Warm Forming of Aluminum Alloys using a Coupled Thermo-Mechanical Anisotropic Material Model

Nader Abedrabbo*, Farhang Pourboghrat* and John E. Carsley†

*Department of Mechanical Engineering, Michigan State University, East Lansing, MI 48824-1226
†General Motors Research & Development Center, Warren, MI 48090

Abstract. Temperature-dependant anisotropic material models for two types of automotive aluminum alloys (5754-O and 5182-O) were developed and implemented in LS-Dyna as a user material subroutine (UMAT) for coupled thermo-mechanical finite element analysis (FEA) of warm forming of aluminum alloys. The anisotropy coefficients of the Barlat YLD2000 plane stress yield function [1] for both materials were calculated for the range of temperatures 25°C–260°C. Curve fitting was used to calculate the anisotropy coefficients of YLD2000 and the flow stress as a function of temperature. This temperature-dependent material model was successfully applied to the coupled thermo-mechanical analysis of stretching of aluminum sheets and results were compared with experiments.

INTRODUCTION

Weight reduction has long been identified as a key enabler for improving automotive fuel economy. Aluminum can be a suitable replacement for steel in the structure and body of an automobile, which can lead to significant mass reduction, although its room temperature formability is generally less than that of typical sheet steel [2]. One way to overcome this is to raise the forming temperature of the sheet. The elevated temperature leads to decreased flow stress and increased ductility, which enables deeper drawing and more stretching to form panels without design modifications to the stamped steel product. Numerical analysis is critically important to understanding the complex deformation mechanics that occur during sheet forming processes. Finite element analysis (FEA) and simulations are used in automotive design and formability processes to accurately predict deformation behavior during stamping operations. Although commercially available FEA codes offer a library of material models simulating a variety of applications, they often do not offer highly specialized material models appropriate for simulating the thermo-mechanical forming processes of anisotropic materials such as aluminum sheet alloys. The process becomes increasingly complicated when materials exhibit anisotropic behavior. The importance of using an appropriate material model for aluminum during hydroforming processes using Barlat’s YLD96 model has been emphasized recently [3]. In previous papers [4], a temperature dependant material model using Barlat’s YLD96 was developed for AA3003-H111 and implemented into LS-Dyna. The model accurately predicted warm forming behavior of this aluminum alloy. In this paper, a new plane stress yield function developed by Barlat et al. [1], YLD2000, for AA5754-O and AA5182-O was fully characterized at elevated temperatures and implemented into FEA.

Material Model

Accurate simulation of aluminum sheet forming processes depends on the use of a constitutive model that precisely describes the behavior of the material [3]. For that purpose, Barlat’s YLD2000 anisotropic yield function [1] was used to simulate the coupled thermo-mechanical forming process with LS-Dyna.

Although YLD96 [5] is one of the most accurate anisotropic yield functions for aluminum and its alloys, there are some challenges with respect to FE simulations that inhibit implementation of the material model. Therefore, Barlat et al. [1] developed a better incompressible anisotropic plasticity formulation that
can guarantee convexity, simplify FE implementation and application, and take \( \sigma_0, \sigma_{45}, \sigma_{90}, r_0, r_{45}, r_{90} \) and \( \sigma_b \) into account for plane stress conditions.

The yield function for plane stress plasticity can be reduced to the general form:

\[
\phi = \phi' + \phi'' = 2\sigma'', \tag{1}
\]

where

\[
\phi' = \left| X'_1 - X'_2 \right|' , \tag{2}
\]

and

\[
\phi'' = \left[ 2X''_2 + X''_1 \right]' + \left[ 2X''_1 + X''_2 \right]' . \tag{3}
\]

The coefficients \( X'_1 \) and \( X'_2 \) are the principal values of the linear transformation equations:

\[
X' = L' \sigma \tag{4}
\]

\[
X'' = L'' \sigma ,
\]

where \( L' \) and \( L'' \) are linear transformation matrices:

\[
\begin{bmatrix}
L'_{11} \\
L'_{12} \\
L'_{21} \\
L'_{22} \\
L'_{66}
\end{bmatrix} = \begin{bmatrix}
2/3 & 0 & 0 \\
-1/3 & 0 & 0 \\
0 & -1/3 & 0 \\
0 & 2/3 & 0 \\
0 & 0 & 1
\end{bmatrix}
\]

and

\[
\begin{bmatrix}
L''_{11} \\
L''_{12} \\
L''_{21} \\
L''_{22} \\
L''_{66}
\end{bmatrix} = \begin{bmatrix}
-2 & 2 & 8 & -2 & 0 \\
1 & -4 & -4 & 4 & 0 \\
4 & -4 & -4 & 1 & 0 \\
-2 & 8 & 2 & -2 & 0 \\
0 & 0 & 0 & 0 & 9
\end{bmatrix}
\]

The independent coefficients \( \alpha_k \) (for \( k: 1 \) to \( 8 \)) are all that is needed to describe the anisotropy of the material, where they reduce to 1 in the isotropic case.

These coefficients are determined from seven experimental tests as described by Barlat [1].

In order for the constitutive model to accurately represent aluminum alloys AA5754-O and AA5182-O at elevated temperatures, the anisotropy coefficients \( \alpha_1, \alpha_2, \alpha_3, \alpha_4, \alpha_5, \alpha_6, \alpha_7 \) and \( \alpha_8 \) were fully characterized with uniaxial tension tests in multiple directions at several elevated temperatures. Tables 1 and 2 show a sample of the results for AA5754-O and AA5182-O, respectively, as a function of temperature.

**TABLE 1. AA5754-O YLD2000 anisotropy coefficients as a function of temperature. (°C)**

<table>
<thead>
<tr>
<th>( \alpha )</th>
<th>( 1.06 - 5.98E-03T + 8.57E-05T^2 - 4.66E-07T^3 + 8.46E-10T^4 )</th>
<th>( -4.66E-07T^2 )</th>
<th>( + 8.46E-10T^4 )</th>
</tr>
</thead>
</table>

**TABLE 2. AA5182-O YLD2000 anisotropy coefficients as a function of temperature. (°C)**

<table>
<thead>
<tr>
<th>( \alpha )</th>
<th>( 0.95 + 6.76 E-03<em>T - 9.99E-05</em>T^2 + 5.63E-07<em>T^3 - 1.05E-09</em>T^4 )</th>
<th>( 5.63E-07*T )</th>
<th>( - 1.05E-09*T^4 )</th>
</tr>
</thead>
</table>

**Flow Stress**

Flow stress \( \sigma \) represents the size of the yield function during deformation. As an appropriate constitutive equation describing changes in the flow stress of the material, the power law flow rule was modified to include temperature effects, assuming isotropic hardening behavior as follows:

\[
\sigma(\varepsilon, \dot{\varepsilon}, T) = K(T)(\varepsilon^p + \varepsilon_0)^n(T) \left( \frac{\dot{\varepsilon}}{\varepsilon_{S0}} \right)^m(T) \tag{7}
\]

where \( K \) (strength hardening coefficient), \( n \) (strain-hardening exponent) and \( m \) (strain-rate sensitivity index) are material constants. \( \varepsilon^p \) is the effective plastic strain, \( \varepsilon_0 \) is a constant representing the elastic strain to yield and \( \varepsilon_{S0} \) is a base strain rate.
Table 3 shows a sample of the flow rule coefficients in Equation (7) for AA5754-O and in Table 4 for AA5182-O, both shown as a function of temperature.

**TABLE 3. Flow stress parameters as a function of temperature for AA5754-O. (Temperature in °C)**

<table>
<thead>
<tr>
<th>Hardening Parameter</th>
<th>Rolling Direction, 0°</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$K(T)$</td>
<td>$558.500 - 1.4152*T$</td>
<td>MPa</td>
</tr>
<tr>
<td>$n(T)$</td>
<td>$0.35977 - 0.0009727*T$</td>
<td></td>
</tr>
</tbody>
</table>


<table>
<thead>
<tr>
<th>Hardening Parameter</th>
<th>Rolling Direction, 0°</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$K(T)$</td>
<td>$621.350 – 1.6627*T$</td>
<td>MPa</td>
</tr>
<tr>
<td>$m(T)$</td>
<td>$0.00106*\exp(0.0174*T)$</td>
<td></td>
</tr>
</tbody>
</table>

**Results and Discussion**

**Experimental Setup**

Limiting dome height (LDH) tests were conducted using a modified Interlaken 75-ton double action servo press. Detailed information about the press and the forming procedure were reported previously [3]. The experimental setup was used to form 101.6 mm (4 in) diameter hemispherical cups from 1 mm (0.04 in) thick, 177.8 mm (7 in) diameter blanks of both materials under pure stretch conditions. The blank was placed over a draw bead and clamped with a blank holding force (BHF) of approximately 267 kN (60000 lbf) to prevent material from drawing-in during the pure stretch experiments.

Heating elements with an active control device were added to the LDH machine in order to elevate the temperature of the dies and the blank. The active control was achieved with two thermocouples linked to the die and blank system. Additional thermocouples were installed to directly measure the temperature of both the blank and the punch during the forming process. These critical measurements were needed to perform accurate numerical analysis of the experiment.

The experimental procedure at a specific elevated temperature is as follows. The blank was clamped between the dies with three heating element bands wrapped around the perimeter of the dies, which were insulated to minimize heat loss to the environment. The desired temperature was set and maintained for about 20 min or until a constant and isothermal condition was achieved. Temperature was monitored using several thermocouples within the system. The temperature of the punch was not controlled independently, and for the current research, the punch temperature was found to be cooler than the blank. With an isothermal blank condition, the punch was then actuated to stretch the blank while recording the punch force-displacement curve. This process was repeated several times for each temperature to establish repeatability. Pure stretch experiments were performed at several elevated temperatures in the range of 25°C – 204°C (77°F – 400°F).

**Coupled Thermo-Structural Finite Element Model**

Finite element analysis (FEA) was performed using the commercial explicit finite element code LS-Dyna to understand the deformation behavior of the aluminum sheet during the thermoforming process. The UMAT option was used to build the user material subroutine in FORTRAN (COMPAQ VISUAL FORTRAN PROFESSIONAL EDITION 6.6B®), which was then linked with the library files supplied by LSTC. The finite element model used in the simulations was first created using Unigraphics® and imported as IGES (Initial Graphics Exchange Specification) files. Hypermesh® was used to create the finite element mesh, assign the boundary conditions and to build the LS-Dyna input deck for the analysis. The full size finite element model used approximately 55,000 four- and three-node shell elements. The punch, die, and blank-holder were created using rigid materials (Material 20 in LS-Dyna).

The thermal analysis was performed first, during which the temperature of each element was calculated and supplied as input to the UMAT. Using the temperature value for each element, the temperature-dependent anisotropic material model coefficients were calculated. Before every structural iteration step, two thermal analysis steps were performed with a controlled time step to insure that the temperature update was adequate.

A linear, fully implicit transient thermal analysis was performed with the diagonal scaled conjugate gradient iterative solver type in LS-Dyna. The die and blank materials were assumed to behave with isotropic thermal properties. Table 5 shows the thermal properties defined in the analysis for the die and blank.
TABLE 5. Thermal properties of material used in FEA

<table>
<thead>
<tr>
<th>Material</th>
<th>Density kg/m$^3$</th>
<th>Specific Heat Capacity J/kg.K</th>
<th>Thermal conductivity J/m.K</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rigid (FE)</td>
<td>7.85 E3</td>
<td>450.0</td>
<td>70.0</td>
</tr>
<tr>
<td>Blank (AL)</td>
<td>2.71 E3</td>
<td>904.0</td>
<td>220.0</td>
</tr>
</tbody>
</table>

The lower die, blank holder and punch were assigned a constant temperature boundary condition, while the blank was given an initial temperature boundary condition equal to the upper and lower dies. The temperature of the punch was set to the lower temperature based on experimental data (see Table 6). Thermal properties were assigned to contact surfaces to enable heat transfer at appropriate areas of contact between the blank and die during the analysis. Subsequently, areas of the blank that made contact with the punch lost heat to the punch while the unsupported regions of the blank remained at higher temperatures.

**Failure Criteria**

Failure criteria used in the analysis were based on forming limit diagrams (FLD). FLDs for AA5754-O and AA5182-O at multiple temperatures were calculated with the Marciniak-Kuczynski (M-K) model [6] using Barlat’s YLD2000 anisotropic yield function and appropriate coefficients for each temperature. In the current process, it was assumed that the loading path was sufficiently linear to justify use of a strain-based FLD for accurate assessment of failure in the sheet. For a general forming process in which the loading path may not be linear, it would be necessary to either integrate the M-K model into the FEA to assess each element separately according to its loading path, or to use a stress-based FLD that is less sensitive to strain path [7].

FLD curves used in this study were based on the Voce hardening law, which offers a more conservative prediction of failure compared to the power law. Figure 1 shows the FLD curves for AA5754-O at different temperatures. As seen from this figure, the forming limit curves increase with temperature, suggesting that the materials can be stretched to higher levels before failure occurs. A similar figure was also generated for the AA5182-O material.

![Fig 1. FLD curves for AA5754-O at different temperatures.](image-url)

**Fully Coupled Thermo-Mechanical Analysis**

Finite element analysis of pure stretch experiments was performed with the thermo-structural finite element model described previously using the temperature-dependant UMAT of Barlat’s YLD2000 anisotropic yield function. The purpose of the numerical analysis was to verify the accuracy of both the FEA model and the UMAT to predict failure in the aluminum sheet at elevated temperatures.

Temperatures of the dies, punch, and blank were measured with thermocouples as listed in Table 6. Since the material model is temperature-dependant, these temperature values were input to the numerical model to insure accurate analysis corresponding to the experimental results.

**TABLE 6. Measured temperatures of dies, punch and blank during experiments.**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>25</td>
<td>25</td>
<td>25</td>
</tr>
<tr>
<td>38</td>
<td>38</td>
<td>38</td>
<td>38</td>
</tr>
<tr>
<td>93</td>
<td>94</td>
<td>91</td>
<td>93</td>
</tr>
<tr>
<td>149</td>
<td>147</td>
<td>122</td>
<td>140</td>
</tr>
<tr>
<td>177</td>
<td>172</td>
<td>141</td>
<td>174</td>
</tr>
<tr>
<td>204</td>
<td>205</td>
<td>171</td>
<td>202</td>
</tr>
</tbody>
</table>

Figures 2 and 3 show the experimental and the fully coupled thermo-mechanical simulation results of pure stretching of AA5182-O at room temperature (25°C) with failure punch depth and failure locations indicated. Numerical simulation accurately agrees with the experimental observations of forming depth and failure location.
Figures 4 and 5 show the experimental and fully coupled thermo-mechanical simulation results of pure stretching of AA5182-O at a temperature of 149°C (300°F) with failure punch depth and failure locations indicated. Again the experimental results accurately agree with the numerical prediction of failure location and forming depth at this elevated temperature.

Figure 6 shows a comparison between experimental and numerical results of the punch load vs. punch depth curve at several elevated temperatures for AA5182-O. The fully coupled thermo-mechanical model was capable of accurately predicting the punch load curves.

Figures 7 and 8 show the experimental results and the fully coupled thermo-mechanical simulation of pure stretch for AA5754-O at a temperature of 177°C (350°F), indicating the failure punch depths and failure locations, which confirm that the simulation agrees with observations.
Finite element analysis with the developed thermo-mechanical constitutive model accurately predicted both the deformation behavior and the failure location in the blank and compared favorably to the experimental results. The current research shows the importance of using both thermal analysis and an anisotropic temperature-dependant material model in a fully coupled mode in order to quantitatively model the warm forming process.

Although the current thermoforming analysis was only verified for biaxial stretching, its application to more complex parts is expected to yield accurate results. This is because the accuracy of the YLD2000 yield function at room temperature has already been thoroughly verified by many researchers. Therefore, to expect a similar performance at elevated temperature is not unreasonable. However, to further verify the accuracy of this model’s prediction, the authors plan to conduct thermoforming of a deep-drawn automotive part in the near future and compare the results with numerical predictions.

ACKNOWLEDGMENTS

The authors would like to thank General Motors for financial support of this research project. The authors especially would like to thank Drs. Paul Krajewski and Anil Sachdev from the GM R&D Center for their assistance and helpful discussions in support of this research.

REFERENCES