Sheet Metal Forming Simulation of Automotive Body Panels

Michael Y. Sheh and *Longwu Wu*, Cray Research, Inc., Eagan, MN

Abstract

Stamping parts constitute a major portion of an automobile body structure and surface panels. Designing the tooling to produce these stamping parts is one of the major tasks in the vehicle design and manufacturing processes. Historically, the tooling development is experiential-based; many physical try-outs are needed to determine the final tooling and process configurations. The use of modern computer simulation technology helps reduce the amount of tryouts and points the design to the right direction. This significantly reduces the cost and time in stamping tooling development.

Introduction

There are about two to four hundred stamping parts in a typical passenger car construction. Most of them, such as brackets, have simple geometry and are relatively easy to manufacture. However, a good portion of the body frame structure and panels requires more sophisticated manufacturing process to meet their quality requirement. These includes medium size stamping parts such as rail sections and large parts such as door inner and outer panels, front fenders, and rear quarter panels. They require certain levels of precision, or dent resistance, or surface quality, or all of above. Tooling design for these parts are generally labor and material intensive, not to mention the time spent on the various tasks involved.

The tooling development begins with the part design, usually in the form of computer CAD data or engineering drawings. The manufacturing engineer then lays out a preliminary stamping process based on experience or documented guidelines. Using the part design and stamping process information, the tooling engineers then create the punch and die surfaces, design the addendum and binder rings, and determine the blank size. Draw bead and/or lock beads are also defined, if they are needed. The tooling design is then send to the tool and die shops to develop the prototype tooling.

Based on the tooling design, tool and die shops create a series of wood and foam patterns. They are later used in loss-foam zinc alloy casting to produce the tool set: punch, die and binders. The zinc alloy is a relatively soft and machinable material compared to tooling steel. It is a common material for prototype tooling. The punch and die surfaces are further smoothed using a milling machine. Manual grinding is also typical to spot the tool set. The finished tooling, so called soft tool, is then placed on a stamping press to check if it can produce an acceptable part.

Several criteria are used in determining the part manufactuability, they include: split or fracture, wrinkle, metal thinning, and surface quality. If the try-out part is not acceptable, changes are needed to reach a solution. These could be modification of the tooling design, or part design, or the stamping process configuration, and sometimes, change of the sheet material, or all of the above. Either way, changes are made and the process repeats until an acceptable try-out part is produced. This iterative, try-an-error process, as shown in Figure 1, is time consuming and costly.

Advanced computer simulations replace a large amount of physical try-outs by simulating the sheet metal forming process on a computer without building test dies and going through the labor-intensive, try-and-error process. As depicted in Figure 1, tooling engineers can now build finite element models of the tooling and perform nonlinear analysis of the forming process to determine the part formability. This results in significant savings in dollar and time spent on try-outs.

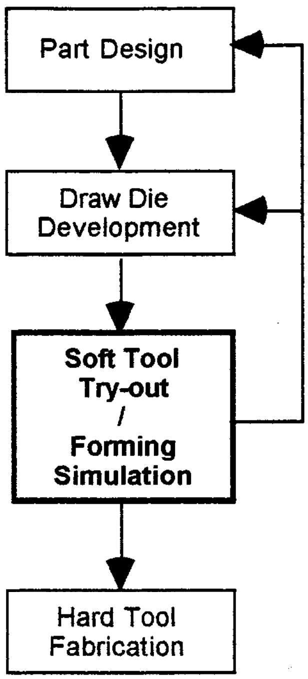


Figure 1. Tooling development Process

The ability to assess manufactuability using computer simulation also allows manufacturing planning and R&D to begin early in the vehicle design process without investing heavily in try-out tooling. It also enables concurrent engineering in vehicle development.

Sheet Metal Forming Simulation

During the past few years, advanced numerical techniques have been developed to simulate sheet metal forming process. In fact, the same technology has been used in vehicle crash simulation since the mid-80's [1,2]. However, forming simulation requires a higher level of accuracy than crash simulation. These compute-intensive calculations are only possible with the availability of high performance computing equipment.

There are many forming simulation algorithms developed over the years [3-7]. In general, each approach has its own purposes and provides different levels of reliability and accuracy. In recent years, nonlinear explicit finite element (FE) techniques has gained wide acceptance in the forming simulation communities because their success in simulating vehicle crash events [8-10]. Detailed description of the technique can be found in many published references.

Typical sheet metal forming simulation involves the following parts: punch, die, upper and lower binders, and the metal sheet. This, for example can be seemed in Figure 2, where a fender model is shown. Except the metal sheet, all other parts are assumed to be rigid, i.e., non-deformable. The simulation begins with closure of the binders, followed by the movement of punch pressing the sheet metal into the die cavity, as shown in Figure 3. Tens of thousands of small displacement increments were used in completing the entire punch stroke. As a result, the complete forming history can be simulated and most importantly, visualized.

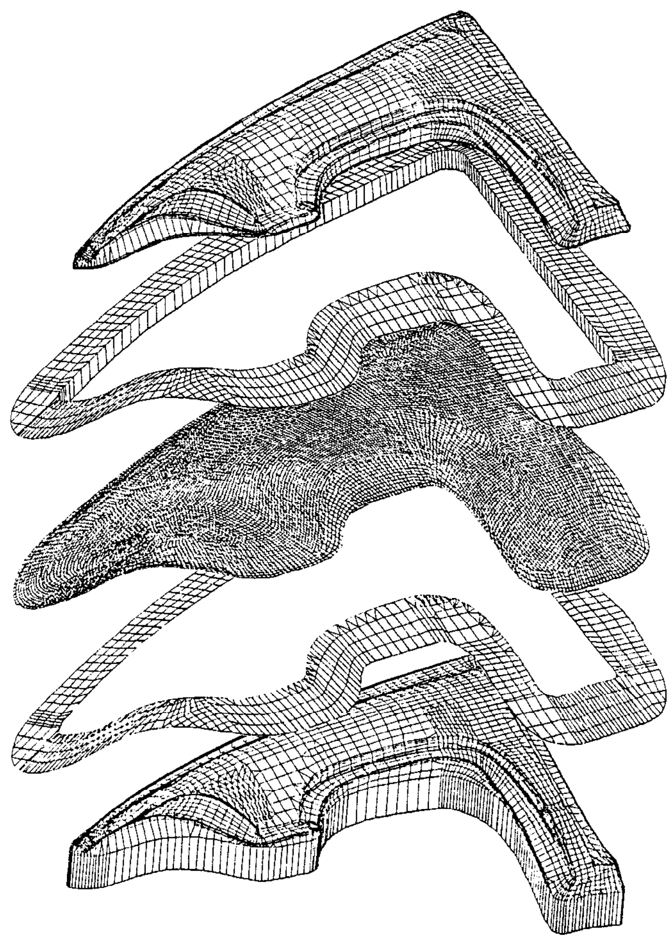


Figure 2. Configuration of a sheet metalforming simulation model.

The ability to see how wrinkles are formed, how sheet metal is stretched, and how fracture occured , allows tooling engineers to see what they were unable to see in the physical try-outs. For example, shown in Figure 4 is a thinning map of a fender panel. Because the insight gained from simulation, engineers can begin to modify the stamping designs on the computer and determine the best possible way to produce a good part.

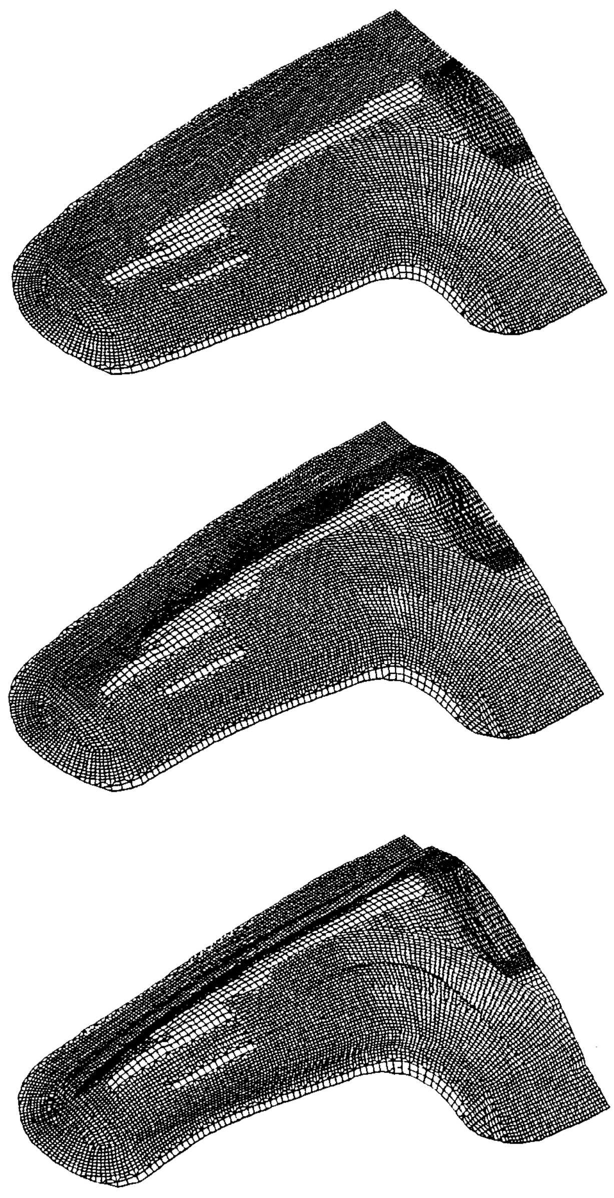


Figure 3. Simulation of sheet metal forming process.

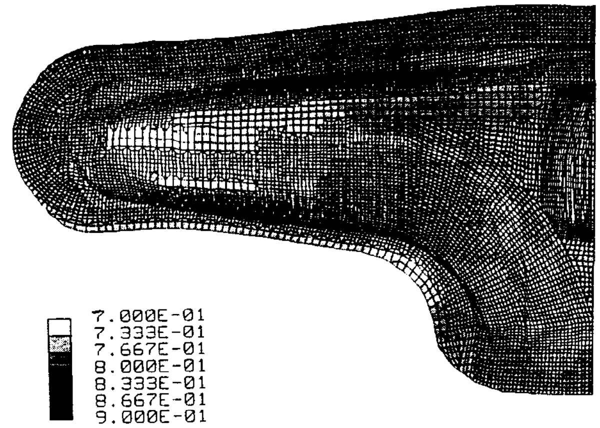


Figure 4. Contour plot of metal thinning

To correctly calculate the metal stretch and material flow, it requires accurate description of the tooling (punch, die and binders) geometry and realistic representation of:

- friction between the metal sheet and tooling,
- material nonlinear behavior,
- draw bead,
- finite element formulation, and
- contact algorithm.

To date, most of them can only be partially approximated by mathematical models. Thus, careful preparation of the data is essential to obtain reasonable results. So far, the engineering communities have accumulated sufficient knowledge of how to use the limited capabilities of these numerical techniques in solving forming problems. For instance, most of the outer and inner panels and frame body stamping parts can be reasonably simulated [11]. At the same time, scientific communities are also looking for better and more accurate models and methodologies to represent these complicated physical properties.

Current Issues

Though in recent years, computer simulations have been used to solve stamping problems, many issues, such as the ones discussed earlier, remain to be investigated to assure accurate and reliable results. There are other issues that are relate to a broader scope of the sheet metal stamping processes. They are discussed below.

- **Binder Wrap**

In many sheet metal stamping processes, the blank is clamed on the perimeter by upper and lower binders before the punch starts to move. The shape of the binders are designed to obtain the optimal tooling contact and material flow in the forming process. This is achieved by pre-deforming the blank into a wrapped shape, called binder wrap. Binder wrap can be simulated using commercially available software using explicit (LS-DYNA3D and PAM-STAMP) or implicit (MARC and ABAQUS) FE methods. However, either approach has inherent difficulties.

Use of the explicit FE routines usually requires increase of the punch and the binder speed by about an order, compared to physical process, to reduce the computer CPU consumption. As a result, the simulated forming process occurs in less than one-tenth of a second. The increase of binder speed introduces an impulse loading to the metal sheet, and causes 'ringing' of the metal sheet at unsupported area. In addition, they could also cause problems in the dynamic contact-impact calculations. Complicating the issues, it was also found that different binder closing speeds could produce different binder wrap shapes. Although artificial treatment of this problem by damping out the dynamics as implemented in many commercial codes, it also brought new controversy.

Implicit algorithms, based on more rigorous mechanics principles also have difficulties in binder wrap and forming simulation . Among many other problems, implicit algorithms have difficulty to converge because of the complicated contact conditions. Continued research in these areas are needed and should be the first priority in future manufacturing simulation.

- **Springback Simulation**

One of the most important issues of sheet stamping is springback. Springback is a phenomena that a formed part undergoes extra deformation after it is removed from the die. This extra deformation is caused by the release of the remaining stress at the end of the forming process. Stamped parts with significant springback deformation have difficulty to mate with other parts in the welding and/or vehicle assembly processes.

To simulate springback, it requires accurate stresses calculation at the end of the forming simulation. In addition, a reliable stress-strain/displacement algorithm is also needed for calculating the displacement field after unloading the stresses in springback [12].

Accurate forming stresses, however, are difficult to calculate. This, again, is due to the increase of punch velocity in the forming simulation. The additional momentum produces higher stresses. It is generally accepted that punch velocity should not be too high when stress distribution is of interest. So far, there is no general guideline on how high the punch velocity should be in the forming simulation.

- **Springback Correction**

With special care, success has been reported on springback prediction [12]. In collaboration with Chrysler, Cray Research, Inc., has developed a procedure to generate the complete die surfaces with springback compensation taken into consideration.

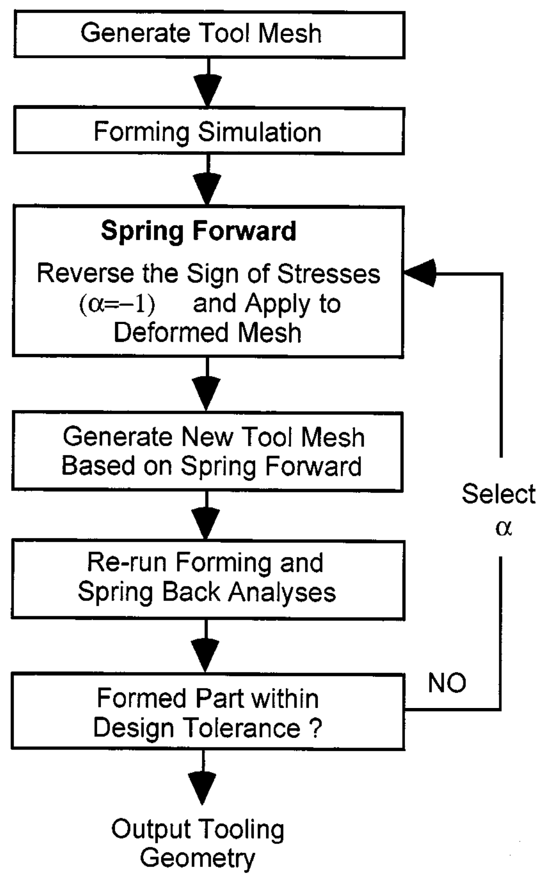


Figure 5. Springback correction procedure

As illustrated in Figure 5, the concept is to impose negative forming stresses to deform the formed part in a manner opposite to springback, referred to as spring-forward. The deformed part geometry from spring-forward calculation is used to generate a new die design. With springback compensated, the new die design shall be able to produce a better part, i.e., closer to the part design.

Iterations may be needed to control critical dimensions within tolerance. Results of using the procedure on an example part are shown in Figures 6 and 7. Figure 6 shows the formed part at the end of the forming simulation. Figure 7(a) shows the sprung-back part, a rail cap, after initial springback calculation. The z-direction displacement is as high as 9 mm. Figure 7(b) is the z-displacement after one sping forward iteration and it has been reduced to less than 3 mm. The z-displacement are further reduced to less than 1 mm after two iterations, as shown in Figure 7(c). The applicability of the algorithm to parts with subsequent trimming operation is to be further investigated.

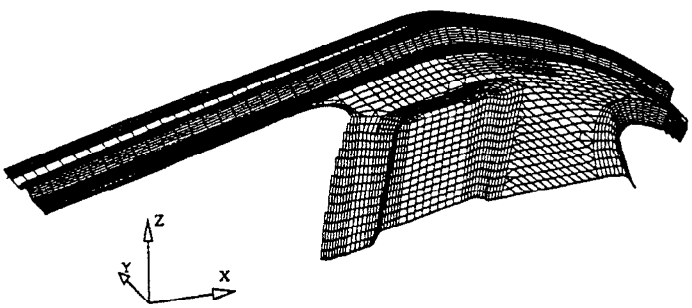
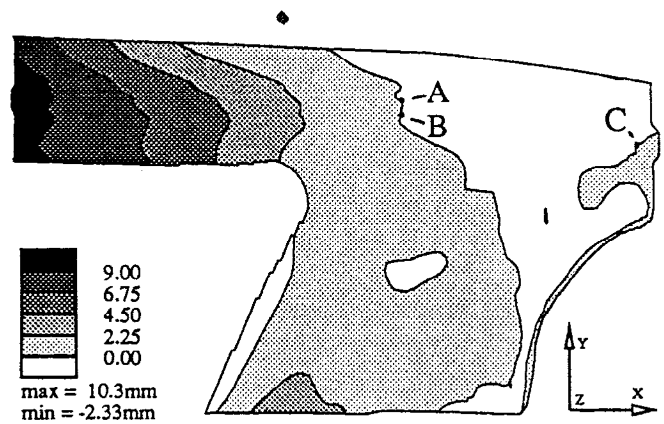


Figure 6. Simulated rail cap geometry after forming simulation

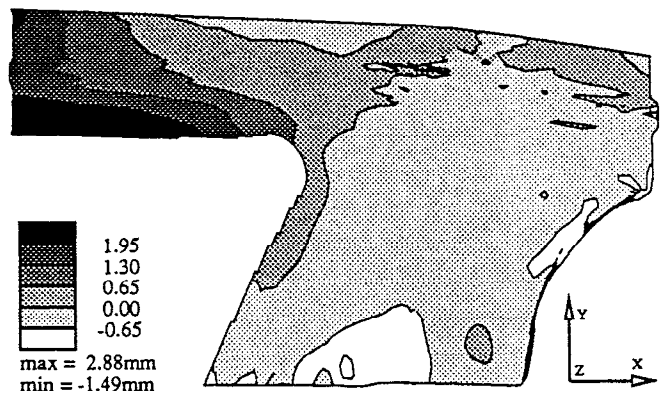
- **Draw Bead**

In order to control the material flow and to apply more stretch to the stamped part to reduce springback, draw beads are usually used on the binders. Several approaches have been employed to deal with draws beads when conducting sheet stamping simulations.

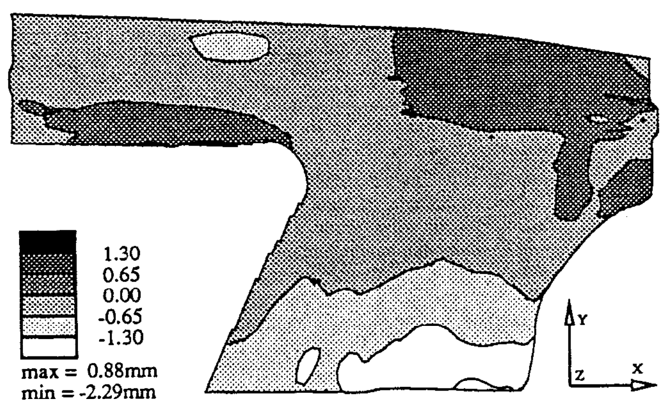
The first approach is to model the detailed draw bead geometry and simulate the material flow over the beads. This approach uses very detailed, and large models [13]. As a result, significant compute resources are needed. The second method is to replace draw beads by applying tangential forces on the edges of the blank. The magnitude of the forces is taken from the experimental data. This approach is relatively simple to implement. However, an experiential database of the draw bead forces is needed.



(a) Initial springback



(b) After 1 iteration of spring forward



(c) after 2 iterations of spring forward

Figure 7. Z-direction springback displacement.

A third approach, as appeared in LS-Dyna3d [14], uses contact segments under the pre-defined draw bead location to control the amount of tangential forces being generated by the draw beads. However, this feature is still under development. In summary, current approaches to model draw bead are at best approximations and requires further development.

- **Computing**

Simulation technologies today can carry out a forming analysis of a 40,000 element-model in less than 6 hours on a Cray C90 supercomputer and use about 150 Mbyte memory. Here, a punch speed of about an order faster than the physical process is prescribed in the model. The springback calculation of a similar size problem will consume 2-3 hours of CPU time and requires approximately 400 Mbyte memory if an implicit algorithm is used.

Parallel computing can reduce the elapsed computing time significantly. As shown in Table 1, a forming analysis used 10,227 Cray C-90 CPU seconds with 1 processor can finish the analysis in about 4,300 seconds using 4 processors simultaneously, which represents a speed up of 2.3. Better performance could be achieved with improved parallel algorithms. At present, parallel computing holds the promises to greatly shorten the turnaround time of an analysis.

Number of CPU	MFLOP	Elapse Time (sec)	Speed Up
1	201	10227	1.0
1	338	6107	1.67
4	482	4300	2.34

Table 1. An example sheet metal forming problem using multiple CPU on a Cray C90 supercomputer.

Future Trends

In the past few years, computer simulation of metal forming has gained a great deal of attention from researchers in both industry and academic communities. The technology has been gradually matured especially with forming simulation. The research and application interest will shift from simulations of larger and detailed models to more accurate results. To achieve this, more reliable and sophisticated material models, friction models, and contact algorithms are needed. To successfully implemented in a production environment, new algorithms need to be developed to better bridge the CAD and CAE processes. As a result of these, computer recourses also need to be adequately configurated to realize the potential of significant savings in time and cost.

Reference

1. Hung, E. Arnaudeau, F., Dubois, J., de Rouvray, A., and Chedmail, J.F., "Static and Dynamic Finite Element Analysis of Structural Crashworthiness in the Automotive and Aerospace Industries," Structural Crashworthiness, Ed. Jones and Wierzbicki, Butterworth & Co (Pub.), 1983.

2. Vander Lugt, D.A., Chen, R.J., and Deshpande, A.S., "Passenger Car Frontal Barrier Simulation Using Nonlinear Finite Element Methods" SAE paper 871958, 1987.

3. Liu, S.D., and Karima, M., "A One Step Finite Element Approach for Product Design of Sheet Metal Stamping," NUMIFORM'92, pp.497-502, 1992.

4. Chappuis, L.B., Tang, S.C., Chen, X.M., and Wu, J.H., "A Numerically Stable Computer Model for Sheet Metal Forming Analysis by 2D Membrane Theory," *Sheet Metal and Stamping Symposium*, SAE Publication SP-944, SAE paper No. 930518, pp155-164, 1993.

5. Balun, T., Tang, S.C., Chappuis, L.B., and Wu, J.H., "Application of Mechanics Methods to Evaluation of Forming and Process Design," *Sheet Metal and Stamping Symposium,* SAE Publication SP-944, SAE paper No. 930518, pp181-187, 1993.

6. Schweizerhof, K., Weimer, K., Hallquist, J.O., and Stillman, D.W., 'Improving Standard Shell Elements, Friction Models, and Contact Algorithm for the Efficient Solution of Sheet Metal Forming Problems with LS-Dyna3D," in VDI Berichte 894, FE-Simulation of 2D Sheet Metal Forming Process in Automotive Industry, 1991.

7. Aita, S., El Khaldi, F., Fontaine, L., Tamada, T., and Tamura, E., "Numerical Simulation of a Stretch Draw Autobody: Part I - Assessment of Simulation Methodology and Modeling of Stamping Components," *Autobody Stamping/Applications and Analysis*, SAE Publication No. SP-897, SAE paper No. 920639, pp.193-205, 1992.

8. Saha, N.K., Mahadevan, S.K., Midoun, D.E., and Yang, J.S., "Finite Element Structure-Dummy System Model for Side Impact Simulation", *Symposium on Crashworthiness and Occupant Protection in Transportation Systems*, ASME Winter Annual Meeting, pp 169-180, 1991.

9. Sheh, M.Y., Reid, J.D., Lesh, S.M., and Cheva, W., "Vehicle Crashworthiness Analysis Using Numerical Methods and Experiments," SAE P-258, *The 8th Vehicle Structural Mechanics and CAE Conference*, pp 119-128, 1992.

10. Calso, S.M., Saha, N.K., Faruque, M.O., and Balabhadra, K.S., "Simulation of Offset and In-line Car to Car Rear Impacts", *Symposium on Crashworthiness and Occupant Protection in Transportation Systems*, ASME Winter Annual Meeting, pp 149-160, 1993.

11. "First Impression", Cray Resaerch Inc. A video interview of Chrysler Advanced Stamping Technology Department, 1994.

12. Changqing Du, Zhang, Li, and Wang, Neng-Ming, "Springback Prediction in Sheet Forming Simulation", *Analysis of Autobody Stamping Technology,* SAE Publication No. SP-1021, SAE Paper No. 940937, pp. 113-124, 1994.

13. Galbraith, C., and Finn, M., "Supercomputer Simulation of Automotive Sheet Metal Forming," in Cray Channels, Vol 16, No. 2, pp. 16-21. 1994.

14. LS-Dyna3d User Manual (1993 Version), Livermore Software Technology Corp., 1993.