Modeling of Forging and Other Bulk-forming Processes

Mike Foster, Senior Engineer
EWI

Metal forming processes are often divided into two categories: bulk forming and sheet forming. The difference between the two deals with the extent of plastic deformation involved. In a bulk-forming process, the workpiece undergoes extensive plastic deformation. Forging and extrusion are examples of bulk-forming processes. Sheet forming, on the other hand, locally bends the metal to form the piece into shape. As a result, sheet forming involves significantly less plastic deformation than bulk forming.

The images in Figure 1 illustrate the forging of a steel crankshaft from a cylindrical billet. Here, the starting billet has a shape and cross-section that are completely different from that of the final component. This is typical in a bulk-forming process.

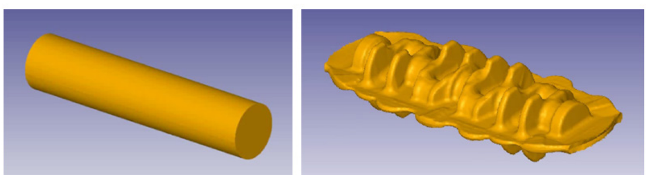

Figure 1: Initial billet and final forging for a crankshaft

The Benefits of Process Simulation

Finite element analysis (FEA) can be used to model bulk-forming processes. These simulations provide information that is difficult or impossible to obtain from experimental trials. Simulations can show how the material flows in the dies and whether defects will occur in the part. Laps and unfill are two defects that are easily observed in simulations. Laps occur when the deforming metal folds over on top of itself. Unfill occurs when the die cavity is not completely filled by the flowing material. Modeling can also provide insight into whether the dies will crack due to excessive loading.

These simulations can be done prior to the machining of any real dies, or running of time-consuming forging trials. For new parts under development, simulation significantly speeds the development cycle and reduces cost. For existing parts, modeling can be used to troubleshoot production problems. The following examples illustrate EWI's expertise and show some of the benefits of bulk-forming process modeling.

Example 1: Optimizing a Forging Process to Save Material

A forging company in Texas wanted to produce a component that had two independent lug-like features. The shape of the part required that multiple cavities be used to distribute the material during the forging process. The company wanted to optimize the process so that the part was forged using minimal material. The final part shape could not be changed, but the starting billet size and any intermediate cavity shape could be modified. An acceptable forging was one that fully filled the die cavity without any forging defects, and had a small amount of flash surrounding the part.

Three analyses were run. The initial simulation used a relatively small-diameter billet. Figure 2 shows several snapshots from the forging analysis. In this process, the initial billet is hit several times in the roll cavity on the left side of the die, with a 90° rotation between each blow. The part is then moved to the blocker cavity on the right side of the die and hit once. After a 180° flip, the forging is hit once in the finish cavity in the center of the die. The simulation showed that if this small-diameter billet were used, the large lug would not fill completely.

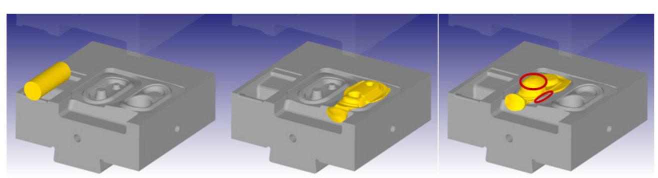

Figure 2: Forging process simulation showing unfill due to small-diameter billet

Both sides of the part would also have some unfill due to inadequate flash in that region. Red highlights are used in Figure 2 to denote the unfill.

A second simulation was run using a slightly larger-diameter billet. The roll cavity was also modified to improve material distribution. The analysis showed that there was some improvement, but unfill was still present in the same locations.

The billet diameter was increased again for the third analysis. This simulation showed that enough material was now available to completely fill the large lug and no unfill was present on the sides of the component (Figure 3). Adequate flash was also generated around the part, a requirement for a robust forming process. The company was able to take this process into production knowing that a good forging would be produced with minimal material waste.

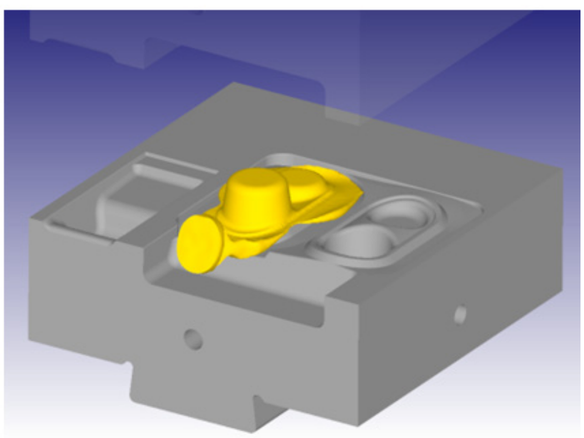

Figure 3: Forging process simulation with final billet diameter

Example 2: Eliminating Cracking Failures in an Aluminum Extrusion Die

A company that specializes in aluminum extrusion tooling was experiencing cracking failures in one of their die holder designs and wanted to run an FEA to understand the root cause. Figure 4 provides an illustration of the die assembly, with the die holder shown in light gray on the left. On the right, the die holder is transparent so the internal cartridges can be seen.

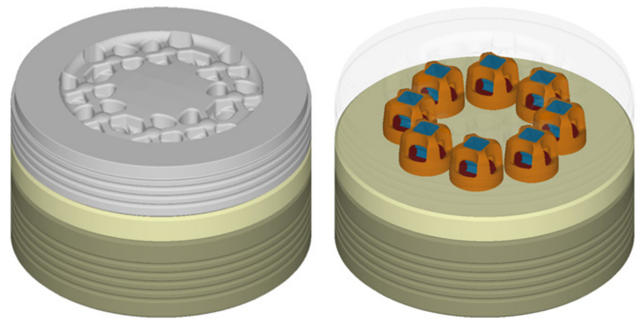

Figure 4: Original design for the extrusion die

A die stress analysis was needed to determine the cause of the cracking. This analysis required modeling of the extrusion process to accurately determine the loading on the die. Figure 5 shows the load vs. stroke curve, along with images of the deformed workpiece at various points in the process. Load increased as the four material flows filled the welding chamber of the die. When the flows merged together and the extrudate started to exit the die, the load was highest.

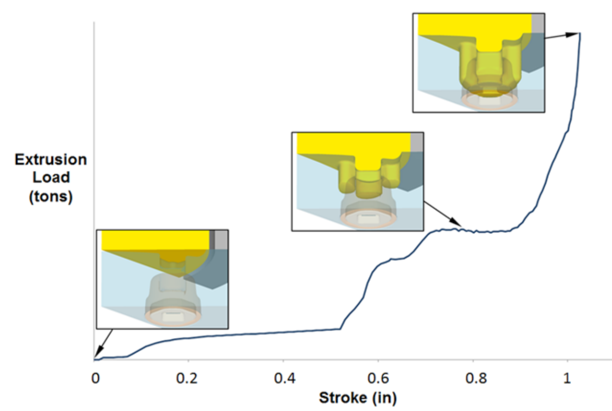

Figure 5: Load vs. stroke curve for the original die design

A simulation showing the maximum principal stresses in the die holder is provided in Figure 6, with tensile stresses shown in red, and compressive stresses shown in green. The fractured die holder is pictured as well, clearly illustrating that it cracked where the tensile stresses were highest. A magnified deflection plot showed that significant bending was taking place at these locations, generating the high tensile stress. This analysis indicated that tensile stress was the cause of the cracking failures, and that maximum principal stresses from the analysis could be used to identify where cracking may occur.

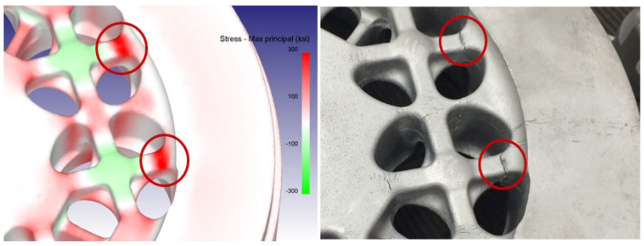

Figure 6: Locations of high tensile stress match the cracks observed in the holder

Once the root cause of the fracture was identified, a new holder was designed to eliminate detrimental bending stresses and improve die life. In the modified die design shown in Figure 7, the bridges previously integrated into the holder have been transferred to the internal cartridges.

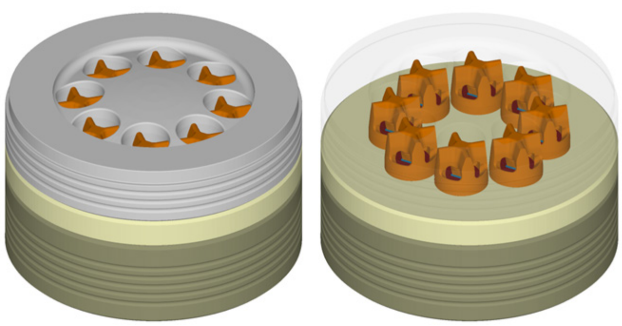

Figure 7: Modified modular extrusion die design

The extrusion simulation was then rerun using the modified die design. As with the previous design, a die stress analysis was performed at the point in the process when the loads were highest. Figure 8 shows the maximum principal stress in the modified die assembly. Tensile stresses were observed on the bottom of the cartridge bridges, but the magnitude was much reduced compared to the original design.

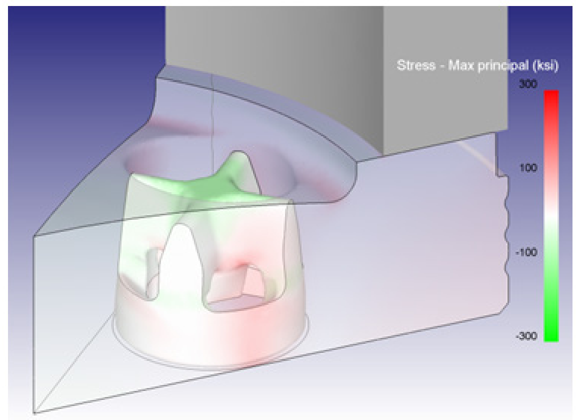

Figure 8: Maximum principal stress in the new die assembly

It is expected that cracking will no longer be a problem for this new die design. In addition, in the case that cracking does occur, repair will be more economical since only the small modular cartridges will need to be manufactured and replaced, instead of the entire monolithic holder.

The modeling discussed in this article was done in collaboration with Scientific Forming Technologies Corporation, the developers of the DEFORM process-modeling software package.

***Mike Foster** is a senior engineer in the EWI Structural Integrity and Modeling group where he focuses on simulating and optimizing metal forming processes. His work ranges from sheet metal forming to the analysis of bulk forming processes like forging. He has additional expertise in troubleshooting die cracking failures that occur during these processes. Mike has published numerous papers on the optimization of bulk forming processes such as forging, extrusion and drawing, as well as on die stress analysis.*