Advances in LS-DYNA® for Metal Forming (I)

Xinhai Zhu, Li Zhang, Yuzhong Xiao, and HouFu Fan
Livermore Software Technology Corporation

Abstract

The following will be discussed:

- *Enhancements in *CONTROL_FORMING_ONESTEP*
- *Smoothing of strain ratio (β) for failure prediction under nonlinear strain paths, with *CONTROL_ FORMING_TOLERANC*
- *"Soft 6" contact improvement for gage pin contact*
- *Weld line mapping with *INTERFACE_WELDLINE_DEVELOPMENT*
- *Improvements in *BOUNDARY_SPC_SYMMETRY_PLANE (SET)*
- *Improvements in springback compensation*
- *Improvements in *ELEMENT_LANCING*

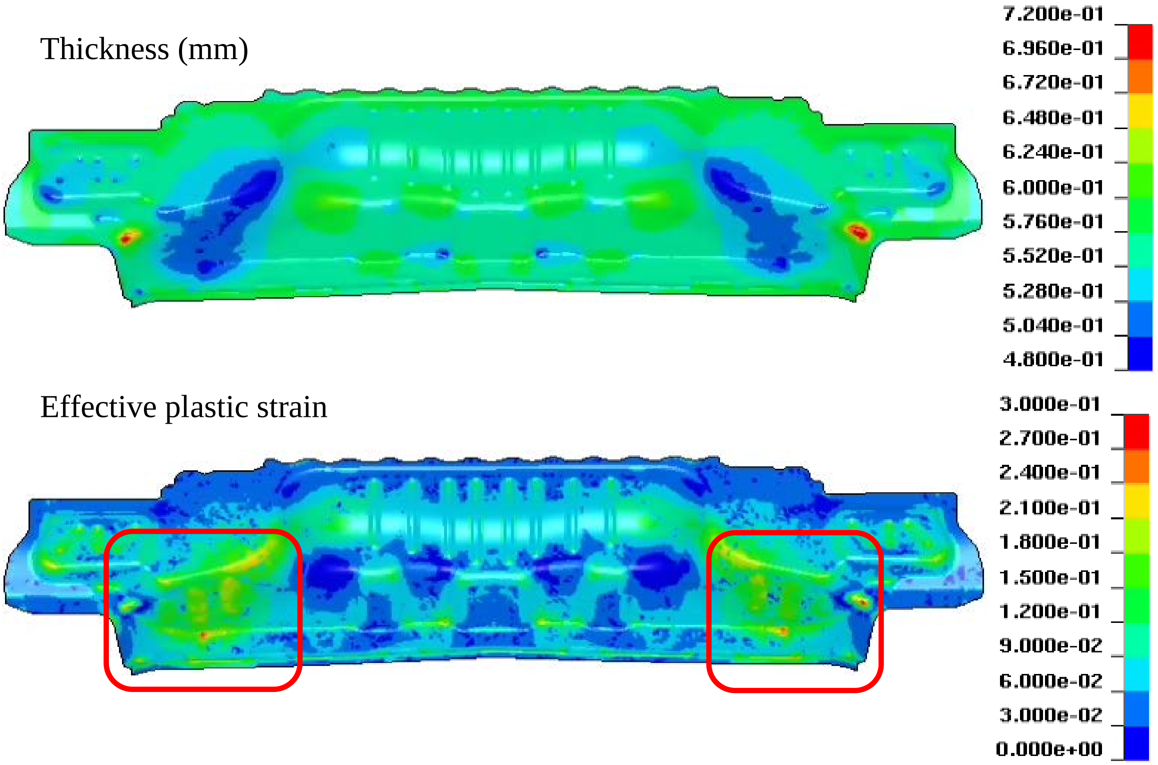
Enhancements in *CONTROL_FORMING_ONESTEP

When one-step forming method was first introduced back in 2011, all quadrilateral elements in the model were split into two triangular elements internally for calculation. As of Dev112682, this original formulation (with no option) is set as option TRIA. A new option QUAD (one integration point in plane, Dev112071) is now available supporting quadrilateral elements, and with improved algorithm in various areas, which leads to better results. In addition, this option greatly improves calculation speed under multiple CPUs in SMP mode. Another new option QUAD2 (four integration points in plane) improves upon the option QUAD with fully integrated element formulation, which further improves results in terms of thinning and plastic strain with slightly longer CPU times.

Results in Figure 1, Figure 2 and Figure 3 show thickness and effective plastic strain contour plots for option TRIA, QUAD and QUAD2, respectively. In the areas marked, both thickness and plastic strains are better in QUAD2 than the other options.

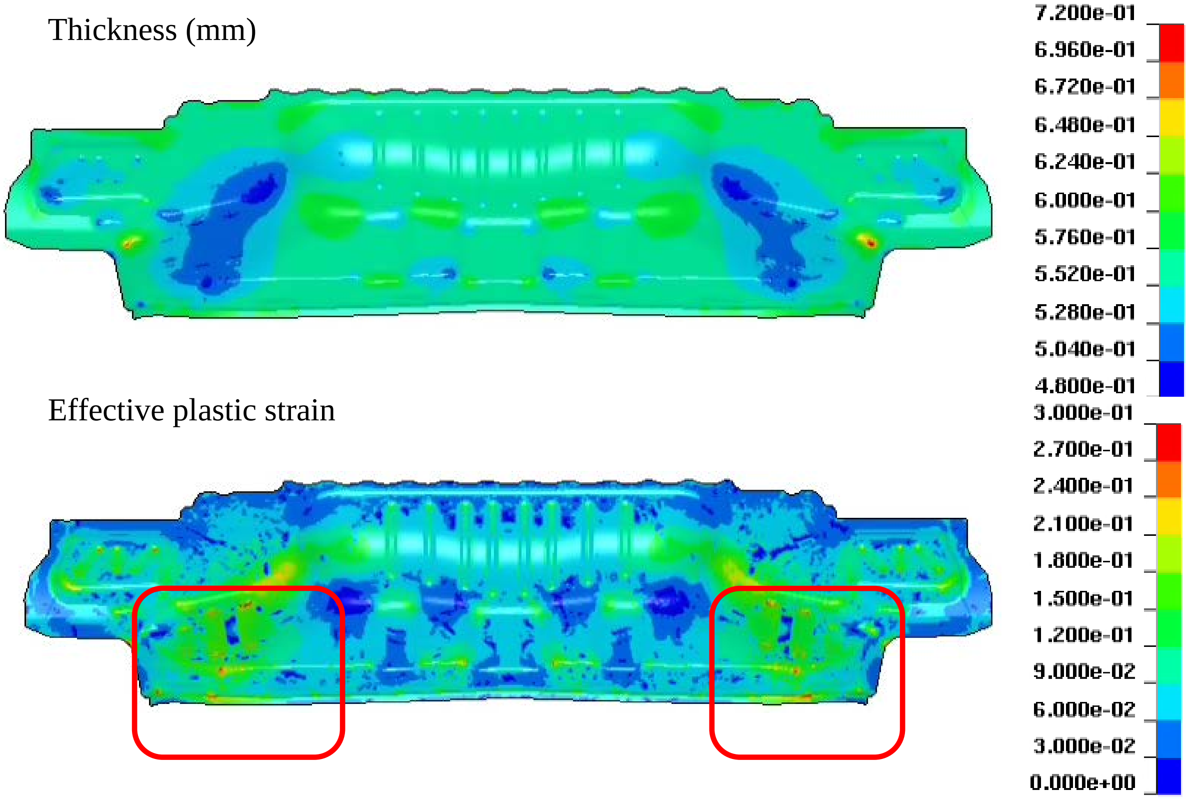
Performance comparisons among the three options can be found in Table 1. The option QUAD2 is set as the default option as of Revision 112682 and is the recommended option.

On a general note, more stretch in the part could be achieved by using the option _DRAWBEAD. In addition, process parameters, such as blank size (final blank shape) should be carefully considered to achieve a realistic one-step simulation results.


Thickness (mm)

Effective plastic strain

Figure 1 Results from the original option TRIA.


Thickness (mm)

Effective plastic strain

Figure 2 Results from option QUAD

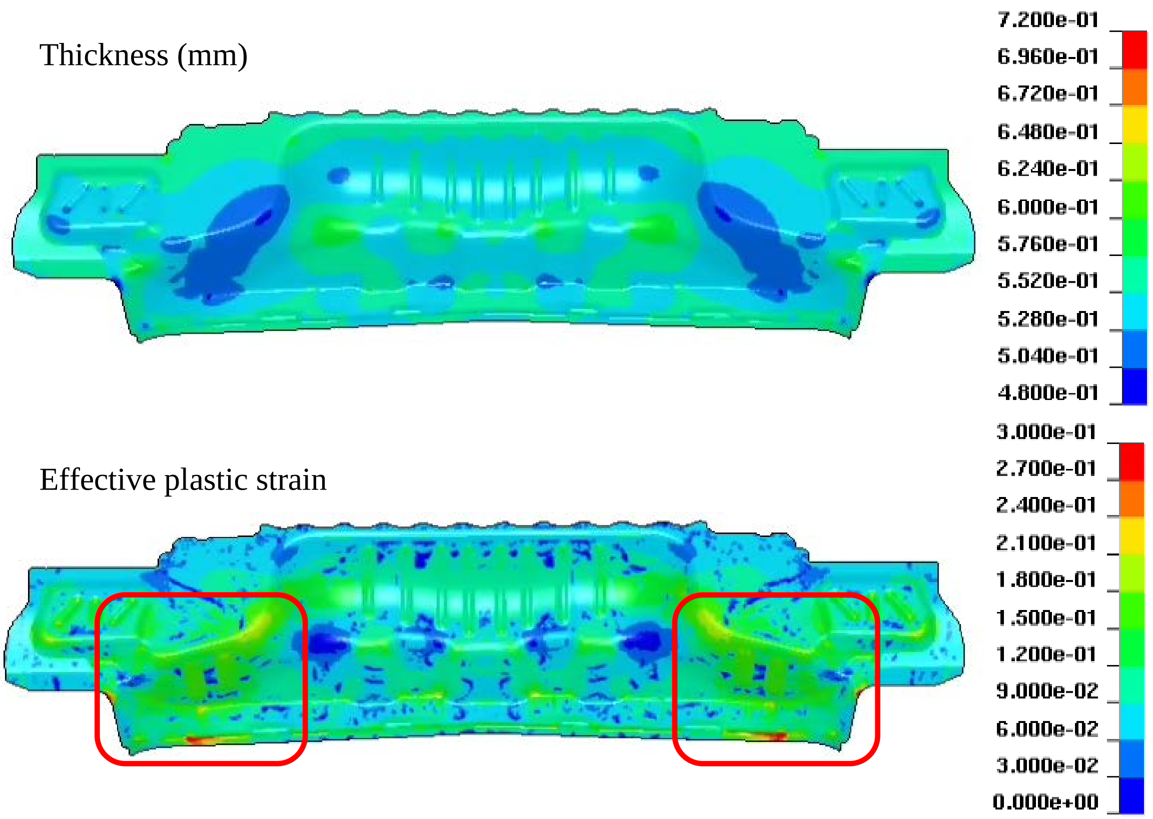


Figure 3 Results from option QUAD2

Table 1 Computing performance comparison of various options.

	Number of elements	Calculation speed (D.P. SMP Rev.112720, 8 CPUs)		
		Option TRIA	Option QUAD	Option QUAD2
A hat shaped part	71000	21.0 min	14.1 min	16.6 min
An upper dash panel	61700	24.5 min	11.5 min	17.2 min

Smoothing of Strain Ratio (β) with *CONTROL_FORMING_TOLERANC

Formability Index (F.I.) is used to assess sheet metal formability (necking failure). Explicit dynamic analysis introduces noises in strain ratio (minor/major strain increment) calculation affecting the F.I. values. A new keyword *CONTROL_FORMING_TOLERANC was developed jointly with the Ford Motor Company, to address the issue.

The incremental change of in-plane major and minor strains are smoothed according to the following formula:

$$\Delta\epsilon 1_{(n-1)} * (1-\alpha) + d\epsilon 1_{(n)} * \alpha$$
$$\Delta\epsilon 2_{(n-1)} * (1-\alpha) + d\epsilon 2_{(n)} * \alpha$$

where, $d\epsilon 1_{(n)}$ and $d\epsilon 2_{(n)}$ are incremental changes of $\epsilon 1$ and $\epsilon 2$ in the current time step n, $\Delta\epsilon 1_{(n-1)}$ and $\Delta\epsilon 2_{(n-1)}$ are incremental changes of $\epsilon 1$ and $\epsilon 2$ in the previous time step *n-1*. The weighting coefficient α regulates the smoothness of the incremental changes in $\epsilon 1$ and $\epsilon 2$.

Strain ratio β results from smoothed incremental major and minor strains and stored in history variable #2 along with additional information (see Table 2 for details) in ".o" file if running in a batch queueing system, or directly dumped onto the screen if running in an interactive window.

$$\beta = \frac{\Delta\epsilon 2_{(n-1)} * (1-\alpha) + d\epsilon 2_{(n)} * \alpha}{\Delta\epsilon 1_{(n-1)} * (1-\alpha) + d\epsilon 1_{(n)} * \alpha}$$

The upper limit of β is set at 1.0 while the lower limit is:

$$-\frac{\bar{r}}{1+\bar{r}}$$

where $\bar{r}$ is the anisotropic parameter:

$$\bar{r} = \frac{r_0 + 2r_{45} + r_{90}}{4}$$

where r_0, r_{45} and r_{90} are Lankford parameter in the rolling, diagonal and transverse direction, respectively.

The keyword is applicable to *MAT_036, *MAT_037, *MAT_125, *MAT_226, with the nonlinear strain path option (NLP). History variables are summarized below for *MAT_036, *MAT_037, *MAT_226:
#1 - F.I.
#2 - β (strain ratio)
#3 - effective plastic strain

For *MAT_125, history variables are designated as follows:
#24 - F.I.
#25 - β (strain ratio)
#26 - effective plastic strain

As shown in Figure 4, beta smoothed using smoothing algorithm is much better than unsmoothed one. Most importantly, a plot of strain path (Figure 5) in the traditional FLD space ($\epsilon 1$ vs. $\epsilon 2$) confirms the terminal beta is approximately 0.9, which is much closer to the smoothed beta value (Figure **Error! Reference source not found.**) at the end of the simulation.

Table 2 ".o" file output information and positions. Note only the mid-IP information are output.

Output items	IP #	Element ID	Time	β	$\epsilon 1$	$\epsilon 2$
Columns	1st to 8th	9th to18th	19th to 29th	30th to 40th	41th to 51th	52th to 62th

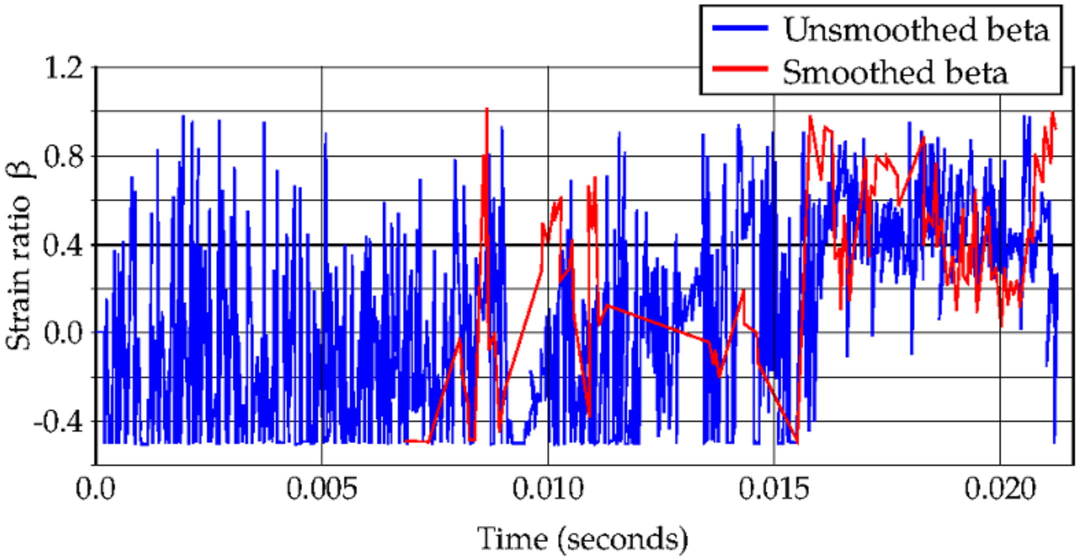


Figure 4 Effect of smoothing on strain ration β

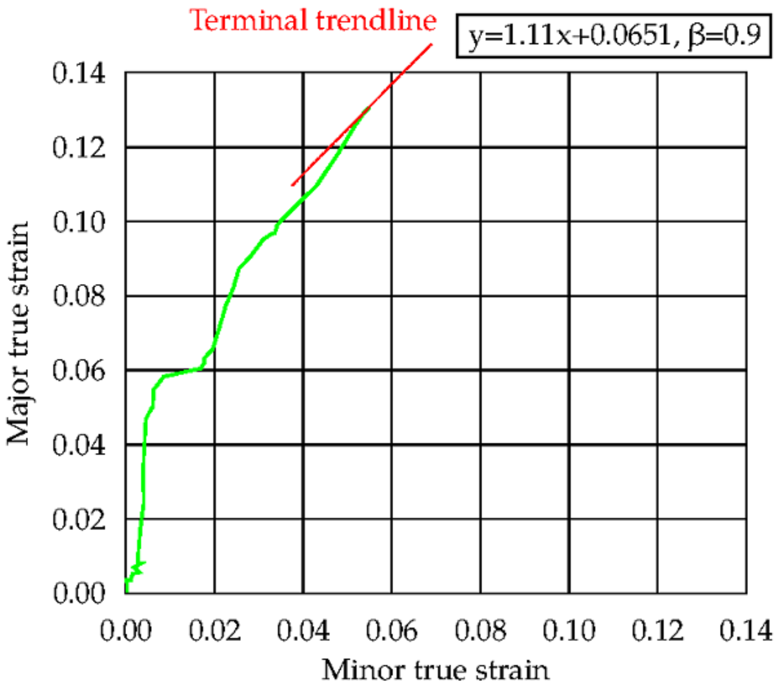


Figure 5 Strain path and terminal strain ratio β value.

"Soft 6" Contact Improvement for Gage Pin Contact

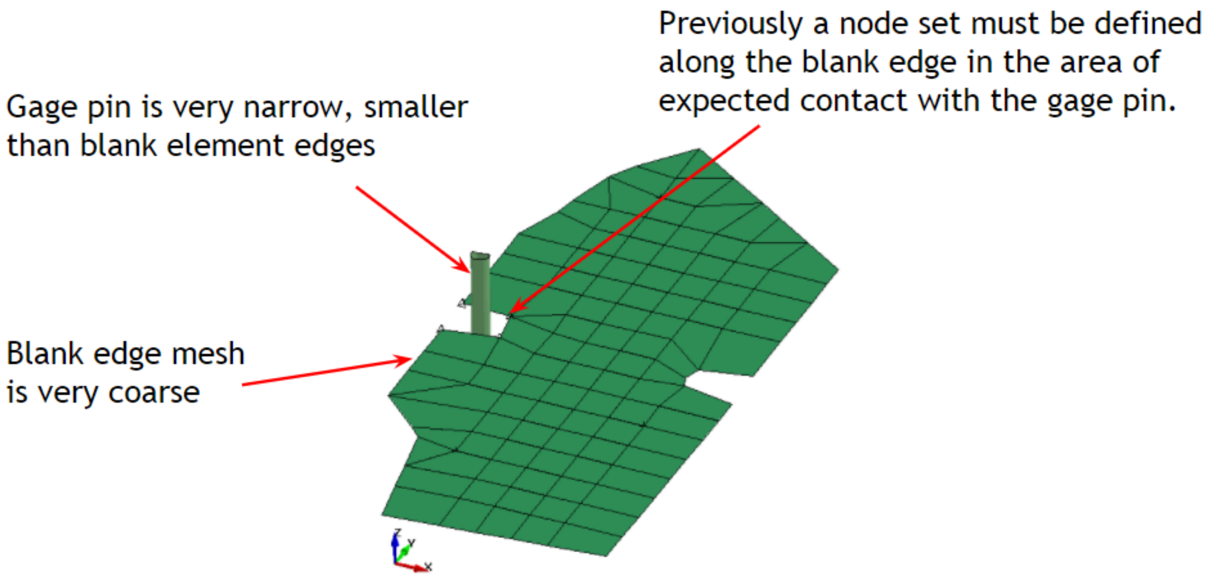
Contact option SOFT=6 can be activated in *CONTACT_... Optional Card A, using the variable SOFT. This specialized contact treatment addresses contact issues in situation where blank gage pins are narrow or small and blank mesh are coarse (Figure 6), leading to missing contact in some cases. This feature applies to gravity loading and forming of sheet blank with mesh, and for use with *CONTACT_FORMING_NODES_TO_SURFACE only. In addition, the variable ORIENT in *CONTROL_CONTACT must be set to "4". Currently this feature is available in double precision, SMP only, starting in Revision 81297 and later releases.

Nodes along the entire or a portion of the blank edge to be contacted with gage pins must be included in a node set for the variable SSID in *CONTACT_FORMING_NODES_TO_SURFACE (Figure 7 left). The nodes in the node set must be listed in a consecutive order, as defined "*by path*" in *LSPP4.3*, under *Model* → *CreEnt* → *Cre* → *Set Data* → **SET_NODE*. Note there is no thickness existing for either the blank edge or the gage pins. *Starting in Revision 110072, SSID in *CONTACT_FORMING_NODES_TO_SURFACE can be input as part ID of the blank, making it much easier to use SOFT=6.*

In an example shown in Figure 7, a gage pin goes right through the blank edge without the use of the option. Using the option, the blank edge stops the gage pin correctly, in Figure 8.

IMPROVEMENT #1 (Revision 109342). Instead of defining a node set using SSTYP=4, an SSTYP of 3 (a part PID, **not the part set ID**) can also be used for the SSID, simplifying the definition of contact interfaces for SOFT=6.

IMPROVEMENT #2 (Revision 118623). Certain coarse blank mesh causes an initial contact penetration between blank gaging hole edge and the gaging pin mesh. Additional nodes will be created on the blank along the hole edge and then moved to clear the initial penetration with the gage pin, as shown in Figure 9.



Previously a node set must be defined along the blank edge in the area of expected contact with the gage pin.

Gage pin is very narrow, smaller than blank element edges

Blank edge mesh is very coarse

Figure 6 Gage pin contact issue.

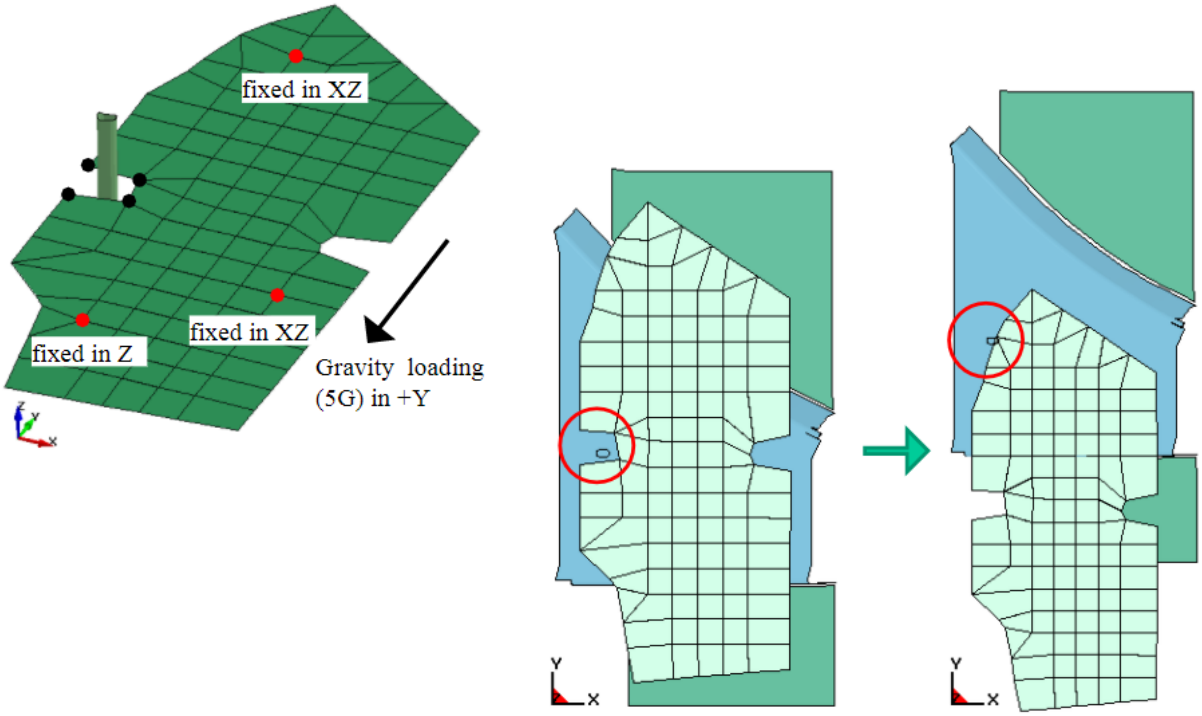


Figure 7 Missing contact without SOFT=6.

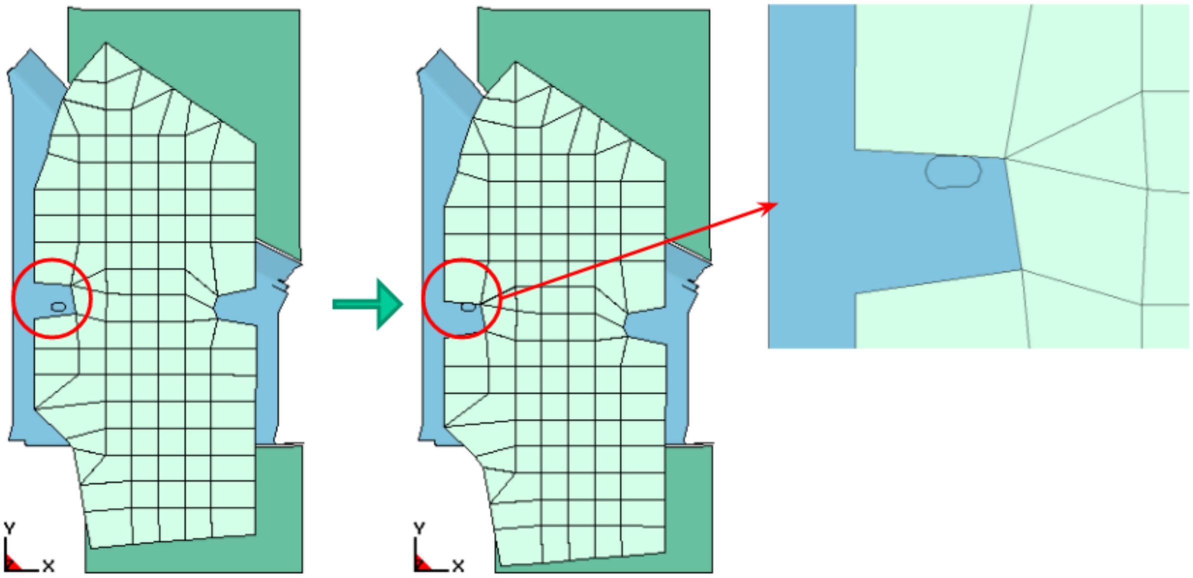


Figure 8 Contact captured with SOFT=6.

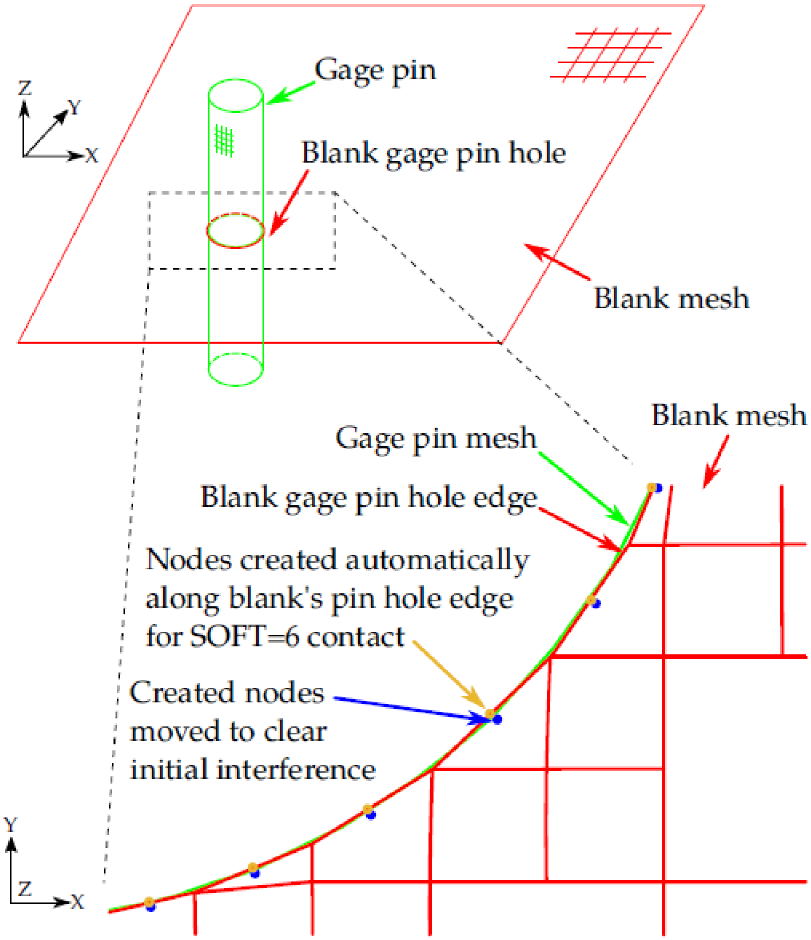

Gage pin
Blank gage pin hole
Blank mesh
Z
Y
X
Blank mesh
Gage pin mesh
Blank gage pin hole edge
Nodes created automatically along blank's pin hole edge for SOFT=6 contact
Created nodes moved to clear initial interference
Y
Z
X

Figure 9 Removal of initial penetration between gage pin and blank edge.

Weld Line Mapping with *INTERFACE_WELDLINE_DEVELOPMENT

In forming of tailor welded blanks, an initial straight weld line could become a curve on the formed part. The amount of deviation of the formed weld curve from its initial line depends on the part shape and forming conditions. Sometimes the formed weld curve is not desirable; so a correction to the initial weld curve is needed. This feature allows for determination of the final weld curve given the initial weld curve and vice-versa without performing another simulation. Mesh with adaptivity for the initial blank and final part is supported.

Preparation of the final (formed) weld curve may involve its projection onto the final blank mesh if it does not exactly lie on the mesh surface. This can be done with LS-PrePost® 4.2 via the menu option *GeoTol* → *Project* → *Project*, select *Closest Projection*, select *Project to Elements*, then define the destination mesh and source curves, and hit *Apply*. Sometimes the target curve may need enough points before projection; the points may be added via menu option *Curve* → *Spline* → *Method (Respace)* → *by number*. To write the curve out in *DEFINE_CURVE_TRIM_3D format, use *Curve* → *Convert* → *Method (To DEFINE_CURVE_TRIM)* → *To Key*, then write out the keyword using *FILE* → *Save Keyword*.

Computed weld curves are written with *DEFINE_CURVE_TRIM_3D keyword into a file called weldline.ibo (or, weldline_f.ibo, depending on the IOPTION). The format of this file follows the keyword's specification. LS-PrePost 4.0 can convert the computed curve to IGES; see the procedure in the manual pages for *INTERFACE_BLANKSIZE_DEVELOPMENT. After hitting the button *Apply*, the curves will show up in the graphics window, and *File* → *Save as* → *Save Geom as* can be used to write the curves out in the IGES format.

As shown in Figure 10, given the initial (initialblank.k) and final blank (finalblank.k) configuration and a final formed weld curve (finalweldingcurve.k), the following input calculates a new initial weld curve on the initial blank. In this case, the final weld curve is specified as straight in the drawn panel.

The output is the initial weld curve in the file weldline.ibo, Figure 10. Nodes of element edges that intersect the initial weld curve are output in affectednd_i.ibo; while nodes of element edges that intersect the final formed weld curve are output in affectednd_f.ibo, Figure 11.

To verify the predicted weld curve, the initial blank can be re-meshed according to the curve. A draw simulation can be performed again to confirm the final weld curve as straight, Figure 12.

Likewise, if given an initial weld curve (initialweldingcurve.k) and a final weld curve (weldline_f.ibo) can also be calculated.

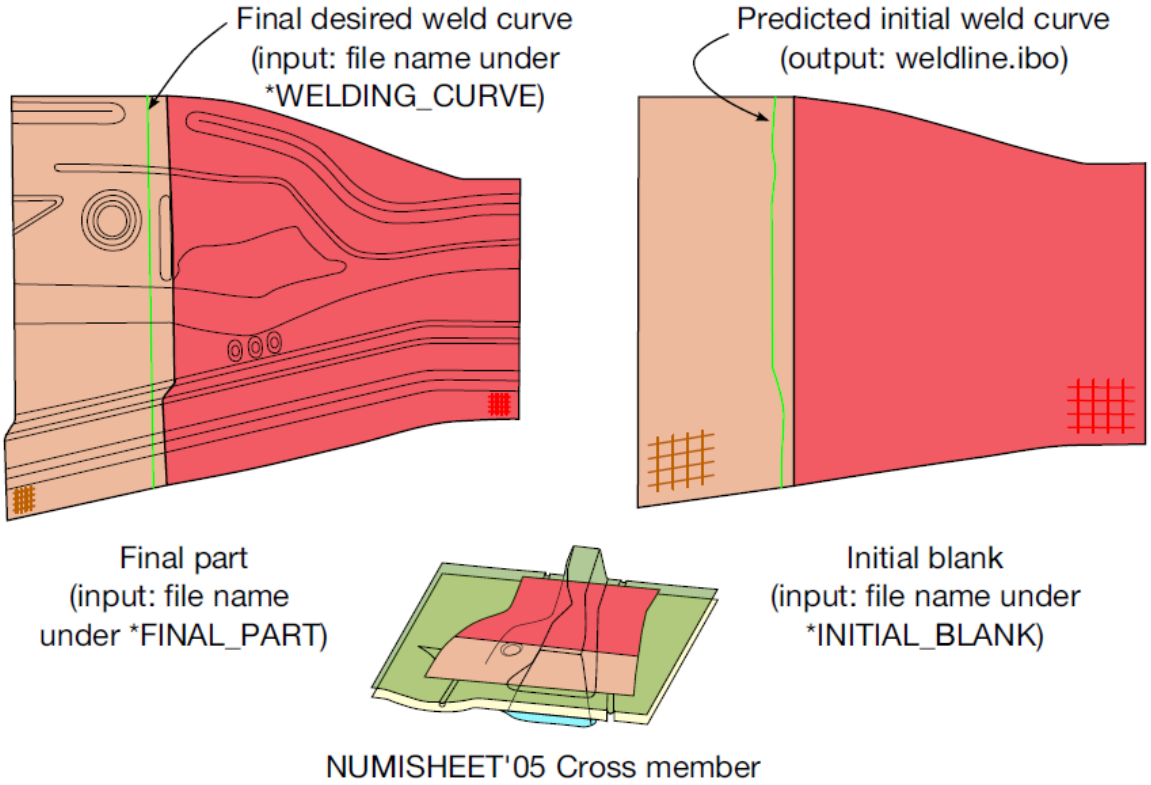
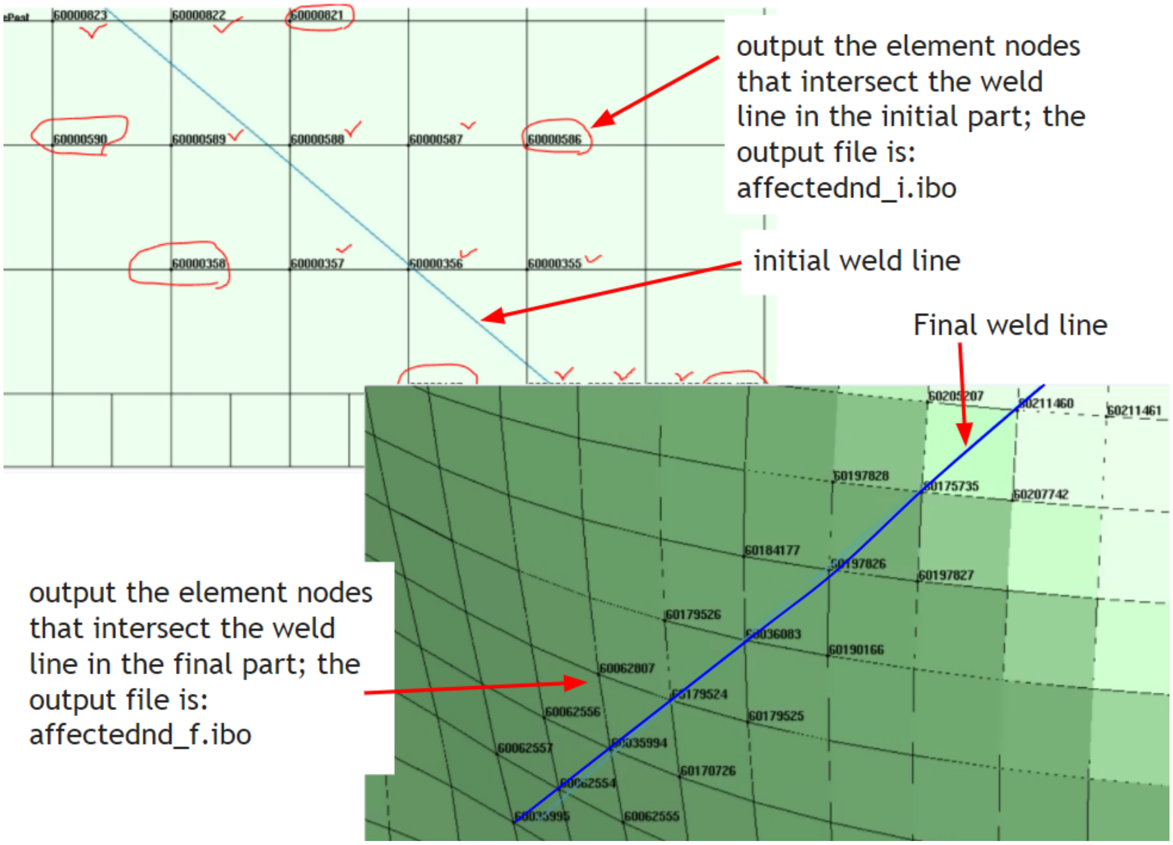


Figure 10 Calculating initial weld curve from final weld curve.



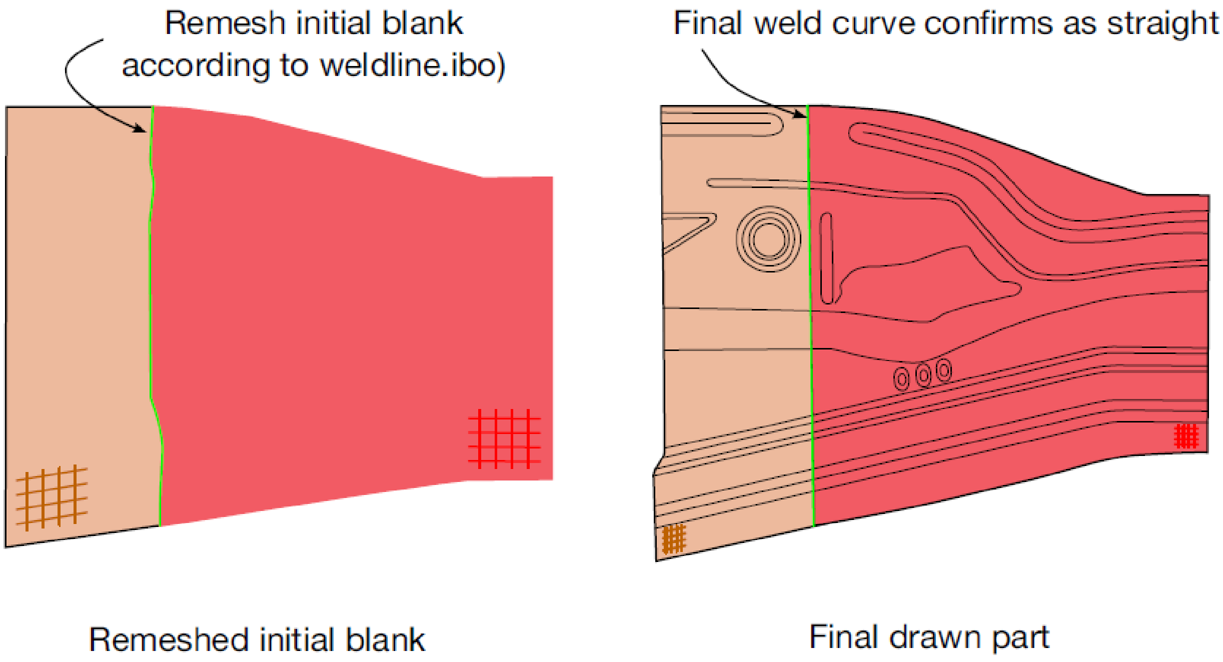
output the element nodes that intersect the weld line in the initial part; the output file is: affectednd_i.ibo

initial weld line

Final weld line

output the element nodes that intersect the weld line in the final part; the output file is: affectednd_f.ibo

Figure 11 Output of affected nodes.


Remesh initial blank according to weldline.ibo)

Final weld curve confirms as straight

Remeshed initial blank

Final drawn part

Figure 12 A verification simulation may be needed.

Improvements in *BOUNDARY_SPC_SYMMETRY_PLANE (SET)

This keyword constrains nodes that are within some distances (a tolerance) of a plane to have no motion orthogonal to that plane. This keyword is usually used for a geometric symmetry plane, so that the full geometry does not have to be modeled; and it supports both h- and r-adaptivity.

As shown in Figure 13, a symmetric plane's position is defined using a point's coordinates X, Y and Z, and its orientation is defined by a vector component VX, VY and VZ. All nodes within a specified tolerance with the variable TOL will be constrained in motion orthogonal to the plane.

Originally designed for metal forming simulation only, the feature has garnered much interests from the users to the point where many of the initial limitations are removed so it can be applied in many other simulation fields. Two of the eliminated restrictions are:

1) Only two such (symmetry plane) definitions can be used when the symmetry planes are not parallel to the global XY, YZ, or ZX planes, and,
2) For multiple symmetry planes, position coordinates defined by the variables X, Y, and Z on all symmetry planes must have the same values for these fields.

Starting from Dev Revision 124122, multiple symmetric planes can be defined even if they are not aligned with the global coordinate systems. Each symmetric plane can be defined with distinctive point coordinates.

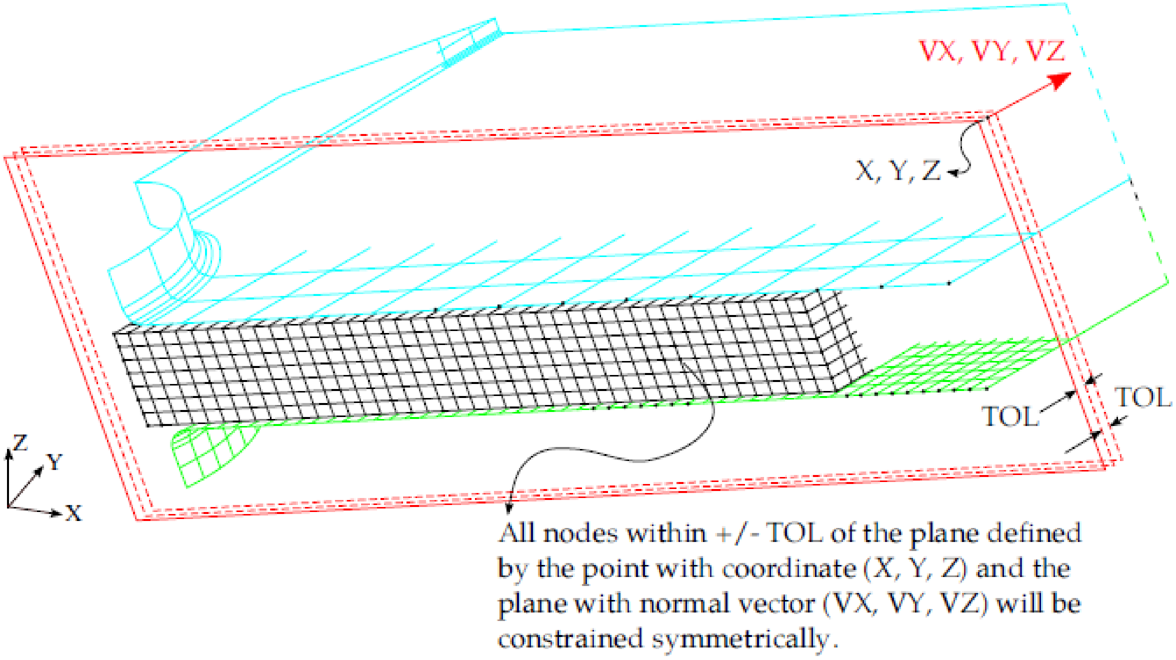

VX, VY, VZ
X, Y, Z
TOL
TOL
Z
Y
X
All nodes within +/- TOL of the plane defined by the point with coordinate (X, Y, Z) and the plane with normal vector (VX, VY, VZ) will be constrained symmetrically.

Figure 13 Defining *BOUNDARY_SPC_SYMMETRY_PLANE_SET

Improvements in Springback Compensation

New keywords and additional options are made to improve the springback compensation convergence. These keywords are:
*INTERFACE_COMPENSATION_3D (new variable TANGENT),
*INTERFACE_COMPENSATION_3D_REFINED_RIGID,
*INCLUDE_COMPENSATION_TRIM_NODE,
*INCLUDE_COMPENSATION_TANGENT_CONSTRAINT.

Compensation of flanging tools is now possible with the keyword *INTERFACE_COMPENSATION_ FLANGE.

Trim Lines Alignment

As shown in Figure 14 (top) and Figure 15 (top middle), rigid bodies that represent stamping tools often are meshed across the trim lines; as such, trim lines could lie right in the middle of large elements, and this is especially true in small curvature areas. In this situation, since there are no nodes right along the trim lines there, the compensated mesh do not conform to the trim lines. This leads to errors in springback error (Figure 15 bottom) in the subsequent dies and therefore affecting the compensation convergence. The following keyword *INTERFACE_COMPENSATION_3D_REFINE_RIGID
refines the rigid body elements and aligns the nodes in the tool mesh (Figure 14 bottom, Figure 16 top and Figure 17 bottom) with the provided trim curves defined using *DEFINE_CURVE_TRIM_3D. The refined rigid tool mesh will be output as rigid_refined.tmp, which can be used to start the iterative springback compensation process, leading to a much successful compensation. The aligned nodes will also be output in a file named bndnd0.tmp.

Maintain Tangency

To improve the tangency that must be maintained between the addendum and binder (Compensation Method 7, refer to Keyword User's Manual I under *INTERFACE_COMPENSATION_3D) in the draw die, and tangential transitions immediately off the rigid tool at the trim curve area to the addendum, a new variable TANGENT is added, and it must be set to "1". In the keyword example below, the rigid_refined.tmp (a resulting file from the option REFINE_RIGID) is used as both the current and the original tool. The aligned nodes file bndnd0.tmp is used under the new keyword *INCLUDE_COMPENSATION_TRIM_NODE, which is used for tangency calculation. Note the variable TANGENT replaces the keyword *INCLUDE_COMPENSATION_TANGENT_CONSTRAINT, which was initially developed to do a similar job.

In Figure 15 (top left), compensated die (in yellow) is not tangent to the binder at the die radius. Using the option TANGENT, the same area has maintained tangency after the compensation, in Figure 16 (left) and Figure 17 (left). Similarly, Figure 16 (top right) shows addendum transition off the trim lines is not smooth (kinky); while the tangency is fully maintained in the same area using the option TANGENT, in Figure 16 (bottom) and Figure 17 (bottom right).

Compensation of Flanging Process with option FLANGE

This new capability is developed specifically to compensate multiple flanging tools for springback in a flanging process. Currently a maximum of two flanging steels in the same process can be defined. Ash shown in Figure 18, each flanging steel's moving direction can be defined by a vector component [Vx, Vy, Vz]. The flanging post can be selected as the master tool surface mesh and the flanging steels are compensated while maintaining the necessary tool gaps as in the original, uncompensated tool mesh. Note in the compensation of flanging process the requirement on the even tool gap everywhere is removed, so flanging steels do not have to follow the flanging post, for example in the flanging steel radius area, etc.

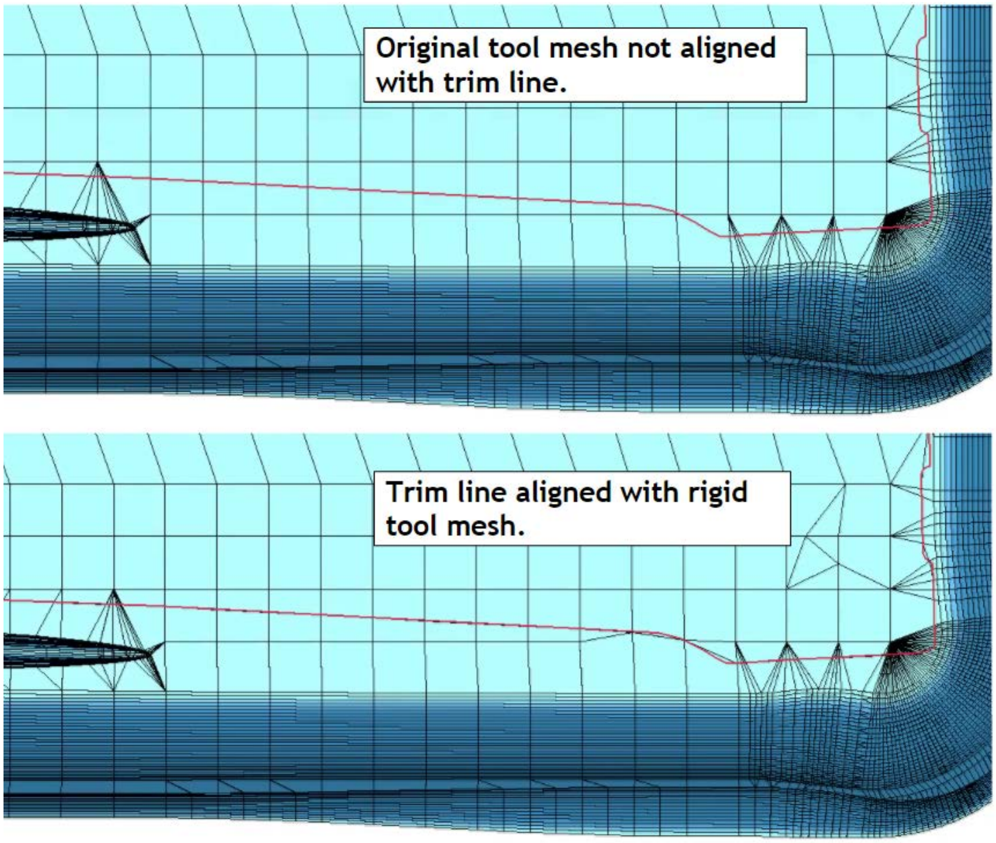
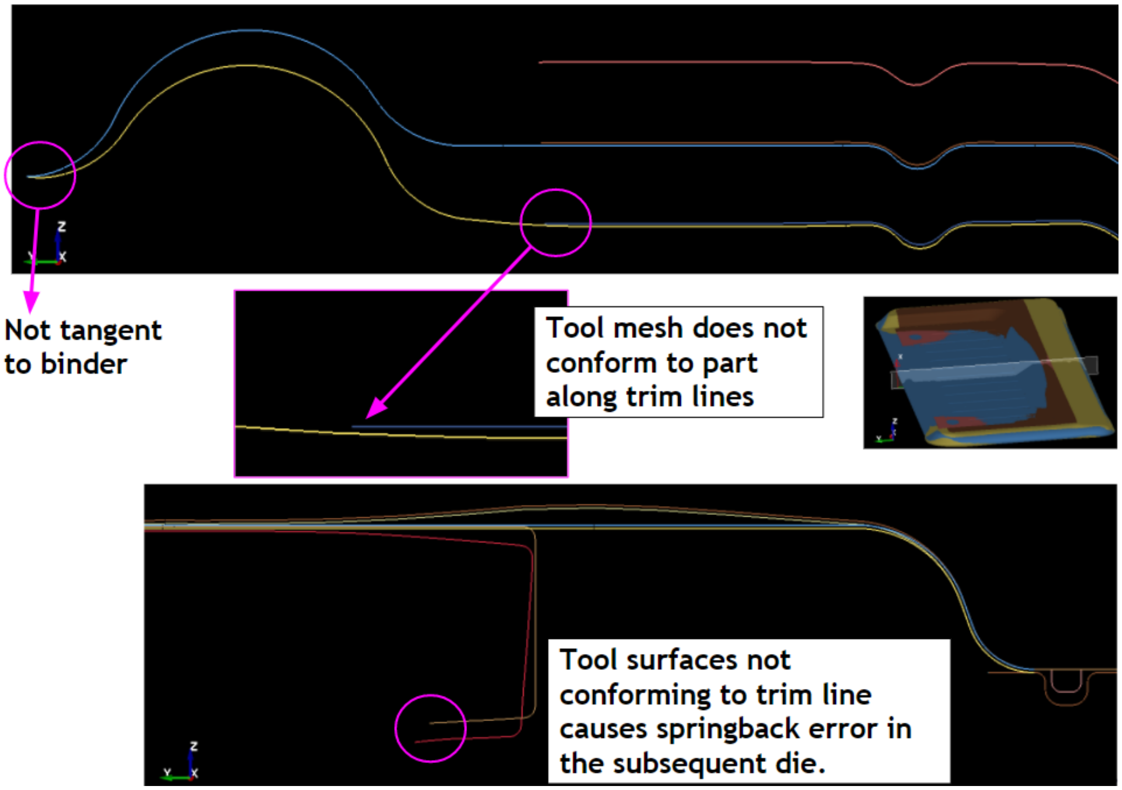

Original tool mesh not aligned with trim line.
Trim line aligned with rigid tool mesh.

Figure 14 Rigid tool mesh do not align with trim lines.

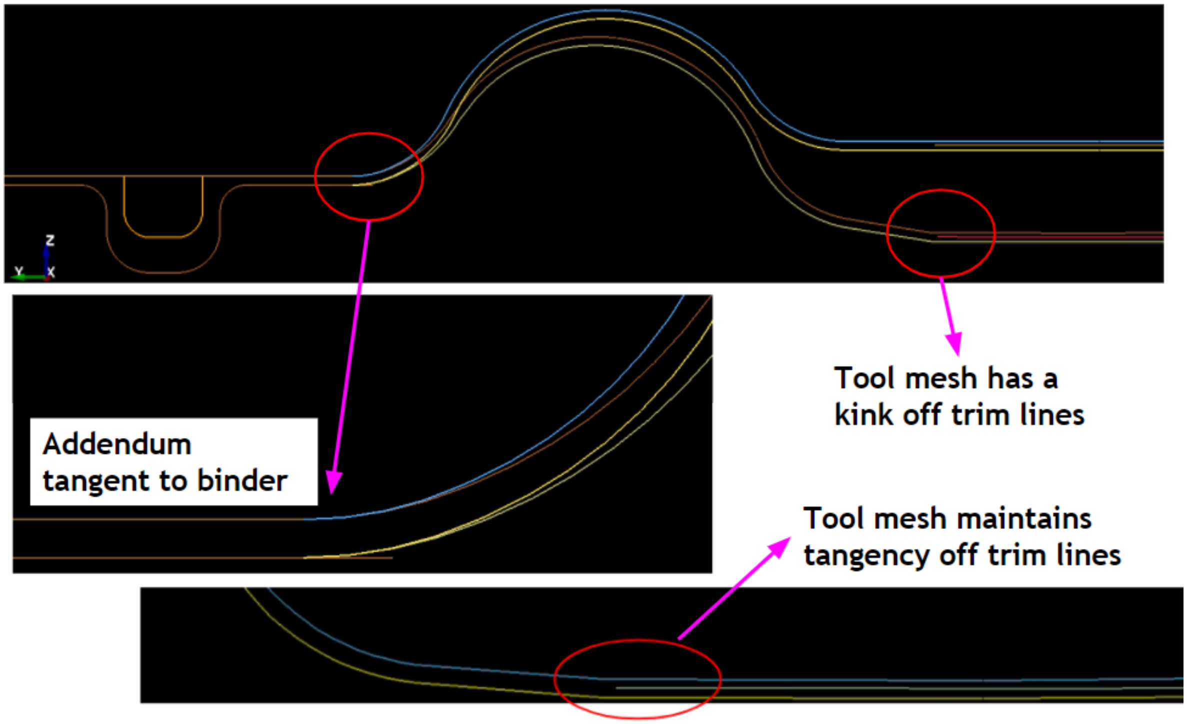


Not tangent to binder

Tool mesh does not conform to part along trim lines

Tool surfaces not conforming to trim line causes springback error in the subsequent die.

Figure 15 Tool mesh does not conform to nominal surface along the trim lines, leading to springback error in the subsequent die.



Addendum tangent to binder

Tool mesh has a kink off trim lines

Tool mesh maintains tangency off trim lines

Figure 16 Rigid tool mesh do not cross trim lines and addendum remains tangent to the binder.

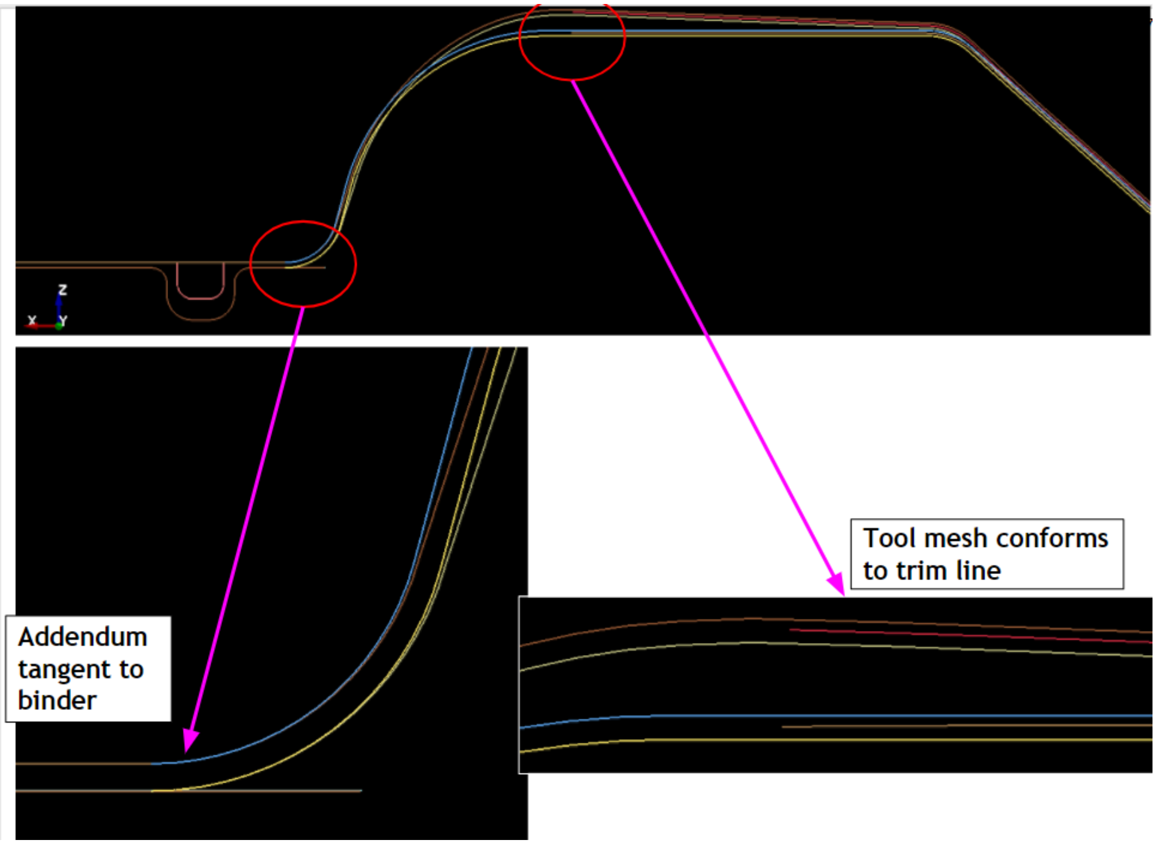


Figure 17 Improved rigid tool compensation geometry

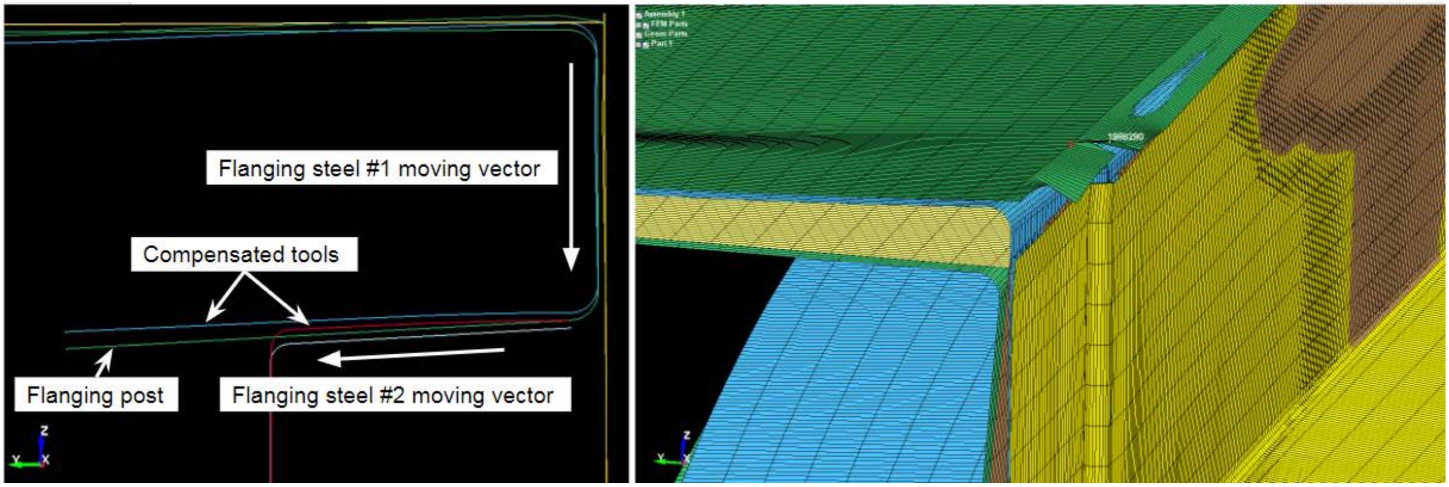


Figure 18 Compensation of flanging die using option FLANGE.

Improvements in *ELEMENT_LANCING

Keywords: *ELEMENT_LANCING, and *DEFINE_CURVE_TRIM_3D can be invoked to conduct lancing operation. With keyword *ELEMENT_LANCING, both instant and progressive lancing can be simulated. The keyword *DEFINE_CURVE_TRIM_3D can be used to define lancing curves. There are several limitations with the original implementation:

1) Only 1 part (PID) is allowed,
2) Only X, Y, Z data pairs are allowed to define lancing curves,
3) No adaptive refinement along lancing route,
4) Lanced scraps cannot be removed,
5) Onset of lancing cannot be defined directly using distance from punch bottom.

Enhancements are made to remove the limitations:

1) A part set is now allowed, which enables lancing across tailor-welded blanks, shown in Figure 21,
2) IGES format curves can now be used as input to define the lancing route,
3) Mesh along the lanced boundary is now automatically adapted to provide a smooth edge, in Figure 20, and Figure 21,
4) Trimming now can be defined after lancing to remove the scrap. This is done by define a seed point indicating the remaining part after trimming, with the new keyword *DEFINE_LANCE_SEED_POINT_COORDINATES, Figure 19.
5) Lancing activation distance can now be defined using a new variable CIVD.

As additional requirements arises from our users, this feature will be continuously improved in the future.

References

LS-DYNA User’s Manual I.

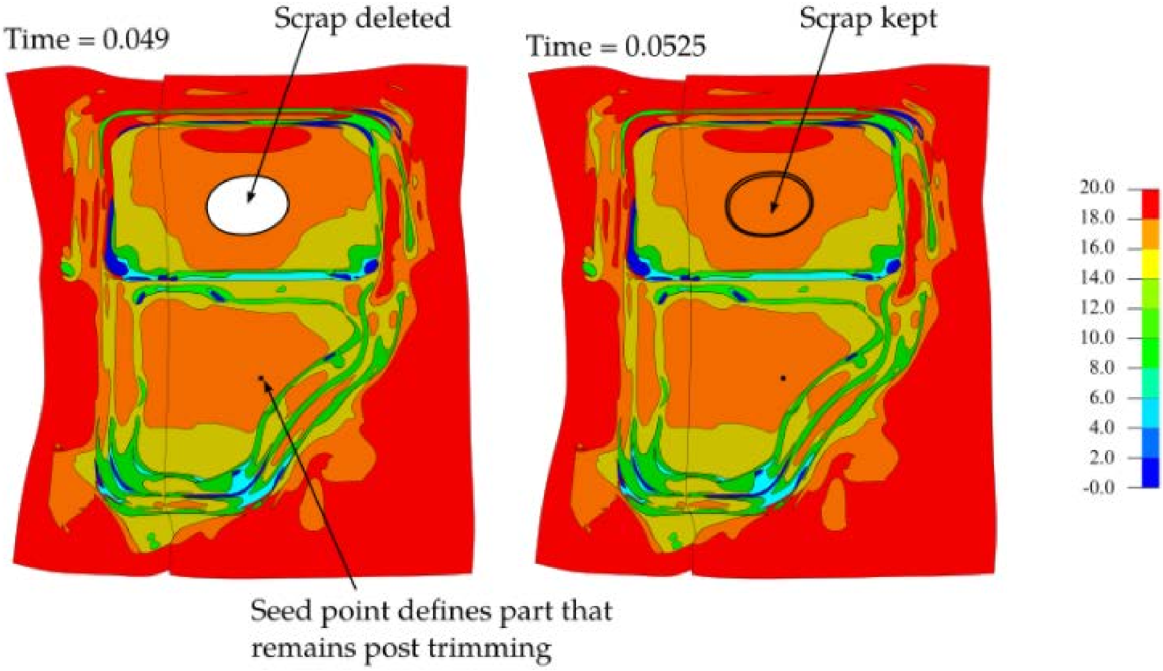


Figure 19 Lancing with trimming.

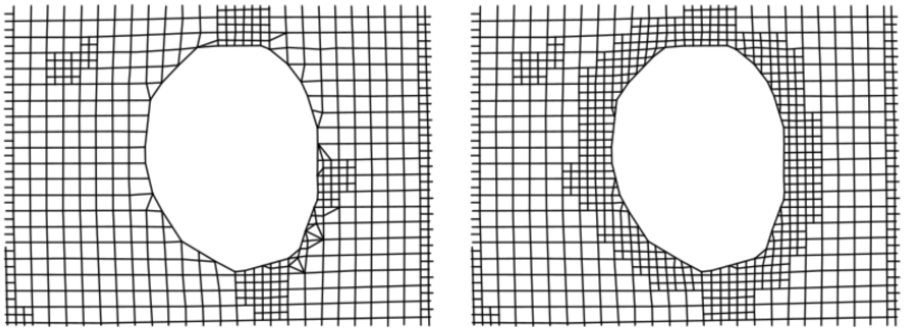
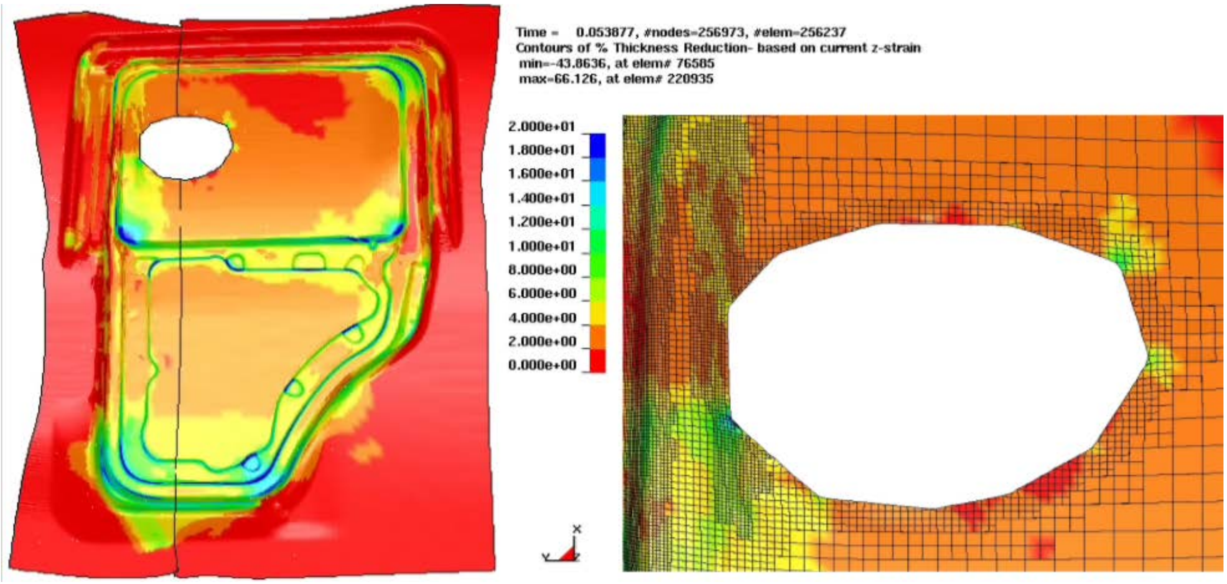


Figure 20 Automatic mesh refinement along lancing route.



```
Time =    0.053877, #nodes=256973, #elem=256237
Contours of % Thickness Reduction- based on current z-strain
 min=-43.8636, at elem# 76585
 max=66.126, at elem# 220935
```

Figure 21 Lancing across laser welded line and automatic mesh refinement along lancing route.