Advance in Sheet Metal Forming
- One-step Solution, Multi-Beads, Gravity Prebending, Auto Nets, and Local Compensation

Xinhai Zhu & Li Zhang
Livermore Software Technology Corporation

Abstract

Some of the new features developed since the last conference will be discussed.

1) *A fast one-step sheet metal forming program*
    Developed mostly for blank size estimation and forming effect initialization in crash simulation, this powerful feature allows hundreds of parts to be simulated in a very short time.
2) *An improvement to the contact draw bead*
    Developed together with Ford Motor Research and Advanced Engineering Laboratory, multiple beads can now be generated automatically based on the base draw bead definition, to represent the actual bead width and forces. This more realistic approach allows for a better blank size prediction and utilization.
3) *Pre-bending for gravity simulation*
    Developed together with Ford Motor Company, blanks can now be pre-bent to simulate the various scenarios of stamping press destacker loading of the sheet blank to the draw die. This new approach produces a more realistic gravity-loaded blank shape, resulting in shorter die travel and better stamping quality.
4) *Automatic fixture nets for free-standing springback simulation*
    This feature allows for automatic generation of fixture nets of user-specified sizes, in user specified locations, with contacts automatically established, for a free-standing springback simulation.
5) *Springback compensation for localized multi-regions*
    Developed together with Chrysler LLC, local compensation of stamping dies can be extended to multiple regions.

A Fast One-Step Sheet Metal Forming Program

A series of keywords, *CONTROL_FORMIING_ONESTEP_*OPTION* [1] were developed to address some requirements of the forming and crash application:
1) Blank size estimate;
2) 'Quick and rough' formability assessment;
3) Forming effect initialization for crash/occupant safety application.

The one-step solver utilizes the existing implicit solver. Below are some highlights of the one-step program:
1) **INPUT** – In addition to the usual input for part, material and physical properties, final product mesh in keyword format is needed. Any trimmed-out holes need to be filled. The filling can be done in LS-PrePost v3.2, by selecting *Mesh/EleGen/Shell/Shell by Fill_Holes/Auto Fill*. It is noted that the usual mesh used for the tooling surface representation is not suitable for implicit calculation. A regular, size-controlled mesh, generated using *MESH/AUTOMESH/SIZE* is recommended. Furthermore, although the results are better with die face included, the input model does not need it. Additionally, tipping to proper draw axis is not required.

2) **OUTPUT** - Results are stored in an ASCII file named '*onestepresult*'(dynain). The final estimated blank size (in its initial, flat shape) is output through d3plot files;

3) **Draw beads**
   - 'Auto Beads' applies a user supplied lock percentage (30% default) for a variable *AUTOLOCK* (Card1, 3rd field in *CONTROL_FORMING_ONESTEP) to all nodes along the part boundary automatically. This option is the easiest and sufficiently good;
   - Specify a node set along the part boundary and a lock percentage using the _DRAWBEAD option;

4) **Friction** can be accounted for, by using a new option _FRICTION. The frictional force is based on a "binder pressure", and is a percentage of the input pressure;

5) **Boundary conditions** are automatically applied using a new option _AUTO_CONSTRAINT to prevent any rigid body motion;

The Taurus fender outer model from NCAC of George Washington University was used to illustrate the typical simulation results. A total of 3500 fully integrated shells were used, and it took 15 seconds on a Linux, with 8 Intel Xeon E5520 CPUs. In Figure 1, trimmed-out holes are automatically filled by LS-PrePost (left), with mesh generated by AutoMesh via 'Size' controlled method (right). In Figures 2 and 3, thickness and plastic strain contours are displayed, which seem to capture the essence of the formability. Finally, the predicted, unfolded blank size is shown in Figure 4.

It is noted that although 'one-step' simulation provides a quick formability assessment, it is not recommended to be used for making product change decisions. Simulation with incremental solution is more suitable for that purpose.
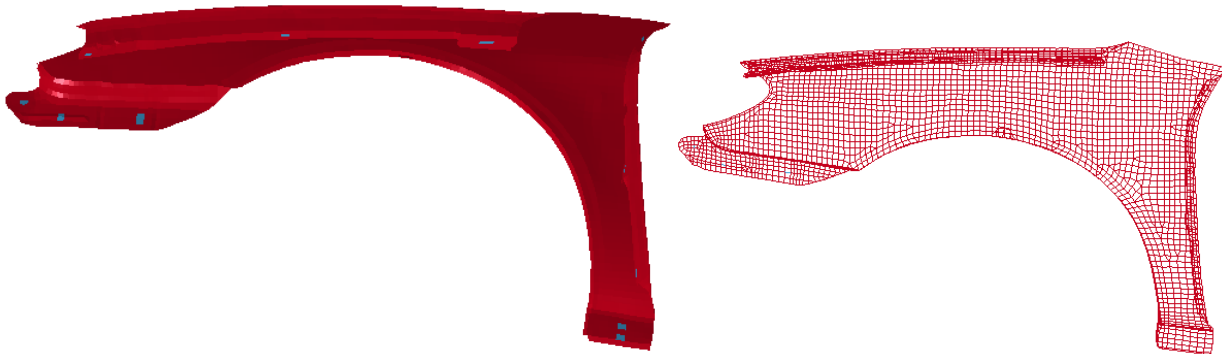
Figure 1. Holes are auto-filled (left); Mesh by 'Size' controlled AutoMesh (right).
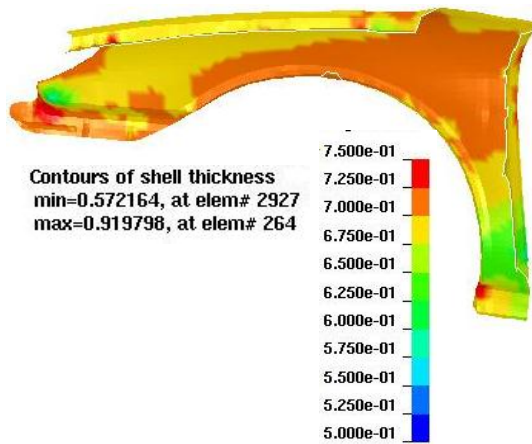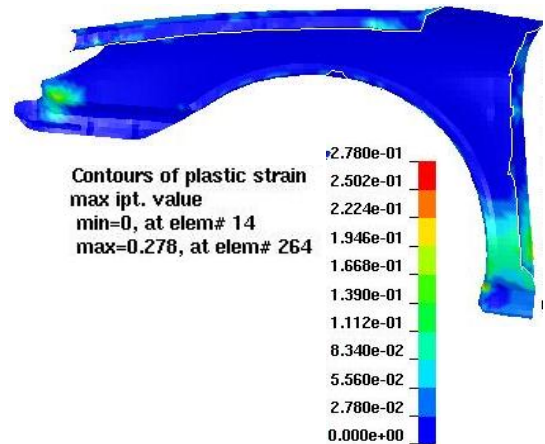*(model courtesy of NCAC)*

Figure 2. Predicted Thickness   *Figure 3. Predicted Plastic Strains*
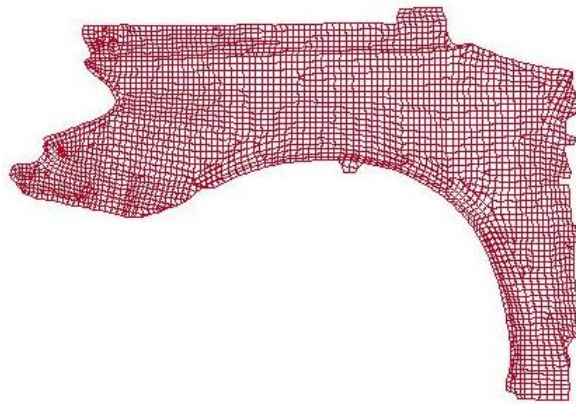

Figure 4. Predicted final unfolded blank size

An improvement to the contact draw bead

The improvement is directed at distributing the strain/stress more evenly and more realistically in the line bead modeling. This is important in cases where the final blank edge is ended in and near the draw bead region. New variables in *CONTACT_DRAWBEAD [1] automatically generate a number of multiple, parallel beads, and redistribute the intended bead forces evenly among the multiple beads. As shown in Figure 5, the total bead width, defined by the variable WIDTH, is the distance between the inner and outer most beads. The number of beads is specified by the user, and always in odd integer. An effective bead height, represented by a variable *EFFHGHT* defined by the user, controls the timing of bead force. Currently, two nodes on each of the binders need to be specified to track this height. Restraining forces starts to take effect only when binder gap is less than *EFFHGHT*.

As shown in Figure 6, from RCFORC plot file, bead forces are distributed evenly among 3 multiple beads, the total of which is equivalent to the force generated by 1 regular bead. In Figure 7, a strip of blank, clamped by the upper and lower binders, is being pulled through a 3 beads configuration, with an initial binder gap of 12mm. Clearly, from the pulling force vs.

binder closure distance plot, bead forces start taking effect gradually at 8mm binder gap, specified by the *EFFHGHT*.

As shown in Figure 7, while the mean stresses are more evenly distributed for the multiple beads in the beads area, no significant differences are found inboard of the bead area between the multiple beads and regular bead. In the bead area, stress level for multiple beads is about 1/3 of the regular bead.
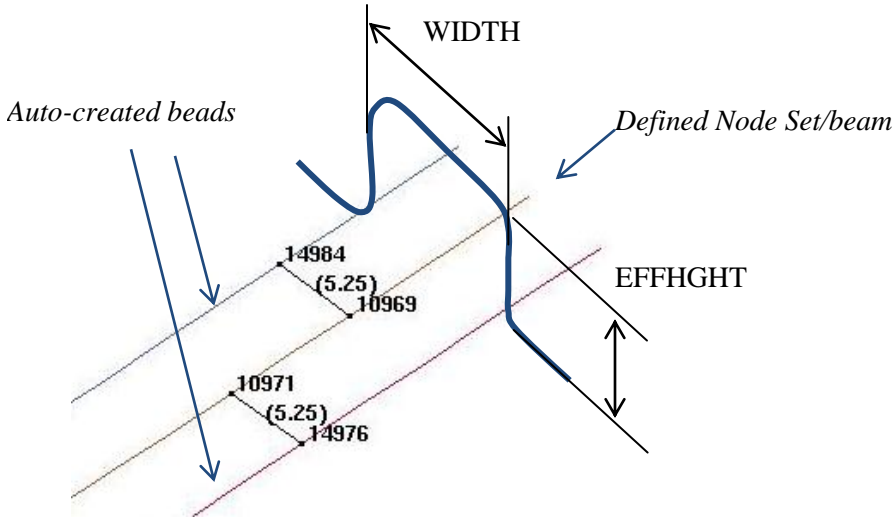
This feature was developed in conjunction with Ford Motor Company,

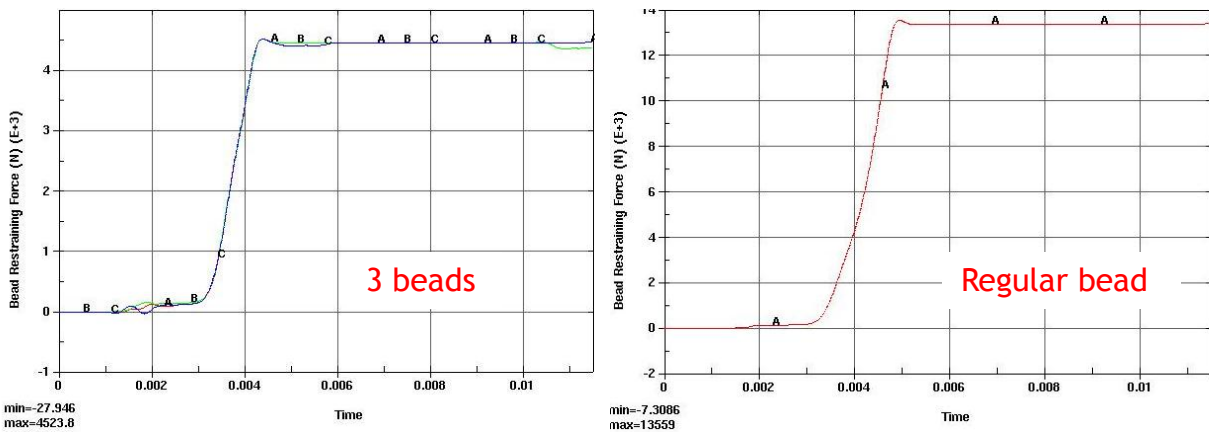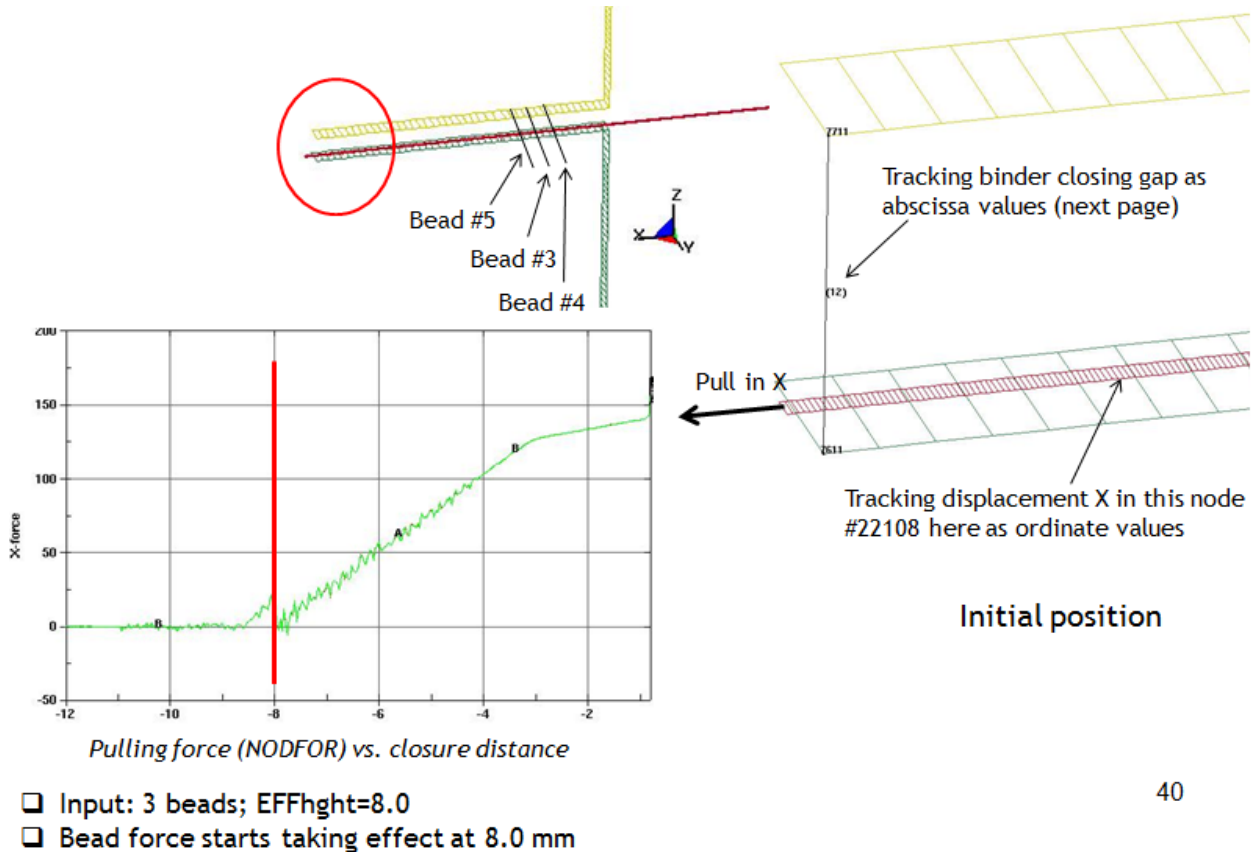*Figure 5. Illustration of the new contact beads.*

Figure 6. Bead force verification.

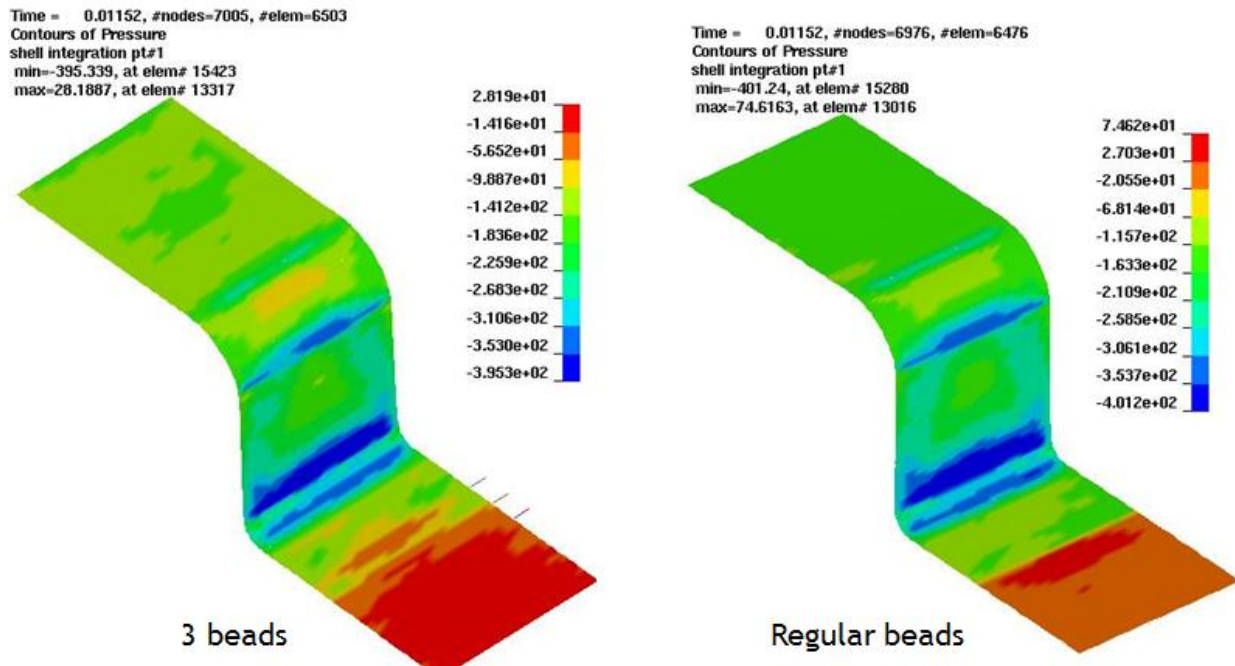*Figure 7. Verification of the variable EFFHGHT.*

Figure 8. Mean stress comparison.

Pre-bending for gravity simulation

From a flat shape, all blanks have two bending modes: either in concave or convex shape. Once the blank takes one shape, it is impossible for it to go into the other shape without external perturbation. Most blanks sent to a draw die in air are made sure it is in convex shape, by manually pre-bending, suction cups, or mechanical arms, etc. The convex shape is further reinforced with some sort of lower support. Without the pre-bending, in some cases even with the support, blanks do not take the convex shape as desired. Blanks in toggle draw tend to have a naturally concave shape, conforming and favorable to the cavity-shape lower die. This new feature allows for pre-bending of blank in either convex or concave shape prior to gravity loading onto the tools, just as it would in real stamping press going from destacker to the draw die with the boom or suction cup. The keyword responsible for this simulation is *CONTROL_FORMING_AUTO_NET [1].

As shown in Figures 9, 10 and 11, a vector can be used to define a bending axis, its location relative to the sides of the blank is determined by the signs of the radius R and blank normal direction. A negative R puts the bending axis on the opposite side of the blank normal, while a positive R assigns the axis on the same side of the blank normal. An optional set of coordinates defines the center of the most-bent location, which is the center of gravity if left undefined. The pre-bent blank, prior to gravity loading, is observed in the 1ˢᵗ state of the d3plot files, in Figure 11. The last d3plot state shows the final result, in Figure 12. Compared with no pre-bent case, Figure 13, which has a sagging shape in the middle of die cavity despite the maximum allowable punch support, the pre-bent case produces a blank much closer to the lower binder in one corner by some 35mm. Furthermore, sagging in the middle of a class-A panel during gravity loading may result in potential surface lows during forming.

The pre-bending feature is now available in eZ Setup under Metal Forming Application (under *BLANK* tab) in LS-PrePost v3.2.
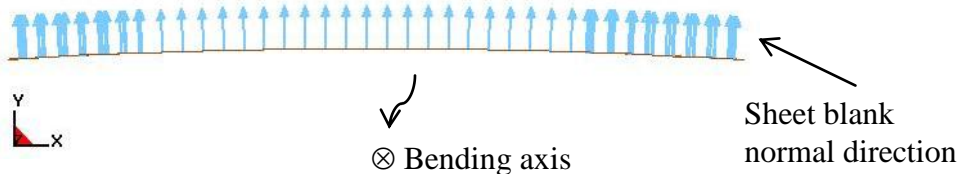

⊗ Bending axis

Sheet blank
normal direction

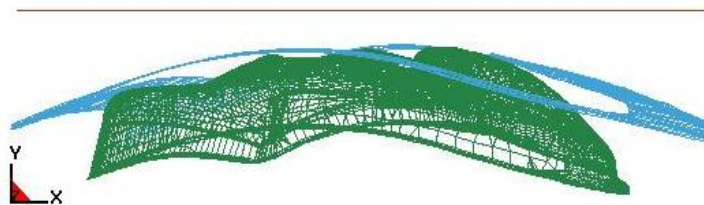*Figure 9. Defining a bending axis and normal direction*

Figure 10. Initial model setup

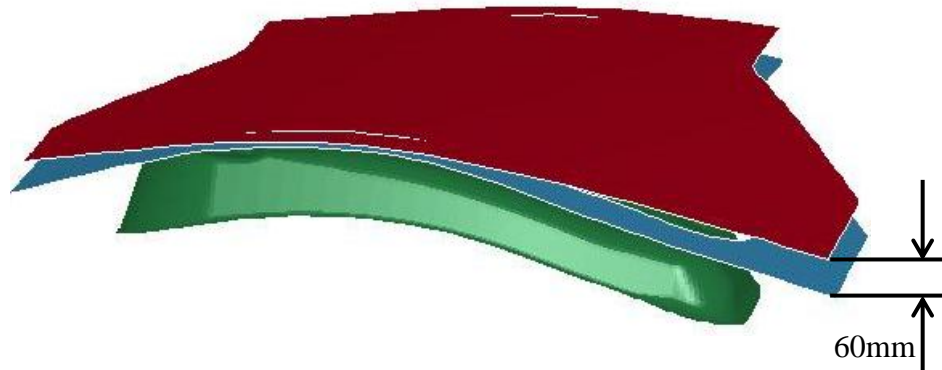*Figure 11. Pre-bending result with R=-10000.0mm (1st state of D3plots)*

60mm

Figure 12. Gravity loaded shape with pre-bending (last state of D3plots)

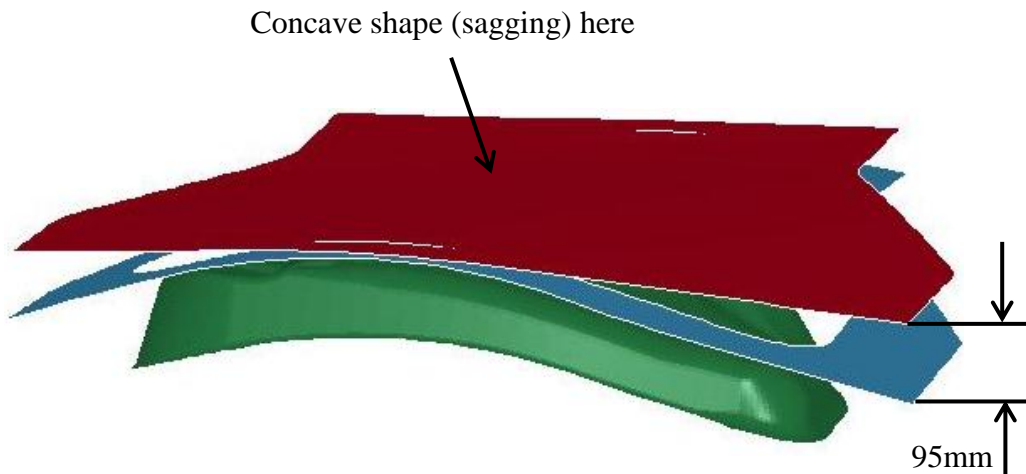Concave shape (sagging) here

95mm

Figure 13. Gravity loaded shape with pre-bending (last state of D3plots)

Automatic fixture nets for a free-standing springback simulation

Frequently, stamped panels are in need of springback prediction on resting nets of a checking fixture, without any constraints. In addition to model each individual nets, which are used for contact with a to-be predicted panel, new part ID, section/material ID, and contact pairs need to be defined between the nets and panels. With the keyword *CONTROL_FORMING_AUTO_NET [1], the process is made much easier.

This keyword allows users to define the center location of each net position, the size of the net, and optionally, the direction to which the net surface normal is aligned to. The nets so defined are created in the code and contact definitions are automatically established. Each net defined will have its own individual part ID and contact ID so contact condition can be analyzed. Users also have the option of defining an initial gap between the nets and the panel. With gravity applied, the panel will undergo springback calculation, taking into account the contact conditions imposed by the nets. The result is a constraint-free springback panel on the checking fixture nets. In Figure 14, four locations along the flanges of the NUMISHEET2005 Cross Member were chosen. A single shell element (10x10mm) is generated centered on each net location coordinates specified, Figure 15. The created nets are about 7mm below the panel initially, Figure 16. Springback results are shown in Figure 17. In this 5-step implicit static springback simulation, contact condition can be determined using the RCFORC file. As plotted in Figure 18, two upper nets and one left net are in contact with the panel after springback, as contact forces exist at these net locations.

Figure 14. Net locations –

NUMISHEET '05 Cross Member

Figure 15. Four elements are
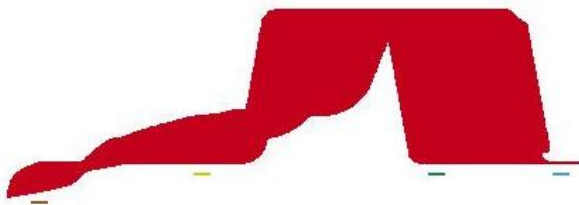*created in the locations specified*

Figure 16. Initial position of the
nets about 7 mm below the panel
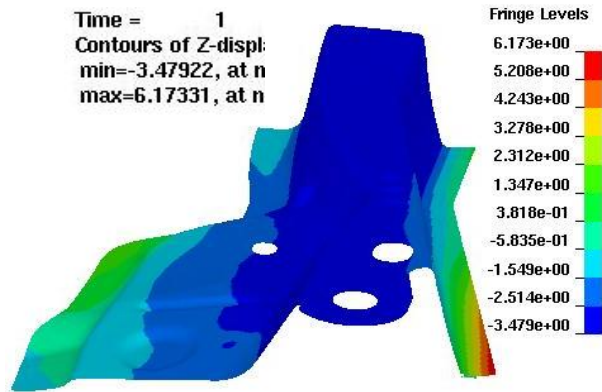
Figure 17. Panel springback
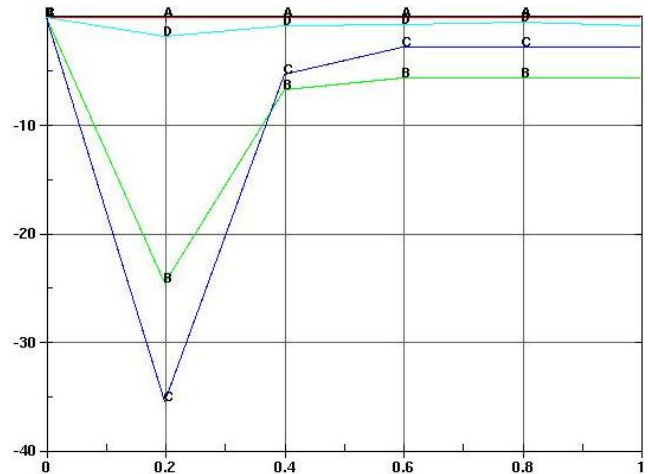
Figure 18. Panel springback

Figure 19. Contact condition determination - Force(N) vs. Time (steps)

Springback compensation for localized multi-regions

Developed together with Chrysler LLC, this feature extends the previous localized tooling compensation capability within LS-DYNA. Localized regions are defined by two self-enclosed curves – a 'begin' curve and a 'end' curve, which form a transition region. These curves are defined in X,Y,Z format (LS-PrePost3.2) using keywords *DEFINE_CURVE_COMPENSATION_CONSTRAINT_BEGIN [1] and *DEFINE_CURVE_COMPENSATION_CONSTRAINT_END [1], and are referenced by keyword *INCLUDE_COMPENSATION_CURVE [1] in the regular springback compensation input file.

In Figure 20, two concentrated loads are applied to a deformable surface of a partial sphere, causing localized deformation, as shown in Figure 21. This deformed sphere surface is used as a surface after springback. The 'begin' curve is drawn based on the deformation contour around the edge of the deformed zone in each region. The 'end' curve is drawn about 15mm outside of the 'begin' curve. A transition region is thus formed by the curves, Figure 22. The compensated tooling geometry is superimposed with target geometry (a perfect sphere surface), Figure 23. In Figure 24, a section view illustrates the result of one compensation with a factor of 0.7. This new compensated tool surface can be used to continue the springback compensation iterations.
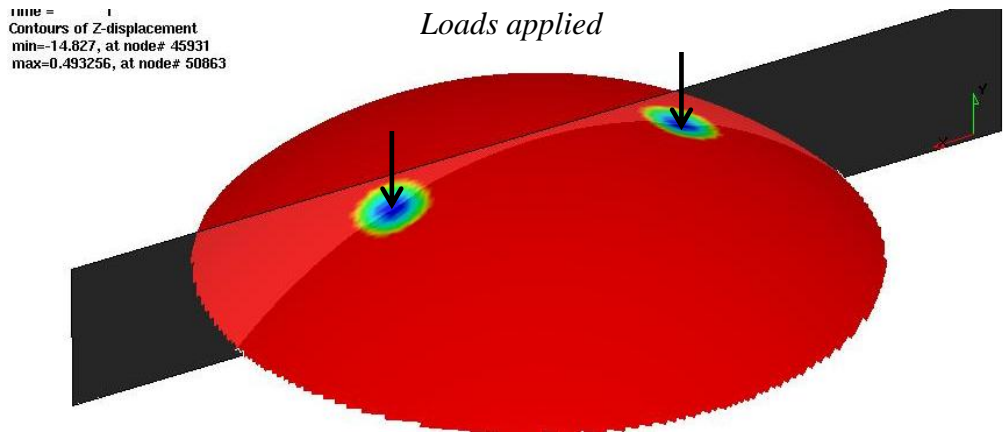
Time =   1
Contours of Z-displacement
min=-14.827, at node# 45931
max=0.493256, at node# 50863

Loads applied

Figure 20. Concentrated loads applied on an sphere surface

Contours of Z-displacement
min=-14.827, at node# 45931
max=0.493256, at node# 50863

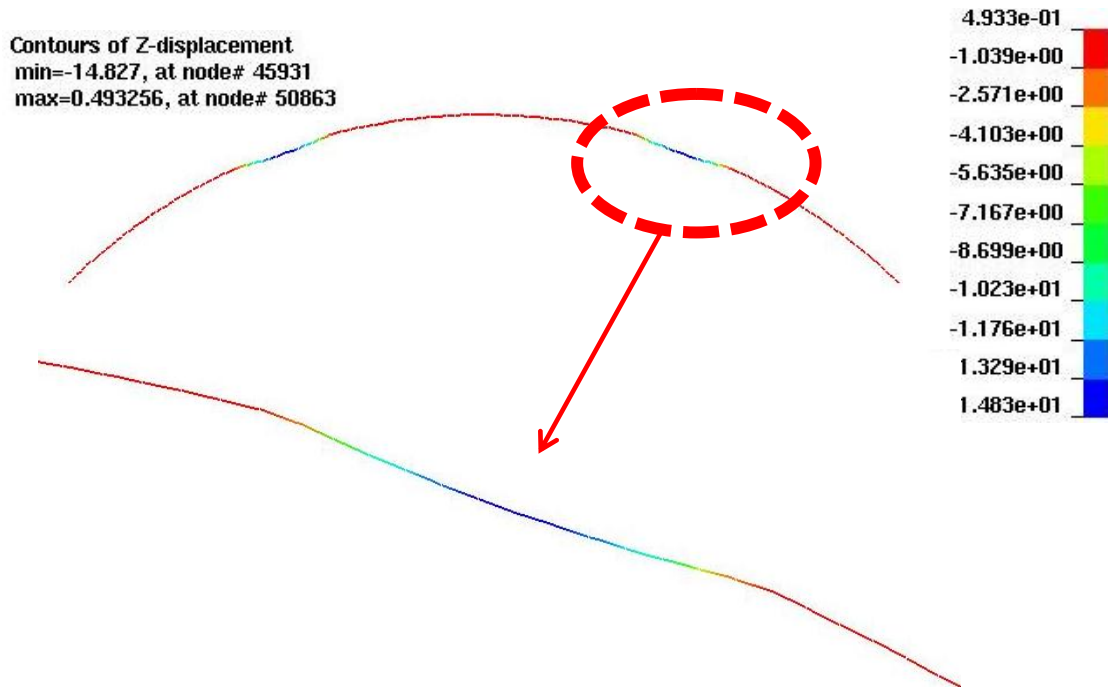| |
| --- |
| 4.933e-01 |
| -1.039e+00 |
| -2.571e+00 |
| -4.103e+00 |
| -5.635e+00 |
| -7.167e+00 |
| -8.699e+00 |
| -1.023e+01 |
| -1.176e+01 |
| 1.329e+01 |
| 1.483e+01 |

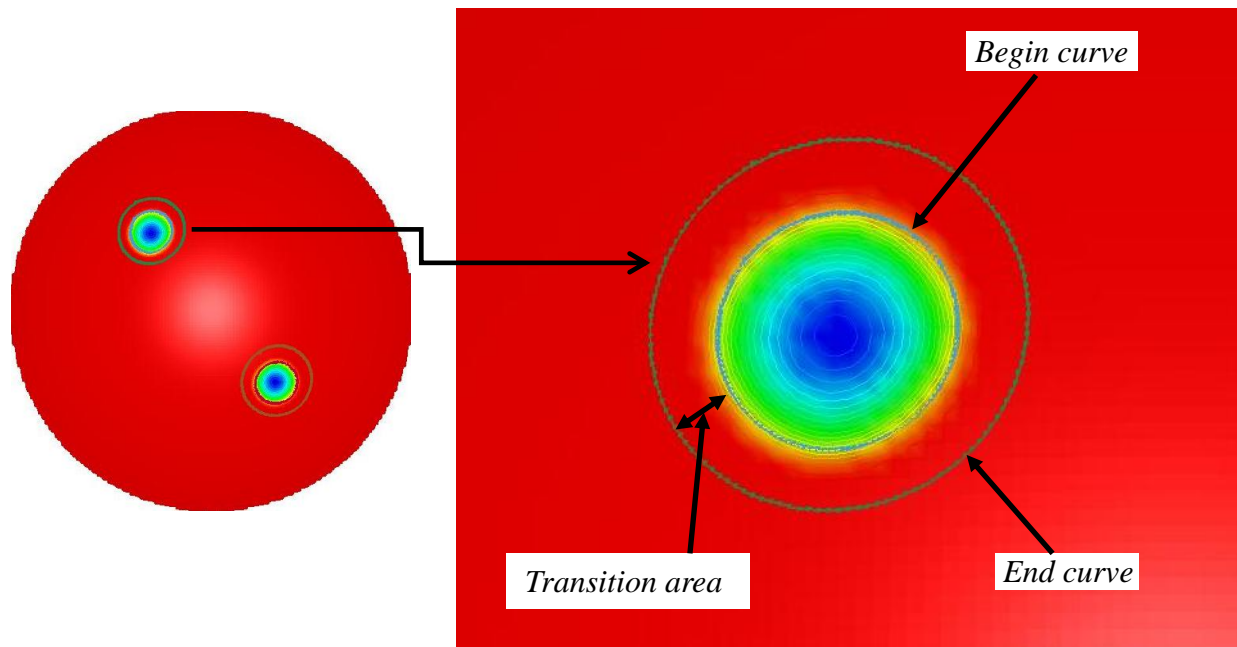*Figure 21. Deformed sphere surface caused by the loading – section view*

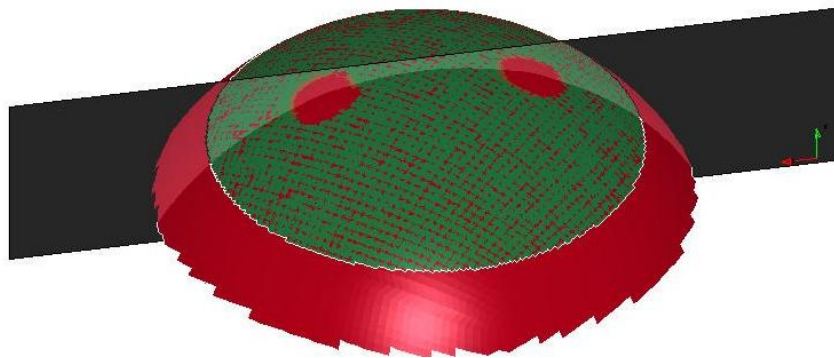*Figure 22. Deformed sphere surface caused by the loading – section view*

Figure 23. Compensated sphere surface (red) superimposed with target surface(green)
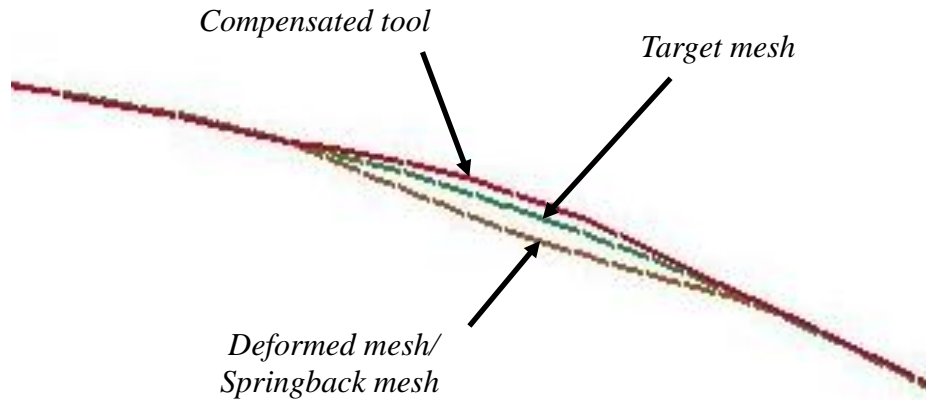
Compensated tool

Target mesh

Deformed mesh/ Springback mesh

Figure 24. Compensated tooling surface (red) with compensation factor of 0.7

Conclusion

A fast one-step sheet metal forming program was developed to address needs in blank unfolding and in crash/safety stress and strain initialization. A new feature in draw beads allows users to model bead forces in a distributed fashion, throughout the width of a bead, allowing partial bead forces when the blank edge has moved inside the bead width. Blank pre-bending prior to gravity loading leads to a more realistic gravity loaded blank shape, potentially improve class-A surface quality. Automatic fixture nets enable users to conduct a constraint-free springback on a checking fixture. New feature in die surface compensation provides multiple regions tooling surface compensation at the same time. At LSTC, we are committed to advancing sheet metal forming simulation technology, benefiting our current and future users.

References

1) LS-DYNA User's Manual R6.0.0 (Volume I), ftp://user:computer@ftp.lstc.com/manuals.