On thermal compensation of Hot-Form-Quench stamping die

D Szegda1,*, M Mohamed1 and M Ziane2

1Impression Technologies Ltd, Unit E Lyons Park, 46 Sayer Dr, Coventry CV5 9PF, United Kingdom
2ESI Group, 100-102, Avenue de Suffren, 75015 Paris, France

Abstract. Today, most premium and EV vehicle body in white structures rely heavily on lightweight materials and lightweighting technologies to offset vehicle mass. Increased utilisation of lightweight metals is also being observed in mainstream vehicles. Press hardened steels have gained significant use, however high and ultra-high strength aluminium alloys are starting to follow as their usability is enhanced and exploited. Both material groups require hot forming technologies to be used in manufacture of vehicles components. Thermal expansion of aluminium alloys is approximately twice those of steel, and hence leads to significant material contraction during in-die quenching. This is especially pronounced for large area or long components. Therefore, stamping dies employed in the Hot-Form-Quench (HFQ) process used to manufacture lightweight components from aluminium alloys are required to be compensated for thermal expansion/contraction. This paper presents simulation methodology developed in PAM-STAMP and used to verify the level of die compensation required and the effect of the compensation on the part dimensional conformance after HFQ forming.

1. Introduction
The main source of shape distortion and geometric tolerance noncompliance in cold sheet metal forming is springback. This major drawback for cold forming is a phenomenon that occurs when the forming force applied to the blank is released. The deformed material tends to partially return to its original shape as an attempt to release the internal elastic residual stresses. Computer simulations relying on Finite Element Method coupled with advanced material constitutive models were developed to tackle the springback problem. This allowed to predict springback when accurate material model was established and ultimately use the simulation environment, such as PAM-STAMP, to design compensated tool faces. However, in hot stamping, the issue proved to be more complex than in cold forming [1-3]. At elevated temperatures the material response becomes viscoplastic, i.e. rate dependent, whilst the process is inherently non-isothermal due to contact with cold tools and ambient air. Moreover, quenching phase is defined by mechanical boundary condition applied through contact which is characterised by contact pressure distribution leading to non-uniform cooling and, hence, non-uniform thermal contraction. The complex interactions between the thermal and mechanical conditions can often lead to difficulties in accurate simulation of the process leading to inaccuracies in prediction of the thermo-mechanical distortion. Without accurate distortion prediction effective tool compensation is impossible.

This paper describes a modelling approach developed for prediction of the complex thermo-mechanical distortion for HFQ process which enables effective design methodology for tool compensation.
2. Mathematical Background To Prediction of Thermo-Mechanical Distortion Using Thermo-Viscoplastic Material Model with Damage.

2.1. The Analytical Continuum Damage Mechanics (CDM) model

The CDM Material is a viscoplastic-damage isotropic constitutive model that takes the mechanisms of dislocation-driven evolution processes such as hardening, dynamic and static recovery and damage into account [4, 8]. The basic principles and main equations of the complete CDM are presented below.

\[
\bar{\sigma}_{ij} = \frac{\sigma_{ij}}{(1 - \omega)} = D_{ijkl} \varepsilon_{kl}^p
\]

\[
\dot{\varepsilon}_{ij}^p = \frac{3}{2f} s_{ij}
\]

\[
\dot{\rho} = \left( \frac{2}{3} \dot{\varepsilon}_{ij}^p \dot{\varepsilon}_{ij}^p \right)^{1/2}
\]

\[
R = B \rho^{n_1}
\]

\[
\dot{\rho} = A (1 - \rho) \dot{\rho} - C \rho^{n_2}
\]

\[
X(\sigma) = \frac{\alpha_1 J_0(\sigma) + \alpha_2 J_1(\sigma) + \alpha_3 J_2(\sigma)}{\sigma_1 + \alpha_2 + \alpha_3} J_2(\sigma)
\]

\[
\dot{\omega} = \Delta X^p \eta_1 \dot{\rho}^{n_2} \frac{1}{(1 - \omega)^{n_3}}
\]

Where \( p \) is the equivalent viscoplastic strain, which could be also written \( \varepsilon_{eq}^p \), \( f \) is the equivalent Von Mises stress, (also written as \( \sigma_e \)), \( s_{ij} \) is the deviatoric stress tensor. \( \omega \) is damage variable, \( D_{ijkl} \) is elasticity tensor in which the Young modulus is assumed to depend on temperature. The evolution of dislocation density \( \rho \) is related to the equivalent viscoplastic strain rate, it includes the dynamic recovery and the static recovery which appear at high temperature [5]. Parameters \( A, B, C, K, n \) are functions of temperature. The damage criterion is a combination of the three invariants of the stress tensor \( J_0(\sigma), J_1(\sigma), J_2(\sigma) \), which are respectively the maximum principle stress: \( J_0(\sigma) = \sigma_1 \), the first invariant: \( J_1(\sigma) = tr(\sigma) = 3 \sigma_H \), the second invariant: equivalent stress \( J_2(\sigma) = f = \sigma_e \). The parameters \( \alpha_1 \) is temperature dependent \( \alpha_2 \) is strain rate dependent and \( \alpha_3 \) is constant. The three invariants together enable the representation of two different damage mechanisms, namely grain boundary damage and ductile damage. The parameters \( \eta_1, \eta_2 \) are assumed to be functions of temperature and parameters \( \eta_3, \phi, \Delta \) are assumed to be temperature independent. The damage parameter defined in Equation (7) is assumed to be equal to 0 at the initial state of the deformation. When the damage level reaches 0.7, it is assumed that failure takes place in the material [10]. Due to the exponential nature of the damage accumulation, as the damage increases from 0.7 to 1.0, the strain increment is negligible and can be omitted.

2.2. Coupling material resolution with dynamic-thermal analysis

In this paper we present a general concept for the numerical treatment of thermomechanical transient equations. We consider a weak formulation of the general thermomechanical equations. The heat equation and the relations governing the stress field are considered separately, i.e. weak coupling.

The interaction of temperature and deformation during plastic flow appears in various forms. A thermal field influences the material properties, modifies the extent of plastic zones, but also the deformation induces changes in the temperature distribution [5-7].

The weak form of thermal equations is applied in thermal analysis:
\[ \int_V T^* (\rho_m \dot{H} - \text{div}(\lambda \text{grad} T) - r) \, dV + \int_{S_q} T^* (\lambda \text{grad} T \cdot n - q) \, dS = 0 \quad (8) \]

\( T \) is temperature, \( T^* \) is the test function, \( H \) is the material enthalpy, \( \lambda \) is thermal conductivity, \( r \) is the heat source inside the volume of the part, for example due to plastic strain \( r = \sigma : \dot{\varepsilon}^p \), \( q \) is the heat exchanged on external surface \( S_q \) with normal vector \( n \), for example air convection or thermal contact. Integration by parts allows to lower the order of the derivative of the temperature field.

\[ \int_V T^* \rho_m \dot{H} \, dV + \int_V \text{grad} T^* \lambda \text{grad} T \, dV = \int_V T^* r \, dV + \int_{S_q} T^* q \, dS \quad (9) \]

We assume that temperature may vary along the thickness of the shell elements. The spatial discretization of the parts assigns several temperature degrees of freedom along the thickness to each node inside one shell element. We use the shape functions in shell elements \( N(x, y, z) = F(x, y) \cdot G(z) \) with a separate dependence of temperature in the plane \( x, y \) of the shell and along thickness \( z \), \( G(z) \) defines a piecewise linear interpolation. We also use the shape functions \( F(x, y) \) in surface elements, \( N_c \) in the thermal contact elements, the matrix \( B^T \) which defines the temperature gradient \( \text{grad} T \), the degrees of freedom \( T^e \) (odd number 3 to 9 at each node) in the relations \( T(x, y, z) = N(x, y, z) \cdot T^e \). \( T^* \) is the global vector of degrees of freedom:

\[ T^* (C^s \cdot \dot{T} + K \cdot T - Q_v - Q_s - Q_c) = 0 \quad \forall T^* \]

\[ C^s \cdot \dot{T} + K \cdot T - Q_v - Q_s - Q_c = 0 \quad (10) \]

\( C^s \) is the global specific heat matrix assembled from the local matrix in each element. As in dynamics, \( C^s \) is diagonalized to save computation time.

\[ C^{se} = \int_{\Omega^e} N^T N \rho_m \frac{dH}{d\Theta} \, dV \quad (11) \]

\( K \) is the global conduction matrix assembled from the corresponding local matrix:

\[ K^e = \int_{\Omega^e} B^T \lambda B^T \, dV \quad (12) \]

\( Q_v, Q_s \) are the global thermal loads obtained from the assembly of \( Q_v^e, Q_s^e \) :

\[ Q_v^e = \int_{\Omega^e} N^T r \, dV \quad (13) \]

\[ Q_s^e = \int_{S^e} F^T q \, dS \quad (14) \]

\( Q_c \) is the global heat due to contact resulting from the assembly of the thermal contact elements.

\[ Q_c^e = \int_{S^e} N_c^T q_c \, dS \quad (15) \]
With:

\[ q_c = h(\delta, P_c) (T - T_{Tool}) \]

The heat transfer coefficient \( h \) depends on the gap \( (\delta) \) and the contact pressure \( (P_c) \). We note that the material resolution fixes the internal source load \( r \), the dynamic resolution fixes the geometry used in the thermal analysis \( V, S_q, S_c \) and the contact heat flux \( q_c \). Inversely the temperature field \( T \) fixes the value of material properties and the thermal strain \( \varepsilon^{th} \).

3. Model verification

Implementation of the thermo-viscoplastic model with damage was verified using simple simulation setup in PAM-STAMP 2020.0. A straight strip with dimensions of 1000x200x2 mm was used to simulate the strip cooling. Two simulations were setup, one allowing the strip to contract freely and another with mechanical boundary condition preventing the strip from longitudinal contraction arising from the specimen cooling.

In the first model, the strip was fixed in the longitudinal direction along the left edge, and in addition, the top left corner was also fixed in the transversal and normal direction to prevent free body motion. The opposite end of the strip was unconstrained as shown in Figure 1a. The strip was cooled from initial temperature of 520°C down to 20°C. Thus, as expected, the strip contracted without building any residual stresses. As is shown in Figure 1b the total strain was simply equal to thermal strain and since no mechanical (elastic) strain was accumulated the resultant stress level was also zero. This exercise allowed verification of calculation of thermal strains exchanged between the implemented material constitutive model and PAM-STAMP solver.

In the second model, the strip was mechanically constrained along the longitudinal direction and cooled. Longitudinal boundary condition was applied to left- and right-hand side edges of the strip. The longitudinal strain was zero but some mechanical, plastic and elastic, strain was accumulated during cooldown from the initial temperature of 520°C to 20°C, as is shown in Figure 2. Stress level built up as a result of elastic strain. It is the so-called “thermally induced stress”.

![Figure 1](image1.png)

**Figure 1.** Unconstrained strip cooling model.
Figure 2. Mechanically constrained strip cooling model.

4. Model Validation
This section presents the validation process of the implementation of the viscoplastic model. In this work, a lab scale U-shape part was formed under two different conditions to provide a data set for model validation. The first test involved forming of the U-shape part at low speed followed by reduced-time in-tool quenching under load and the second test was to form the part under high speed followed by longer-time quenching.

4.1. Case 1: Low forming speed followed by reduced time quenching.
As previously mentioned, the thermomechanical distortion is strongly influenced by the quenching stage and is assumed to be driven by the temperature gradients in the panel. This hypothesis is based on the uneven material shrinkage due to varying temperatures at different locations of the panel. It is expected that forming the panel with low speed followed by quenching with short time increase the temperature gradients and magnitude upon part release. This, in turn, increases uneven distribution of the arising strains and, hence, residual stresses in the part and ultimately, as is shown in Figure 3, distortion of the part. Detailed comparison of shape distortion between the simulation and experiment is shown in Figure 4 which shows overlay of the scan of the formed part with the simulated part. Three sections have been cut to examine the agreement of the two parts in different locations. As can be seen in Figure 4 a very good agreement between the simulation predicted shape distortion and the actual part scan was achieved using the thermo-viscoplastic material model with damage.

Figure 3. U shape part with low forming speed and short time quench, a) Simulation, b) Experiment.
4.2. **Case 2: High forming speed followed by typical time quenching.**

Increasing the forming speed and quenching time has a significant effect on reducing the temperature gradient on the panel which results in reduced uneven distribution of stresses. Therefore, the distortion of the part is significantly decreased as shown in Figure 5. Similar to the previous case, the scanned part is overlayed with the simulated part and the three sections have been cut showing a good agreement of the two parts in different locations as shown in Figure 6.

**Figure 4.** Overlay of Scan physical part and FEM predicted model (low forming speed and short time quench).

**Figure 5.** U shape part with high forming speed and long time quench, a) Simulation, b) Experiment

**Figure 6.** Overlay of Scan physical part and FEM predicted model (high forming speed and long-time quench).
5. Conclusions

This paper presented implementation of the thermo-viscoplastic material model with damage developed for simulation of HFQ process and its use for prediction of thermomechanical distortion which may arise during HFQ forming of aluminum parts. The material model together with the dedicated simulation methodology established in PAM-STAMP for prediction of distortion were verified and validated against lab scale U-shape component. It was also demonstrated that part distortion depends strongly on process conditions and when process conditions deviate from the recommended settings distortion can increase significantly. This becomes particularly obvious when forming speed is too low or quenching is not well controlled giving rise to significant temperature gradients and residual stresses.

The simulation approach discussed here was tested against two test cases with very different process parameters and very good correlation was found in both. This confirms the model versatility and wide range of applicability spectrum.

Further publications will present the capability of the model to compensate tool faces and discuss the achieved dimensional control and conformance against specification.

References

[1] Alistair F, Mohamed M, Jianguo L, and Trevor D 2008 An investigation of lubrication and heat transfer for a sheet aluminium heat, form-quench (HFQ) process (Steel Res. Int, 79-11-VII) pp 133–140.
[2] Mohamed M 2010 An investigation of Hot Forming Quench process for AA6082 aluminium alloys (PhD Thesis, Imperial College London).
[3] Mohamed M, Alistair F, Jianguo L, Daniel B, and Trevor D 2012 Investigation of deformation and failure features in hot stamping of AA6082: Experimentation and modelling, Int. J. of Machine Tools and Manuf. Vol. 53 pp27-38.
[4] Jianguo L, Mohamed M, Daniel B, and Trevor D 2014 The Development of CDM-based theories for predicting FLD for hot stamping applications, Int. J. of Damage Mechanics. Vol. 23 (5), pp. 707 – 724.
[5] PAM-STAMP; User Guide, 2020.
[6] Thomas J R Hughes The Finite Element Method Linear Static and Dynamic Finite Analysis, Prentice-Hall, INC, Englewood Cliffs, New jersey 07632, ISBN 0-13-317025.
[7] Lugnberg M, and Netz T Investigation of thermal spring back of a hot formed 2MnB5 A-pillar with tailored properties (Dissertation). Retrieved from http://urn.kb.se/resolve?urn=urn:nbn:se:hy:diva-10108.12.
[8] Mohamed M, Norman D, Petre A, Melotti F and Szegda D 2018 Advances in FEM Simulation of HFQ® AA6082 tailor welded blanks for automotive applications, IOP Conference Series: Materials Science and Engineering, v (418).