thermal effects in the material. Johnson and Cook [5] have proposed the following constitutive equation that relates the flow stress to the yield stress (\sigma_0), strain (\varepsilon), strain rate (\dot{\varepsilon}), and temperature (T):

\[
\sigma = (\sigma_0 + B \cdot \dot{\varepsilon}^n) \cdot \left(1 + C \cdot \ln \left( \frac{\dot{\varepsilon}}{\dot{\varepsilon}_R} \right) \right) \cdot \left(1 - \left( \frac{T - T_{room}}{T_{melt} - T_{room}} \right)^m \right) \tag{1}
\]

Reference [2] has conducted all experimental testing without considering changes in temperature and thus has only considered the first two terms (enclosed in brackets) in the constitutive law. Aune et al. have found that for the AM50A magnesium alloy the values of \sigma_0, B, n, and C are 88 MPa, 599 MPa, 0.5966, and 0.019 respectively. The magnitude of C implies that only a small sensitivity to rate effects is present in this magnesium alloy.

This research investigates the deformation of the AM50A magnesium alloy through experimental techniques and proposes a material model, which can be
utilized in the nonlinear finite element code LS-DYNA. The testing methodology of this investigation first involves the experimental tensile testing of AM50A magnesium alloy specimens. Unfortunately, rate effects could not be investigated as the necessary experimental facilities were unavailable. However, information from [2] suggests that only a slight rate dependency of the AM50A alloy exists. Analysis of the data acquired from the tensile test is then used for input in the FE model for the numerical simulation of the tensile tests. Further material model verification is conducted by investigating, both experimentally and numerically, the deformation of charpy impact specimens, under standard charpy testing conditions, and impact testing of magnesium alloy steering wheel armatures.

2 Uniaxial tensile testing

2.1 Experimental uniaxial tensile testing

Eight standard die-cast tensile test samples were made in accordance with the American Standard of Testing Materials (ASTM) designation B 557N-94 [6]. The tensile test procedure was performed in accordance with the above ASTM standard (B 557M). The test was performed using calibrated apparatus, on a United Test machine as illustrated in Figure 1. The cross sectional diameters of each specimen were measured to the nearest 0.025 mm three times and the average value was calculated and recorded into a computer test file. The specimen was set into the grips and the extensometer, with a 50.8 mm gauge length, was mounted within the gauge region of the tensile test specimen to acquire strain data. The tests were conducted with a crosshead speed of 0.42 mm/sec. The tensile test was performed until the specimen had reached a strain of 5%, at which point the extensometer was removed and the test was continued until failure of the specimen occurred. The automated data collection system generated a load versus engineering strain curve and determined values for load at material yielding, the peak load, the load at specimen failure, the yield stress of the material, the maximum stress, the fracture stress, percent elongation, and percent reduction in area. Table 1 provides a summary of the most relevant material properties.
Table 1. Summary of Most Significant Material Properties

<table>
<thead>
<tr>
<th>Material Property</th>
<th>Value with Appropriate Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>1770 kg/m³</td>
</tr>
<tr>
<td>Elastic Modulus</td>
<td>31.26 GPa</td>
</tr>
<tr>
<td>Yield Stress (0.2% Offset)</td>
<td>104.6 MPa</td>
</tr>
<tr>
<td>Strain to Failure</td>
<td>10.35 %</td>
</tr>
</tbody>
</table>

2.2 Finite element modeling of the tensile test

An axisymmetric FE model of the tensile test specimen was developed using Finite Element Model Builder (FEMB) based upon geometry provided in the ASTM tensile testing standard. The FE model consisted of 2627 nodes and 2304 elements. The numerical simulation was conducted in a similar fashion as the experimental test. Nodes at one extreme end of the FE test specimen were fully constrained from motion, while nodes at the other extreme end of the FE model were displaced approximately 10mm. This amount of displacement provided enough strain in the specimen to reach the failure strain within the gauge region. The numerical material model used to describe the stress/strain behaviour for the AM50A magnesium alloy in the FE software LS-DYNA was material type 24, which is a piecewise linear plasticity model. This isotropic material model requires as input the material's density, elastic modulus, Poisson's ratio, yield stress, strain rate parameters (if simulating rate effects) and a series of data points which presents the yield stress versus effective plastic strain. This material model was selected for its ease of implementation and its widespread use in the automotive industry.

Material information from the experimental tensile test was used to obtain all the necessary information for the material model. The true stress/strain data was calculated (using equations (2) and (3)) from the engineering stress/strain information acquired in the experimental tests. The effective plastic strain was then determined using equation (4).

\[
\varepsilon_{\text{true}} = \ln\left(1 + \varepsilon_{\text{eng}}\right) \tag{2}
\]

\[
\sigma_{\text{true}} = \frac{F}{A_0} \cdot \left(1 + \varepsilon_{\text{eng}}\right) \tag{3}
\]

\[
\varepsilon_{\text{plastic}} = \varepsilon_{\text{true}} - \frac{\sigma_{\text{yield}}}{E} \tag{4}
\]

Although equations (2) and (3) are only valid up to the onset of necking they were utilized for the entire range of stress and strain since no significant amount of necking was observed in all the test specimens.
A mesh sensitivity analysis was also conducted on the FE model to investigate the effects of mesh density on the simulation results. Another FE model was developed with four times the number of axisymmetric elements compared to the first original FE tensile test model. No significant changes in the simulation results were observed indicating that the mesh density of the original FE model was acceptable.

2.3 Experimental and numerical comparison of the uniaxial tensile tests

The experimental testing observations, acquired from the United Test tensile machine, were provided as tensile load versus engineering strain curves. Similar load versus strain curves were developed based upon the numerical simulation results. Figure 2 presents the tensile load versus engineering strain up to approximately 10.3% strain. The stress/strain relationship determined using the Johnson-Cook constitutive equation (1) is also presented in Figure 2 to compare the results from other research.

Comparison of the results from Figure 2 illustrate that the numerical material model correlates well with the experimental findings in both the elastic and plastic regions of deformation during the tensile test. The Johnson-Cook relation and parameters used from reference [2] do not correlate as well as the numerical model developed in this research, especially in the elastic region of the load/strain curve.
3 Charpy impact testing

3.1 Experimental charpy impact testing

Twenty-six un-notched charpy impact test specimens were die cast according to American Standard of Testing Methods E23-98 (ASTM-E23) [7]. Sixteen samples were tested using the charpy impact testing apparatus at KS Centoco in Windsor, Ontario, Canada, following the ASTM E23 charpy impact test procedure (Figure 2). The recently calibrated impact machine was inspected for damage and wear, and the zero position of the machine was verified. Testing was conducted at room temperature. The specimen was centered on the specimen supports and flush against the anvil. The pendulum was raised to the latched position (specified in ASTM E23 for the magnesium test specimen) and the energy indicator was set to zero. The pendulum was released and data was recorded using the automated data collection system which measured the impact velocity, impact energy, maximum load, energy at maximum load, and total energy.

The data collected from this experiment was compared with data received from Norsk Hydro ASA [8]. The total energy values from the charpy impact tests performed at KS Centoco did not correlate with the information provided by Norsk Hydro; the impact load results from the KS Centoco testing equipment were significantly lower.

To verify the initial test data, ten additional charpy tests were conducted at the University of Windsor. The same test method in ASTM-E23 was utilized with the exception of the automated data collection. For the charpy impact tests conducted at the University of Windsor the total energy values were obtained from a dial indicator.

3.2 Finite element modeling of the charpy impact tests

Finite element models of the charpy impact testing specimen, impactor, and rigid supporting anvil were developed using TrueGrid based on geometry provided in ASTM E23. Both the impactor and the supporting anvil utilized a rigid material model and to properly discretize the geometry into a finite element mesh a large number of elements were utilized in both the impactor and support anvil finite element models. All nodes of the supporting anvil were fully constrained from motion and the nodes of the impactor were constrained to move in the direction
of the impact velocity. The FE model of the impactor contained 2832 solid hexahedral elements and the FE model of the support anvil contained 2986 hexahedral elements and 38 wedge elements. An underintegrated element formulation was selected for both the impactor and supporting anvil FE models. Since material failure was to be simulated in this numerical investigation the FE model of the charpy impact testing specimen had to be very highly discretized. 79054 hexahedral elements employing an underintegrated element formulation with Flanagan-Belytschko stiffness hourglass control were utilized for the FE model of the specimen. No constraints were placed on any of the nodes within the specimen; since an explicit time integration scheme is being utilized in this simulation (and hence no inversion of the stiffness matrix is conducted) it is not necessary to fix or constrain any nodes on the specimen to ensure numerical convergence. A finer nodal distribution towards the centre of the FE model of the test specimen was used to better simulate the failure procedure. The smallest edge length of the elements within the model of the test specimen was 0.145 mm, the largest dimension of any elements within the model was 1.5 mm towards the edge of the specimen where element size is not significant. Figure 4 illustrates all FE models used in the numerical investigation.

Two different modes of failure were investigated in the in the numerical charpy test simulations. The first method required the definition of an effective plastic strain to failure. When the effective plastic strain (within each finite element) reaches 10.35% (see Table 1) the finite element is deleted from the simulation. One of the main difficulties with using this technique is that the model assumes the material behaves identically in tension and compression. When failure occurs in compression (and under bending conditions) the finite elements on the other sides of the failed elements have a tendency to ‘pass through’ each other, Figure 5 illustrates this effect. Another technique, using contact algorithms, which requires surfaces in contact to be ‘tied’ together until a failure condition is satisfied was also used. The failure condition is based upon equation (5).

\[
\left( \frac{\text{max}(0.0, \sigma_{\text{normal}})}{S_{f,\text{normal}}} \right)^2 + \left( \frac{\sigma_{\text{shear}}}{S_{f,\text{shear}}} \right)^2 - 1.0 > 0.0
\]

In equation (5), \(S_{f,\text{normal}}\) and \(S_{f,\text{shear}}\) are the normal and shear stresses at failure, which were determined from the experimental stress/strain relationship. The
shear stress at failure was assumed to be $\frac{1}{2}$ of the normal stress at failure. Figure 5 illustrates the failure of the FE charpy specimen using the two different failure modes.

Figure 5. Two failure modes of charpy specimen investigated.

Two contact algorithms were used in the simulation of the charpy impact testing. The NODES_TO_SURFACE contact algorithm in LS-DYNA was used for modeling contact between the specimen/supporting anvil and the specimen/impactor. The TIED_SURFACE_TO_SURFACE_FAILURE contact algorithm was utilized for the failure of the specimen.

### 3.3 Experimental and numerical comparison of the charpy impact tests

Figure 6 presents the experimental and numerical load versus time observations acquired during the impact. The experimental results were provided from reference [8]. Results from the simulations investigating the two different modes of failure are presented in Figure 6. The failure mode using equation (5) better predicts the experimental results. Furthermore, the total energy absorbed during the charpy test for the different testing methods are summarized in Table 2.
Table 2. Summary of the total energy absorbed during the charpy impact test.

<table>
<thead>
<tr>
<th>Testing Method</th>
<th>Energy Absorbed (J)</th>
</tr>
</thead>
<tbody>
<tr>
<td>K.S. Centoco Testing</td>
<td>9.22</td>
</tr>
<tr>
<td>University of Windsor Testing</td>
<td>23.63</td>
</tr>
<tr>
<td>Norsk Hydro ASA [8]</td>
<td>18.30</td>
</tr>
<tr>
<td>Failure based on equation 5</td>
<td>15.6</td>
</tr>
<tr>
<td>Failure based on strain</td>
<td>9.38</td>
</tr>
</tbody>
</table>

4 Experimental impact testing on steering wheel armatures

Approximately sixteen different impact tests have been conducted with AM50A magnesium steering wheel armatures. These impact tests involve the use of a rigid plate impacting a steering wheel armature which is supported at its hub. Figure 7 illustrates the test setup.

A limited number of numerical simulations, investigating impact of a rigid plate on the armature have been completed utilizing the AM50A material model developed in this research. The observations from the tests are presented as a load versus wheel armature displacement curve. Both variables are considered in the direction of the impact velocity.

Figure 8 provides an experimental and numerical comparison of the results from a steering wheel armature impact test. An acceptable correlation between the two testing methods has been observed, especially for the degree of deformation associated with these types of impact tests.
5 Summary and conclusions

The focus of this research is on the development of a material model for the AMSOA magnesium alloy to be used in nonlinear finite element software (LS-DYNA) investigating structures subjected to large deformation. Experimental tensile tests provided stress/strain information for the magnesium alloy and rate effects were not considered. A good correlation between experimental and numerical procedures for tensile testing, charpy impact testing, and armature impact testing has been found.

Furthermore, simulation of the failure for the charpy impact specimen is best conducted by implementing a tied surface to surface contact algorithm in LS-DYNA.

References


