CONTROL_FORMING_ONESTEP_*OPTION

Purpose: This keyword activates a one-step solution, also called 'inverse method', which can be applied in initial blank size/material cost estimation, and initialization of forming effects such as plastic strains and blank thickness for crash simulation. The input is a formed or trimmed part (with holes filled) and the output is a corresponding flat blank in an un-deformed state, along with all the stress/strain information on the formed or trimmed part. This feature utilizes some of the existing implicit static features to iterate and to arrive at a converged solution. Other keywords related to blank size development are *CONTROL_FORMING_UNFLANGING, and *INTERFACE_BLANKSIZE_DEVELOPMENT.

Available options include:

<BLANK>

AUTO_CONSTRAINT

DRAWBEAD

FRICTION

Option **AUTO_CONSTRAINT** allows nodal restraints to be applied automatically in implicit calculation to prevent rigid body motion. Option **DRAWBEAD** is used for application of extra draw bead forces, in addition to the AUTOBD below. Option **FRICTION** applies friction along the periphery of the part, based on the binder tonnage input by the user.

Card 1 for no option, <BLANK>.

Card 1	1	2	3	4	5	6	7	8
Variable	OPTION	UNUSED	AUTOBD	TSCLMIN	EPSMAX			
Type	I		F	F	F			
Default	none		0.3	none	none			

Card 1 for option AUTO_CONSTRAINT.

Card 1	1	2	3	4	5	6	7	8
Variable	ICON							
Type	I							
Default	none							

Card 1 for option DRAWBEAD.

Card 1	1	2	3	4	5	6	7	8
Variable	NDSET	LCID	TH	PERCNT				
Type	I	I	F	F				
Default	none	none	0.0	0.0				

Card 1 for option FRICTION.

Card 1	1	2	3	4	5	6	7	8
Variable	NDSET	BDTON	FRICT					
Type	I	F	F					
Default	none	0.0	0.12					

Card 2 for no option <BLANK>.

Card 2	1	2	3	4	5	6	7	8
Variable	flatname							
Type	A							
Default	none							

VARIABLE	DESCRIPTION
OPTION	Options to invoke one-step solution methods which account for finished part undercut: EQ.6: One-step solution with unfolded blank (flat) provided by LS-PrePost (see remarks). Card #2 is required. EQ.7: One-step solution with blank unfolded in LS-DYNA. Card #2 is a blank line.
UNUSED	Unused – leave blank.
AUTOBD	Apply a fraction of a fully locked bead force along the entire periphery of the blank. The fully locked bead force is automatically calculated based on a material hardening curve input. LT.0.0: Turns off the “auto-bead” feature. EQ.0.0: Automatically applies 30% of fully locked force. GT.0.0: Fraction input will be used to scale the fully locked force.
TSCLMIN	If not zero, it defines a thickness scale factor limiting the minimum thickness reduction. For example, a value of 0.8 will be used here if the minimum thickness allowed is 0.6mm for a blank with initial thickness of 0.75mm. All computed thicknesses that are less than 0.6mm in the sheet blank will be set to 0.6mm. The scale factor is useful for tailored welded blanks application.
EPSMAX	If not zero, it defines the maximum effective plastic strain allowed. All computed effective plastic strains that are greater than this value in the blank will be set to this value.
ICON	Automatic nodal constraining option to eliminate the rigid body motion: EQ.1: Apply.

VARIABLE	DESCRIPTION
NDSET	Node set ID along the periphery of the part, as defined by keyword *SET_NODE_LIST.
LCID	Load curve ID that defines the material hardening curve.
TH	Thickness of the unformed sheet blank.
PERCNT	Draw bead lock force fraction of the fully locked bead force.
BDTON	Binder tonnage used to calculate friction force.
FRICT	Coefficient of friction.
FLATNAME	File name of the initial unfolded blank by LS-PrePost (see remarks). This is needed only for the OPTION=6. Leave a blank line for OPTION=7.

About One-Step forming solution:

One-step solution employs the deformation theory as opposed to the incremental theory. In the deformation theory, the stress is related to the total strain in contrast to the small, incremental strain in the incremental theory. It is therefore strain path-independent, forming history independent, and stamping process independent. The final product geometry is used to calculate backwards for the initial un-deformed, flat state. The one-step solution results can get close to the incremental results only in forming cases where the strain path is linear and deformation is either monotonically increasing or decreasing, however, in most forming cases the results cannot match those from incremental forming.

There are many advantages of using the one-step forming solution. For example, binder and addendum geometry are not required, saving a lot of effort or knowledge needed in creating these geometries; stamping die processes (including part tipping) are not of concern; there is no need for contact treatment since there are no tools and dies involved. Currently, the one-step solution is mostly used in advance formability studies where approximate but fast estimate of forming conditions are needed to compare different design alternatives, in blank size cost estimate, and as a first guess in an accurate blank size development process (see *CONTROL_FORMING_UNFLANGING, and *INTERFACE_BLANKSIZE_DEVELOPMENT). It is also widely used to initialize forming stress and strains in crash and occupant safety analysis.

Input details:

In addition to the usual input required for part, material and physical properties, product part mesh in keyword format is needed. The mesh used for one-step simulation should not be like those used for tooling mesh. For example, along the part bend radius, there is no need to build six elements along the arc length like one would do for the punch/die radius; rather, two elements may be enough there. Essentially, uniformly distributed, quadrilateral shell elements of square are the most ideal for the input mesh. In *LS-PrePost4.0*, this kind of mesh can be created using *Mesh/AutoM/Size*. Since this is an implicit static solution, the mesh size does not affect the computation time; but rather, the number of elements does. Furthermore, it is important to note that if one wants to obtain forming results that are closer to the incremental forming results, the part in the one-step input should be similar in size to the final formed blank shape in the incremental forming (before trimming).

Any trimmed-out holes can be filled (but not necessary). The filling can be done semi-automatically in *LS-PrePost 4.0*, by selecting *Mesh/EleGen/Shell/Shell by Fill_Holes/Auto Fill*. The filled area of the part can be saved in a different part, as multiple parts (PID) are allowed. The forming results may be different between part with and without holes.

For OPTION=6, the unfolded blank can be obtained from *LS-PrePost* via menu option *EleTol/Morph*, select *Type* as *Mesh_Unfolding*, and then click on *Unfold*. The unfolded mesh can be saved as a keyword file and used as input for the variable FLATNAME in Card 2. In effect, OPTION 7 also uses this unfolded blank within LS-DYNA for the solution.

Both shell element types 2 and 16 are supported. Since implicit solver is used here, type 16 shell is strongly recommended, which helps the convergence rate and reduces CPU times significantly. Results are output on all integration points based on the variables ELFORM and NIP specified in *SECTION_SHELL.

Currently, *MAT_024, and *MAT_037 are supported in the solution. Material hardening curve must be input through the variable LCSS in *MAT_024 and HLCID in *MAT_037. The variable ETAN for bi-linear hardening in both material models will be supported in a future release. Additionally, in *MAT_024, strain rate is not accounted for even if the variables C, P are present. Tables in *MAT_024 are also supported.

The primary 'boundary/loading condition' for the one-step solution is the draw bead forces, specified using the option **DRAWBEAD**. Draw bead forces are applied on a user defined node set along the periphery of the part. A fraction of the full lock force, determined by the tensile strength and sheet thickness, can be applied. The larger the fraction the less the metal flow into the die, resulting in more stretch and thinning. The alternative way of applying a boundary condition is to use 'Auto Beads' (*AUTOBD*), which allows users to specify a fraction of the fully locked bead force, and the code will apply the force to all the nodes along the part boundary automatically. In most cases, default lockage of 30% from 'Auto Beads' achieves sufficiently good results for crash/occupant safety engineers. The last important but often overlooked 'boundary condition' is the part shape. For example, larger flange area of an oil pan would contribute to greater thinning in the part wall, and vice versa. To obtain results that are closer to those from the incremental forming, additional materials may need to be added to the final part geometry in cases where the sheet blank is not 'fully developed', meaning no trimming is required to finish the part.

Optionally, friction can be accounted for, by using the option **FRICTION**. The frictional force is based on an expected binder tonnage applied, and is a percentage of the input force. It is noted that the binder tonnage input here is for friction force calculation only, and is not actually applied on the binder as a part of the boundary conditions.

Nodal constraints used in implicit calculation to prevent rigid body motion are automatically applied using the option **AUTO_CONSTRAINT** by setting the variable ICON to 1.

All other implicit cards, such as *CONTROL_IMPLICIT_GENERAL, _SOLUTION, _SOLVER, _ AUTO, _TERMINATION, etc., are used to set the convergence tolerance, termination criterion, etc. It was determined, that the two important variables controlling the solution convergence, *DELTAU* in *CONTROL_IMPLICIT_TERMINATION, and *DCTOL* in *CONTROL_IMPLICIT_ SOULTION, can be set to 0.001 and 0.01, respectively, to obtain the most efficient and best results. In addition, four total implicit steps are usually sufficient by setting *DT0* in *CONTROL_IMPLICIT_GENERAL and *ENDTIM* in *CONTROL_TERMINATION accordingly.

Card #2 for no option <**BLANK**> is a blank card, but must be present.

Output:

Results are stored in an ASCII file named ‘onestepresult’. It is a dynain file with forming thickness, stress and strain tensors on the part, which can be plotted with *LS-PrePost*. One quick and useful forming results display is the ‘formability contour’ map, where simple colors of ‘cracks’, ‘severe thinning’, ‘good’, and ‘wrinkles’, etc. are presented. This formability map is accessible through *Post/FLD/Formability*. Additionally, the final estimated blank size (in its initial, flat state) can be viewed and output through *D3PLOT* files, along with intermediate shapes stored for each implicit step. The final blank mesh in its flat state can be output using *LS-PrePost* via menu option *Post/Output/Keyword*, check the box to include *Element* and *Nodal Coordinates*, move the animation bar to the last state, and click on *Curr* and *Write* for the last state. Alternatively, blank outlines can be created with menu option *Curve/Spline/From Mesh (Method)*, check *Piecewise*/*byPart*, select the blank, click on *Apply*, and finally save the curves as IGES format from the *File* menu at the upper left corner.

Application example:

The following example provides a partial input file with typical control cards. It will iterate for four steps, with ‘Auto Beads’ of 30% lock force applied around the part boundary, and with automatic nodal constraints.

```
*CONTROL_TERMINATION
$   ENDTIM
       1.0
*CONTROL_IMPLICIT_GENERAL
$   IMFLAG       DT0
          1     0.25
*CONTROL_FORMING_ONESTEP
$   OPTION              AUTODB
         7

*CONTROL_FORMING_ONESTEP_AUTO_CONSTRAINT
$     ICON
         1
*CONTROL_IMPLICIT_TERMINATION
$   DELTAU
     0.001
*CONTROL_IMPLICIT_SOLUTION
$  NSOLVR    ILIMIT    MAXREF      DCTOL      ECTOL
         2        11      1200       0.01       1.00
*CONTROL_IMPLICIT_SOLVER
$   LSOLVR
```

```
         4
*CONTROL_IMPLICIT_AUTO
$    IAUTO    ITEOPT    ITEWIN     DTMIN     DTMAX
         0         0         0       0.0       0.0
```

Additional cards below specify extra bead forces of 45% and 30% applied to node sets 22 and 23 along the part periphery, respectively. Also, the resulting friction forces with friction coefficient of 0.1 and binder tonnage of 10000.0 N used for friction force are applied on the same node sets.

```
*CONTROL_FORMING_ONESTEP_DRAWBEAD
$   NDSET      LCID        TH    PERCNT
       22       200       1.6      0.45
*CONTROL_FORMING_ONESTEP_DRAWBEAD
       23       200       1.6      0.30
*CONTROL_FORMING_ONESTEP_FRICTION
$   NDSET     BDTON    FRICT
       22   10000.0      0.1
*CONTROL_FORMING_ONESTEP_FRICTION
$   NDSET     BDTON    FRICT
       23   10000.0      0.1
```

The one-step forming results of a NCAC Taurus model firewall is shown in Figure **0-1**. With average element size of 8 mm across the blank, the trimmed part (with holes filled) consists of 15490 elements. A BH210 material properties with *MAT_024 was used. On a 1 CPU Xeon E5520 Linux machine, it took 4 minutes to complete the run with 4 steps total. Thickness, plastic strain, and blank size prediction were reasonable, as shown in Figures **0-2**, **0-3** and **0-4**.

The variables TSCLMIN and EPSMAX provide limits so the minimum thinning and maximum plastic strains can be specified. In a product development process, product designs in the early stage are typically not feasible to form, resulting in very large strains and very thin thickness (not useable for crash simulation); however, the designs are certain to evolve later so eventually the final design can be formed. Limiting these resulting values is a convenient way to perform the crash simulation with realistic forming effects, but without resorting to physical modifications of product shape, which may require substantial amount of change time. In the keyword below for the firewall, TSCLMIN and EPSMAX are defined as 0.8 and 0.3, respectively, as follows:

```
*CONTROL_FORMING_ONESTEP
$   OPTION              AUTODB   TSCLMIN    EPSMAX
         7                           0.8       0.3
```

The thickness and effective plastic strain plots are shown in Figures **0-5** and **0-6**, respectively. The minimum value for thickness contour plot and maximum value for plastic strain contour plot at the upper left corner correspond to what are specified in TSCLMIN and EPSMAX, respectively.

In Figure **0-7**, the thickness contour plot of the one-step calculation results of the firewall with holes is shown. It is noted that the minimum thickness is slightly higher than the one in the case above with holes filled, a result of some amount of material feeding from the inner holes. In reality, thicknesses with holes filled are likely more accurate, since forming is done with holes mostly filled, followed by a trimming process (which does not change the strain states), removing material occupying the holes. On the other hand, it is important to realize that not all the holes are filled in a draw panel. Some holes are cut inside the part (but not all the way to the trim line) in draw process in order to feed the material in areas that are difficult to form to avoid split.

Revision information:

This feature is available starting in Revision 67778 SMP and double precision only. Outputs of stress and strain tensors start in Revision 73442 and 75156, respectively. Both variables THINPCT and EPXMAX are available starting in Revision 75854. Holes are allowed starting in Revision 76709.

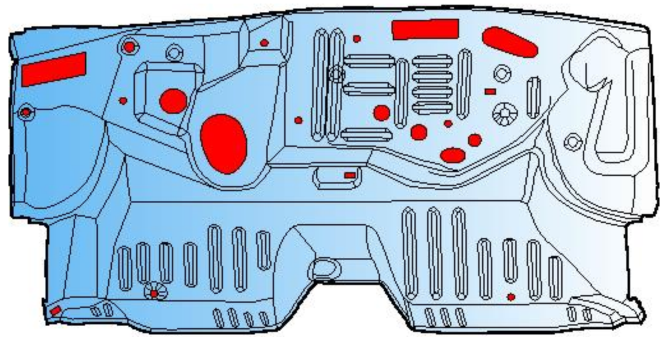

Figure 0-1. A trimmed dash panel (firewall) with holes auto-filled using LS-PrePost 4.0 (original model courtesy of NCAC Taurus crash model).

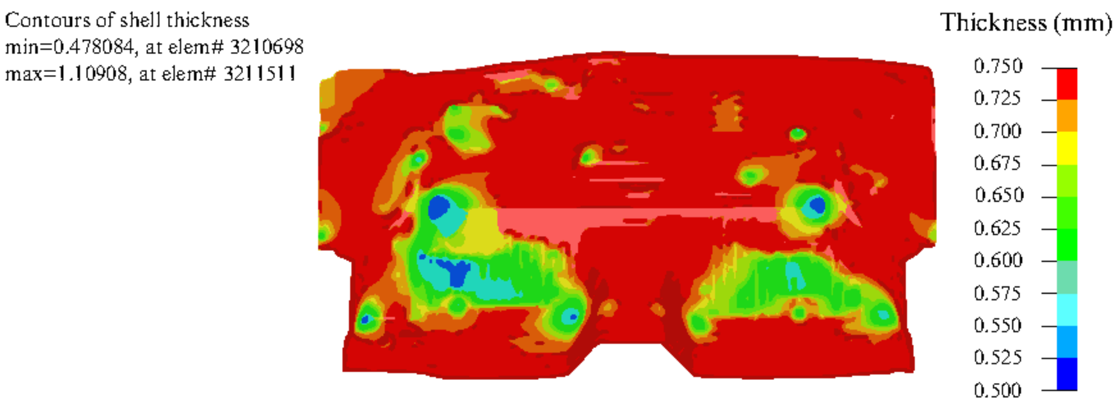

Figure 0-2. Shell thickness prediction (t_0=0.75mm).

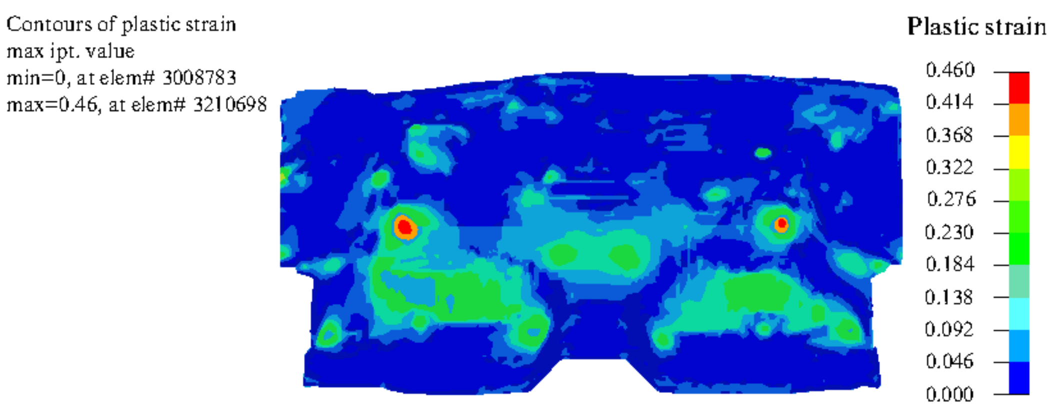

Figure 0-3. Effective plastic strain Prediction.

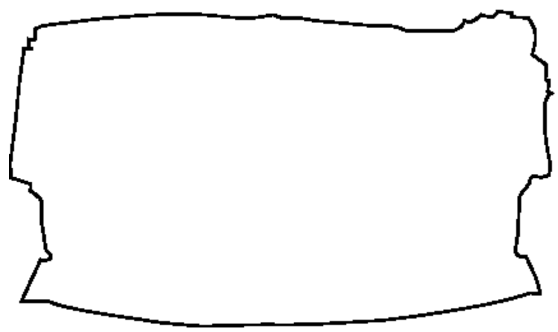

Figure 0-4. Initial blank size prediction (flat, not to scale).

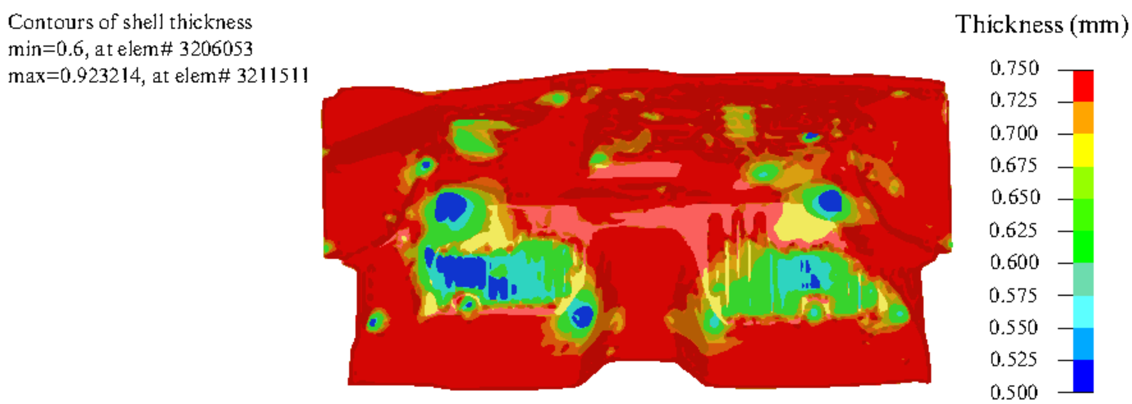

Figure 0-5. Blank thickness prediction with TSCLMIN=0.8.

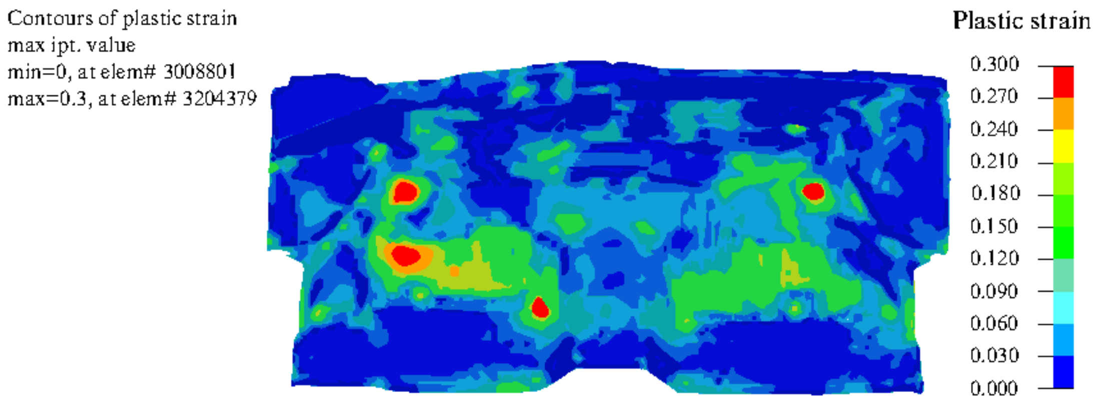

Figure 0-6. Effective plastic strain with EPSMAX=0.3.

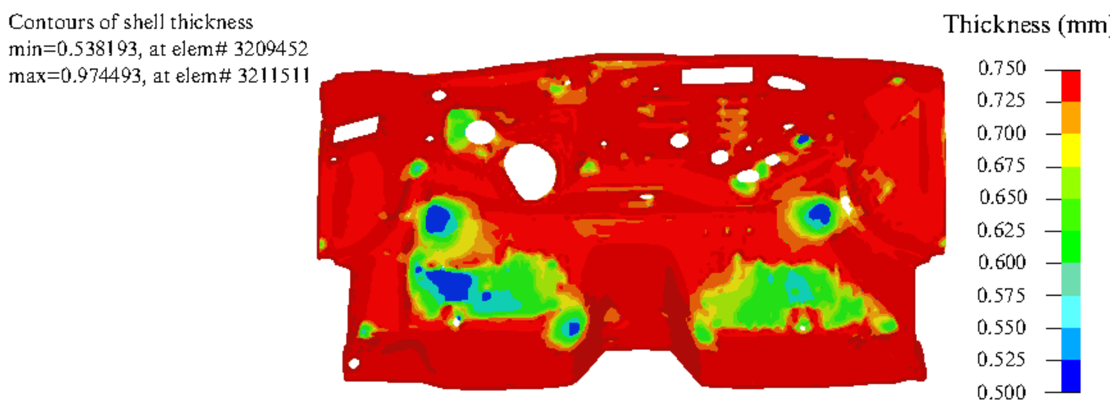

Figure 0-7. Blank thickness with trimmed holes (t_0=0.75mm).