Simulation results of a 70-degree SPIF formed cone

Carlos Felipe Guzmán∗
Department ArGENCo, Université de Liège, Belgium

June 3, 2013

Contents

1 Description 2
  1.1 Geometry and material 2
  1.2 Finite element model 2
  1.3 Constitutive law 2

2 Results 3
  2.1 Shape 4
  2.2 Thickness 6
  2.3 Forces 6

3 State variables 11
  3.1 Coarse mesh, Set2 12
  3.2 Fine mesh, Set2 14
  3.3 Analysis 15

References 18

∗cf.guzman@ulg.ac.be
1 Description

1.1 Geometry and material

The Fig. 1 shows a 70° wall angle cone formed by SPIF. The sheet material is an Aluminum AA3003-O of 1.2 mm thickness. The angle is chosen because it is near the failure angle, which is 71° for this particular geometry and material [Duflou et al., 2008]. The tool is a 10 mm sphere, and the tool path is composed of 60 contours with a 0.5 mm stepdown, obtaining a 30 mm final depth.

![Figure 1: Target geometry.](image)

1.2 Finite element model

The 8-node SSH3D element is used, with 4 integration points in the plane and 5 through the thickness. The element uses 24 EAS modes and the second version of ANS. Two meshes are used: a coarse mesh with 4484 nodes and 3064 elements and a fine mesh with 4492 elements.

As the cone is clamped, the nodes of the outer circumferential part are completely fixed. In the other edges, rotational boundary conditions are assumed.

1.3 Constitutive law

A $J_2$ isotropic plasticity model plus mixed isotropic/kinematic hardening is used during the simulations. Two sets of parameters are used, being the isotropic hardening and the identification procedure the difference between them.

The first set uses a Swift hardening with parameters identified using classical tests (tensile test and monotonic and Bauschinger shear test), while the second set uses a Voce law and parameters using also the classical tests,
but plus an indent test [Henrard et al., 2010]. The Table 1 shows the used parameters.

Table 1: Set of parameters used during the simulations [Henrard et al., 2010].

<table>
<thead>
<tr>
<th></th>
<th>Set2</th>
<th>Set8</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Swift/Ziegler</td>
<td>Voce/Ziegler</td>
</tr>
<tr>
<td>$K$</td>
<td>146.7</td>
<td>89</td>
</tr>
<tr>
<td>$n$</td>
<td>0.229</td>
<td>22.5</td>
</tr>
<tr>
<td>$\epsilon_0/\sigma_0$</td>
<td>1.5e-3</td>
<td>20</td>
</tr>
<tr>
<td>$C_A$</td>
<td>240.6</td>
<td>11.2</td>
</tr>
<tr>
<td>$G_A$</td>
<td>83</td>
<td>0</td>
</tr>
</tbody>
</table>

2 Results

Despite the long CPU time, the simulation with Set2 did not present any convergence problems. Nevertheless, the Set8 was surprisingly slower and a lot of convergence problems arose. So only the results for the coarse mesh of Set8 are shown, even if they are not complete.

Figure 2: Finite element mesh depicting the three selected elements.
2.1 Shape

There is no influence of the material parameters on the shape prediction. The results are obtained from the nodal displacement at a transversal cut (Fig. 3), and they are shown in Fig. 4.

Figure 3: Transversal cut used to obtained the nodal displacements.
Figure 4: FEA results (blue) and experimental (red). The Fig. 4(c) and Fig. 4(d) shows a zoom into the wall zone, depicted in a different scale for better analysis.
The shape match very well in both the upper and bottom layer with the experimental measurements.

2.2 Thickness

The thickness is calculated following a simple procedure, shown in Fig. 5. This consists in a normal vector from the upper layer towards the bottom layer, and taking the magnitude as the thickness. The results are shown in Fig. 6.

![Figure 5: Thickness calculation method used for solid elements.](image)

The results match quite well for both meshes. Nevertheless, and because of the mesh refinement in the deformed zone, the fine mesh is more accurate when predicting the minimal thickness. The predicted minimal thickness is 0.27 mm, while the experimental is 0.28 mm.

2.3 Forces

The force evolution is obtained as the reactions forces acting on the tool during the process. To avoid the noise due to edge effects and the modified tool motion adapted to the symmetry conditions, an average value of the force at the central third part of the pie is used and depicted in Fig. 9. The evolution is shown in Fig.
Figure 6: Comparison of the thickness distribution in a transversal cut between experimental measurements and numerical predictions, for two different meshes.

(a) Coarse mesh.

(b) Fine mesh.
Figure 7: Zone used to obtain the average values of the force.

The analytical formula proposed by Aerens gives values $F_s = 382.3 \text{N}$ (steady state) and $F_p = 596.9 \text{N}$. The finite element simulations are far lower as was previously observed for this same element for the line test and the two-slope pyramid. Any further work about SPIF and the SSH3D should address this issue.
Figure 8: Tool reaction during the process.
Figure 9: Average tool reaction during the process.
3 State variables

The evolution of the triaxiality (Eq. 1) and the normalized third invariant (Eq. 2) are analyzed for the three elements depicted in Fig. 2. Three integration points (being IP1 closer to the bottom surface) are plotted.

\[ T = \frac{\sigma_h}{\sigma_{eq}} \]  

\[ X = \cos 3\xi_{Lode} = \pm \frac{27}{2} \frac{J_3}{\sigma_{eq}^3} - 1 \leq X \leq 1 \]  

From this last equation, by assuming \( T = 0 \) \( X = \pm 1 \) is somewhat similar to an axisymmetric stress state while \( X = 0 \) is closer to in-plane shear. In plane stress conditions, there is a direct relation between triaxiality and the third invariant:

\[ X(T) = \cos 3\theta = -\frac{27}{2} T \left( T^2 - \frac{1}{3} \right) \]  

Because of the noise introduced by the edge effects of the symmetry assumptions, the results are between the edges depicted in Fig. 7. To know in what stage of the deformation the elements are, the equivalent plastic strain is plotted in Fig. 10. A new element is added (element 4) that will be explained later, while in element 1 the plastic strain is negligible. Only results from set2 are shown hereafter.

![Figure 10](image123.png)  

**Figure 10:** Equivalent plastic strain for three selected elements.
3.1 Coarse mesh, Set2

Figure 11: Results for triaxiality.
(a) Element 1.

(b) Element 2.

(c) Element 3.

Figure 12: Results for normalized third invariant.
3.2 Fine mesh, Set2

Figure 13: Results for the triaxiality.
3.3 Analysis

For the elements far from the tool contact, the triaxiality remains near zero meaning that is a deviatoric state. During the tool contact, the triaxiality varies much more compare with elasticity, which remains steady. For a deeper analysis, in Fig. 15 the results for the fine mesh, are compared with the equivalent plastic strain. It is observed a sudden increase of the

**Figure 14:** Results for the normalized third invariant.
triaxiality \((\text{triaxiality peak})\) when the tool is leaving the contact zone. This peak is also observed in the coarse mesh (Fig. 11(c)).

**Figure 15:** Triaxiality and equivalent plastic strain evolution for element 3, and the position of the element during the deformation.

The normalized third invariant exhibits high variance for all elements during one contour, making it difficult to get conclusions. This variance also implies a variable stress state, which depends in both in the tool position and the geometry (because the element in the flange is subjected to different stress state compared to the element under the tool). If the the Lode angle is obtained from triaxiality like in Eq. 3 (i.e., plane stress), and plotted in Fig. 16, it is remarkable how similar are both curves concluding that in elasticity the stress state is closer to plane stress, while in plasticity (or under the tool contact) is triaxial.

As element number 3 is deformed plastically too early during the process while element 2 start too late (Fig. 10), a new element (element 4) between these two is selected and analyzed during ten contours. The results for the triaxiality are depicted in Fig. 17 showing that even if the level of deformation is the same that for element 3, the triaxiality is could be much higher during one contour.
Figure 16: Lode angle plane for element 3.

Figure 17: Triaxiality and equivalent plastic strain evolution on element 1756 during 10 contours.
References
