Paper 113, ENG 106

Post-Formed Mechanical Properties Prediction for CAE Crash Applications

Payam H. Matin and Ali Eydgahi
Department of Engineering and Aviation Sciences
University of Maryland Eastern Shore
phmatin@umes.edu and aeydgahi@umes.edu

Abstract

Computer Aided Engineering (CAE) has been widely used by automakers for Crash Simulation of new cars during design and development process. Finite Element Method (FEM) is used for CAE Crash Simulation, where motion of elements is determined over very short periods of time during the plastic deformation caused by an impact. Mechanical Properties of structural components need to be specified for the FEM-based Crash Simulation software before simulation starts. These components mostly stamped or hydro-formed have very different mechanical properties than those of as-received sheet metal before forming operation. Common practice is to determine these properties at the locations of interest by tension testing of tensile coupons cut out from these locations on the components. There are many difficulties associated with this common practice such as recognizing critical locations, marking coupons, efficiently cutting samples from complicated areas, machining samples, and etc. All these complexities lead to higher cost and longer time of development. To avoid these difficulties, an analytical model/method is developed and proposed in this work to deliver post-formed mechanical properties of formed components. The model takes the as-received mechanical properties of the component and the level of the plastic strain that each element on the component has experienced during forming process to calculate post-formed mechanical properties for each element on the component. The strain data can be extracted from forming simulation of the component during early stage of die development, and be verified by circle grid analysis of the formed prototype during early stage of die tryout. Trends predicted by the proposed model match well with the experimental evidence. This work provides promising outcomes that significantly reduce the experimental efforts needed for post-formed mechanical properties required for Crash Simulation.

Introduction

Computer Aided Engineering (CAE) has been widely used by automakers for Crash Simulation of new cars during the design and development process. A crash simulation is a virtual recreation of a destructive crash test of a car using computer simulation. Crash simulation is used to examine the level of safety of the car and its occupants. The data obtained from a crash simulation indicate the capability of the car structure to protect the occupants in the event of collision. The important results of a crash simulation are the deformations of passenger compartment and the decelerations felt by the occupants [1].

These values should meet the safety regulations set by the federal government to assure safety of the cars and their occupants. To comply with these federal regulations, auto-manufacturers extensively use crash simulation for different modes of collisions such as frontal impact, side impact and rollover. In the recent years, more advancement has been made since virtual models of crash test dummies and passive safety devices such as seat belts and airbags have been incorporated in the crash simulation models. All these different crash simulations are beneficial since they produce the results without actual destructive testing of a new car. Simulation can be performed quickly and inexpensively giving the opportunity to the design team to modify the design before the actual prototype of the car has been manufactured. Pitfalls and problems can be explored in the early stage of design and development leading to saving of huge amounts of money and time. While simulation provides all these benefits, it may endanger the future of a product if the results delivered are not accurate enough. The importance of accurate results is even more crucial these days as auto-manufacturers highly desire to design lower-weight vehicles to improve fuel efficiency. Thus the efforts that promise to improve the accuracy of crash simulation would be highly desirable.

CAE Crash simulation is commonly based on Finite Element Method (FEM), where motion of elements is determined over very short periods of time during the plastic deformation caused by an impact [1]. The mechanical properties of structural components need to be specified for the FEM-based crash simulation software before simulation starts. These structural components are mostly stamped or hydro-formed. These formed components have very different mechanical properties than those of as-received sheet metal before forming operation. That is because of strain hardening phenomenon that the components have undergone during the forming operation. The mechanical properties are even different from point to point on each component due to the fact that different levels of plastic strain and consequently different levels of strain hardening each component has experienced at different locations. This non-homogeneity of material properties should be incorporated in the crash simulation model to improve the accuracy. Common practice is to determine these properties at the locations of interest by tension testing of tensile coupons cut out from these locations on the components as shown in Figure 1. There are many difficulties and disadvantages associated with this common practice such as:

1. For each individual part, it has to be decided at what locations the tensile coupons should be cut out to represent the properties in a corresponding region. Recognizing critical locations may need engineering calls, which may not be straight forward all the time.
2. Even if the locations of interest are known, it is not always feasible to cut out a sample. That may be due to the geometry of the part at the location of interest or the space available surrounding the location for the cut out. For instance no tensile sample could be cut out from curvilinear locations on a part.
3. So many preparatory efforts need to be taken to have the tensile coupons ready for tension testing. The locations should be accurately marked. Then the coupons should be cut out. Appropriate cutting methods such as laser cutting should be utilized to avoid the undesired material properties change due to heating up the metal. Next the coupons cut should be machined carefully to remove the stress raisers before testing starts.

4. Tension testing needs to be conducted carefully; the results should be collected and validated before being used in simulation.
5. One sample from one location on a particular part is not statistically enough.
6. There are several locations per part that need to be tested.
7. There are many structural parts that need to be tested with the same process.
8. An auto-manufacturer manages many active car programs that need to be supported with similar effort.

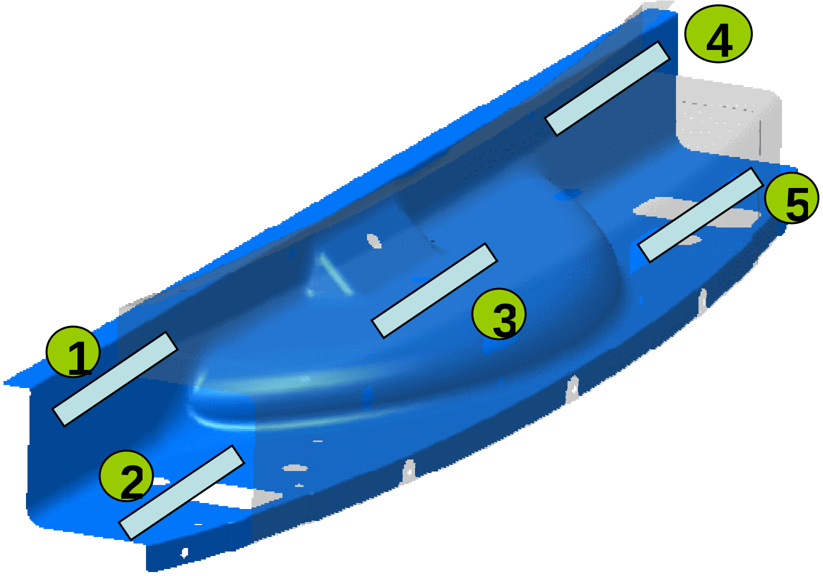

Figure 1: Tensile coupons cut out from the locations of interest for mechanical properties determination.

All these difficulties and disadvantages lead to higher costs and longer development time. An alternative analytical/computational method is highly desirable to replace these experimental efforts.

Objective

The objective of this study is to propose an analytical/computational method/model to calculate material properties of the formed parts for crash simulation applications to avoid and minimize the disadvantages associated with the traditional experimental method.

As shown in Figure 2, the proposed model takes the as-received mechanical properties of the sheet metal before forming operations along with the level of strain that a particular point on the formed part has experienced to deliver the mechanical properties of that particular point on the part. This process may repeat locally from point to point to cover the mechanical properties of all points on the part. The calculated properties of the formed part are then inputted into the CAE crash simulation model for more accurate simulation.

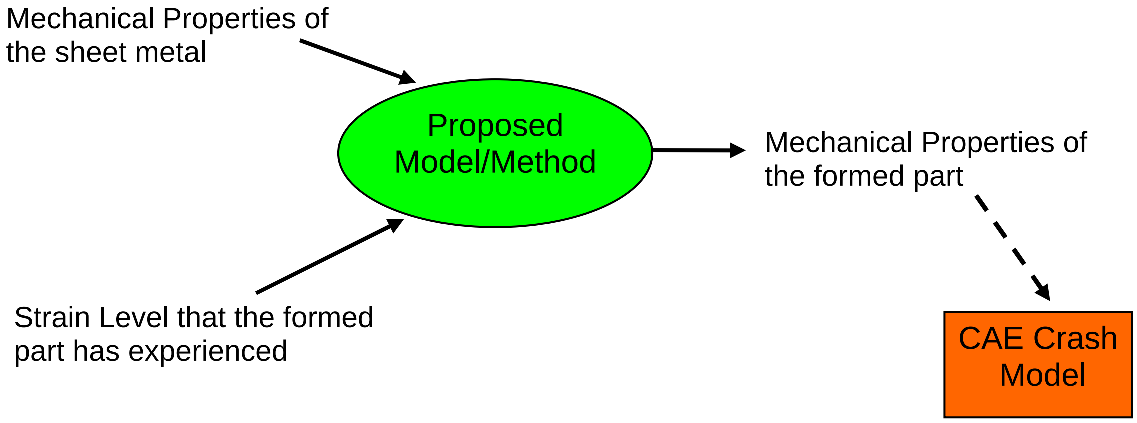

Figure 2: Mechanical properties of the sheet metal along with the level of the strain the formed part has experienced are inputted into the proposed model for post-formed mechanical properties estimation for CAE crash simulation.

Modeling Approach

The material model of as-received sheet metal before forming operation is shown in Figure 3. Depending upon whether the stress exceeds the yield strength of the material, the deformation can be divided into two different deformation zones that are called Elastic and Plastic deformation zones. As shown in Figure 3, the material model may be expressed as [2]:

$$\sigma = \begin{cases} E\varepsilon & \text{where } 0 < \sigma < \sigma_y \quad \text{(Elastic zone)} \qquad \text{(1-a)} \\ K\varepsilon^n & \text{where } \sigma_y < \sigma \quad \text{(Plastic zone)} \qquad \text{(1-b)} \end{cases}$$

$\sigma =$ True stress
$\varepsilon =$ True strain
$E =$ Modulus of elasticity (Yong Module)
$\sigma_y =$ Yield strength of as-received sheet metal (before forming operation)
$K =$ Strain hardening coefficient (K-value)
$n =$ Strain hardening exponent (n-value)

In the elastic deformation zone, the material behaves linearly as opposed to the plastic deformation zone where the material behaves non-linearly.

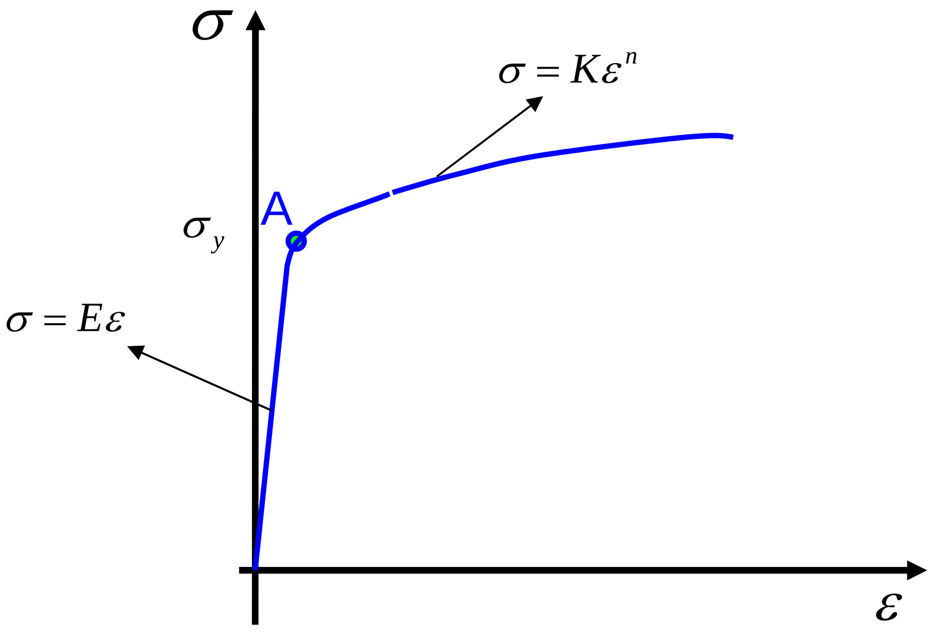

Figure 3: Material model representing elastic and plastic deformation zone

In the Elastic deformation zone, removing the loading leaves no permanent deformation. Conversely, in the Plastic deformation zone, permanent deformation remains after the loading is removed [2].

As shown in Figure 4, when forming operation begins, sheet metal starts to deform linearly in the elastic deformation zone (from point O to point A). As the deformation continues, and the stress level increases, the material enters the non-linear plastic deformation zone (beyond point A), once the stress level exceeds the yield strength of the material σ_y. At the end of the forming operation (at point B) after the part gets released from the die, the loading is completely removed linearly parallel to the elastic line (to point O') leaving permanent deformation behind on the part. This permanent deformation corresponds to a plastic strain ε_P remaining on the formed part as shown in Figure 4.

Now, if the formed part were loaded as a result of a crash incident, the material would not behave as it used to along the curve OAB. Instead, it would start to deform elastically along load-releasing line O'B, and enter the plastic deformation zone at point B, at some higher level of yield strength σ'_y. Then it would continue to deform in the plastic deformation zone to higher level of stress (such as point C). Increase in the yield strength is due to strain hardening phenomenon; The material becomes stronger as a result of the plastic deformation during the forming process. Now that the material strain hardened, a post-formed material model, which lies down on O'BC should be specified for crash simulation model. It should be noted that the hardened material also becomes more brittle as it undergoes repeated cycles of this work-hardening.

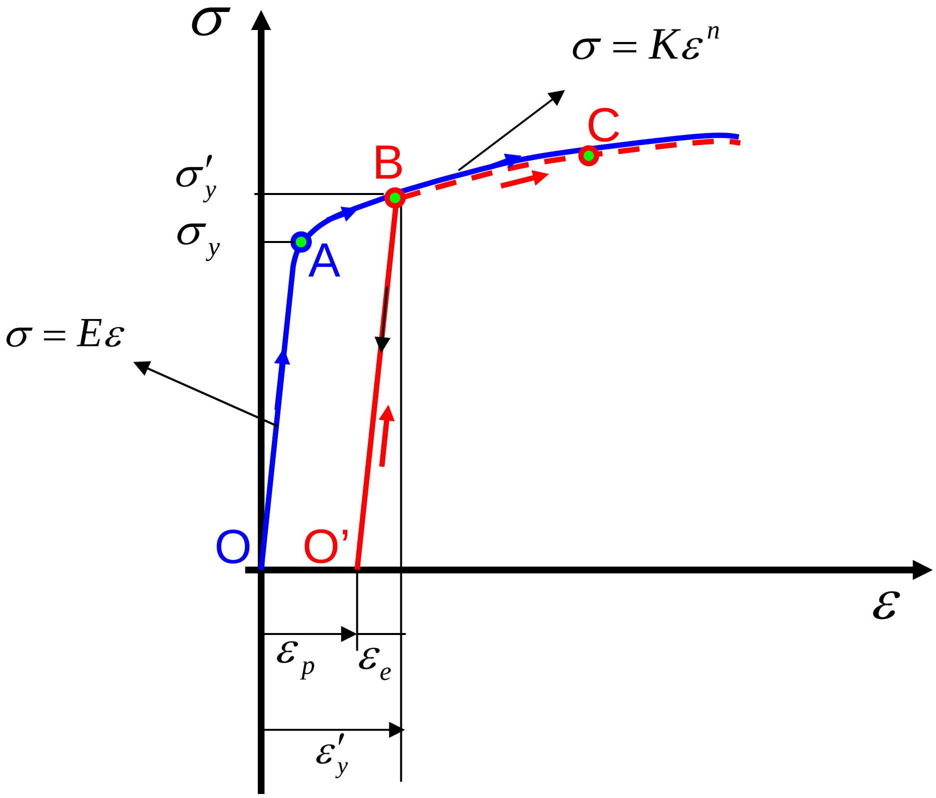

Figure 4: Forming operation occurs along OAB (OA and AB represent elastic and plastic deformation during forming, respectively; Loading removes along BO' at the end of forming process when the formed part releases from the die; Post-formed deformation (such as crash) occurs along O'BC (O'B and BC represent elastic and plastic deformation during crash deformation, respectively)

To estimate the post-formed material properties, first the new yield strength should be estimated. On one hand, point B is on the plastic deformation zone of the as-received material model. That leads to

$$\sigma'_y = K\varepsilon'^{\,n}_y \qquad (2)$$

or

$$\varepsilon'_y = \left(\frac{\sigma'_y}{K}\right)^{\frac{1}{n}} \qquad (3)$$

where

σ'_y = Post-formed yield strength (new yield strength after forming operation)

ε'_y = Total strain corresponding to the post-formed yield strength

On the other hand, point B is right at the end of elastic deformation zone of the post formed material model. That leads to

$$\sigma'_y = E\varepsilon_e \qquad (4)$$

or

$$\varepsilon_e = \frac{\sigma'_y}{E} \qquad (5)$$

where

ε_e = Elastic strain corresponding to the post-formed yield strength σ'_y

As shown in Figure 4, the total strain at point B consists of the plastic strain and elastic strain. Thus

$$\varepsilon'_y = \varepsilon_P + \varepsilon_e \qquad (6)$$

where

ε_P = Plastic strain corresponding to the post-formed yield strength
= Plastic strain the formed part has experienced

Substituting (3) and (5) into (6) leads to

$$\left(\frac{\sigma'_y}{K}\right)^{\frac{1}{n}} = \varepsilon_P + \frac{\sigma'_y}{E} \qquad (7)$$

or

$$\sigma'_y = K\left(\varepsilon_P + \frac{\sigma'_y}{E}\right)^n \qquad (8)$$

Equation (8) contains the post-formed yield strength (new yield) σ'_y, material properties of the as-received sheet metal (E, K, n), and the plastic strain the formed part has experienced ε_P. The as-received mechanical properties of the sheet metal are known. The plastic strain the formed part has experienced may be obtained through circle grid analysis [3] during the die tryout of the part. Alternatively, the plastic strain may be obtained from the results of the forming simulation, which may be available before the die tryout in the early stage of die design and development. Either way, the plastic strain is known. The only unknown is the post-formed yield strength σ'_y. Since equation (8) is non-linear with respect to σ'_y, it should be solved numerically. Since the level of plastic strain ε_P is different from point to point on the formed part, equation (8) needs to be solved for a range of plastic strains to estimate the corresponding post-formed yield strengths σ'_y at different points. Although it is possible to solve equation (8) numerically for σ'_y, there is an alternative solution. Equation (8) may be algebraically re-organized as

$$\varepsilon_P = \left(\frac{\sigma'_y}{K}\right)^{\frac{1}{n}} - \frac{\sigma'_y}{E} \qquad (9)$$

Since it is due to strain hardening, σ'_y is always greater than original yield strength σ_y, a different plastic strain may be calculated for a range of incremented σ'_y (starting from the original yield strength σ_y) using equation (9) as follows

$$\begin{array}{l} \text{Let } \sigma'_y = \sigma_y \quad \ldots\ldots. \quad \varepsilon_{\text{p}} = 0 \\ \text{Let } \sigma'_y = 1.1\sigma_y \quad \ldots\ldots. \quad \varepsilon_{\text{p}} = ? \\ \text{Let } \sigma'_y = 1.2\sigma_y \quad \ldots\ldots. \quad \varepsilon_{\text{p}} = ? \\ \quad \vdots \\ \text{Let } \sigma'_y = \sigma_{TS} \quad \ldots\ldots. \quad \varepsilon_{\text{p}} = ? \end{array} \qquad (10)$$

where $\sigma_{TS} = \text{Tensile strength}$

Plotting out the different values of yield strengths σ'_y versus the different values of plastic strains ε_P calculated, the variation of post-formed yield σ'_y is estimated with respect to the plastic strain ε_P. (Such curve is presented in the results and validation section in Figure 6.)

Now that the post-formed yield strength has been determined, the whole new material model should be estimated. As mentioned earlier, a post-formed deformation (such as crash) occurs along O'BC (O'B and BC represent elastic and plastic deformation during crash deformation, respectively). Although the formed part is pre-strained with the amount of the plastic strain ε_P, any post-formed deformation is considered fresh independent deformation, which starts from zero strain. That means a post-formed deformation treats a formed part as an as-received sheet metal by measuring the strain from the origin as the deformation starts. That implies that the post-formed material model should shift to the left with the amount of the plastic strain ε_P to start the deformation from the origin as shown in Figure 5. A shift to the left with the amount of ε_P coincides the new and original elastic lines O'B and OA respectively. However, the new yield strength σ'_y is positioned at point B', which is at a higher level than the original yield strength as discussed previously. The shift causes the plastic deformation zone simply shifts to the left with the same amount. Using Figure 5, the material model of the post-formed deformation is achieved as:

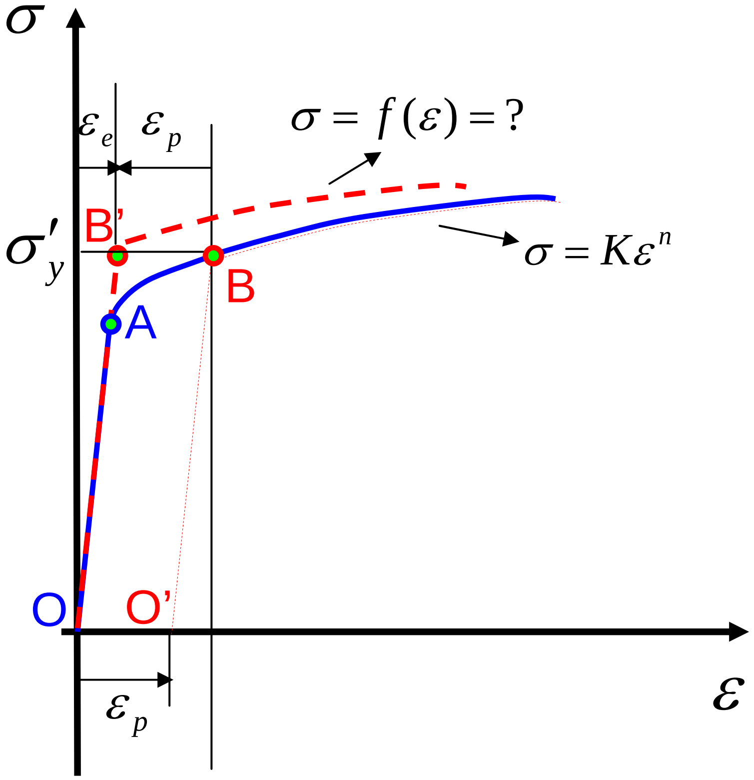

Figure 5: Post-formed material model starting from the origin for crash simulation

$$\sigma = \begin{cases} E\varepsilon & \text{where } 0 < \varepsilon < \varepsilon_e \qquad \text{(Elastic zone)} \qquad (11\text{-a}) \\ K(\varepsilon + \varepsilon_P)^n & \text{where } \varepsilon > \varepsilon_e \qquad \text{(Plastic zone)} \qquad (11\text{-b}) \end{cases}$$

where ε_e is given in (5) as $\varepsilon_e = \frac{\sigma'_y}{E}$, and σ'_y is calculated based on the method explained earlier.

This model should be calculated for different levels of plastic strain ε_P to deliver the material model for each point on the formed part that has experienced such plastic strains during forming operation. (Such an effort is presented in the result and validation section in Figure 7.)

Results and Validations

A vehicle structural part has been selected to estimate the post-formed properties based on the model proposed. The material is coated medium-strength steel with the following mechanical properties:

$E = 200$ Gpa

$\sigma_y = 216$ Mpa
$K = 600$ Mpa
$n = 0.2$

Using equation (9) and the method explained in (10), the post-formed yield strengths σ'_y have to be estimated for different levels of plastic strains ε_P as shown in Figure 6. As expected, the model predicts higher amounts of yield strength for higher plastic strains. That is due to the strain hardening phenomenon where higher plastic strains lead to more hardening. The model also predicts no hardening where there is no plastic strain.

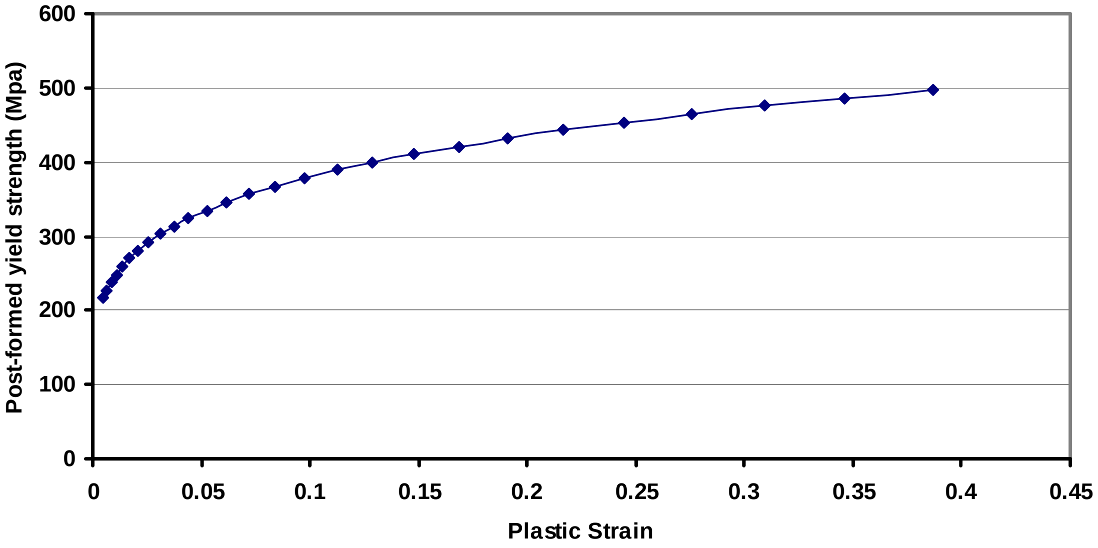

Figure 6: Calculated post-formed yield strength σ'_y as a function of plastic strain ε_P

For the same material, using equations (11-a) and (11-b), and the calculated function of yield strengths with respect to the plastic strain shown in Figure 6, the post-formed material models have been estimated for different levels of plastic strain. These results are shown in Figure 7. The curve with 0% plastic strain corresponds to the original as-received sheet metal before forming operation takes place. As seen in Figure 7, for the higher levels of the plastic strain ε_P, the post-formed material models (stress-strain curves) have been shifted to the left exhibiting higher levels of post-formed yield strength. As a trade off the plastic deformation zone will be smaller for higher ε_P if it is assumed the material can only sustain up to the maximum stress level that the original as-received sheet metal can sustain as discussed in [4-8]. These results can be directly fed into the crash simulation model depending on the level of the plastic strain the formed part has experienced at different points.

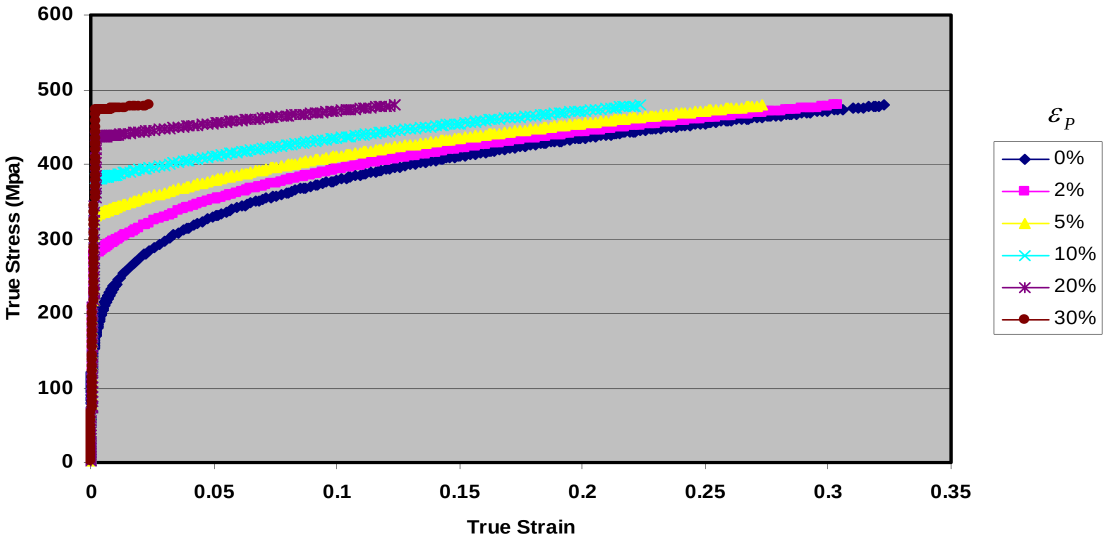

Figure 7: Calculated post-formed material models for different levels of plastic strain ε_P

Some level of validation has been conducted on the model/method proposed using existing data previously collected for other purposes. An L/H (left-handed) structural part shown in Figure 8 was selected.

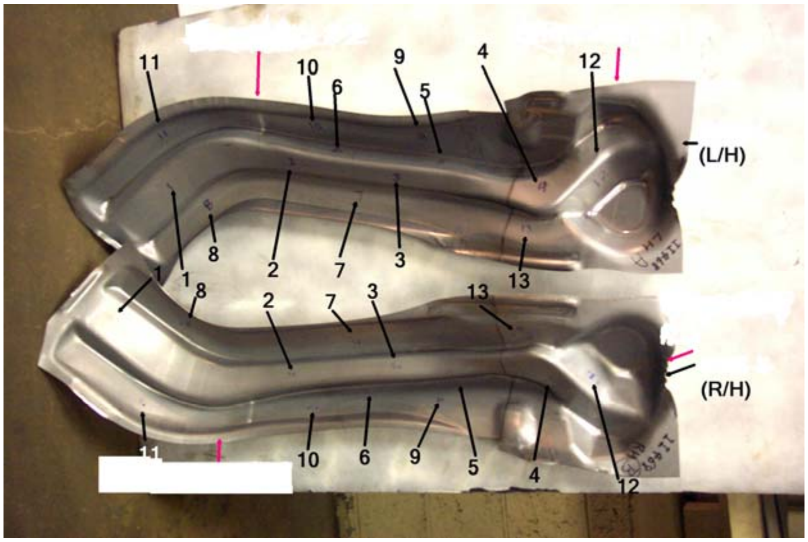

Figure 8: L/H structural part used for testing and validation of the proposed model.

The existing data was available as a result of tension testing conducted on the tensile coupons cut out from different locations of the part. These locations have been shown with different numbers on the part in Figure 8. The material is Dual phase steel with 590 MPa of tensile strength (DP590). The results of the tension testing have been depicted in Figure 9. Similar trends, as compared to what the model predicted is seen in Figure 9. For the areas of the part that experienced less plastic strain (less stretch), lower yield strength and larger plastic deformation zones are observed. These areas are where coupons # 11, #1 and #2 are located. In contrast, for the high stretch areas, where higher levels of plastic strain have been experienced, higher levels of yield strength are observed. As a trade off the plastic deformation zones have become smaller. Furthermore, it is seen that all the curves have limited to approximately the same level of stress. This behavior is similar to the prediction trends of the proposed model. That could be a good indication that the assumption that the material is able to sustain only up to the maximum stress level that the original as-received sheet metal can sustain is a fair assumption. Another set of existing data have been presented in Figure 10 with trends similar to that of the proposed model.

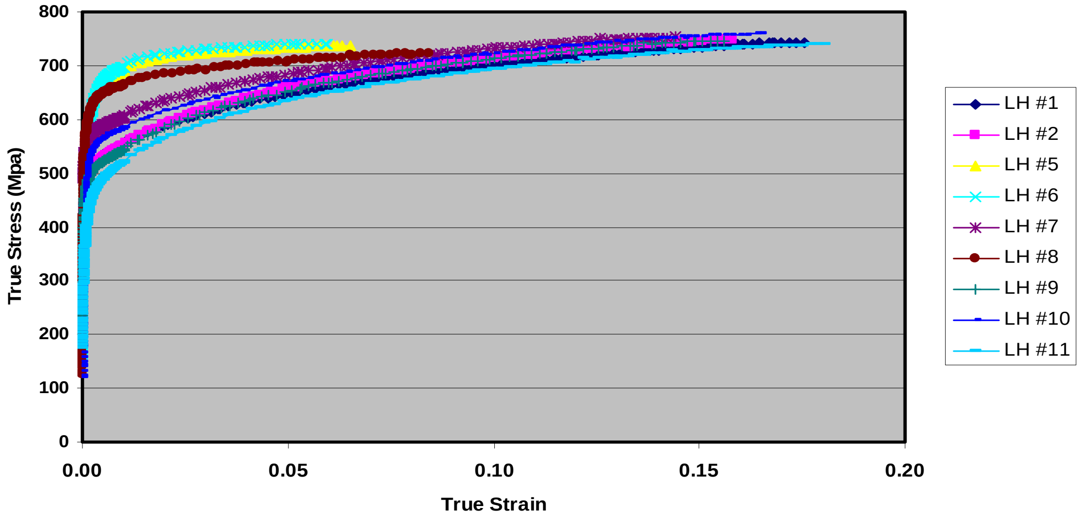

Figure 9: Tension test results conducted on different coupons cut out from a structural part made of DP590 showing similar trends as the proposed model predicts

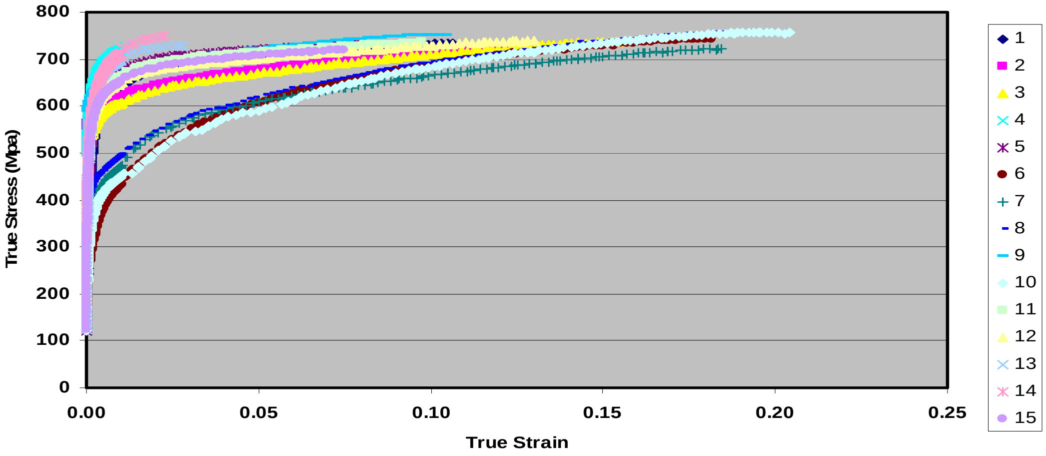

Figure 10: Tension test result on another structural part with similar trends

Conclusion

A model/method has been proposed for prediction of post-formed mechanical properties of structural parts for CAE crash simulation. The model takes the mechanical properties of the as-received sheet metal before forming operation along with the level of strain the formed part has experienced during forming to calculate the post-formed mechanical properties of the part. The trends predicted by the model are similar to the trends obtained by the existing experimental data. The prediction results of the model are promising. The broader impact of the model is to minimize costly time-consuming experimental efforts that auto-manufacturers utilize as a common practice to estimate the post-formed material properties for crash simulation application.

Reference

[1] Haug, E., "Engineering safety analysis via destructive numerical experiments," EUROMECH 121, Polish Academy of Sciences, Engineering Transactions Vol. 29, No. 1, pp. 39–49, 1981.

[2] Hosford, W., Caddel, R., Metal forming, Mechanics and Metallurgy 2nd Ed., Prentice Hall, 1993.

[3] Keeler, S., Circular Grid System – A Valuable Aid for Evaluating Sheet Metal Formability. In: SAE# 680092, 1965.

[4] Stoughton, T.B., “A General Forming Limit Criterion for Sheet Metal forming. International Journal of Mechanical Sciences,” Vol. 42, pp. 1 –27, 2000.

[5] Smith, L.M., Averill, R.C., Lucas, J.P., Stoughton, T.B., Matin, P.H., “Influence of transverse normal stress on sheet metal formability,” International Journal of Plasticity Vol. 19, No.10, pp.1567-1583, 2003.

[6] P.H. Matin and L.M. Smith, “Practical limitations to the influence of transverse normal stress on sheet metal formability,” International Journal of Plasticity, Vol. 21, No. 4, pp. 671-691, 2004.

[7] P.H. Matin, L.M. Smith, and S. Petrusevski, “Methods for stress space forming limit diagram construction for aluminum alloys,” Journal of Materials Processing Technology, vol. 174, pp. 258-265, 2004.

[8] Matin, P.H., “Sheet metal formability: thickness stress and construction in stress space”, Ph.D. Dissertation, Oakland University, 2005.

Biography

PAYAM MATIN is currently an assistant professor at the department of Engineering and Aviation Sciences at the University of Maryland Eastern Shore. Dr. Matin’s research interests are materials processing and manufacturing, experimental mechanics, computational mechanics, manufacturing simulation and mobile robot automation.

ALI EYDGAHI is a Professor and Chair of Engineering and Aviation Sciences Department, Director of the Center for 3-D Visualization and Virtual Reality Applications, and Technical Director of Space Vehicle Mission Planning Laboratory at the University of Maryland Eastern Shore. Dr. Eydgahi has received a number of awards including the Dow Outstanding Young Faculty Award from American Society for Engineering Education in 1990 and the Silver Medal for outstanding contribution from International Conference on Automation in 1995. Dr. Eydgahi has served as a session chair and a member of scientific and international committees for many international conferences and has published more than hundred papers in refereed international and national journals and conference proceedings.