RECENT ADVANCEMENTS IN 'STATE OF THE ART' ALUMINUM EXTRUSION SIMULATIONS

Mike Foster[1], Tushar Bakhtiani[2] and Deepu Joseph[1]

[1]Scientific Forming Technologies Corp., 2545 Farmers Dr., Columbus, OH, 43016, mfoster@deform.com

[2]VIP Tooling, 739 E. Franklin, Shelbyville, IN 46176, tbakhtiani@viptooling.com

Keywords: Extrusion simulation, Updated Lagrangian, die design, die stress analysis

Introduction

Process modeling software based on the finite element method (FEM) is a valuable tool in analyzing metal forming processes [1,2]. Extrusion is one such process where important information can be gained by using simulation [3,4]. The material flow during extrusion can be analyzed, giving insight into the initial transient behavior in the die, weld seam formation, etc. Stresses in the die components can also be obtained using simulation, allowing die modifications to be analyzed to see if they would improve the stress state and make die cracking less likely.

Several techniques can be used to simulate the stresses in the extrusion dies. One option is to model the extrusion process using rigid dies that do not deform. A subsequent stress analysis can then be run at a particular step of the extrusion simulation, after switching the tooling components from rigid to elastic. This provides the stresses in the tools at that one particular instance in time. This is called a decoupled stress analysis.

This paper deals with a second technique called coupled die stress analysis. Some extrusion dies are designed to elastically deflect during the extrusion process. This deflection closes gaps between the tooling components that were initially present. In order to model the extrusion process in these dies, a coupled analysis is needed so that the die deflection can be accurately captured during the forming. This type of simulation needs to use the Updated Lagrangian (UL) finite element formulation so that the transient flow and die deflection can be captured. All UL simulations performed in this study used the process modeling software DEFORM-3D. These coupled extrusion simulations are very computationally intense and are considered to be 'state of the art'.

Also discussed in this paper is a novel method for viewing material flow in Arbitrary Lagrangian Eulerian (ALE) extrusion simulations. ALE extrusion analyses, as implemented in DEFORM, use a mesh that does not update in the extrusion direction. Functionality has been developed to track the metal flow in these ALE simulations in a transient manner. This allows the location of weld seams to be predicted in either Updated Lagrangian or ALE simulations.

Original Design - Coupled Die Stress Analysis

VIP Tooling is a manufacturing company that specializes in aluminum extrusion tooling. They were having some die cracking issues when extruding an aluminum hollow square profile. Figure 1 shows the construction of the die stack that was having problems and Figure 2 shows where the die holder (top component in the die stack) cracked in production.

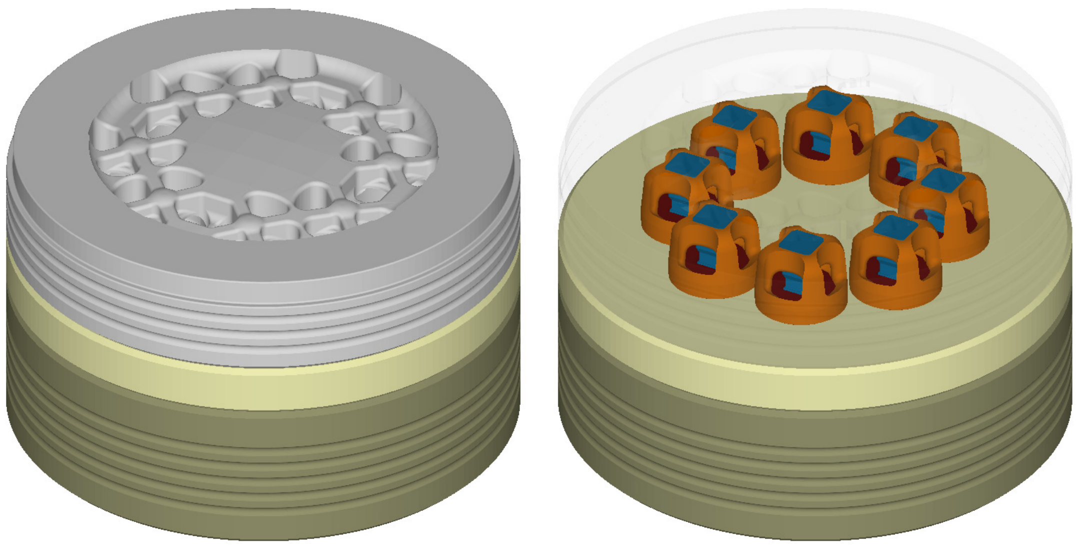

Figure 1: Original design for a modular hollow tube extrusion die. The top holder was made transparent on the right so that the internal cartridges could be seen.

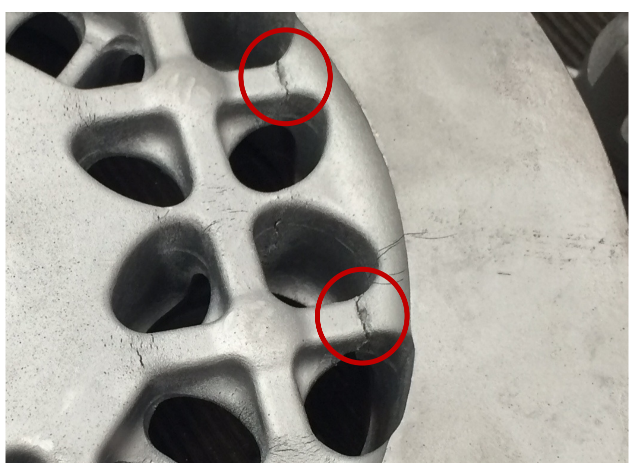

Figure 2: The holder consistently cracked at the locations highlighted.

In order to understand the root cause of this die failure, a finite element simulation needed to be run. Often for these type of die stress studies, simulations can be run with the billet extruding through rigid tooling. At the end of this extrusion analysis, the tools are converted from rigid to elastic and the loads from the extrusion simulation are interpolated onto the deformable tools. The output of this simulation is a one step solution for the stresses in the dies.

This decoupled approach to die stress would not work for this case. That is because VIP Tooling designed the tooling components to deflect during the assembly process and during extrusion. Figure 3 shows a sliced view of one of the die cartridge assemblies. The holder (gray), cartridge (orange), die plate (red) and mandrel (blue) are all shown.

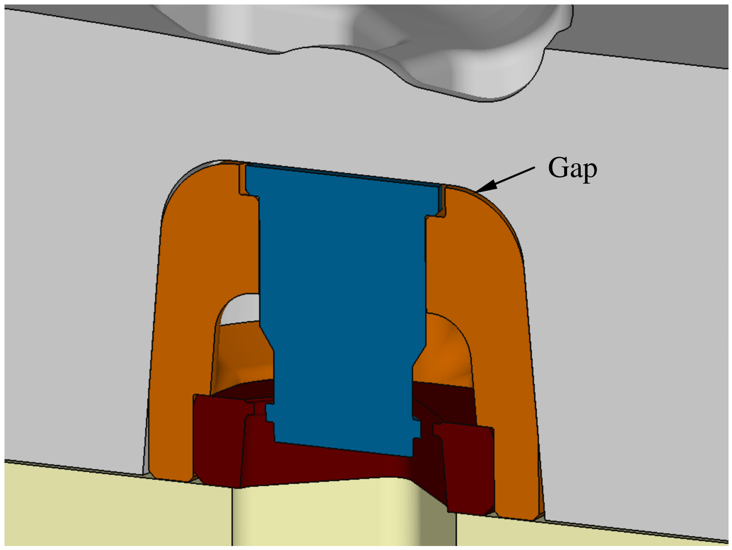

Figure 3: Cross-section through the cartridge assembly of the original die design.

In production, the die stack is assembled by pressing the cartridges into the holder. During this process, the gap between the cartridge assembly and the holder is engineered to partially close. The remainder of the gap closes during the extrusion process. For this reason, all of the tooling shown in Figure 3 needs to be deformable during the extrusion simulation so that correct deflection and stress is calculated in the tools. Figure 4 shows the finite element mesh that was used for the holder, cartridge, die plate and mandrel.

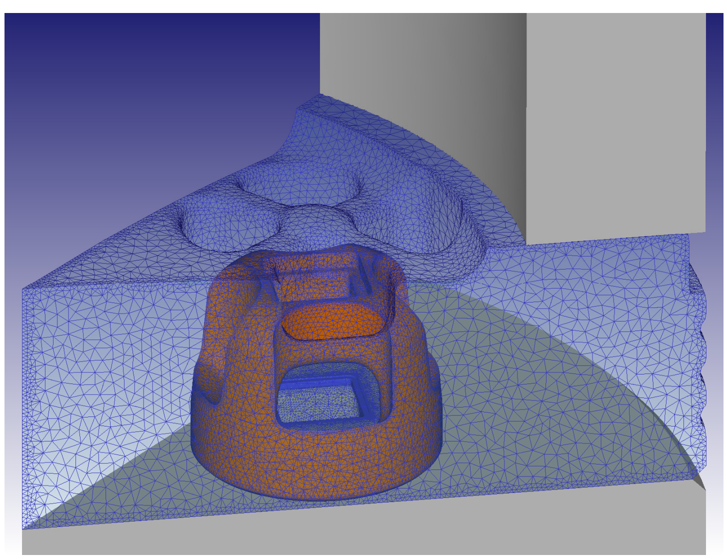

Figure 4: Mesh system used on the die components in the coupled die stress analysis.

The billet being extruded was Al-1100 and the die components were H-13. For the simulation, the mechanical and thermal properties for these materials were taken from the DEFORM material library. Due to the symmetry of the model, only one of the eight die ports was modeled. The process parameters used in the simulation are listed in Table 1.

Table 1: Process parameters used in the simulation

Billet length [in]	4
Billet diameter [in]	7
Initial temperature of billet [°F]	450
Initial temperature of tools [°F]	400
Friction between billet and dies	0.7 Shear
Friction between die components	0.05 Coulomb
Ram speed [in/s]	0.25
Total stroke [in]	1

The transient Updated Lagrangian simulation was divided into two parts. First, the cartridge assembly was pressed into the holder. After this was completed, the billet was extruded through the deformable die stack. Figure 5 shows the load vs. stroke curve for the process, along with images showing deformation of the billet. Each of the four elastic dies had between 150,000 and 200,000 elements and the billet had approximately 300,000 elements by the end of the simulation. Mesh windows were used for the billet to keep the mesh in the bearing zone fine and the mesh in the container more coarse.

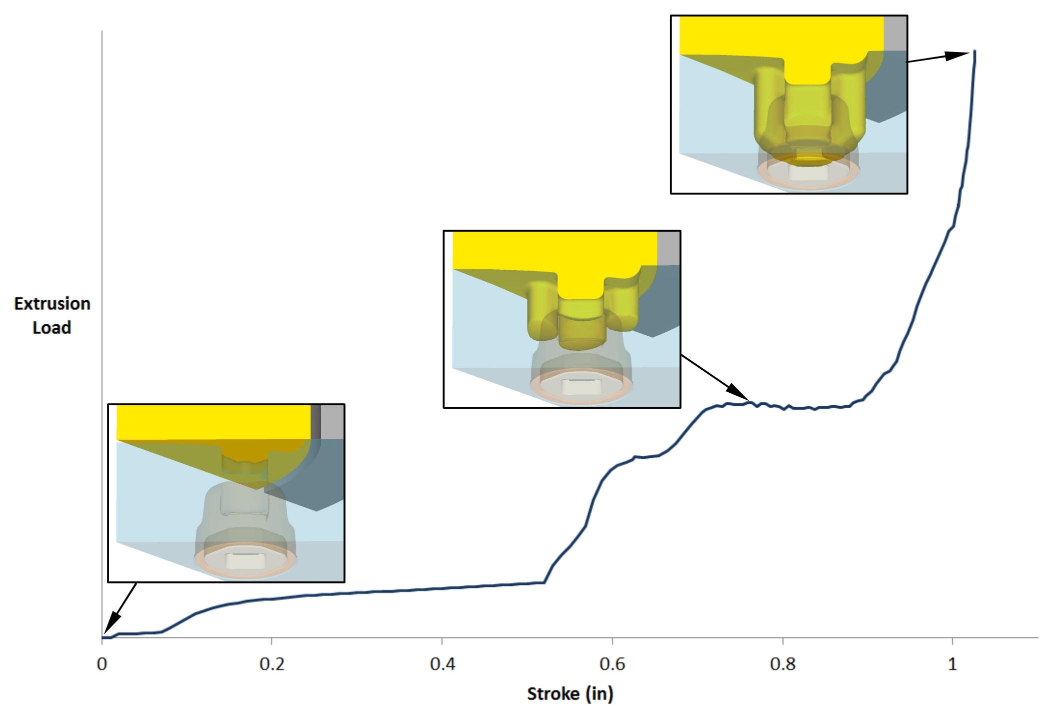

Figure 5: Load vs. stroke curve for the original die design, as well as images showing the progressing deformation.

Since this was a fully coupled analysis, the stresses in the dies could be viewed at any step of the simulation. As the extrusion progressed, the load increased and the stresses in the die components went up. Figure 6 compares the real cracked component with the stress in the holder at the end of the simulation. Maximum principal stress is shown (red is tension, green is compression), and the highest tensile stress in the holder is at the same location where cracking occurred. Further investigation showed that there was a large amount of bending that occurred at this location of the die.

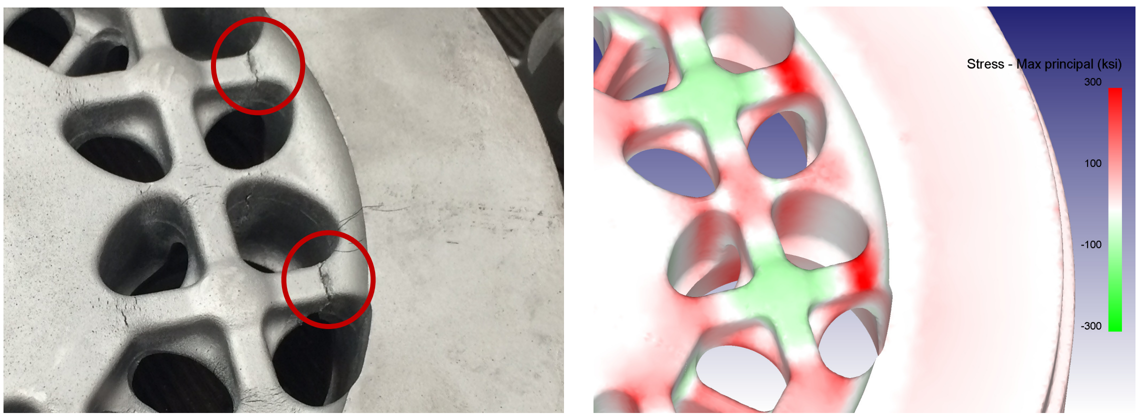

Figure 6: Locations of high principal stress (red is tension) match the cracks observed in the holder.

Modified Design - Coupled Die Stress Analysis

Now that the root cause of the cracking had been identified, a new holder was designed to try to improve the die life. The bridges in the holder were transferred to the cartridge to get rid of the previously observed high bending stress. Figure 7 shows the new die design. A sliced view of one of the new die cartridge assemblies is showed in Figure 8.

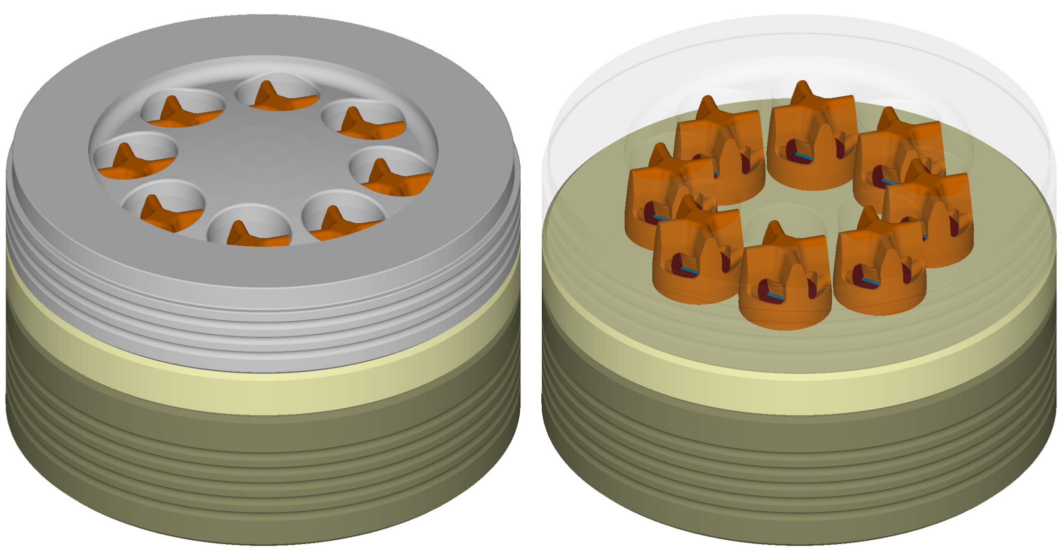

Figure 7: Modified modular extrusion die design. The bridges on the holder were removed and added to the internal cartridges.

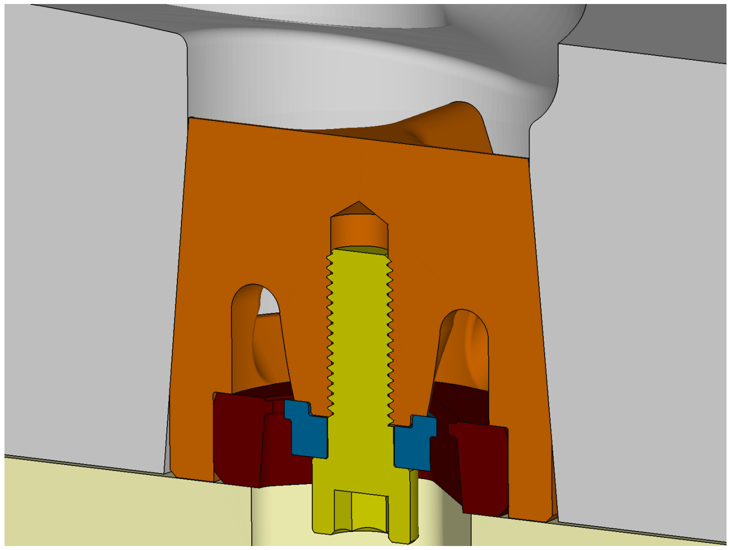

Figure 8: Cross-section through the cartridge assembly of the modified die design.

The same two part extrusion simulation was run for the modified die design. The cartridge assembly was first pressed into the holder and then the billet was extruded through the elastic die stack. At the end of the analysis, the extrusion load was highest, and Figure 9 shows the stresses in the elastic dies at that point.

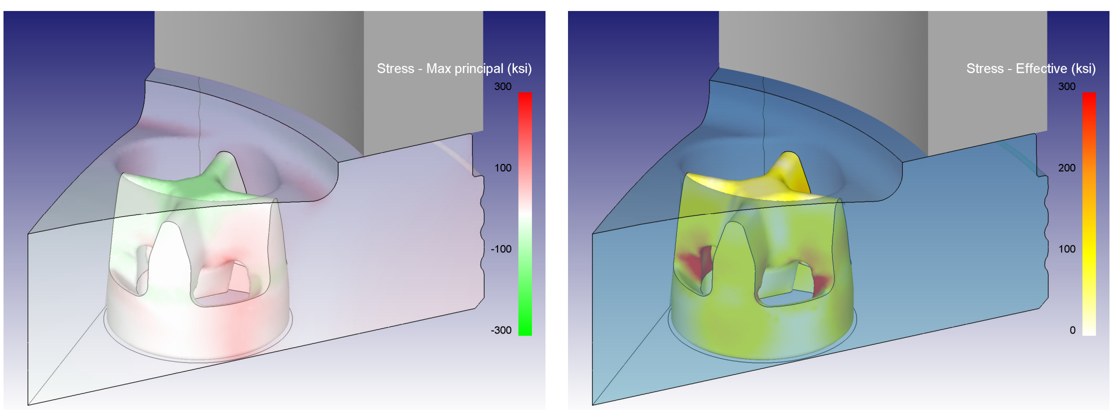

Figure 9: Locations of high principal stress (left) and effective stress (right) in the new die assembly.

In the new design, the stresses in the holder were very low. A high tensile stress is observed on the bottom of the bridges of the cartridge. The magnitude of this stress is reduced compared to that seen on the original cracked holder, though. Figure 9 shows that there is a large effective stress in the arms of the cartridge. This stress exceeded the yield stress of the H-13 die material, so some plastic deformation should be expected in this area. VIP Tooling is still working to create the optimal design for these type of extrusion dies.

Updated Lagrangian simulations such as these are transient in nature. Figure 5 showed what the material flow looked like at various points during the extrusion process. One thing that is of interest to die designers is the location of the weld seams in the extrudate. Figure 10 shows the difference in weld seam location for the two die designs. The original dies produced weld seams that are not exactly at the corners of the square extrudate.

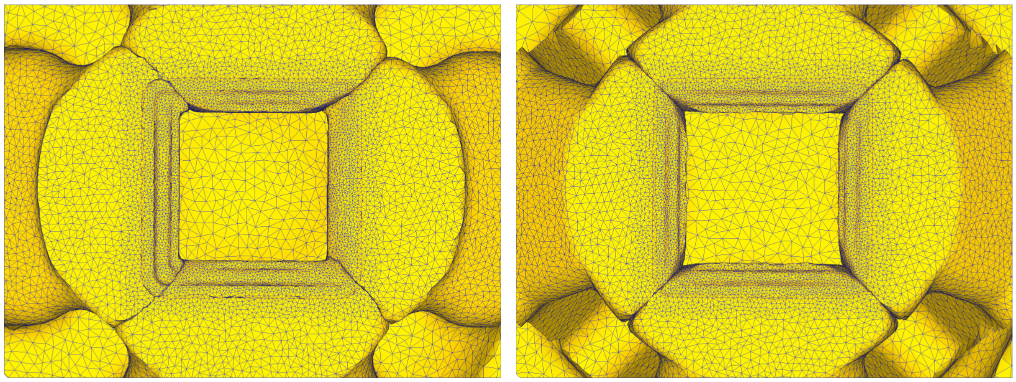

Figure 10: Weld seam locations using the original (left) and new (right) die design.

ALE Extrusion Simulation - Flownet

Aluminum extrusion dies typically employ a sharp 90 degree entry into the bearing zone. Frequent remeshing is often required in Updated Lagrangian extrusion simulations as material flows around this sharp die feature. This significantly adds to the total simulation time. In order to overcome this issue, DEFORM -3D has additional techniques, such as the Arbitrary Lagrangian Eulerian (ALE) method, to simulate extrusion processes. In the ALE approach, the material flow and the mesh update are decoupled and the mesh is only updated in the direction perpendicular to the extrusion direction. A convective correction is applied to the state variables to account for the different velocities of the mesh and the material. This essentially eliminates the need for remeshing. This ALE approach has been successfully applied to industrial extrusion problems [5].

One of the drawbacks of the ALE technique in simulating extrusion is the inability to track the actual material flow through the dies. Due to this, determining the location of weld seams in the extrudate has not been possible using ALE. This has now been addressed in DEFORM-3D by the development of a new flownet capability for ALE extrusion. A user-supplied flownet can now be tracked in an ALE extrusion simulation to help visualize the material flow. Figure 11 gives an example of this new flownet capability.

Tracking the flownet involves two steps. First, the velocity field is interpolated from the ALE workpiece and used to update the flownet geometry. Second, the flownet is remeshed to prevent excessive distortion. This flownet remeshing is carried out at a user-specified step interval and is performed from within FEM without stopping the simulation. The addition of the flownet remeshing does not add any significant overhead to the computation and therefore does not slow down the simulation.

Extrusion simulations using either the Updated Lagrangian or ALE methods can now predict the location of seam welds. The results from the two analysis types should match. Figure 12 compares the seam welds predicted in UL and ALE simulations, and it is seen that the locations where the flows merge together are the same.

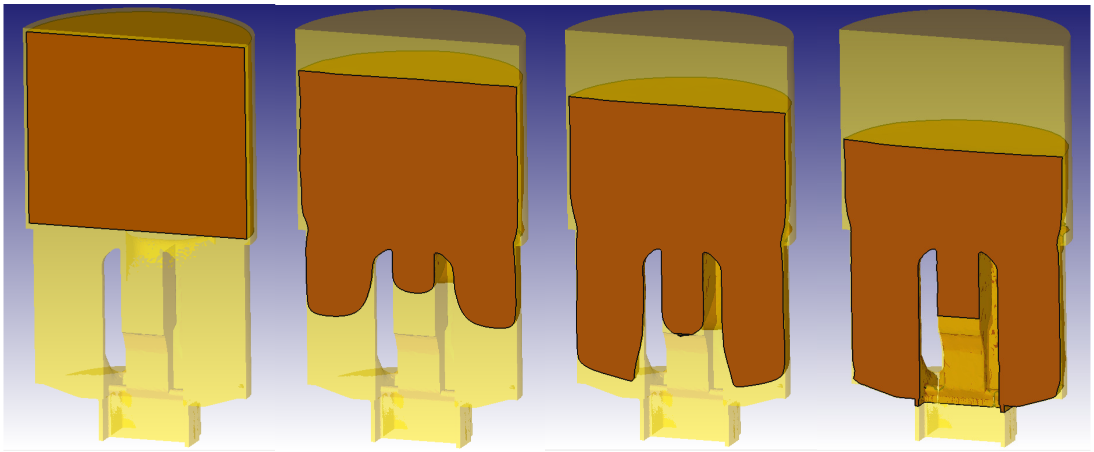

Figure 11: Sequence showing the new flownet updating in an ALE extrusion simulation.

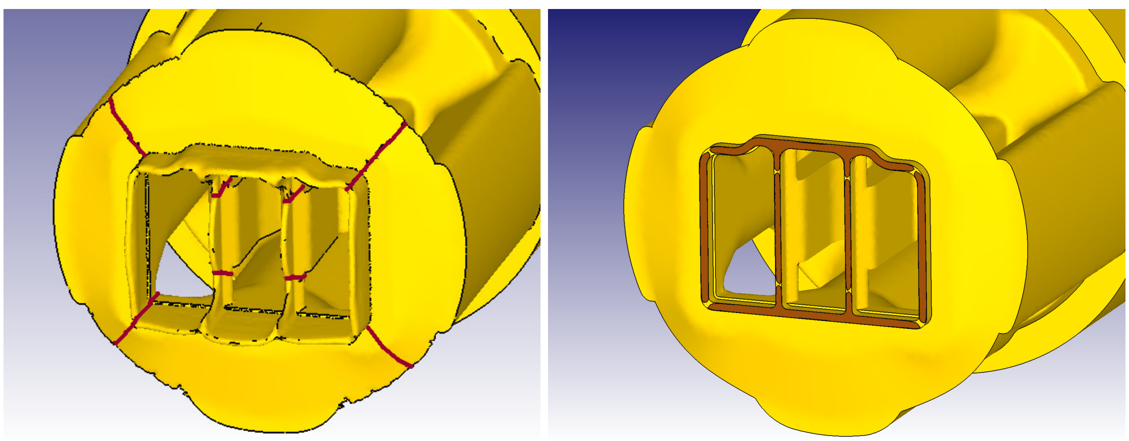

Figure 12: Weld seams predicted in Updated Lagrangian (left) and ALE (right) analyses. Both approaches show the flows merging at the same locations.

References

[1] M. Foster, C. Fischer: *Using process simulation to optimize forging processes,* FORGE Magazine (January 2007), p. 10-12.

[2] J. Miller, J. Walters: *Practical process simulation of multiple operations sequences,* Forging Magazine (July 2014), p. 20-22.

[3] J. Walters, M. Foster, A. Bandar: *The 'state of the art' in aluminum extrusion simulation using the finite element method,* Extrusion Technology 2012 Conference.

[4] M. Foster, D. Joseph, M. Fiderer: *Using finite element modeling to better understand extrusion material flow and die stress,* 2013 International Conference on Extrusion and Benchmark.

[5] D. Joseph, J. Yang, J. Oh, M. Foster, W. Wu, J. Lim, K. Kim, S. Park: *Aluminum Extrusion Simulation using Finite Element Method,* 2011 International Conference on Extrusion and Benchmark.