Tips about Springback and compensation with ETA/Dynaform

DYNAFORM Team
June, 2015
Simulation Basics
- Mesh
- Implicit and Explicit
- Time step
- Contact

- Material Model
  - Yielding Surfaces
  - Isotropic and Anisotropic
  - Kinematic Hardening
What is Mesh?

Mathematical representation of surfaces

Surfaces

Mesh
Explicit and Implicit

- **Explicit**
  - Easy formulation
  - Time step stability limited

- **Implicit**
  - Matrix
  - Unconditional stable
During the solution, program loops through elements and determines a new time step size by taking the minimum value over all elements.

\[
\Delta t_{\text{new}} = \alpha \ t_{\text{min}} \ \{ \Delta t_1, \Delta t_2, \ldots, \Delta t_N \}
\]

where \( N \) is the total number of elements.

For stability reasons the scale factor \( \alpha \) is typically set to a value of 0.9 or smaller. To reduce solution time, use of the largest possible stable time step size is desired.
Critical (or minimum) time step size:

$$\Delta t_{\text{min}} = \frac{\ell_{\text{min}}}{C}$$

where $C$ is the sound wave propagation speed in 3D-continuum:

$$C = \sqrt{\frac{E(1 - \nu)}{(1 + \nu)(1 - 2\nu)\rho}}$$

- $E$ = Yong’s modulus
- $\nu$ = Poisson’s ratio
- $\rho$ = specific mass density
- Accurately catch critical tool geometry, such as radii or curved regions, using small elements.
- Efficiently mesh the tool surface, such as flat regions, using larger elements.
- Mesh quality requirements must be met. Collapsed or overlapping elements should be corrected.
- Prefer square elements in uniform pattern
- If necessary, triangular elements are placed around the outer edge area
- No free boundary or gap inside the blank
- Mesh quality criteria must be met
  - No collapsed elements
  - No overlapping elements
Theory of Penalty Contact

- Placing normal interface springs between all penetrating nodes and the contact surface.

No penetration
No Contact force

With penetration
with contact force

Contact force is proportional to the penetration
Penetration due to improper contact setup
Penalty Method

Contact surface conditions imposed by penalty.

Restoration stiffness for solids is prescribed as follows:

\[ k = \alpha aKA \frac{A}{V} \]

Where \( a \) is the penalty scale factor,

\( K \) is the material’s Bulk Modulus,

\( A \) is the segment Area,

\( V \) is the element volume.

With the penalty method, nodes actually penetrate through the surface, impenetrability violated.

Penalty method straightforward and simple.
Large penetration cause bad result

- Details of the tools can not be obtained
  Correlation with experimental measurement will not be good
- Over-predict sheet metal formability
  Large safety margin should be used
- Bad stress distribution
  Cause bad springback prediction
Tooling Speed in Stamping Simulation

To obtain the simulation results in a reasonable time requires to apply higher tooling speed in simulation analysis, various study shows,

for a good strain prediction:

- binder speed about 1000mm/s to 2000 mm/s
- Punch/Die speed about 5000mm/s to 10,000mm/s

For better stress prediction, tooling speed needs to be reduce further.
Commonly used elements in Sheet Metal Forming Simulation

Under Integrated Element

Fully Integrated Element
Shell integration through the thickness

Importance of integration points in bending problems

Complicated stress profiles mean high number of IP is required to simulate springback
To obtain the simulation results in a reasonable time requires to apply higher tooling speed in simulation analysis, various study shows,

for a good strain prediction:

- binder speed about 1000mm/s to 2000 mm/s
- Punch/Die speed about 5000mm/s to 10,000mm/s

For better stress prediction, tooling speed needs to be reduce further.
Tips about Springback and compensation with ETA/Dynaform

For a accurate springback prediction in eta/dynaform:

- Provide the correct material parameters

- Perform the draw analysis with care:
  - Enough elements to catch the stress in the curvature area
  - Avoid stress oscillation, limit the mass scaling

- When calculating the springback, constrain is essential
Tips about Springback and compensation with ETA/Dynaform

For an accurate springback compensation with ETA/Dynaform:

- Finer mesh for the tool

- Build a compensation scale factor library
Simulation Limitations

- Tonnage prediction is very sensitive to setups and draw conditions
- Dynamic effect with explicit calculations
- Strain rate issue
- Mass scaling
Simulation Limitations

- Simulation has strength and limitations
- Effectiveness of the simulation relay on the understanding of the technology properly
Simulation has strength and limitations

Effectiveness of the simulation relay on the understanding of the technology properly
Current and Future Development

- Hot Forming and Die Cooling
- Draw Analysis with Implicit Approach
- Optimizations
- Forming Analysis with Flexible Tools
Execution time primarily depends on:
- material properties
- mesh size
- number of elements
- contacts
- speed of computer

CPU estimation
- Time step $\Delta t = \text{minimum } \Delta x/c$
- number of cycles = termination time / $\Delta t$
- CPU time = ($\#$ cycles)(# elements)(time per zone cycle)
- correction is needed for time step reduction
- correction is needed for number and size of contacts
Yield Criteria and Plasticity Theory

Plasticity theory addresses three components:

- **Yield criteria** (or yield function) that defines the initial inelastic response of materials
- A flow rule that relates the plastic strain increments to the stress increments after initiation of the inelastic response
- A hardening role that predicts changes in the yield surface due to plastic strain.
With kinematic hardening, the yield surface may shift as a function of plastic strain.
Typical Draw Beads – Parametrically Defined

Round Bead

Square Bead

Male Bead & Female Bead
Optimization Based on (6) FLD Zones

Forming Limit Diagram (FLD)

Define the Constrain/Objective function as the ratio of elements in a particular Zone # / total elements
Case 1, Sink, Deep Draw Optimization Setup

Bead 1, bead 2, bead 3 and bead 4 should be symmetrical, and the bead 2, bead 3 and bead 4 are defined as dependant of bead 1. The baseline Values are changed incrementally in a range from 5% to 90%. The base line of each variable is 25%. The incremental drawbead rate is 5%.

Objective and constraints are all “Crack” and “Safe”. The allowable thinning 20%. The number of Evaluations is 100. Due to the dependency, it reduced to 18.
Case 1: Optimization Result

SHERPA returns with (5) best/optimized Cycles – 1 Line Bead Rates due to symmetry.

Higher Positive SAFE values indicate better optimized results. Negative SAFE values indicate a bad cycle.

<table>
<thead>
<tr>
<th>Bead ID</th>
<th>Base Line</th>
<th>Optimized</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>25</td>
<td>70</td>
</tr>
</tbody>
</table>
Case 1: Optimization Performance – Before / After Comparison

Full model 25% Drawbead Force Lock Rate
Before Optimization, FLD Plot

Full Model 70% Drawbead Force Lock Rate
After Optimization, FLD Plot
Implement Optimization Results

<table>
<thead>
<tr>
<th>Bead ID</th>
<th>Base Line</th>
<th>Optimized</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>25</td>
<td>70</td>
</tr>
</tbody>
</table>
Case 2: Deep Draw setup, Full Blank, Full Tools, Adaptive Level 4

Draw Setup Parameters:

- Material: DC04
- Thickness: 1.2mm
- Friction: 0.125 0.125 0.125 0.125
- Blank size: 16
- Adapt mesh levels: 4
Case 2: Optimization Setup

7 Line Beads were setup for optimization, and the other beads are defined as dependant of them. The baseline values are changed incrementally in a range from 5% to 90%. The base line of each variable is 25%. The incremental rate is 5%.

Objective and constraints are all “Crack” and “Safe”. The allowable thinning 20%. The number of iterations is 130. The C-Crack is set as 0.01.
SHERPA returns with (5) best/optimized Cycles – 7 Line Bead Rates. The first one (Cycle 119) is as shown in the table below.

<table>
<thead>
<tr>
<th>Bead ID</th>
<th>Base Line</th>
<th>Optimized</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>25</td>
<td>35</td>
</tr>
<tr>
<td>2</td>
<td>25</td>
<td>5</td>
</tr>
<tr>
<td>3</td>
<td>25</td>
<td>30</td>
</tr>
<tr>
<td>6</td>
<td>25</td>
<td>20</td>
</tr>
<tr>
<td>8</td>
<td>25</td>
<td>55</td>
</tr>
<tr>
<td>10</td>
<td>25</td>
<td>15</td>
</tr>
<tr>
<td>11</td>
<td>25</td>
<td>55</td>
</tr>
</tbody>
</table>
Case 2: Optimization Performance – Before / After Comparison

Full model 25% Drawbead Force Lock Rate Before Optimization, FLD Plot

Full Model, Drawbead Force Lock Rate After Optimization, FLD Plot
# Case 2: Oil Pan, Computing Resources

<table>
<thead>
<tr>
<th>CASE 1</th>
<th>Objective and Constraint</th>
<th>Iteration</th>
<th>CPU</th>
<th>Calculate Time (Hours)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Oil</td>
<td>Crack, Safe</td>
<td>130</td>
<td>Intel(R) Core(TM) i5-3570 CPU @3.40GHz 3.40 GHz 1 CPU/4 cores/4 threads</td>
<td>42</td>
</tr>
</tbody>
</table>
Case 3: Cross Member (NUMISHEET05), Draw Setup

Draw Setup Parameters:

- Material: DP600
- Thickness: 1.6mm
- Friction: 0.125 0.125 0.125 0.12
- Blank size: 16
- Adapt mesh levels: 4

Optimize (20) Line Beads
Case 3: Optimization Setup

10 Line Beads were setup for optimization, and the other beads are defined as dependant of them. The baseline values are changed incrementally in a range from 5% to 90%. The base line of each variable is 60%. The incremental rate is 5%.

Objective and constraints are all “Crack” and “Safe”. The allowable thinning 20%. The number of iterations is 150. “C-Crack” is set as 0.01.
SHERPA returns with the (5) best/optimized parameters – 10 Line Bead Rates. The Cycle 113 is shown in below table.

<table>
<thead>
<tr>
<th>Bead ID</th>
<th>Base Line</th>
<th>Optimized</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>60</td>
<td>90</td>
</tr>
<tr>
<td>2</td>
<td>60</td>
<td>75</td>
</tr>
<tr>
<td>3</td>
<td>60</td>
<td>45</td>
</tr>
<tr>
<td>4</td>
<td>60</td>
<td>5</td>
</tr>
<tr>
<td>5</td>
<td>60</td>
<td>75</td>
</tr>
<tr>
<td>6</td>
<td>60</td>
<td>75</td>
</tr>
<tr>
<td>7</td>
<td>60</td>
<td>90</td>
</tr>
<tr>
<td>8</td>
<td>60</td>
<td>55</td>
</tr>
<tr>
<td>9</td>
<td>60</td>
<td>90</td>
</tr>
<tr>
<td>10</td>
<td>60</td>
<td>85</td>
</tr>
</tbody>
</table>
Case 3: Optimization Performance – Before / After Comparison

Full blank 60% Drawbead Force Lock Rate
Before Optimization, FLD Plot

Full blank, Drawbead Force Lock Rate
After Optimization, FLD Plot
### Case 3: Cross member, Computing Resources

<table>
<thead>
<tr>
<th>CASE 1</th>
<th>Objective and Constraint</th>
<th>Iteration</th>
<th>CPU</th>
<th>Calculate Time (Hours)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cross member</td>
<td>Crack, Safe</td>
<td>150</td>
<td>Intel(R) Core(TM) i5-3570 CPU @3.40GHz 3.40 GHz 1 CPU/4 cores/4 threads</td>
<td>40</td>
</tr>
</tbody>
</table>
Case 4 DeckLid (NUMISHHET05), Stretch Forming

Draw Setup Parameters:

- **Material:** BH180
- **Thickness:** 0.8mm
- **Binder Force:** 133400.00N
- **Friction:** 0.125 0.125 0.125 0.125
- **Adapt mesh level:** 4
- **Initial element size:** 16mm 1mm
- **Lower Binder Travel:** 65mm
11 Line Beads were setup for optimization, and the other beads are defined as dependant of them. Baseline values are changed incrementally in a range from 5% to 90%. The base line of each variable is set at 60%. The incremental rate is 5%.

Objective and constraints are all “Crack” and “Safe”. The allowable thinning is 20%. The number of iterations is 162. “C-Crack” is set as 0.01.
SHERPA returns with the (5) best/optimized parameters – 11 Line Beads Rates, Cycle 149 is presented

<table>
<thead>
<tr>
<th>Bead ID</th>
<th>Base Line Rate</th>
<th>Optimized</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>60</td>
<td>50</td>
</tr>
<tr>
<td>3</td>
<td>60</td>
<td>50</td>
</tr>
<tr>
<td>4</td>
<td>60</td>
<td>70</td>
</tr>
<tr>
<td>7</td>
<td>60</td>
<td>85</td>
</tr>
<tr>
<td>8</td>
<td>60</td>
<td>50</td>
</tr>
<tr>
<td>9</td>
<td>60</td>
<td>30</td>
</tr>
<tr>
<td>10</td>
<td>60</td>
<td>90</td>
</tr>
<tr>
<td>11</td>
<td>60</td>
<td>50</td>
</tr>
<tr>
<td>12</td>
<td>60</td>
<td>45</td>
</tr>
<tr>
<td>13</td>
<td>60</td>
<td>35</td>
</tr>
<tr>
<td>14</td>
<td>60</td>
<td>60</td>
</tr>
</tbody>
</table>
Case 4: DeckLid, FLD Plot Before and After Optimization

After the Design Optimization, based on FLD presentation, the Crack/Split area is significantly reduced.
## Case 4: DeckLid, Computing Resource

<table>
<thead>
<tr>
<th>CASE</th>
<th>Objective and constraint</th>
<th>Iteration</th>
<th>CPU</th>
<th>Calculate Time (Hours)</th>
</tr>
</thead>
<tbody>
<tr>
<td>DeckLid</td>
<td>Crack, Safe</td>
<td>162</td>
<td>Intel(R) Core(TM) i7-2600 CPU @3.40GHz 3.70 GHz 1 CPU/4 cores/8 threads</td>
<td>74</td>
</tr>
</tbody>
</table>
Case 5: Fender (NUMISHEET02), Stretch Forming

Draw Setup Parameters:

- Material: HSS_FENE
- Thickness: 0.7mm
- Binder Force: 1200000N
- Lower Binder Travel: 130mm
- Friction: 0.125 0.125 0.125 0.125
- Adapt mesh level: 4
- Initial element size: 16mm
  1mm
Case 5: Fender, Optimization Setup

5 Line Beads were setup for optimization, and the other beads are defined as dependant of them. The baseline values are changed incrementally in a range from 5% to 90%. The base line of each variable is set at 60%. The incremental rate is 5%.

Objective and constraints are all “Crack” and “Safe”. The allowable thinning is 20%. The number of iterations is 130. “C-Crack” is set as 0.01.
Case 5: Fender, Optimization Results

SHERPA returns the (5) best/optimized parameters – 5 Line Beads Rates. The Cycle 95 is as shown in below table.

<table>
<thead>
<tr>
<th>Bead ID</th>
<th>Base Line Rate</th>
<th>Optimized</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>60</td>
<td>15</td>
</tr>
<tr>
<td>2</td>
<td>60</td>
<td>20</td>
</tr>
<tr>
<td>4</td>
<td>60</td>
<td>75</td>
</tr>
<tr>
<td>5</td>
<td>60</td>
<td>30</td>
</tr>
<tr>
<td>15</td>
<td>60</td>
<td>25</td>
</tr>
</tbody>
</table>
Case 5: Fender, FLD Plot, Before and After Optimization

After the Design Optimization, based on FLD presentation, the Crack / Split area is significantly reduced.
## Case 5: Fender, Computing Resource

<table>
<thead>
<tr>
<th>CASE</th>
<th>Objective and constraint</th>
<th>Iteration</th>
<th>CPU</th>
<th>Calculate Time (Hours)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fender</td>
<td>Crack, Safe</td>
<td>130</td>
<td>Intel(R) Core(TM) i5-3570 CPU @3.40GHz 3.40 GHz 1 CPU/4 cores/4 threads</td>
<td>37.5</td>
</tr>
</tbody>
</table>
Conclusions and Discussions

- Sherpa and INCSolver based Formability Optimization is practical for the evaluation of drawbead layouts, and determining the rates in terms of the drawbead Lock Rates (100%)

- “Black Box” setup enables average tooling users to quickly adopt the optimization approach for forming applications

- SMP & Multiple CPUs / Cores computing is recommended for demanding optimization calculations

- Accept the multiple solution scheme of forming a part

- Determines the best solution per tooling design requirements

- Follow the “Best Practices” to use INCSolver and OP Module
  - See appendix for the said “Best Practices”

- Validation cases are provided for review
  - Training & Tutorial cases are available
Outlines of Local definition:

- Add local definition and allow the user to define local region for optimization.
- Add Restart button, and allow the user to recalculate the job.
Description of New Feature

- Make the Response definition as Group, named “Global” as default. It means the definition for all elements. Once adding region definition, the newly group will be put behind the “Global”.

- Active: Allow the user to define the region groups for optimization, not include Global group. This option is used to control if the Global group is active. If the Global group is not active, the definition information cannot be viewed, and the setup of this group cannot be exported.
Description of New Feature

“+”: Click on this button, the program will generate new group named “Region 1”, as shown figure.

“-“: Click on this button, the program will remove the generated region group.
Definition of New Feature

- **Define**: Allow the user define the curve to select wanted region. The programs will popup the **Curve Editor** to define the Curve or select existed line. The user can define the curve from the Tools elements, and the program will project the curve to Blank elements in the Post. The Post will extract the forming result of the selected elements in the defined Curve.

After defined the Curve, the curve will be highlight in the display window. One region group is allowed to define one closed curve.

- **Delete**: Allow the user remove the defined curve. After **Delete** the curve, the Define button will be in red.
Description of New Feature

The user can define several region groups, named as “Region 1”、 ”Region 2”、 ”Region 3”, etc, as shown figure.
**Description of New Feature**

**RESTART**
After the job is submitted for running, this button will be activated. If the job aborts suddenly, the user can re-calculate the current job.

**Note:**
1. If the user has changed the variables, the program will automatically pop up the eta/Dynaform Question illustrated in shown figure when clicking this button.

**YES**
The program will delete the calculated results and rerun the job according to the modified variables, and it is very risky.

**NO**
The program will return to the Optimization GUI.
**Description of New Feature**

**RESTART**

After the job is submitted for running, this button will be activated. If the job aborts suddenly, the user can re-calculate the current job.

**Note:**

1. When the variables keep unchanged and the user modifies the Response, the program will automatically pop up the eta/Dynaform Question illustrated as shown figure when clicking this button.

**YES**

The program will save the calculated results and rerun the job according to the modified responses, and it is very risky.

**NO**

The program will return to the Optimization GUI.
Description of New Feature

**RESTART**
After the job is submitted for running, this button will be activated. If the job aborts suddenly, the user can re-calculate the current job.

**Note:**
1. When the variables and Response keep unchanged and the user only aborts the calculation temporarily, the program will automatically pop up the eta/Dynaform Question illustrated as shown figure when clicking this button.

**YES**
The program will continue the calculation based on the calculated results, which saves certain computation time.

**NO**
The program will return to the Optimization GUI.
EFLD in eta/DYNAFORM 5.9.2

Aug 29, 2014
If “Show EFLD Area Only” is toggled on, it only plots FLD result in EFLD area and the other regular FLD area will be treated as no data and plotted in gray color.

The EFLD compensation will be used in EFLD area. For example in the marked area, crack is compensated.
If EFLD is toggled on and the “Show EFLD Area Only” is toggled off, it plots FLD result in EFLD area and the regular FLD area to make a complete FLD plot.
EFLD is set as a FLD value filter like “FLD Region”. The work procedure is set as the following.

1. Select entry line and exit line of the drawbead to define the EFLD region. This lines should be imported into eta-Post.

2. User set a reference frame (default as frame 1).
   Use entry line and exit line as drive lines to calculate skid mark between the reference frame and the current frame. The skid area of exit line defines the limit of EFLD applicable region.

3. For each element in the EFLD applicable region, calculate FLD compensated value base on decrease of major strain value. The crack can be reduced and the compensated result is closer to the actual phenomena.

4. Plot FLD result in the EFLD region.
1. Added a EFLD button in FLD dialog.
2. It will enter EFLD dialog when selecting the EFLD button.
3. The EFLD on option is used to set the EFLD filter on/off.
   If it is toggled off, the Show EFLD Area Only option will be deactivated. Otherwise, it will be activated.
4. The Shows EFLD Area Only option is used to set the display mode of EFLD area.
   If it is on, it only plots EFLD elements to FLD map, other elements are treated as no data.
   If it is off, it plots EFLD elements with other elements in regular FLD result.
5. The Reference Frame slider is used to set the reference frame.
6. The Select Entry Line button will invoke the selection of EFLD entry line.
7. The Select Exit Line button will invoke the selection of EFLD exit line.
8. The Apply button will apply the current EFLD setting and re-plot FLD result.
Method to Find EFLD Elements

After defining the exit line in EFLD dialog, the line will be used as the
drive line to find skid mark between the current frame and the
reference frame, and find the skid area illustrated in the left
picture (area between blue lines). The elements located in the
skid area are defined as EFLD elements.

If the entry line or exit line is enclosed, 2
enclosed skid lines will be defined. The area
between the 2 skid lines is the skid area.

If the entry line or exit line is not enclosed, 2
opened skid lines will be defined. Connect the
starting points and end points of line 1 and line 2
to define the skid area.
The main process of use EFLD

1. Plot FLD
2. Enter EFLD DIALOG
3. Toggle on “EFLD on”
4. Select Entry Line and Exit Line

If the user plots result on EFLD area only, toggle on “Show EFLD Area Only”.
If the user plots EFLD area with other regular FLD area, toggle off “Show EFLD Area Only”.

5. Click “Apply” to plot
Hot Forming Introduction
New features

- Thermal thick shells for the tools (TSHELL=1)
- Thermal gravity loading simulation
- Thermal press hardening simulation
- Default friction coefficient for hot forming (0.46)
- Initial temperatures for both blanks and tools
- Support of *DEFINE_TABLE_3D for *MAT_106/*MAT_244
- Thermal time step for hot forming
Basic unit system

All the unit will be converted to basic unit [ton, mm, s, N] in the input deck.

Unit in the GUI

Commonly used unit is provided in the GUI, user do not need to convert them to the basic unit system manually in the hot forming setup.
Full processes of hot forming in eta/Dynaform5.9.2
The stages of *gravity loading* and *press hardening* are optional.
Thermal thick shells for the tools

The heat transfer from blank to the tools is proportional to the temperature gradient of the contact partners.

The thermal thick shells give us higher accuracy for longer contact times. This is necessary for gravity phase and slow tool velocity.
Thermal thick shells for the tools

The artificial thickness can be calculated from the thermal effusivity therm. effusivity

\[ b = \sqrt{\rho \cdot c_p \cdot k} \]

with density \( \rho \), capacity \( c_p \) and conductivity \( k \)

**thick vs. thermal effusivity**

\[
y = 1.1138E-06x - 1.3236E-03
R^2 = 9.9665E-01
\]
Thermal gravity loading simulation

Since the blank material is very soft at high temperatures, the bending of the blank during the insertion might be significantly. This could affect the initial blank position before the press hardening process.

The material properties change significantly with the temperature. During the insertion of the hot blank into the press, some areas of the blank get in contact with the tools, some do not. In those areas where contact occurs, the temperature of the blank decreases significantly due to the cooling of the tools. Thus, the initial temperature distribution in the blank before the forming stage is NOT uniform.
Friction

Since there is no lube in hot forming, the friction is very high, the default friction coefficient is 0.46, user can change it if necessary.