Evaluation of hot forming effects mapping for CAE analyses

L Knoerr¹, T Faath², J Dykeman³ and S Malcolm⁴

¹,²thyssenkrupp Steel NA, 22355 W. 11 Mile Road, Southfield, MI 48033 USA; ³,⁴Honda R&D Americas, Inc. 21001 State Route 739, Raymond, OH 43067 USA.

Abstract. Hot forming has grown significantly in the manufacturing of structural components within the vehicle Body-In-White construction. The superior strength of press hardened steels not only guarantee high resistance to deformation, it also brings a significant weight saving compared to conventional cold formed products. However, the benefit of achieving ultrahigh strength with hot stamping, comes with a reduction in ductility of the press hardened part. This will require advanced material modeling to capture the predicted performances accurately. A technique to optically measure and map the thinning distribution after hot stamping has shown to improve numerical analysis for fracture prediction. The proposed method to determine the forming effects and mapping to CAE models can be integrated into the Vehicle Development Process to shorten the time to production.

1. Introduction

The demands for improved vehicle safety and fuel economy have intensified with the development of Advanced to Ultra High Strength Steels. However, increased strength has come at the expense of ductility and formability, making parts more challenging to be defect free and meet required dimensional tolerances. To overcome this, ultrahigh strength parts above 1200MPa in tensile strength are typically hot formed by austenitizing Manganese Boron steel blanks in a furnace to 880-950°C, subsequently forming and quenching in a cooled die. Structural components manufactured in this process can attain tensile strength of up to 1500MPa to 2000MPa, with significantly lower thicknesses. Furthermore, the heat treatment can be tailor-tempered to make crash relevant components that can absorb impact energy and have high deformation resistance. A functional component where intrusion resistance is desired, has martensitic hot formed microstructure that exhibits very little ductility and strain to failure. This behavior can be difficult to predict accurately in a crash performance analysis, where uniform part thickness is assume and a basic material model with no inherent damage accumulation criterion specified. This paper evaluates the benefits of validating the numerical analysis of a hot formed hat profiled section part under quasi-static three pointing bending with the experimental study, by incorporating thickness mapping and damage accumulation criterion. The approach of gridding the blanks to optically measuring the strains after hot stamping, and mapping the thickness over to the finite element mesh model is fully addressed here.
2. Experimental Study
The validation study of the hot formed part under three point bending loading condition was conducted in conjunction with an automotive OEM (Honda of America). The hat profiled parts were hot formed in the thyssenkrupp Steel Europe Research facility in Dortmund, Germany, under series-like conditions. The material used is MBW®1500 (22MnB5) steel grade with Aluminum Silicon (AS) coating of 1.5 mm thickness. The geometrical dimensions of the hat profiled part are shown in figure 1 with a 1.6 mm welded back plate of a High Strength Low Alloy (HSLA) material grade. The assembled part is subjected to a three point bending loading condition in the OEM’s universal testing machine, instrumented to acquire the load and stroke curves of the test setup as illustrated in figure 2.

![Figure 1](image1.png)  ![Figure 2](image2.png)

**Figure 1.** Hat profiled dimensions.  **Figure 2.** Three Point Bending test setup.

2.1. Material Test Specimens
The material selected to investigate the hot forming effect on the crash performance is of the Manganese-Boron steel grade and requires at least two tests, such as uniaxial and plane strain tension for material model calibration. The uniaxial tension test conforms to the Japan Industrial Standard (JIS) Z 2241 under quasi-static strain rate of 0.003/s. The geometry of the test specimen is undersized of 32 mm gauge length and width of 6 mm. The dog bone shaped specimens were cut from material blanks after heat treatment, and prepared with speckled pattern for strain measurement using the Digital Image Correlation (DIC) method as shown in figure 3 (a). Another test specimen configured for plane strain loading as shown in figure 3(b) to evaluate the parameters for fracture model calibration.

![Figure 3](image3.png)

**Figures 3.** (a) JIS uniaxial coupon level test.  (b) Plane strain coupon test.

2.2 Blank Preparation
The rectangular blanks developed for the hot forming of the hat-profiled part is 1.5 mm thick, width of 200 mm and 400 mm in length. The blanks are gridded for optical strain measurements after hot forming using the GOM Argus system. The gridding process as shown in figure 4(a), etched on the blank (b) utilized a patented varnish developed by Sikora et al at thyssenkrupp [1] that will not fade in the hot furnace. The hot stamped part (c) has visible grid markings for strain measurements and mapping over to a numerical model for crash performance evaluation.

![Figure 4](image4.png)

**Figures 4.** (a) Gridding process.  (b) Etched blank.  (c) After hot stamping.
2.3 Experimental Three Point Bending
The bending experiments were conducted on a universal test machine at Honda Research Laboratory with a punch speed of 5 mm/s. Figure 5 illustrates the hat profiled assembled part set on two stationary roller fulcrums at equidistance from the moving hemi-sectional punch of 50 mm radius and 115 mm width under a certain deformation load. The punch load transmitted thru’ the load cell and the stroke via a LVDT (Linear Variable Displacement Transducer), provided the signals to the data logger which synchronized and filtered the noise to output a smooth force-stroke curve as depicted in figure 2 to the maximum distance of 70 mm. The fracture modes of the assembled parts at maximum stroke are displayed in figure 6. The raw force-stroke data acquired correlates to the predicted results of the virtual three point bending simulation using an advanced fracture model for the validation study.

![Figure 5. Hat profiled assembly under load.](image)

![Figure 6. Fractured parts at maximum stroke.](image)

3. Numerical Study
A comprehensive numerical study that incorporates the thinning distribution of a hot stamped component under three point bending, was performed using an advanced material model that can reliably predict fracture under different loading conditions. The constitutive model was formulated on the basis of shear-modified Gurson model [2], by simplifying the governing equations and implementing them in the shell mechanics framework [5, 6, 7]. The shell-based formulation is the only effective approach to analysing large scaled ductile problems, such as vehicle crashworthiness assessment. The calibrated model, was conducted using only two coupons level tests: uniaxial and plane strain tension. The initial analyses conducted with assumed uniform thickness of hat profiled mesh model was compared with thickness mapping from the actual optically measured part. Excellent correlation between the experimental and predicted results was observed on the mesh model with the thickness mapped, validating the fracture prediction methodology utilized.

3.1. Thickness Mapping
The gridded hat profiled part shown in figure 7 is optically measured following the procedural method outlined in GOM Argus system’s manual [3]. The measured 2D strain tensors are processed in the system’s software to compute the thinning distribution which is the more profound effect of hot forming with the limited residual ductility on the finished part. The thickness/thinning mapping workflow as outlined in the manual, includes initial best fit alignment of the measured to the mesh part, then projection followed by interpolation of the nodal thickness values to the mesh model. The resultant mesh model of the mapped distributed thickness values as shown here in figure 8 can be plotted and exported in various commercial finite element solvers’ formats.
3.2. Material Characterization and Model Calibration

Calibration of the ductile fracture constitutive model used in the simulation requires that effective material characterisation after localized necking to failure is performed. This is necessary to account for the influence of multi-axial stress states on ductile fracture. This process can be extensive, considering the numerous coupon level tests typically required for full calibration of ductile fracture models. In this study, only two coupon level tests were used for model calibration. This was possible due to the micromechanically motivated constitutive formulation that explicitly accounts for underlying ductile fracture mechanisms [5, 6, 7]. The resulting fracture locus is as shown in figure 9. Despite limited datasets, notable validation is accomplish as discussed in the following section.

3.3 Fracture Prediction Methodology

The fracture simulation methodology used here allows for reliable prediction of fracture initiation and propagation in thin sheets under multiaxial stress states. The model is based on an advanced dilatational plasticity model with a scalar damage variable governing softening behavior [4, 5]. The elasto-plastic damage criterion for ductile metals is fundamental in capturing the macroscopic response under large strain and varying stress triaxiality. Consequently, accurate experimental material characterization and systematic calibration of the model parameters are imperative to enhance fracture prediction capability. An approach to automate the iterative calibration process was accomplished using custom build software that includes a user-friendly Graphical User Interface [6]. The constitutive equations governing the model formulated in the framework of shell mechanics are suitable for large structural simulation like crashworthiness for which the use of solid elements is time and cost prohibitive. The described formulation as referenced earlier was developed and implemented by principals mentioned in [7, 8]. It is also available as a user-material in ABAQUS and validated against several large-scale shell deformation case studies.
3.4 Analysis Validation
A calibrated model obtained as outlined in the method described above, was utilized for the quasi-static three point bending simulation of a hot formed hat profiled part which is representative of an impact beam under side crash loading condition. Two analyses case scenario of the part mesh model were performed, one with the uniform part thickness assuming no forming effects, and the other with thickness/thinning mapping over the mesh part model. The predicted results for the two case study at maximum stroke are displayed in figures 10(a) and (c), as compared to the actual experimental part (b). The results clearly show the close validation of the latter case with the thickness mapped model that captures the fracture mode behaviour and the force-stroke curve more precisely. A detailed evaluation of the fracture mode revealed that the test part was under complex multiaxial stress states of bending of nonlinear strain paths with dominant crack initiated by in plane shear deformation. Overall, excellent correlation with the experimental test results was achieved.

4. Conclusion
Hot forming with its ability to stamp lightweight components to meet the demands for stringent fuel efficiency and vehicle safety has proliferated since the past decade as the preferred manufacturing process. The resultant ultrahigh strength low ductility however, has prompted application of these manufactured components to be very selective like bumper beams, roof rails, side impact beams etc. Early concept design to evaluate vehicle structural, fatigue and crash assessments can be improved by utilizing advanced constitutive material models that can accurately predict fracture initiation and propagation as investigated in this study. The process of material selection based on forming and performance requirements can be incorporated with a modular software to commercial numerical solvers that calibrates the model parameters for the prediction methodology utilized. Mapping techniques of forming effects from simulation or actual measurements can be integrated seamlessly in the vehicle development process to improve the prediction capability of any numerical analysis.

References
[1] Sikora S et al 2012 United States Patent No. US 8,215,143,B2
[2] Nahshon K and Hutchinson J W 2008 Modifications of the Gurson Model for shear failure European Journal of Mechanics A/Splids 27 1-17
[3] GOM Optical Measuring Techniques 2009 FEA Comparison Argus Manual Chapter 7 v6-2
[4] Voyiadjis G Z and Woelke P 2006 Elasto-Plastic and damage analysis of plates and shells Int. J Solids Structures 2008 Springer -Verlag
[5] Woelke P, Voyiadjis G Z and Perzyna P 2006 Elasto-plastic finite element analysis of shells with damage due to microvoids Int. J Num. Meth. Eng Vol 68 930 pp 338-380
[6] Woelke P, Bentowitz B, Hiriyur B, Knoerr L and Hutchinson J W 2014 Ductile Fracture Prediction for Automotive Applications Int. Automotive Body Congress
[7] Woelke P and Abboud N 2012 Modeling fracture in large scale shell structures J of Mechanics and Physics of Solids 60 12 2044-2063
[8] WAimat Suite: www.waimat.com