

Developments in LS-DYNA® for Metal Forming Simulations

X. Zhu, L. Zhang

Livermore Software Technology Corporation

1 Introduction

Selected major developments for stamping simulation in LS-DYNA are discussed. These developments are:

- Positioning of unfolded blank in one-step simulation (*CONTROL_FORMING_ONESTEP)
- Tooling mesh automatic fix and physical offset (*CONTROL_FORMING_AUTOCHECK)
- Major improvements in scrap trimming (*CONTROL_FORMING_SCRAP_FALL)
- Best fitting of two meshes (scanned STL vs. springback mesh)
- 2D, 3D trimming of solids and laminates
- Formability Index (F.I.) extended for *MAT_036, *MAT_125, *MAT_226

2 Position of unfolded blank in one-step simulation

One-step simulation with keyword *CONTROL_FORMING_ONESTEP has been widely in use in crash and safety for forming stress and strain initialization. It has also being used for initial estimating of blank size in stamping application. One problem stamping users face is the position of the blank after unfolding can be undesirable. Also, the shifted unfolded blank is not easy to align relative to the tooling position. To address this issue, additional entries are now available in keyword *CONTROL_FORMING_ONESTEP_AUTO_CONSTRAINT to specify three node IDs (NODE1, NODE2 and NODE3). The unfolded part will be repositioned and re-aligned according to the nodes specified back to the same nodes in the original part. The transformed blank is written in a keyword file "repositioned.k". As shown in Figure 1, an example of using the NODE1, NODE2 and NODE3 to transform the unfolded blank back to the original part so the same three nodes defined will be coincident. The three nodes are defined around the edge of two holes, as Nodes 197, 210 and 171.

3 Tooling mesh automatic fix and physical offset

Tooling mesh from imperfect CAD surfaces can be time consuming to fix. Some of the problems are so minute that can be easily missed. These bad meshes create problems during forming simulation runs, costing valuable engineering time that can be spent more productively somewhere else. In addition, if one side of tool mesh needs to be physically offset to create another side of tool mesh, extra manual checks and fixes are needed to ensure that offset tool's meshes are problems-free. Now a new feature in LS-DYNA, with keyword *CONTROL_FORMING_AUTOCHECK, allows users to check, as shown in Figure 2, for duplicated elements, overlapping elements, skinny/long elements, degenerated elements, disconnected elements, and inconsistent element normal vectors. It will also make each tool's element normal vectors automatically facing the blank. In addition, offset can be applied to the fixed tool to create another tool.

To check and fix the tool mesh, set the variable ICHECK to "1". The variable of IOFFSET can be use to offset the fixed tool. Meshes will be reoriented correctly towards the blank, and the tool offset is determined by an amount of $0.5 \cdot \text{abs}(\text{MST})$ either on the same or opposite side of the blank, depending on the signs of the MST (Figure 3), where the "MST" is the variable in *CONTACT_FORMING.... A new keyword file, "rigid_offset.inc" file, will be output as fixed, reoriented and offset tooling meshes.

4 Major improvements in scrap trimming

The original *constraint release method* of scrap trimming using the keyword *CONTROL_FORMING_SCRAP_FALL has been in use for quite some times now. This simplified method has the following drawbacks:

1. No scrap trimming – the scrap piece cannot be trimmed directly from a parent piece; an exact scrap piece after trimming must be modeled.

2. Poorly (or coarsely) modeled draw beads in the scrap piece do not fit properly in badly modeled draw beads on the tooling, resulting in initial interferences between the two and therefore affecting the simulation results.
3. For poorly (or coarsely) modeled scrap edges and trim posts, users have to manually modify the scrap trim edges to clear the initial interference with the trim posts.
4. Users must clear all other initial interferences (e.g. between scrap and scrap cutter) manually.

Based on users' feedback, a new method "scrap trimming" (after Revision 91471) has been developed to address the above issues and to, furthermore, reduce the effort involved in preparing the model. The new method (Figure 4) involves trimming scrap from an initially large piece of sheet metal, leaving the parent piece as a fixed rigid body. The trim lines are obtained from the trim steel edge node set NDSET and the trim vector VECTID. The variable EFFSET offset the scrap edge away from the trim steel edge, towards the scrap seed node side, useful to remove initial interference between the trimmed scrap (because of poorly modeled trim steel) and coarsely modeled lower trim post. The variable EXTEND is an amount to extend a trim steel's edge based on the NDSET defined, so it can form a continuous trim line together with a neighboring trim steel, whose edge may also be extended, to trim out the scrap piece. In Figure 5, the variable NDBEAD can be defined to be excluded from initially imposed constraints after trimming. This node set typically consists of nodes in the scrap draw bead region where due to modeling problems the beads on the scrap initially interfere with the beads on the rigid tooling; it causes scrap to get stuck later in the simulation if left as is. In Figure 6, GAP and IPSET are used to remove initial interference between the scrap and other die components.

5 Best fitting of two meshes

In springback prediction and compensation process simulation, there is always a need to assess the accuracy of the springback prediction using physical white-light scanned parts. The keyword *CONTROL_FORMING_BESTFIT is developed for this purpose. Scanned parts are typically given in the STL format, which can be imported into LS-PrePost and write out as a keyword mesh file. The converted scanned keyword file can be used as FILENAME as a target mesh in an input. The predicted springback mesh (with *NODE, *ELEMENT_SHELL, *CONSTRAIN_ADAPTIVITY cards only) can be included in the input file using **Fehler! Verweisquelle konnte nicht gefunden werden..** The best-fit program uses an iterative, least-squares method to minimize the separation distances between the two parts, eventually transforms the springback mesh of the included file (predicted springback mesh) into the position of the target mesh (scan). The separation distances between the two parts are calculated after the best-fitting, and stored as thickness values in a file bestfit.out, which is essentially a dynain file. Color contours of the separation distances between the two parts can then be plotted via FCOMP→Thickness. Both positive and negative thickness are calculated and stored as the Thickness. Positive distance means the included file is above the target in a larger coordinates, and negative distance is below the target in a smaller coordinates. The fitting accuracy is within 0.1mm. The variable IFAST is a computing performance optimization flag and can be used to speed up the computing time. Table 1 lists detailed speed performance data on an industrial stamping part. Figure 7 shows the result of a best fit between a springback mesh and scan data of a decklid inner panel, which has a maximum deviation between 1.23 mm to -1.59mm.

6 2D, 3D trimming of solids and laminates

To trim drawn panels of 3-D solid elements, both 2-D (directional) and 3-D (normal to element face) trimming are now available in LS-DYNA. The usage is the same as one would trim those of shell elements. Three keywords need to be included:

```
*CONTROL_FORMING_TRIMMING, or, ELEMENT_TRIM,
*DEFINE_TRIM_CURVE_NEW, (or 3D), and
*DEFINE_TRIM_SEED_POINT_COORDINATES
```

An example of a 2-D trimming of solids is provided in Figure 8.

In addition, trimming of laminated materials (core of solid elements sandwiched by top and bottom layers of shells) are also available. A new variable of ITYP under *CONTROL_FORMING_TRIMMING needs to be activated to enable the feature:

```
* CONTROL_FORMING_TRIMMING, or, ELEMENT_TRIM,
$ PSID,,$ITYP
```

where, ITYP=1 triggers a laminated material trim. An example of a 3-D trimming is provided in Figure 9.

7 Formability Index (F.I.) extended for *MAT_036, *MAT_125, *MAT_226

Formability Index (F.I.) addresses sheet metal failure under nonlinear strain path. Failure for elements undergoing nonlinear strain paths are different from those with linear strain path, which are handled with the traditional Forming Limit Diagram (FLD). F.I. is based on *critical effective strain method* and is stored in history variable #1 during simulation. Failure occurs when it reaches the value of 1.0. For all these three materials listed above and for *MAT_037, history variables (both in contour and in XY format) available to plot include:

- History variable #1: formability index (F.I.)
- History variable #2: strain ration (minor strain/major strain)
- History variable #3: equivalent plastic strain under planner isotropic assumption

To plot these history variables, it is necessary to set NEIPS under *DATABASE_EXTENT_BINARY to "3". In Figure 10, tests are performed on a single shell element for *MAT_125 to compare the forming limits under three different pre-strain paths to those of the traditional FLC. In Figure 11, the three history variables are continuously plotted under the strain paths assumed. In Figure 12, F.I. contour plot is shown on a cross member after forming. Both strain paths and F.I. evolutions can be plotted for the elements shown close to the critical F.I. value.

8 Summary

Various features related to metal forming have been developed to meet the requirements of our stamping users. Selected developments are discussed. LSTC is committed to working with our stamping users to advance the mteal forming simulation technology, and will continue to improve to stay ahead.

9 Reference

- [1] LS-DYNA User's Manual (I and II).

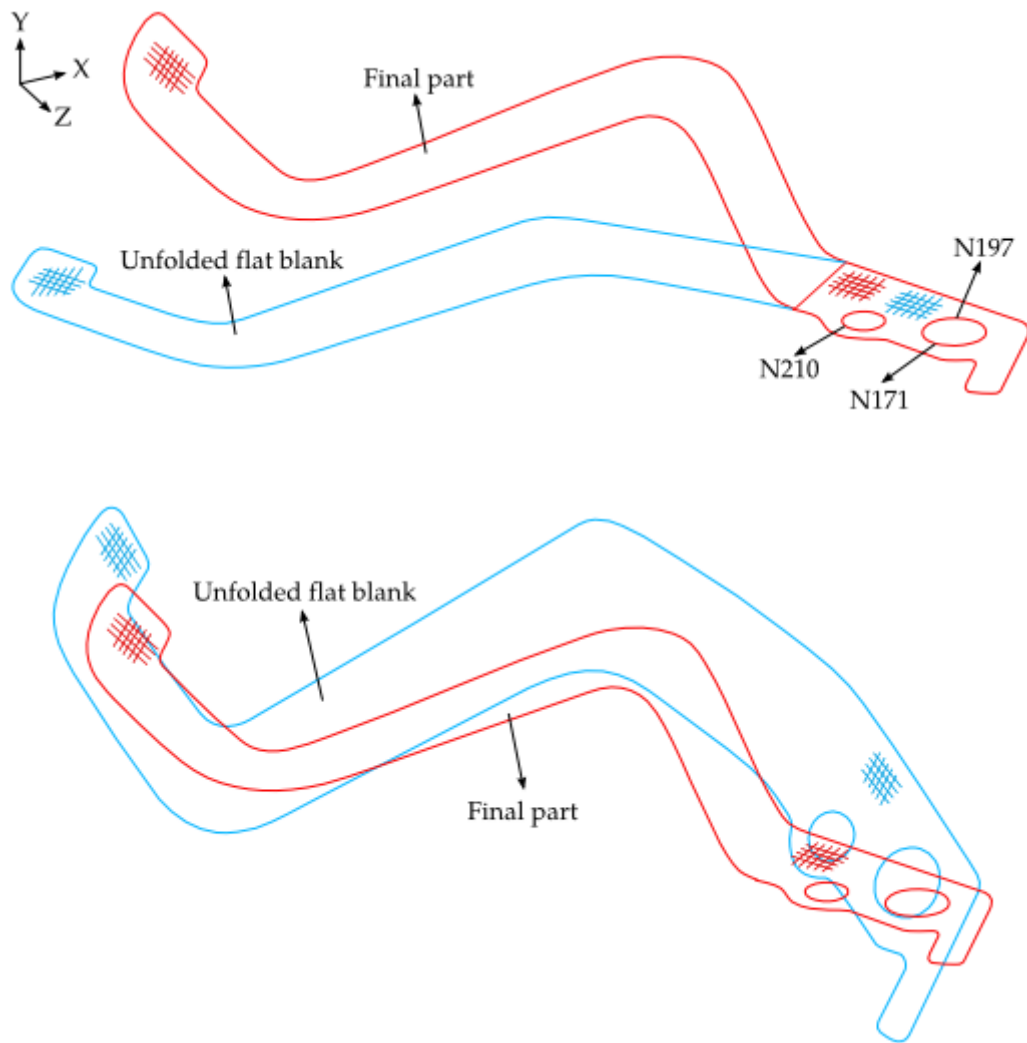


Figure 1 An example of the results when using the NODE1, NODE2, and NODE3 feature (top) and without using the feature (bottom). *Model courtesy of Kaizenet Technologies Pvt Ltd, India.*

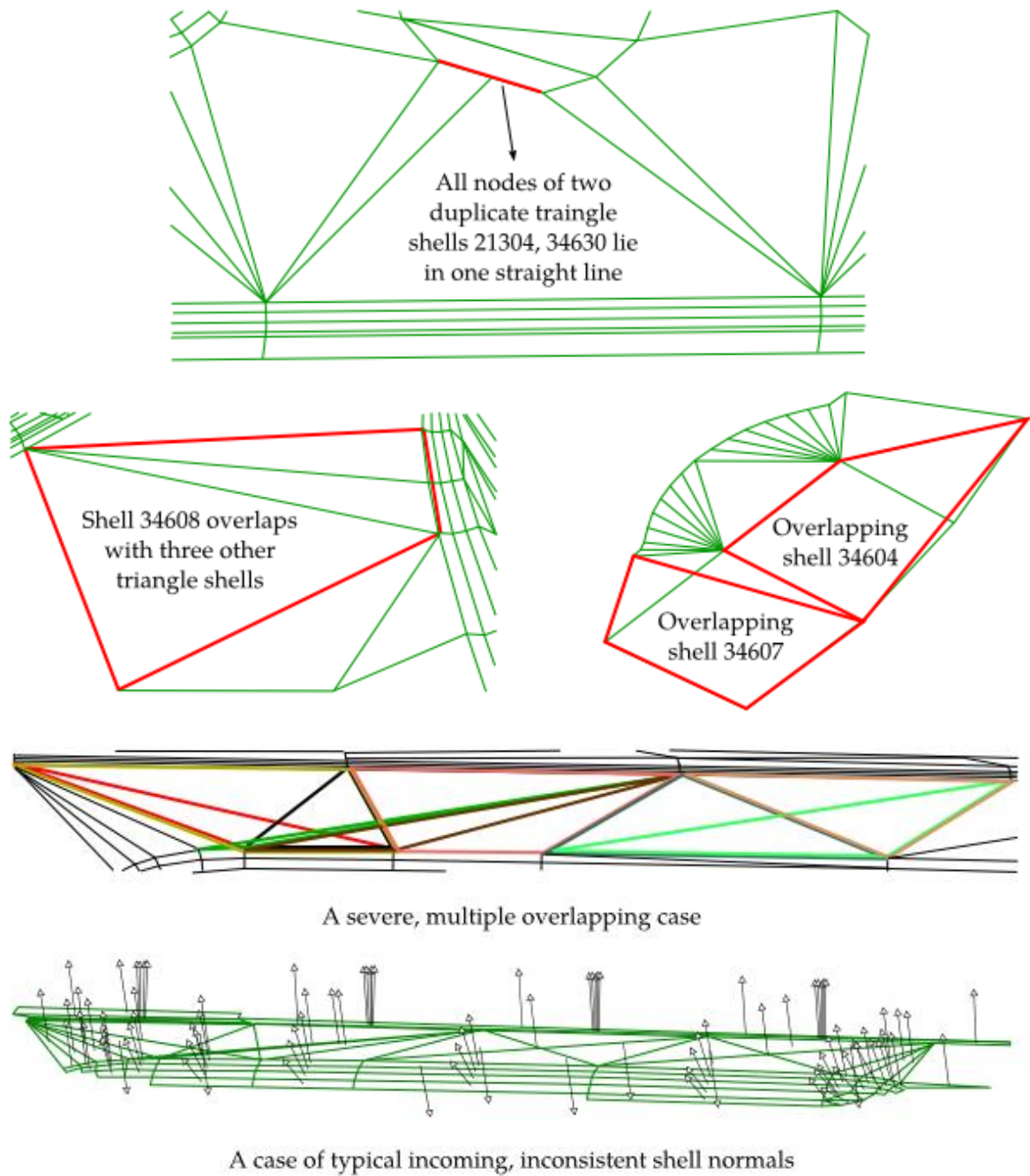


Figure 2 Some of the tool mesh problems fixed by *CONTROL_FORMING_AUTOCHECK.

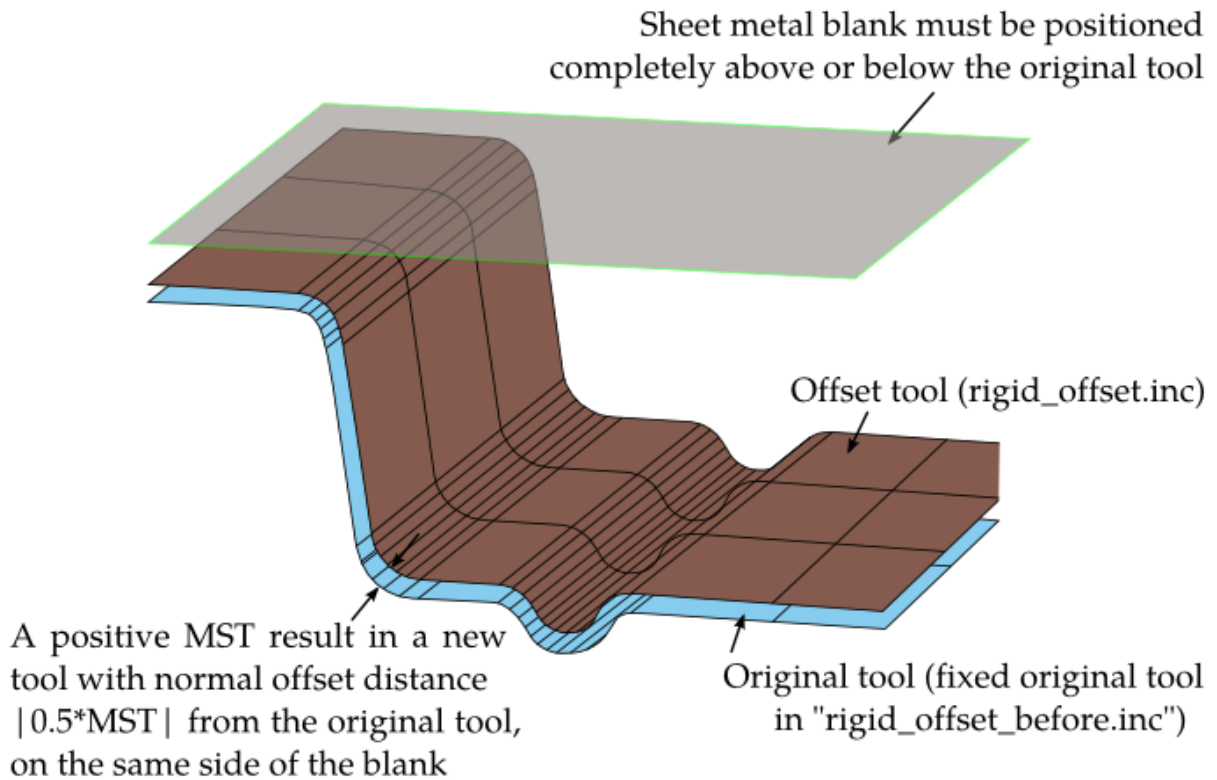
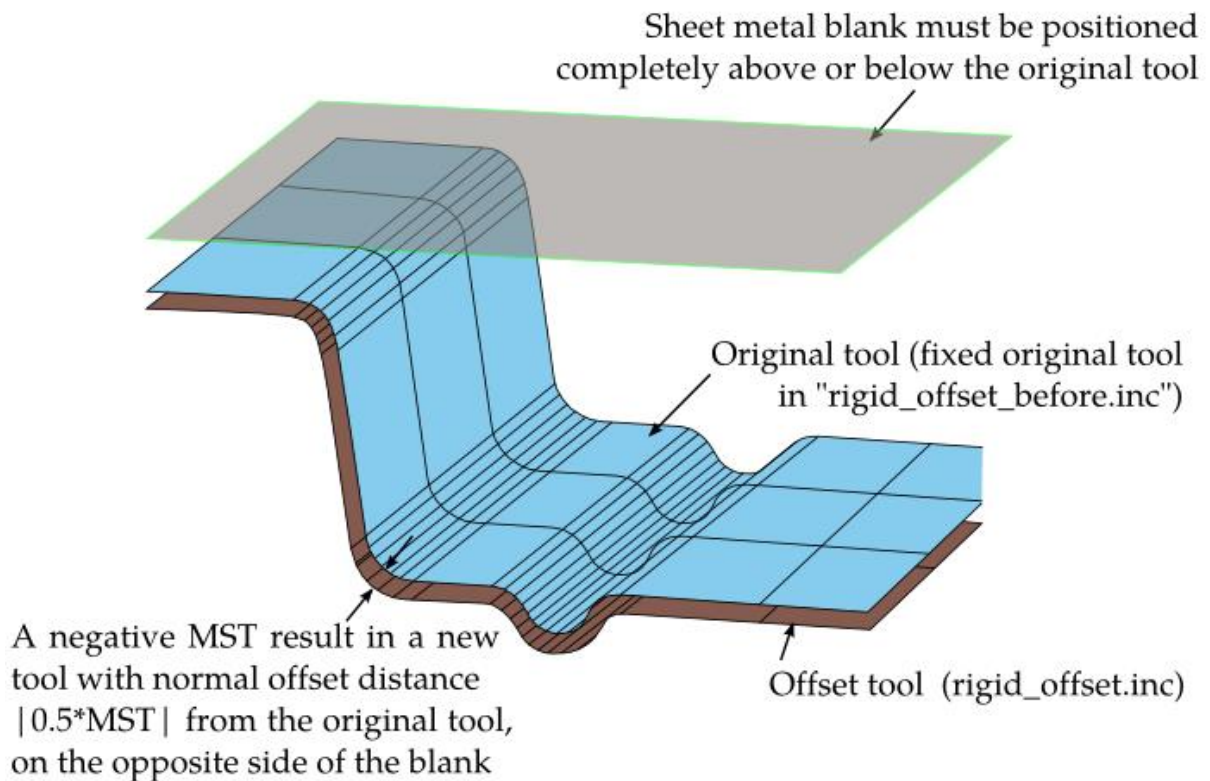


Figure 3 Offset a tool to create another tool with *CONTROL_FORMING_AUTOCHECK.

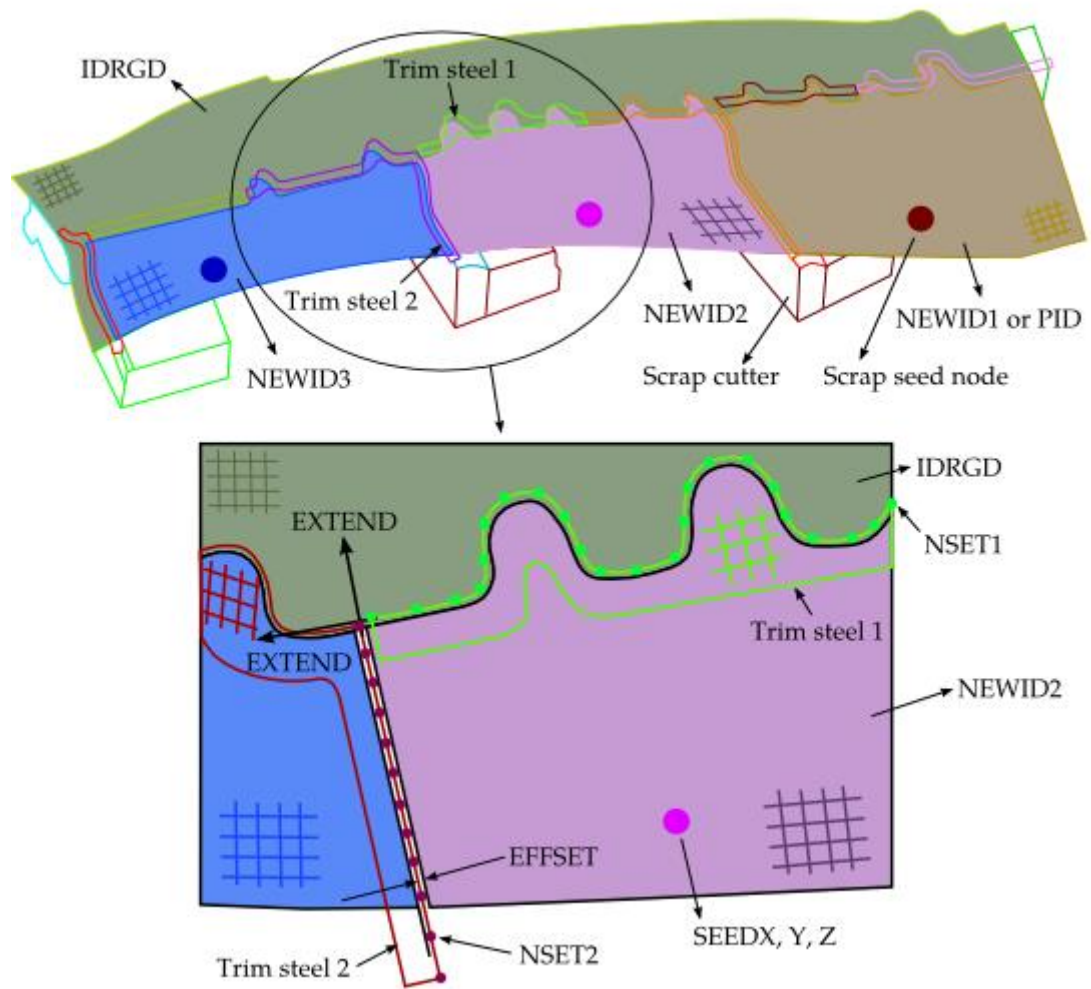


Figure 4 Trimming of multiple scraps and parameter definitions in the scrap trimming method.
Model courtesy of the Ford Motor Company.

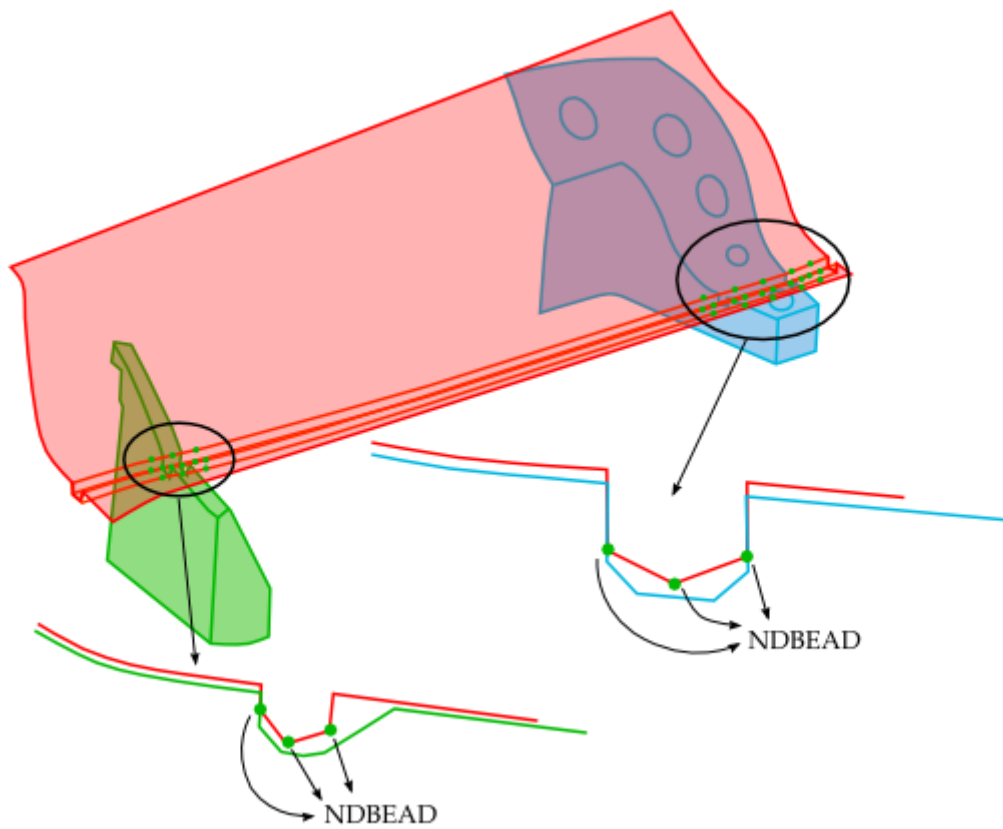


Figure 5 NDBEAD used to clear the initial interference.
Model courtesy of the Ford Motor Company.

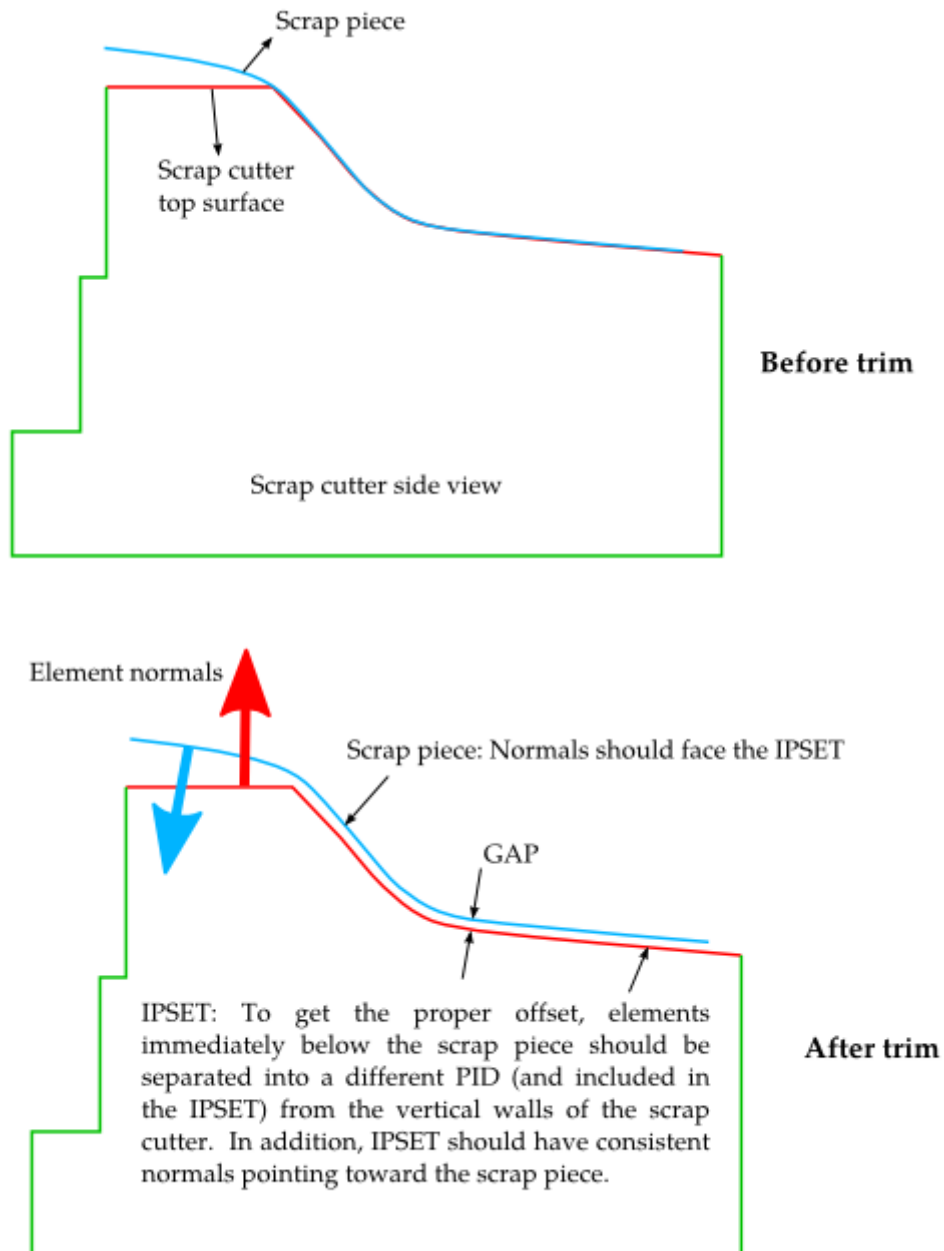


Figure 6 IPSET to automatically offset the scraps from the die surfaces.

Table 1 Computing speed and the max/min deviations from the springback mesh to the target scan for an automotive part, under various combinations of NSKIP and IFAST. All runs were made on a 1 CPU XEON E5520 machine, with 685132 elements on the target scan and 135635 elements on the springback mesh.

NSKIP	IFAST=0		IFAST=1	
	CPU time	Max/Min (mm)	CPU time	Max/Min (mm)
2	10 min 38 sec	1.28/-1.59	4 min 3 sec	1.22/-1.59
5	4 min 49 sec	1.21/-1.59	1 min 59 sec	1.25/-1.61
10	2 min 46 sec	1.27/-1.59	1 min 18 sec	1.44/-1.53
20	1 min 24 sec	1.27/-1.59	59 sec	1.42/-1.64
50	50 sec	1.22/-1.61	40 sec	1.43/-1.67

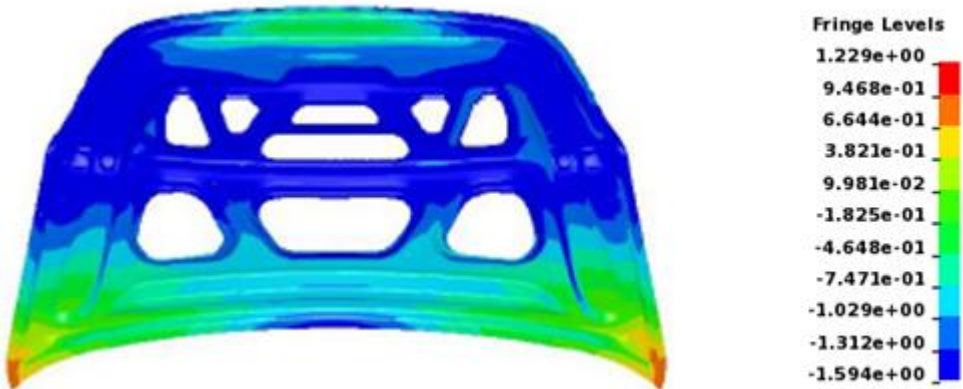


Figure 7 An example of a best fit result.

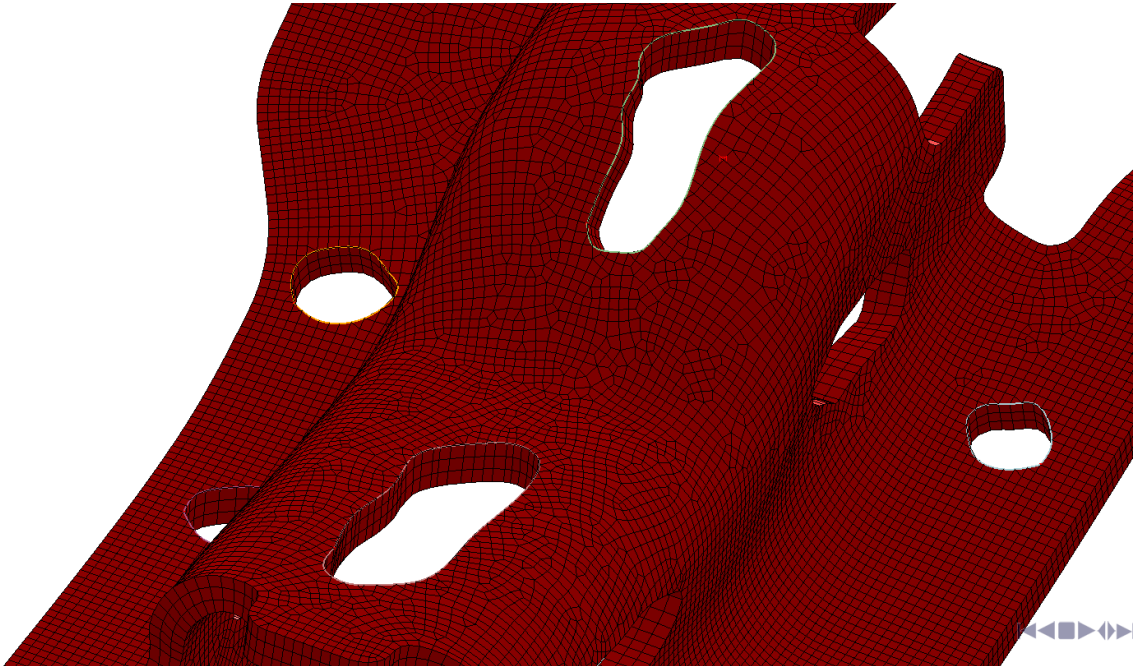


Figure 8 2-D (directional) trimming of 3-layer solid elements.

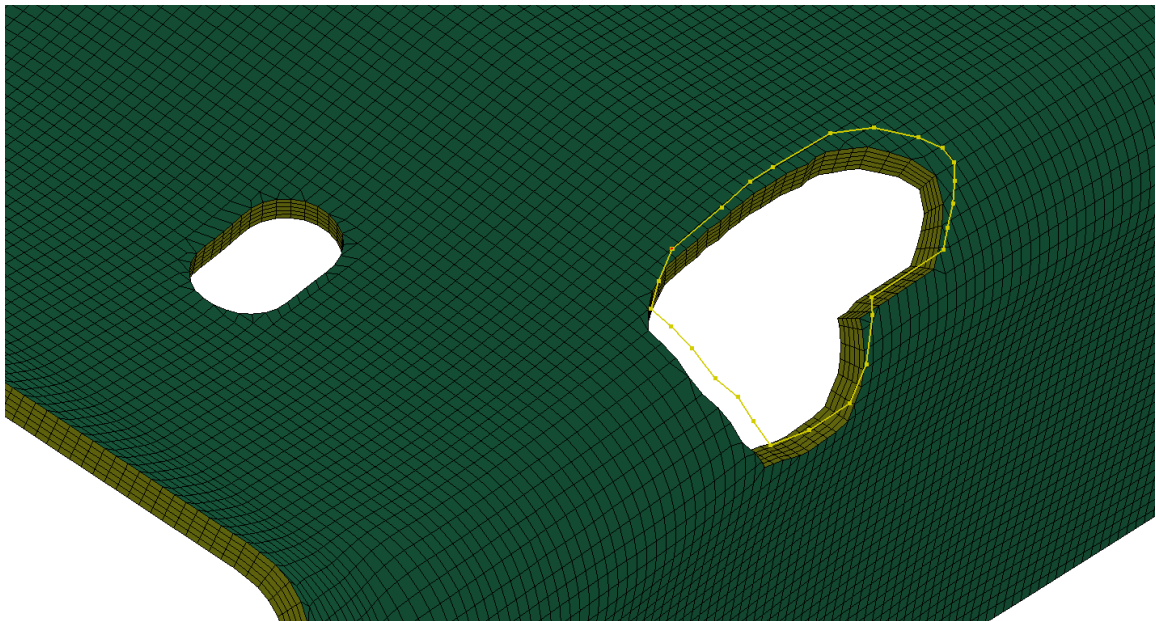
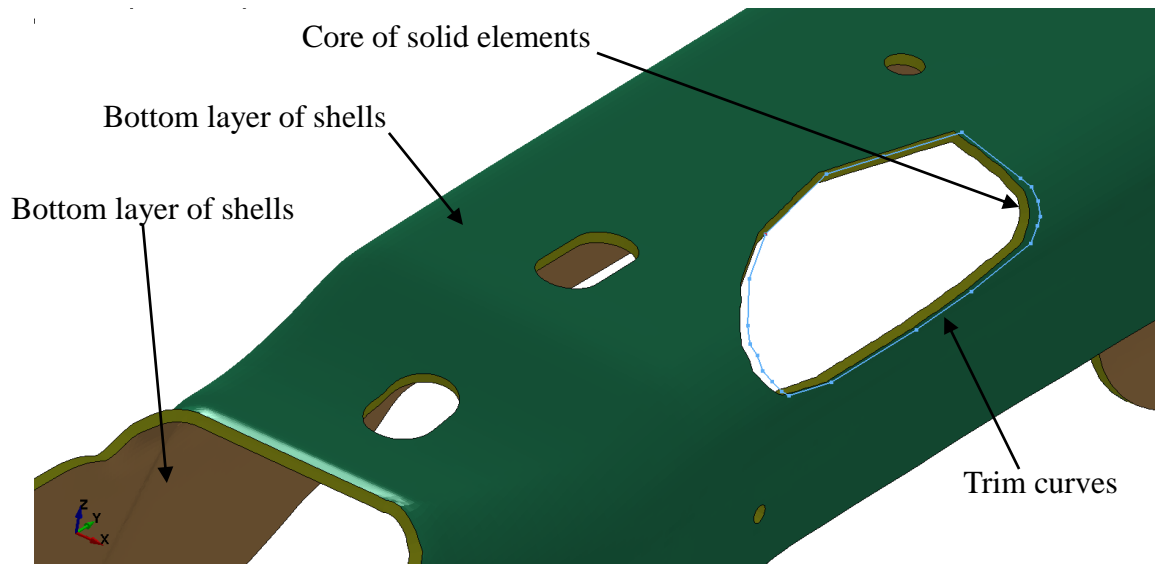


Figure 9 3-D trimming laminates

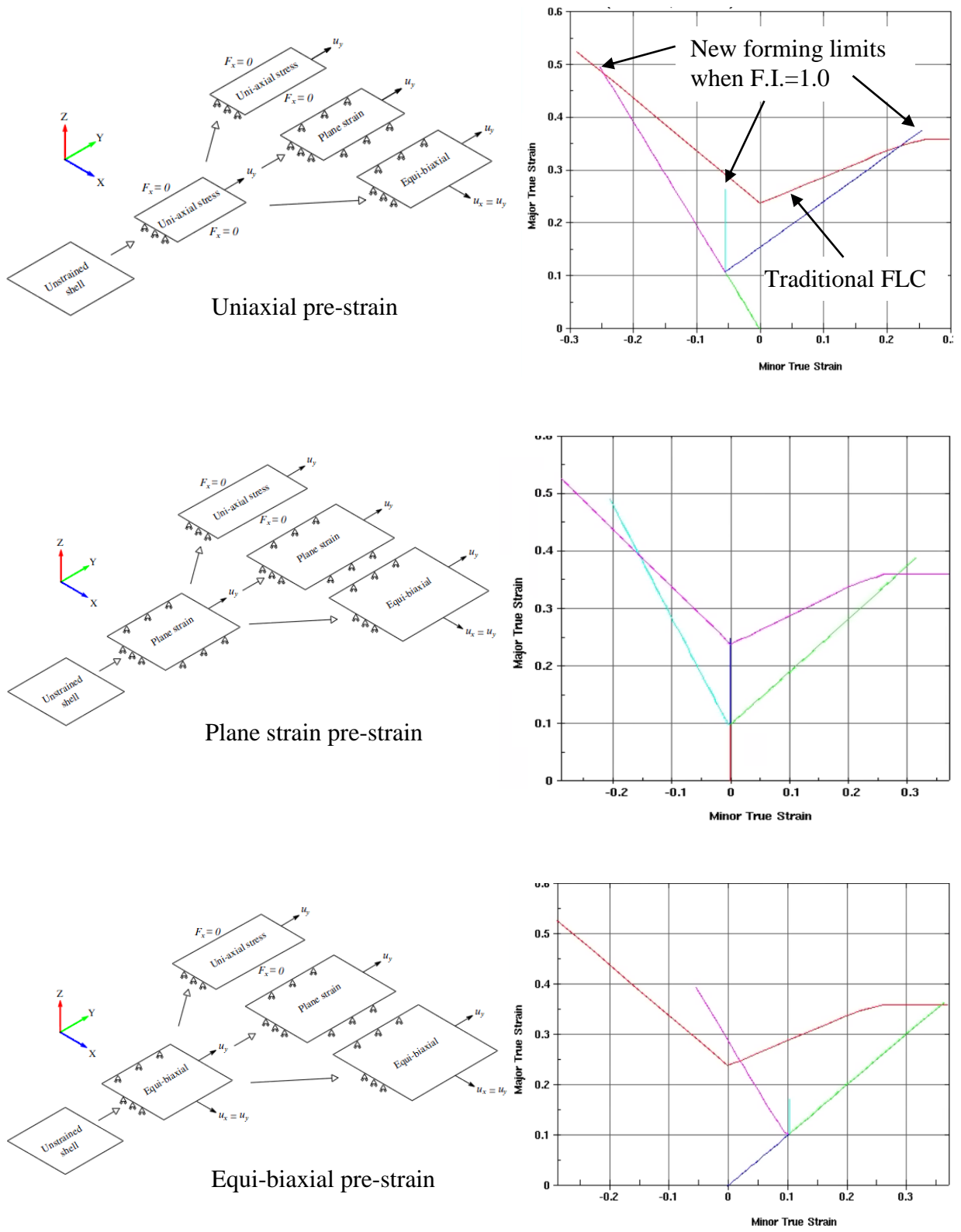
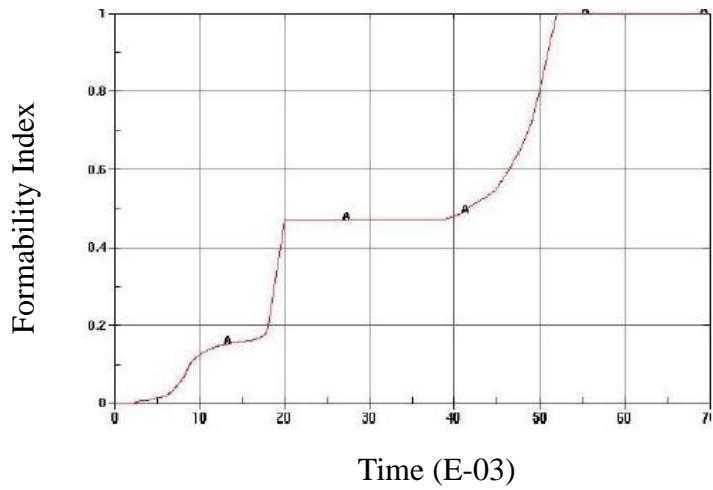
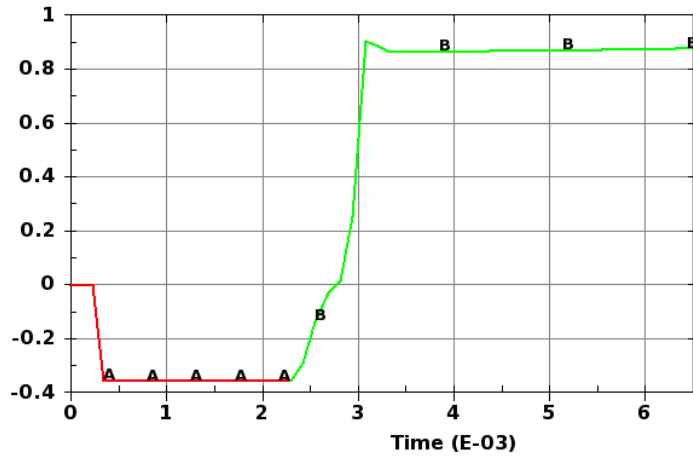


Figure 10 Formability Index (F.I.) tests along three pre-strain paths on a single shell elements for *MAT_125.

History variable #1 (F.I.) history



Strain ratio (history var. #2)



EPS (history var. #3)

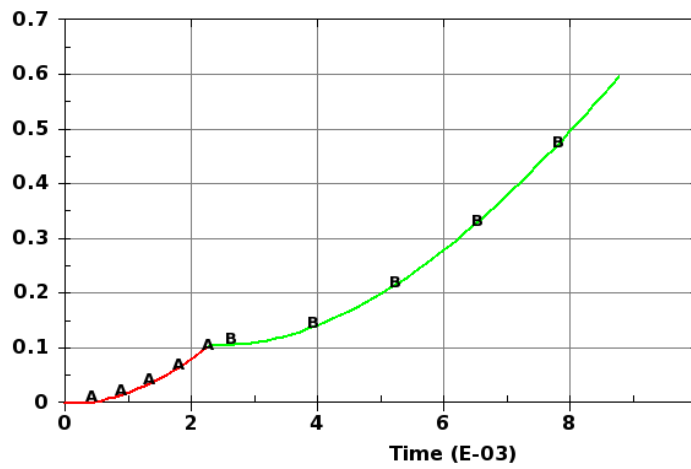


Figure 11 Continuous history variable plots for the single element tests in *MAT_125.

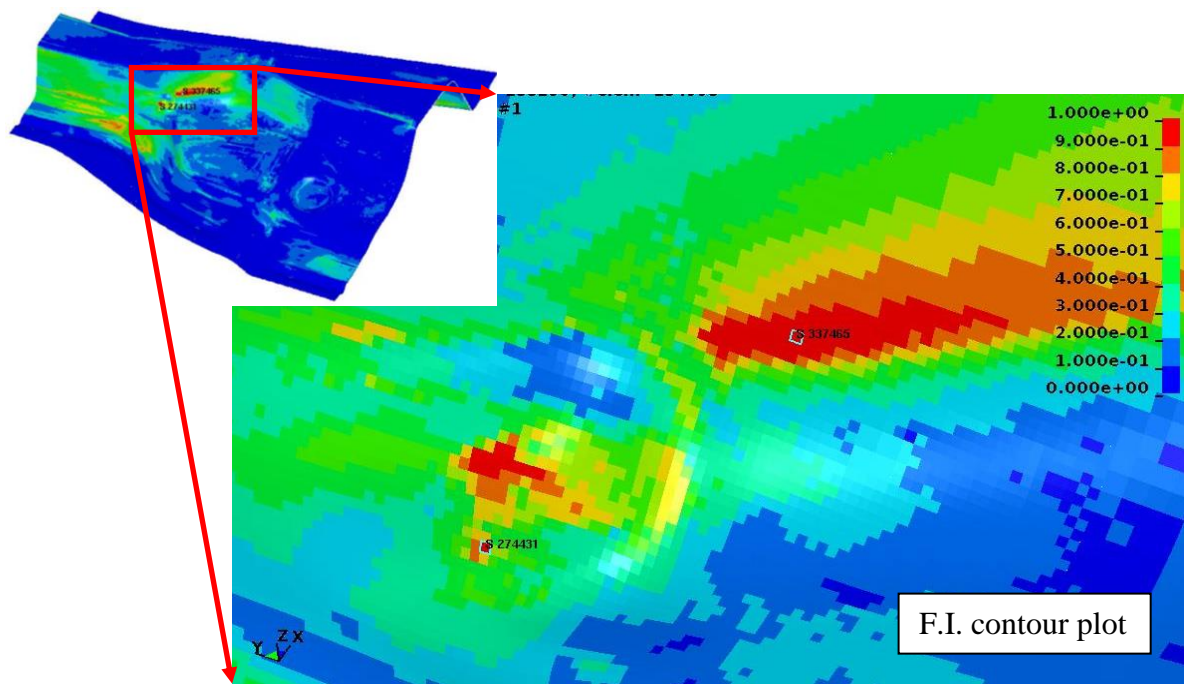


Figure 12 F.I. contour plot on a cross member.