Finite Element Method Applications in Bulk Forming*

Soo-Ik Oh, John Walters, and Wei-Tsu Wu

METALWORKING, with its thousands of years of history, is one of the oldest and most important materials processing technologies. During the last 30 years, with the continuous improvement of computing technology and the finite element method (FEM) as well as the competition for a lower-cost and better-quality product, metalworking has evolved rapidly. This article gives a summary of overall development of the FEM and its contribution to the materials forming industry. Because significant efforts were carried out with great success by many universities and research institutes with a similar objective and application, this article is focused on the overall philosophy and evolution of the FEM for solving bulk forming issues. The program used to demonstrate success is the commercial code named DEFORM (Scientific Forming Technologies Corp.). A number of examples of the application of FEM to various bulk forming processes are also summarized.

This article provides an overview of FEM applications. In this section, a number of applications of FEM are presented in the order they would be used in a typical manufacturing process sequence: primary materials processing, hot forging and cold forming, and product assembly. Material fracture and die stress analysis are covered, and optimization of the design of forming processes is also reviewed.

**Historical Overview**

Lee and Kobayashi first introduced the rigid-plastic formulation in the 1970s (Ref 1). This formulation neglects the elastic response of deformation calculations. In the late 1970s and early 1980s, a processing science program (Ref 2) funded by the United States Air Force was performed at the Battelle Memorial Institute Columbus Laboratories to develop a process model for the forging of dual-property titanium engine disks. These disks are required to have excellent creep and high stress-rupture properties in the rim and high fatigue strength in the bore region. A FEM-based code, ALPID (Ref 3), was developed under this program. Thermo-viscoplastic FEM analyses (Ref 4) were also performed to investigate the temperature variation during hot-die disk-forging processes. The flow stress of thermo-rigid-viscoplastic material is a function of temperature, strain, and strain rate. Approximately five aerospace manufacturers pioneered the use of the code. Based on the same foundation, DEFORM was developed for two-dimensional applications in 1986. Due to the large deformation in the metal-forming application, the updated Lagrangian method always suffers from mesh distortion and consequently requires many remeshings to complete one simulation. Two-dimensional metal-forming procedures became practical for industrial use when automated remeshing became available in 1990 (Ref 5). In the beginning of the 1980s, the PDP11 and the CDC/IBM mainframe computers were used. In the mid-1980s, the VAX workstation became the dominant machine for running the simulations. In the late 1980s, UNIX workstations became the primary computing facility.

Unfortunately, the majority of the metalworking processes are three-dimensional (3-D), where a two-dimensional (2-D) approach cannot approximate reality satisfactorily. The initial 3-D code development began in the mid-1980s (Ref 6). One simulation with backward extrusion in a square container was reported to take 152 central processing unit (CPU) hours on a VAX-11/750. In addition to the need for remeshing, a more complicated process was estimated to take several weeks. Due to the lack of computing speed in the 1980s for 3-D applications, the actual development was delayed until the 1990s (Ref 7). Since then, many ideas to develop a practical 3-D numerical tool were evaluated and tested. The successful ones were finally implemented. After the mid-1990s, significant computing speed improvement was seen in personal computer (PC) technology, coupled with a lower price as compared to UNIX-based machines. For this reason, the PC has become the dominant computing platform. Due to the competition for better product quality at a lower production cost, process modeling gradually became a necessity rather than a research and development tool in the production environment.

Although FEM programs were initially developed for metalworking processes, it was soon realized that metalworking is just one of the many operations before the part is finally installed. Prior to forging, the billet is made by primary forming processes, such as cogging or bar rolling from a cast ingot. After forging, the part is heat treated, rough machined, and finish machined. The microstructure of the part continuously evolves together with the shape. The residual stress within the part and the associated distortion are also changing with time. To really understand product behavior during the service, it is essential to connect all the missing links, not only the metalworking. In the mid-1990s, a small business innovative research program was awarded by the U.S. Air Force and the U.S. Navy to develop a capability for heat treatment and machining (Ref 8). To track the residual-stress distribution, elastoplastic and elastoviscoplastic formulations were used. Microstructural evolution, including phase transformation and grain-size evolution, associated distortion are also changing with time.

During the 1990s, most efforts were focused on the development of the FEM for computer-aided engineering applications. However, the engineer’s experience still plays a major role in achieving a solution to either solving a production problem or reaching a better process design. The FEM solution-convergence speed depends highly on the engineer’s experience, and the interpretation of the results requires complete understanding of the process. As the computing power continues to improve, optimization using systematic search becomes more and more attractive (Ref 9).

In the following sections, a brief overview of the methodologies and some selected representative applications focusing on the bulk forming process are given.
Methodologies

To account for the complicated thermal-mechanical responses to the manufacturing process, four FEM modules, as shown in Fig. 1, are loosely coupled. They are the deformation model, the heat-transfer model, the microstructural model, and, in the case of steel, a carbon diffusion model.

Thermal Mechanical Models

Deformation Model. For metalworking applications, the formulation must take into account the large plastic deformation, incompressibility, material-tool contact, and (when necessary) temperature coupling. To avoid deformation locking under material incompressibility, the penalty method and selective integration method are usually used for the 2-D quadrilateral element and 3-D brick element, while the mixed formulation for the 3-D tetrahedral element is employed. It is generally agreed that the quadrilateral element and brick element are preferred in FEM applications. Due to the difficulty in both remeshing and (frequently) the initial meshing with a brick mesh in most forming applications, a tetrahedral mesh is generally used.

Due to its simplicity and fast convergence, the rigid-plastic and rigid-viscoplastic formulations are used primarily for processes when residual stress is negligible. The elastoplastic and elasto-viscoplastic formulations are important for calculating residual stress, such as in heat treatment and machining applications. However, it is very difficult to accurately characterize residual-stress evolution for forming at an elevated temperature, especially when there is significant microstructural changes, including phase transformation, precipitation, recrystallization, texture changes, and so on.

Because metalforming processes are transient, the updated Lagrangian method has been the primary FEM method for metalforming applications. Using this method for certain steady-state processes such as extrusion, shape rolling, and rotary tube piercing, however, may not be computationally efficient. In these special applications, the arbitrary Lagrangian Eulerian (ALE) method recently has been used with great success.

Heat-Transfer Model. The heat-transfer model solves the energy balance equation. The three major modes of heat transfer are conduction, convection, and radiation. Conduction is the transfer of heat through a solid material or from one material to another by direct contact. Generally speaking, below 540 °C (1000 °F), convection has a much more pronounced effect than radiation. Above 1090 °C (2000 °F), however, radiation becomes the dominant mode of heat transfer, and convection can essentially be considered a second-order effect. Between these temperatures, both convection and radiation play an important role.

In order to predict the temperature evolution accurately during metalworking processes, several important thermal boundary conditions must be considered:

- Radiation heat with view factor to the surrounding environment
- Convection heat to/from the surrounding environment, including the tool contact, free air, fan cool, water or oil quench
- Friction heat between two contacting bodies. It is also noted that frictional heating is the primary heat source in the friction-stir welding process.
- Deformation, latent heat, and eddy current are the primary volume heat sources. Deformation heat is important for large, localized deformation and fast processes, because the adiabatic heat will increase the local temperature quickly, and material is likely to behave differently at elevated temperatures. It plays an important role in metalworking, inertial welding, translational friction welding, and the cutting process. The latent heat comes from the phase transformation or phase change, and eddy-current heat is generated by electromagnetic fields.

Microstructural Model. Grain size is an important microstructural feature that affects mechanical properties. For example, a fine grain size is desirable to resist crack initiation, while a larger grain size is preferred for creep resistance. To obtain optimal mechanical properties, precise control of the grain size is crucial. In order to achieve a desirable microstructural distribution, as-cast materials usually undergo multiple stages of forming, such as billet conversion and closed-die forging, and multiple heat treatment steps, such as solution heat treating and aging.

During thermomechanical processing, a dislocation substructure is developed as deformation is imposed. The stored energy can provide the driving force for various restorative processes, such as dynamic recovery or recrystallization. On the completion of recrystallization, the energy can be further reduced by grain growth, in which grain-boundary area is reduced. The kinetics of recrystallization and grain-growth processes are complex. In order to predict the grain-size distribution in finished components, a basic understanding of the evolution of microstructural evolution during complex manufacturing sequences, including the primary working processes (ingot breakdown, rolling, or extrusion), final forging, and heat treatment, must be obtained. Hence, the development of microstructural evolution models has received considerable attention in recent years. Recrystallization behavior can be classified into three broad categories: static, metadynamic, and dynamic recrystallization (Ref 10).

The description of each recrystallization mode as well as static grain growth is well documented (Ref 11). Sellars’ model has been used for static and metadynamic recrystallization, and the Yamada model has been used for dynamic recrystallization. Microstructural evolution in superalloys is complicated by the precipitation of γ’, γ”, and δ phases (Ref 8). However, the present phenomenological approach neglects the specific effect that such phases have on the mechanisms of microstructural evolution.

Phase transformation is also another important aspect for material modeling (Ref 12). It is not only critical to achieve desirable mechanical properties but also to better understand the residual stress and the associated distortion. Phase transformation can be classified into two categories: diffusional and martensitic. Using carbon steel as an example, the austenite-ferrite and austenite-pearlite structure transformations are governed by diffusional-type transformations. The transformation is driven by a diffusion process depending on the temperature, stress history, and carbon content and is often represented by the Johnson-Mehl equation:

$$\Phi = 1 - \exp(-b\gamma t)$$

where \(\Phi\) is the fraction transformed as a function of time, \(t\), and \(b\) and \(n\) are material...
coefficients. The diffusionless transformation from austenite to martensite usually depends on temperature, stress, and carbon content.

**Primary Materials Processing Applications**

**Cogging**

Ingot conversion, also known as cogging, is one of the most common processes used to break down the coarse, cast micro-structure of superalloy ingots. As shown in Fig. 2, the ingot is held by a pair of manipulators at one of the two ends and is forged between two dies during the conversion process. The primary objective of the conversion process is to produce a fine grain structure for subsequent secondary forging operations. In essence, the process consists of multiple open-die forging (and reheating) operations in which the ingot diameter is reduced and its length is increased. Excessive furnace heating may promote undesirable grain growth. On the other hand, insufficient heating or excessive forging time may result in cracking. Control of the forging temperature, the amount of deformation, the forging time, and the precipitation of second phases is especially important for producing a desirable grain structure. Modeling the microstructural evolution of the ingot during the cogging process has been of great interest in recent years.

In the following example (Ref 13), the billet material was assumed to be nickel alloy 718 with an initial grain size of 250 μm (10 mils) (ASTM 1). The workpiece was taken to be octagonal in cross section (with a breadth of 380 mm, or 15 in., across the flat faces) and 2 m (7 ft) in length. Typical industrial processing conditions were applied. One deformation sequence comprising four passes without reheating was simulated.

Figure 3(a) shows the average grain size at the end of the fourth pass, as predicted by FEM. Predicted microstructures at approximately one-quarter of the workpiece length are shown in Fig. 3(b) to (e). After four passes, the simulation predicted that recrystallization would be rather inhomogeneous, and a number of dead zones would have developed near the surface. These trends are consistent with industrial observations.

**Rotary Tube Piercing**

Tube piercing modeling (Ref 14) illustrates the use of the ALE technique. The rotary piercing of a solid bar into a seamless tube, also known as the Mannesmann process, is a very fast rolling process. In the process, the preheated billet is cross rolled between two barrel-shaped rolls at a high speed, as shown in Fig. 4. The updated Lagrangian approach was first used in the investigation. Due to the dominantly rotational velocity field, the time-step size is limited to a small value, and the whole part must be modeled for better solution accuracy. It therefore increases the computing effort. To reduce the CPU time, a new method with the Eulerian approach was developed. Geometry updating is carried out in the feeding direction, while the nodal coordinates in the hoop direction remain unchanged. With this approach and the rotational symmetry treatment,
only half of the model is simulated, due to the symmetry condition.

During the process, tensile stress is created within the workpiece near the plug tip, and fracture continuously takes place to make the hole as the solid cylinder/tube is pulled through the rollers. The relative plug position with respect to the rollers is an important process design variable that will affect the occurrence of the rear-end defect. Figures 5(a) and (b) show the backend defect of a tube from experiment and simulation, respectively.

Rolling

The following are application examples in shape rolling and FEM evaluation of roll deflection.

Tram Rail Shape Rolling. Voestalpine Schienen GmbH simulated the multipass rolling of a rail section (Ref 15). This type of rail was used for the public tramways in many European cities. There were several passes making up the processing route of this rail section, but the first few were not considered critical. The final four roll passes were simulated.

A 76 cm (30 in.) length of the rail was modeled and initially contained 60,000 elements. After numerous automatic remeshings over the course of the simulations, the mesh had increased to approximately 75,000 elements. All simulations were carried out in nonisothermal mode to allow accurate modeling of any roll chilling and deformation heating effects on the predicted material flow.

Snapshots of the final four passes are shown in Fig. 6 to 9. The actual guide vanes for maintaining rail straightness were included in the simulations as rigid bodies. Without these guide vanes, the rail section could distort quite significantly. In addition, a pusher was applied to obtain the initial feeding of the rail into the roll gap. In all of these figures, the rolls are shown as semitransparent, and the guide vanes and pusher were omitted for clarity. The final two passes included side rolls and can be seen in Fig. 8 and 9. The predicted rail geometry after all rolling operations is shown exiting the final pass rolls in Fig. 9.

The purpose of the side roll in the second-from-last pass was to form the groove in the head of the rail. This was the critical rolling pass. The material flow had to be optimized to give approximately the same pressure or load on the upper and lower faces of the side roll. If this was not achieved, cracking would result in the side roll after a very short service life. Figure 10 shows an end-on view of the groove being formed in the head of the rail.

Elastic Roll Deflection. During flat rolling operations, it is not uncommon to obtain rolled sheet or plate having greater thickness in the center as compared to the edges. This is due to the problem of roll deformation. The material being rolled exerts a reaction force on the rolls. The reaction force bends the rolls, which are supported by bearings at their ends (Fig. 11), and flattens the roll locally due to the contact pressure. The rolls are elastically deformed, and there is less plastic deformation being imparted to the workpiece, resulting in a rolled stock of greater thickness than intended. In order to compensate for the roll deformation and obtain the desired workpiece dimensions, crowned rolls, as shown in Fig. 11, are often used to reduce this effect.

An elastic roll analysis was carried out in a FEM simulation. The analysis was of the ALE type (Ref 14), with a rigid-plastic, aluminum 1100-series alloy rolling stock. The analysis accounted for thermal effects also. The roll was set to a temperature of 425 °C (800 °F), and the roll stock was set to 540 °C (1000 °F). The roll speed was 15 rpm. Half-symmetry was applied, and the rolling configuration is shown in Fig. 12. Figure 12 also shows the predicted roll deformation deflection along its length. The deflection was determined from the brick element nodal coordinates.

Fig. 5 Backend defect of 26.7 cm (10.5 in.) diameter billet. (a) Experimental. (b) Predicted. Source: Ref 14

Fig. 6 Rail section exiting the fourth-from-last pass

Fig. 7 Rail section exiting the third-from-last pass

Fig. 8 Rail section exiting the second-from-last pass

Fig. 9 Rail section exiting the final pass

Fig. 10 End-on view of the rail section being rolled in the second-from-last pass
The predicted elastic roll stress is shown in Fig. 13. The roll is shown sectioned, having been sliced at half-length with contours of y-component stress. The stress at point P1 (Fig. 13) evolves and converges to a steady state, as illustrated in this figure.

**Shape Drawing**

While material being formed always follows the path of least resistance, that path is not always intuitive. Process simulation is a powerful tool in the prediction of material flow, especially in 3-D processes. One such process is shape drawing. When drawing a shape, there are several potential defects. These include die underfill, bending, ductile fracture, peeling at the die entry, and necking after the die exit. To illustrate the capability, a drawing process was analyzed using three input shapes into a shaped-draw die. The process is performed at room temperature. The goal of this process is to draw a shape that matches the exit cross section of the die. The first simulation used a round input material. The result was an underfill on the outside features (Fig. 14). This did not satisfy the final shape requirements of this case. The second simulation used a larger-diameter round input. The result was unstable flow, resulting in peeling (Fig. 15). Peeling is an undesirable effect where material is scraped off the wire into long slivers before entering the input port of the draw die. Because material follows the path of least resistance, it was clear that the round input stock was less than optimal. Finally, a shaped input was simulated with good results (Fig. 16). The hex-shaped initial shape placed more material where it was required to fill the exit cross section. In a process such as drawing, it is difficult to determine whether the input shape will yield a successful output, because the material has several competing directions to flow. Several process parameters, such as friction and temperature, can affect this result. Simulation can give insight into such a process before prototyping a die set.

**Hot Forging Applications**

**Billet-Heating Processes**

Billet heating is an important process in hot forging and heat treatment. The heating time and heat rate are the typical control process parameters. Cracking can occur when an excessive heating rate is used. Long heating time wastes energy and may result in poor microstructural properties. Insufficient heating time can result in high forming load, poor material...
flow, and fracture. It is therefore important to understand the temperature evolution within the workpiece during the heating process. The most frequently used heating methods are induction heating and furnace heating.

In addition to thermal, mechanical, and microstructural models, an electromagnetic model is needed to analyze the induction heating process. The electromagnetic model is first conducted to compute the magnetic field intensity and the eddy-current density. The heat generation based on the ohmic loss is then used to compute the temperature field. Microstructural and deformation information can be computed if necessary.

This method has been successfully applied to heating titanium billets and induction hardening of steel bearings (Ref 16). A scanning induction process (Ref 17, 18) is given to illustrate the methodology. In this process, approximately 300 mm (12 in.) of a 440 mm (17 in.) long, 23 mm (0.9 in.) diameter SAE 1055 steel shaft is induction hardened by moving it through a system comprising a two-turn 20 kHz induction heating coil and a water quench ring.

The FEM model used in the simulation is shown in Fig. 17. Induction heating was applied from the start of the simulation, but there was no cooling or relative movement between the induction unit and the workpiece for the first 2 s. Subsequently, the heating/cooling assembly moved a distance of 300 mm (12 in.) upward with a constant speed of 10 mm/s (0.4 in./s). The workpiece was then allowed to cool to room temperature, specified as 20 °C (70 °F).

In practice, the shaft moves, and the induction unit remains fixed, and the coil and quench ring moved upward.

A power of 15 kW was assumed for heating the workpiece, operating at a frequency of 20 kHz in order to concentrate the heating at the surface. A heat-transfer cooling window, representing the quench ring, was specified a temperature of 20 °C (70 °F) and a convection coefficient of 20 kW/m²·K, which are representative of a water quench. The predicted temperature field is shown in Fig. 18. It is noted that the shaft surface temperature is highest (represented by the square symbol) near the coils. Downward along the shaft surface, the temperature reaches its lowest point due to the water quench. Further down, the temperature increased again due to heat conduction from the hotter interior.

Full details of the overall methodology for induction heating/hardening are contained in Ref 16.

For furnace heating, radiation heat dominates the temperature distribution of the workpiece. Radiative heat energy emitted by a body depends on the emissivity of the body surface. This emissivity depends on the material type and the surface condition, and its value ranges from 0 to 1. If multiple bodies are involved, the net radiant exchange between the bodies depends on the geometry and orientation of the parts as well as the relative distance between the bodies. This net radiant exchange between the multiple bodies is represented by the view factor, F₁₋₂. Modeling radiation with view factor is imperative to achieve accurate simulation results for high-temperature heating and cooling processes.

As an example of this dependence on part geometry and orientation, the heating of nine billets (15 cm diameter by 30 cm high, or 6 in. diameter by 12 in. high) in a 1095 °C (2000 °F) furnace was modeled (Fig. 19). The billets were loaded in three rows and spaced 8 cm (3 in.) apart from each other. From the predicted temperature distribution in the billets, the effect of radiation shadowing can easily be seen. The slower heating rate of the center billet can be seen by comparing a plot...
of temperature versus time for a point sampled with this effect on and off (Fig. 20). If view factor radiation had not been considered in the simulation, all of the billets would have heated the same. Because view factor was incorporated, the difference in temperature between the billets in the corner and in the center of the loading pattern was realistically predicted.

**Axle-Beam Forging**

A major commercial vehicle part manufacturer discovered a problem with an axle-beam forging. A forming lap or fold defect was evident on the finish-forged product, as seen in Fig. 21. History and experience guided the designers to concentrate their efforts on the blocker and finisher stages of manufacture. However, changes to these did not eliminate the lap.

The manufacturing process was simulated. Four discrete operations were involved: roll former, bender, blocker, and finisher operations, as seen in Fig. 22. Because the material behavior can be highly temperature sensitive in a hot forming process, all stages of the process were modeled with full thermal coupling. Also included were intermediate operations, such as transfer times from the furnace to the press and times when the forging was resting on relatively cool dies. In this approach, surface chilling of the workpiece was accounted for.

The simulation results highlighted the fold occurring during the bender operation, as seen in Fig. 23. The defect was carried through to the finish-forged axle beam. After reviewing the simulation result, the designers were able to locate the defect on the actual part, as seen in Fig. 24. The designers modified the pads on the bottom die to revise material flow and eliminate the lap.

In addition to overcoming the forming defect, the bender die-pad modifications resulted in a reduced forging load in the bender, blocker, and finisher operations. The production trials correlated very well with the simulations, which also predicted lower forging loads with the modified pads.

After forging is completed and the flash is removed, the axle beam is heat treated to provide mechanical properties required for its service life. The axle beams are heat treated in batches and are supported on their pads; that is, they are heat treated upside down relative to the orientation on the truck. Distortion during the quenching operation is undesirable. In any case, distortion does occur due to the volume increase associated with the austenite-to-martensite phase transformation. Figure 25 illustrates the comparison between the as-forged and asquenched axle-beam predictions, both at room temperature. The as-forged part is shown in the foreground, and the same part after quenching is shown in the background. Both are at room temperature. Note how the heat treated beam is noticeably longer than the original forging.

![Fig. 18 Steady-state temperature distribution in workpiece 25.4 s into the simulation](image)

Knee-Joint Forging

The forging industry continues to expand with the rest of innovation and often finds new and interesting applications. One such application is the medical implant industry. In this case, an artificial knee implant is considered. The orthopedic surgeon can remove the patella (kneecap), shave the heads of the femur and tibia, and implant the prosthet. Special bone cement is used for suitable adhesion, and the implants can be seen in their locations in Fig. 26.

An analysis was carried out on the hot forging operations of the tibial part of the Ti-6Al-4V knee-joint prosthetic device. In this case, there were three operations: blocker, finisher, and restrick operation. Each of the three operations consisted of a furnace heat, forming operation, and flash trim. Different friction conditions were applied for the extruded part and the coined portion of the prosthet. This was important because, in practice, only the extruded part of the dies is lubricated; the coined section is formed dry. Figure 27 shows the tibial part at the end of the blocker, finisher, and restrick simulations.

Furnace temperatures were specified as 940 °C (1725 °F) for the blocker and 925 °C (1700 °F) for the finisher and restrick operations.

![Fig. 19 Temperature of nine billets (eight are shown) in a 1095 °C (2000 °F) furnace. Note that the proximity between the parts affects the temperature](image)
A rigid-plastic workpiece and rigid dies were used in this analysis. After the blocker-operation simulation, the workpiece was trimmed, as shown in Fig. 28.

The surface curvature weighting was set high in this analysis. As a consequence, the tighter radii of the webs and ribs received a finer element size, whereas the larger, flatter surfaces were assigned coarser elements. This is illustrated clearly in Fig. 29.

**Cold Forming Applications**

**Cold-Formed Copper Welding Tip**

Multiple folds were observed during the production of a copper welding electrode, as seen in Fig. 30. The cause of the defects was not entirely understood. The entire forming process was simulated to gain a better understanding of why the folds were developing (Ref 19).

The actual part underwent a total of five operations to form the finished electrode:

---

**Fig. 20** Temperature plot comparing a point on a billet considering and not considering view factor radiation effects

**Fig. 21** Noticeable fold on a finished axle-beam forging

**Fig. 22** The four operations in the process are initial perform in the forming rolls, after bending (shown in bottom die), after blocker operation (shown in bottom die), and the finished axle beam after flash removal

**Fig. 23** Simulation predicted the fold occurring during the bender operation, as shown

**Fig. 24** Once the fold location was established from simulation, the defect was identified on the actual axle beam after bending (location shown by chalk)

**Fig. 25** The as-forged shape is shown in the front (lighter color), with the heat treated shape behind (sliced and darker color). The phase transformation was the key reason for this distortion

**Fig. 26** After surgery, the femoral and tibial prosthetic parts are in position, securely adhered with bone cement
shearing, squareup, preform, backward extrusion, and final forming (Fig. 31). The sheared rectangular slug was assumed to be the starting material for the simulations, and therefore, the shear and squareup were not simulated. The preform and backward extrusion operations were axisymmetric in nature and were therefore simulated in two dimensions. When the 2-D backward extrusion was finished, the final operation was simulated in three dimensions.

During the preform operation, a stepped die was used to distribute the volume. This sharp step then gets pushed into the tapered die during the backward extrusion, creating a fold on the exterior of the part. The fold develops mid-stroke and moves upward as the extrusion progresses. The location of this external lap predicted in the simulation is identical to that seen on the actual formed electrode.

The final operation involves piercing the extruded slug with a splined punch. At this stage, the outside of the part is close to finish shape, and the punch is used to form the intricate cooling fins on the inside of the electrode. Midway through the finish operation, it is seen that quite a few defects are being formed by the punch (Fig. 32). Simultaneously, folds are created from material smearing onto the interior wall, peeling and eventually smearing onto the bottom internal surface (Fig. 33, 34), and lapping on the inside tip and faces of the cooling fins.

All of the defects observed in the simulations correlated very well with those seen on the actual formed electrodes. The results of the simulations proved invaluable in determining why the various defects occurred.

Pipe-Type Defects in Aluminum Components

A 6061 aluminum suspension component was simulated as an impact extrusion. The part was formed in one operation on a mechanical press. The simulation was performed to test the feasibility of producing a defect-free part in one forming operation. The simulation predicted a pipe-type defect prior to the dies being manufactured. The actual part produced can be seen in
Fig. 35. This piping or "suck" defect occurs due to volume deficiency. As the section between the punch and die becomes thin, an inadequate volume of material is available to feed the extrusion. When this occurs, longitudinal tensile stresses form under the nose of the punch. At that time, surface material is pulled into a cavity as it forms. This defect is shown in Fig. 36.

Another case where a pipe-type defect occurs is in the forward extrusion of a pressure valve (Fig. 37). Although the process is 3-D, the problem was successfully simulated assuming plane-strain deformation (Ref 20) during the late 1980s. In the following discussion, a true 3-D model was used.

The material for the pressure valve is aluminum alloy 6062. Because of symmetry, only one-fourth of the part was simulated. For better resolution of the defect, more elements were placed near the center of the part. The predicted part geometry at different stages of the extrusion process is shown in Fig. 38. From these figures, the defect starts at the center of the part and propagates in the transverse direction where the part is being extruded. This behavior is seen in the flow lines of the extrusion process (Fig. 39). A striking similarity between the predicted flow lines (Fig. 39) and the actual part (Fig. 37) can also be observed.

**Fastener Forming**

In the development of metalforming processes, designers balance many complex parameters to accomplish a workable progression design. These parameters include the number of intended operations, required volumetric displacements, final part geometry, starting material size, available forming equipment, and the material behavior of the workpiece. Frequently, variations have existed between the designer’s concept of the progression and the actual shop trial. When unexpected metal flow occurs, a part with underfill, excessive loads, die breakage, laps, or other production problems can result.
In one case (Ref 21), a fastener manufacturer noted a small defect during the shop trial of an automotive part (Fig. 40). The forming process was simulated, and the simulation reproduced this superficial defect and helped the manufacturer to understand the root cause. Additionally, the simulation (Fig. 41) revealed a severe lap that had originally been overlooked. When the trial parts were cut up, this defect was present as predicted. In this case, the cause of the lap was apparent from the simulation. Each station of a redesigned process was analyzed prior to shop trials. This approach resulted in a lap-free part (Fig. 41).

**Bevel-Gear Forging**

Bevel gears are important components in the automotive industry, such as in transmission differentials. Many of these components are forged at a hot temperature to minimize the amount of load required to form the part. This creates a part as seen in Fig. 42.

There has been some interest in forming these parts at room temperature for a better net shape and an improved surface finish. To study this as a 2-D process, an axisymmetric assumption is used. One tooth is isolated, and the circumferential flow is neglected. The radius of the tooth was specified to consider the volume of the actual part. The flow is seen in Fig. 43 in the case where the top and bottom die move together at the same speed. The flownet result can be compared to different movement conditions, such as when only the top die moves downward, as seen in Fig. 44, or in the case where only the bottom die moves, as seen in Fig. 45. Note that the filling of the material occurs well in the cases where the dies moved together and in the case where the bottom moves upward, but there was folding predicted in the case where only the top die moved downward. In each case, there is a marked difference in the grain orientation after forging, which can be seen from the flow lines. A similar study was performed as a comparison of simulation to plasticine deformation, and it was shown that the simulation was very accurate in predicting the material flow (Ref 22).

**Fracture Prediction**

**Chevron Cracks**

Forward extrusion is a process used extensively in the automotive manufacturing industry. Certain extruded components, such as axle shafts, are considered critical for safe operation of the vehicle and must be free of defects. During the mid-1960s, automotive companies encountered severe axle shaft breakage problems. In addition to the obvious visible external...
defects, internal chevron cracks were also present (Ref 23). The problem was so serious that a number of manufacturers adopted 100% ultrasonic testing procedures, with automatic rejection of suspect shafts.

To avoid 100% inspection, in the early 1970s, the Chrysler Corporation developed conservative guidelines for forward extrusion in conical dies, guaranteeing chevron-free parts (Ref 23). Upper-bound methodologies were used to determine the conditions under which chevrons would form. Based on die-cone angle, process reduction, and friction conditions, Avitzur derived mathematical expressions to describe the central bursting phenomenon during the wire drawing or extrusion of a non-strain-hardening material (Ref 24). To validate this work, experiments were carried out at Lehigh University on AISI 1024 plain carbon steel bars. Drawing was carried out in dies having an 8° semicone angle, with greater than 22% reduction, that is, in the safe region of Avitzur’s curves. No central bursts were reported (Ref 24).

Avitzur derived similar criteria for central bursting in strain-hardening materials (Ref 25) that were later validated by Zimerman and Avitzur (Ref 26). Experimental results are shown (Fig. 46) where it is clear that no central bursting occurred in the safe zone. This work also illustrates the die angles and drawing forces where central bursting occurs. This is overlaid on a schematic of the drawing conditions providing sound flow, dead-zone formation, and shaving (Fig. 47) (Ref 26). DaimlerChrysler implemented Avitzur’s curves in the early 1970s, and the chevron cracking problems were no longer troublesome.

More recently, work has been carried out using FEM in conjunction with ductile fracture criteria to determine the occurrence of central bursting. The parameter damage is a cumulative measure of the deformation under tensile stress and has been associated with chevron cracking. Researchers have evaluated seven different damage or ductile fracture criteria (Ref 27). The various criteria express ductile fracture as a function of the plastic deformation of the material, taking into account the geometry, damage value, stresses, and strain within the workpiece. When the maximum damage value (MDV) of the material exceeds the critical damage value (CDV), crack formation is expected.

A specific damage model proposed by Cockcroft and Latham states that fracture occurs when the cumulative energy density due to the maximum tensile stress exceeds a certain value. This criterion has provided good agreement at predicting the location of the fracture. The Cockcroft and Latham criterion is shown as follows in both dimensional and nondimensional forms:

\[
\int \sigma^* \, d\varepsilon = C_a
\]

\[
\int \sigma^* \, d\varepsilon = C_b
\]

where \(\sigma^*\) is the principal (maximum tensile) stress, \(\sigma\) is the effective stress, \(d\varepsilon\) is the increment of effective strain, and \(C_a\) and \(C_b\) are constant values.

Under ideal drawing or extrusion conditions, the strain distribution across the component cross section would be uniform. However, the occurrence of subsurface redundant deformation causes the strain distribution to become nonuniform. The amount of redundant deformation increases with increasing die angle and can cause extremely high tensile stresses. These internal tensile stresses can in turn lead to micro-voiding and ultimately to cracking (Ref 28). If the die angle/draw reduction combination cannot maintain compressive axial stresses in the drawn component, the center portion will be stretched, and the tensile stress may increase to a level where bursting occurs (Fig. 48). In the first forming operation for a component, the stresses in the component may be high, but the damage will still be quite low because the damage accumulates with deformation. For this reason, fracture does not generally occur until the second or third draw or extrusion operation. A triple extrusion (last image in Fig. 48) was simulated to demonstrate this phenomenon. It is clear that in this case, chevrons do not form until the third reduction.

One method to determine the CDV of a material is to perform poorly lubricated compression and notched tension tests until cracking is detected. After testing, simulations can be performed, matching the geometry and process conditions of the experimental tests, to calculate damage values. The predicted MDV at the instant of fracture is a good representation of the CDV of the material. Because the crack is not visible until after it has formed in the tensile test, a higher estimation of the CDV is likely. Averaging the CDVs calculated from the compression and notched tension tests is a reasonable approach.

A comparison was made between two automotive shaft designs manufactured using a
double extrusion. The only parameter changed was the die semicone angle for the minor diameter. Five-hundred steel shafts (AISI 1024) were produced with a nominal 22.5° extrusion die angle, and 500 were produced with a 5° die angle. Chevron cracks were observed on 1.2% of the shafts made from the 22.5° die, but none were observed in the product produced using the 5° angle. Process simulation was used to analyze both processes. The simulation indicated a higher damage value for the product extruded with the 22.5° die angle than for the parts produced with the 5° angle. The high damage value correlated well with the location of the chevron cracks (Fig. 49).

When damage levels are very high, fracture will occur consistently. When processes are well below the ductility limit, fractures are not expected to occur. A narrow range exists in between these two regions where the chance of cracking is probabilistic and a higher damage prediction can be interpreted as a greater chance of fracture. Because the damage is cumulative in nature, the prior working history of the wire or workpiece is an important factor to be considered for the likelihood of fracture occurrence.

**Fracture during Cold Forming**

During an initial trial of a cylindrical cold-formed part (Fig. 50), the manufacturer observed a severe fracture originating in the inside diameter of the part after the second operation. There was also a die underfill in the area of the fracture. The defect was observed after the end of the forming operation, as shown in Fig. 51. As seen in Fig. 51, the flowlines during the two operations show the grain orientation of the part after forming. Also seen in Fig. 51 are the underfill at both the inner and outer diameters. A plot of damage versus time is shown in Fig. 52, where the separation and contact of the sampled points have been highlighted in the plot. It can be noted that the damage at point 2 remains 0 during the first operation and starts to increase in the second operation. As shown in Fig. 51, in the second operation, point 2 originally resides underneath the punch. The point contacts the punch and moves outward as the punch moves downward. As the material is backward extruded, this point moves upward and loses contact with the punch. The separation between the material and the punch forms an underfill on the part inside diameter surface. The predicted damage value continues to increase, as shown in Fig. 52, when point 2 passed the punch corner, backward extruded, and separated from the punch. At point 1, the damage increases when the material is backward extruded and separated from the bottom die. The damage value at point 1 is significantly lower than point 2.

In order to manufacture this part, subsequent analysis revealed that the damage factor could be reduced. Analysis was used in conjunction with shop trials to eliminate the fracture and die underfill in this part (Fig. 53).

**Die Stress Analysis**

**Four Common Modes of Die Failure**

The four common modes of die failure are catastrophic fracture, plastic deformation, low-cycle fatigue, and wear. **Catastrophic** failure occurs when a die is loaded to stress levels that exceed the ultimate strength at temperature of the die material. This can occur due to gross overloading of the die structure, inadequate support to transmit the load to an adjoining component, or a stress concentration at a sharp feature. Dies that fail in this mode can release enormous amounts of energy, resulting in a serious safety hazard. **Plastic deformation** occurs when the stress on a die exceeds the yield strength at the operating temperature. Plastic deformation can occur as a localized effect or a widespread condition. Examples of a localized effect are rolling the corner of a punch or the initial yielding in a stress concentration prior to a fatigue failure. In these cases, the overall die dimensions do not change. Large-scale yielding occurs when a bolster plate is dished or the entire inside diameter of a shrink ring becomes oversized.
Low-cycle fatigue is a process that can occur when a mechanical or thermal tensile stress is cyclically applied to a die as each part is produced. If the stress intensity or number of cycles is sufficient, a very small crack can initiate. After crack initiation, the crack propagates with each additional cycle. As the defect grows, the die structure weakens, which results in higher stress concentrations and an increased crack growth rate. Eventually, the die fails due to a catastrophic fracture, although the energy released during final fracture is generally small relative to a single-cycle catastrophic failure. Low-cycle fatigue is well understood by the manufacturers of automobiles, aircraft engines, and other critical service systems. Materials used in these applications can be characterized by stress-number of cycles curves that relate the fatigue life to cycles to failure. Typically, these data are very expensive to obtain. Thus, it is extremely rare to find fatigue data for forging or cold heading dies in the literature.

Die stress analysis is used to determine the stress level on a die during service. More than one stress state may be significant. Initially, the effective stress is used to understand the magnitude of stress on the tool. Ideally, the stress should be well below the yield strength at the local service temperature of the die. Additionally, maximum principal stress is used to determine the tensile component of this stress. This can be used to predict the likelihood of a fatigue failure. Stress components are used to quantify the direction of stress. A clear understanding of the stress direction is required to ensure that the correct solution is developed.

Die wear is also a frequent topic of discussion related to coatings, die materials, and lubricants. Considerable research is being performed on the topic of wear relative to a wide range of manufacturing processes.

Die-Insert Fracture

A multistation cold heading process was used to produce a high-volume automotive part from AISI 4037 steel wire. The process involved extruding the body, upsetting, and back extruding the flange area of the part. Piercing out the center finished the part. While this part was not completely axisymmetric, the deviations from this were subtle; thus, using a 2-D analysis was reasonable and very fast (Ref 29).

The first station was primarily a squareup of the cutoff, with a relatively small amount of forward extrusion. The second station was a finished forward extrusion coupled with a small amount of backward extrusion around the punch pin. The third (final) station was a backward extrusion around the punch, combined with upsetting in the flange area. With the deformation analysis results completed, the interface pressure values were interpolated along the tooling surfaces to perform a decoupled-die stress analysis.

This part was run several times in production prior to the investigation. History showed that the die insert was breaking prematurely in the first and third stations, as shown in Fig. 54. Additionally, the punch was failing in the second station, and other tooling components were failing in the piercing station. While all stations were analyzed, the first station is summarized here.

It has been learned that to accurately analyze die stresses, complete tooling geometry must be used, including shrink rings and mounting dies. Analyzing a single die in isolation does not provide meaningful results in cases such as the die insert, where the interaction with the shrink ring is critical. In the present example, the dies were analyzed as elastic bodies during the analysis. In the case of the steel tools, the effective stress was used as a yield criterion, but in the carbide components, maximum and minimum principal stresses were used, along with their components. These low-cycle fatigue failures in the carbide inserts indicated tensile stresses in the crack-initiation site, even though the maximum principal stresses were less than 690 MPa (100 ksi).
Results for the first station tooling assembly can be seen in Fig. 55 and 56. This assembly shows the punch (D40), the insert (D70), and the shrink ring/sleeve assembly (S7 and M300). The ring/sleeve assembly did not indicate any overstress condition and performed well in field service. The insert did indicate a positive maximum principal stress in the region where the fracture was observed. The principal stress direction was predominantly axial. After carefully studying the simulations, it was decided to run a simulation with a two-piece insert, as shown in Fig. 57. The analysis indicated a significantly lower stress in the region of the fracture.

The tooling was modified based on the simulation results. The insert in the first station was split into two components. This resulted in a 550% life improvement. Other tooling modifications resulted in similar improvements in other stations of the process.

Secondary effects influenced tool life on the other components as a result of the aforementioned modifications. For instance, tooling components that were not modified in each station also showed substantial improvements, most of which were above a 150% increase. The net result, based on historical data, showed a 43% reduction in overall tooling costs and a 54% reduction in downtime associated with tooling problems on this product.

**Turbine Spool Die Failure**

A company was experiencing a dimensional problem with a hot-forged turbine disk (Ref 30, 31), as shown in Fig. 58. The forging was undersized on a number of features on an inside diameter. After an investigation of potential causes, the dies were thoroughly inspected. The inside diameter of a die liner and the outside container were oversized. This was somewhat surprising, because the outside diameter of the forging was in tolerance.

A stress analysis indicated that the effective stress exceeded the yield strength at temperature in the region where the die had yielded (Fig. 59). A range of redesign options was developed and analyzed. The redesign that was selected involved a thicker die wall (Fig. 60). Criteria for the redesign included cost, ease of assembly, and structural integrity. It was critical that the die wall possessed sufficient strength to avoid plastically deforming during the forging operation.

This design was successful in that the dimensional deviations were eliminated and there was a significant reduction in die maintenance cost. The root cause of this problem was large-scale plastic deformation. This case is interesting because the original symptom of a problem was a dimensional deviation in the forging. Because this die material was quite ductile, no fracture was observed. In fact, the inside features were actually forged within tolerance, but the outside was forged oversized. During the ejection process, the part was essentially extruded, moving the dimensional deviation from the outside of the forging to the inside, as shown in Fig. 61.

---

**Fig. 50** Photograph of the actual part. Note the severe fracture (dark area) and die underfill (below dark area) on the inside diameter (ID)

**Fig. 51** Flowlines for the forming of the component. Note that the free surfaces are very clear on both the inner and outer diameters

**Fig. 52** Point tracking the damage factor of two points on the surface of the component

**Fig. 53** After the redesign, the inside diameter is free of defects
Product Assembly

Staked Fastener Installation

The development of complex forming processes, such as self-penetrating fasteners, staked studs, and rivets, involves complexities over and above traditional forming operations due to the interactions of multiple plastic deforming bodies. In such cases, fastener installation is influenced by plastic strain (work hardening) induced during prior cold forming operations. Therefore, it is not practical to perform installation trials with machined blanks. The Fabristeel Corporation used computer simulation to develop their self-piercing mechanically staked fasteners for sheet metal parts (Fig. 62). The patented drawform stud was fully developed using simulation. The development process included forming the stud, the installation process itself (Fig. 63), and a pullout test (Fig. 64). Based on damage values in the sheet, the original design was modified to prevent fracture in the panel.

Breakaway Lock Development

Recently, process simulation has been used to develop an aluminum breakaway padlock (Fig. 65). Pull strength was a critical requirement for the intended application. Unlike a typical keyed padlock, this lock was developed to be a one-time-use item.

The lock was developed so that during installation, the bolt is torqued down until the head shears off at an undersized (recessed) diameter. At the installation, the end of the bolt should plastically deform the shackle, resulting in a permanent installation. The grade of aluminum, heat treatment, and geometry of the lock were the primary design variables. Common aluminum alloys were analyzed, including 6061 and 6062, as were different heat treatments for the shackle, bolt, and body. Shackle diameters of 5 and 6 mm (0.20 and 0.24 in.) were modeled to determine the effect of diameter on the pull strength.

The lock shackle was modeled as a rigid-plastic material, and for this portion of the analysis, the bolt was considered rigid. The bolt was twisted and subsequently pushed into the shackle (rotated and translated inward) until the tip of the bolt was fully engaged. The localized deformation can be seen in Fig. 66, where effective strain is displayed in the shackle.

The first question for this lock design was the mode of failure. Localized plastic deformation at the tip of the bolt could occur, as seen in Fig. 67, allowing the shackle to become dislodged. Alternatively, the shackle could unbend or neck, allowing the free end to be pulled out of the lock body. Because the deformation in both the bolt and the shackle was needed to accurately predict the failure, both were analyzed as plastic objects.

Pull tests were performed by assigning a constant (upward) velocity to a rigid object placed inside of the shackle. The bottom of the lock body was fixed using boundary conditions. The contact between the bolt and the shackle was considered, as was the interaction between the shackle and the lock body. It was shown from the simulations that both deformation modes (tip deformation and shackle unbending) occurred during this process. At the start of the pull test, the tip of the bolt plastically deformed, as seen in Fig. 67. After this initial deformation, however, the bolt remained structurally sound, and the shackle started to unbend. The unbending continued until the free end of the shackle pulled free from the lock base, as seen in Fig. 68. This bending was the primary mode of failure observed in all of the simulated pull tests, and it matched prototype tests conducted.

The failure mode and pull strength were studied for each design, as seen in Fig. 69. When using the same-diameter shackle, the lock made from 6061-T6 material demonstrated a higher pull strength than the lock made from 6062. Likewise, for the same material, the simulations showed that the larger the shackle diameter, the higher the pull strength. After this study, 6061-T6 material and a 6 mm (0.24 in.) diameter shackle were selected for the final lock design.

The trend in pull strength that was observed in the pull tests was quite intuitive. Simulation was not only able to confirm the trend, but it was also able to determine the amount of load that each lock could withstand.

The ability to determine how the lock would fail was the real strength of the FEM simulations in this example. Depending on the shackle material and diameter, it was conceivable that some locks would fail due to bolt tip deformation, while others would fail due to shackle unbending. Simulation was able to determine that for the materials and diameters studied, all locks would fail the same way.

Solid-State Welding Process

Inertia welding is a solid-state welding process in which the energy required for welding is obtained from a rotating flywheel. The frictional heat developed between the two joining surfaces rubbing against each other under axial load produces the joint (Ref 32, 33). One of the two components is attached to a rotating flywheel, while the other component remains stationary. During the process, the rotating component is pushed against the fixed one, causing heat generation through contact friction and consequent rise in the interface temperature. As the temperature increases, the material starts to soften and deform. At the final stage,
the flywheel stops as the inertial energy decays to 0, and both sides of the material near the interface get squeezed out of the original contact position and form a complex-shaped flash, as shown in Fig. 70. The quality of the welded joint depends on many process parameters, including applied pressure, initial rotational speed and energy, interface temperature, the amount of upset and flash expelled, and the residual stresses in the joint. Being able to accurately model the process is essential to understand, control, and optimize the process. After forging and machining, several aircraft engine disks can be joined together using this process. Figure 71 shows a cutaway view, with a close-up view of the weld, for a typical aircraft engine disk welding.

A typical comparison of predicted and measured upset versus time is shown in Fig. 72. From this figure, it can be seen that the characteristics of this process include three distinct stages: the upset rate is 0 at the initial stage; the upset rate is increasing first and then decreasing in the middle of the process; and the upset rate approaches 0 at the end of the process. In the first stage, the energy is used to raise the temperature on the contacting surface. The material starts to deform in the second stage. As the flash forms, the contacting surface area increases, the rate of upset starts to slow down, and the inertial energy is eventually consumed in the third stage.

Figure 73 shows a comparison of the predicted temperature at the end of the weld with a macrograph of the weld. There is very good agreement between the predicted temperature field and the observed heat-affected zone and also the flash shape and thickness.

It is an added challenge to weld two dissimilar materials, due to the differences in thermal as well as deformation behavior over a wide range of temperature. Proper process welding conditions are crucial to achieve the desired weld geometry and properties. As shown in Fig. 74, a compressor spool of two adjacent stages made of different materials is joined. Good agreement between the predicted and observed weld shapes is observed.

Optimization of Forging Simulations

In the past, FEM has been primarily used as a numerical tool to analyze forming processes. Decision-making to optimize a process is strongly dependent on the designer’s experience in an actual process and on numerical modeling. Ultimately, it would be very desirable to develop an optimization technique to achieve an acceptable die shape automatically.

In forging operations, for example, intermediate shapes are often used to ensure proper metal distribution and flow. The design of intermediate shapes, also called blockers and preforms, are of critical importance for the success of forging processes. In the following sections, optimization in 2-D and 3-D preform die design is presented.

Two-Dimensional Case

An axisymmetric example to demonstrate the potential application of design optimization is shown in Fig. 75.

In this example, the primary objective was to completely fill the finishing tools during the last operation. A numerical measure of this criterion was obtained by comparing the outline of the desired part and that of the actual forged part. It was also important to obtain homogeneous materials properties within the component; a homogeneous distribution of effective strain provided a satisfactory approximate criterion to achieve this goal.

The shape of a preform die was represented by B-spline curves. The design variables were the control points of B-spline or a piecewise linear curve.

An isothermal forging condition was used in the simulation. The flow stress of the workpiece was assumed to be in the form of \( \sigma = 1000e^{0.25t^{0.3}} \) (ksi). A constant shear friction factor of 0.5 was assumed, as well as a billet radius of 2.5 cm (1.0 in.) and a length of 3.8 cm (1.5 in.). The velocity of both the preform and the finish dies was 2.5 mm/s (0.1 in./s).

Figure 75(a) shows the forging process without using a preform die, and underfill was

![Radial Stress, Hoop Stress, Axial Stress](image)

**Fig. 56** The component stresses were evaluated on the original design to isolate the root cause of the failure. The dark colors represent tension and the light colors compression. Hoop and radial stresses (light color) are essentially compressive. The circle highlights a tensile stress in the axial direction, as shown with the arrows.

![Stress-Max Principal](image)

**Fig. 57** The contours of maximum principal stress on the redesigned insert are shown, with the dark colors representing tension and the light colors representing compression. This design resulted in an extended die life due to the carbide insert remaining in a compressive stress state throughout the process.

![Turbine disk](image)

**Fig. 58** The inside features of a turbine disk forging were undersized. The outside diameter (OD) was in tolerance. OD, inside diameter.
predicted, as indicated by the arrows. The initial guess of the preform die shape prior to the final forging is shown in Fig. 75(b). As indicated in the same figure, an underfill problem remained. Through the optimization iterations, the preform die shape evolved, as shown in Fig. 75(c). It took approximately seven iterations to resolve the underfill problem and to obtain the homogenized strain distribution, as shown in Fig. 75(d) Figures 75(e) to (g) show the distribution of effective plastic strain without preform die, with initial preform die, and with optimized preform die, respectively. It can be seen that the distribution of effective strain became more uniform through the iterative optimization process.

Three-Dimensional Case

Most forming processes are, in fact, 3-D. Tooling geometry is most likely prepared using commercially available computer-aided design (CAD) systems. In the CAD systems, nonuniform rational B-spline surface is a very popular way of representing the complex tooling geometry. In the FEM model, the geometry is generally represented by a set of polygons. At the current stage, the stereolithography (STL) representation is used as a vehicle between CAD and FEM to transfer the geometry definition. Parametric design and feature-based representation, which may be available in the CAD system (dependent on the methodology of the CAD system) during the designing process, are unfortunately lost at the end of the STL transformation. The FEM can only be used as a numerical tool to evaluate the given tooling design. Although the CAD and FEM can conceptually be used as a black box and integrated by a closed-loop optimization procedure, numerous 3-D FEM simulations will be needed to evaluate the sensitivity of design parameters. The procedure can be extremely computational, demanding, and impractical for daily use. Unlike the 2-D approach, being able to generate a reasonable initial design in order to facilitate the design process has been the near-term objective.

In a recent approach, preform design based on the so-called filtering method was used to determine an initial preform shape. For a given final part configuration, there are three steps to generate an initial preform shape (Ref 34–37):

1. **Digitizing process:** The final part geometry is imported into the system as a triangulated surface, in STL or other format. The digitizing procedure is first performed to convert the triangulated surface into a point cloud array.

2. **Filtering process:** The resulting surface will be passed through a filtering procedure to produce a smoothed shape. In the filtering procedure, the geometric domain will be converted into a frequency domain by using a Fourier transformation (Ref 38–40). By using the filtering function, high-frequency regions (sharp corners, edges, or small...
geometric details of a surface) will be removed. An inverse Fourier transformation will then be applied to obtain a smoothed surface.

3. **Trimming process**: The trimming procedure is used to control the boundary shape of a smoothed shape, so as to obtain a realistic initial preform shape.

An aluminum structural part, as shown in Fig. 76(a), is used to illustrate the aforementioned procedure. The overall dimensions of the part are 61.5 by 16.5 by 9.1 cm (24.2 by 6.5 by 3.6 in.). The filtered smoothed shape is shown in Fig. 76(b). After trimming, the designed initial preform is shown in Fig. 76(c).

In order to improve and eventually automate the design modification procedure, a systematic method to evaluate the material flow based on the FEM result must be developed. Because the material flow history is known from the FEM simulation, the defect location can be traced, and modifications can be made accordingly. To illustrate the overall procedure, the forging of the example part was simulated, as shown in Fig. 77.

In this simulation, only the upper part was simulated due to symmetry. Aluminum 6061 was selected as a workpiece material. The initial temperature was assumed to be 480 °C (900 °F) for the workpiece and 205 °C (400 °F) for the die. The total die stroke from the initial contact to the final position was 3.612 cm (1.422 in.). A flash thickness of 2.5 mm (0.1 in.) was used. The estimated maximum load was 6300 kbf. As shown in Fig. 77, the flash shape was not uniform in the areas indicated by arrows, and underfill was predicted, indicated by dashed lines.

To identify the region where modification was necessary, an ideal forged component with flash was defined first in order to compare with the simulation result. At the present stage, the ideal final forged part is defined in such a way that it has uniform flash width and does not have underfill condition. By comparing the ideal forged shape and the predicted shape at the reference step, the location of the underfill defect and the amount of additional volume can be identified. The material flow history information from a FEM simulation can be used to modify the preform shape by adding volume (if material is not sufficient) or by removing volume (if excessive flash was...
Another filtering procedure will be applied to obtain a smooth geometry. An example to illustrate the procedure is shown in Fig. 78. Figure 78(a) is the ideal forged component with uniform flash. The flash width is 25.4 mm (1 in.), and the flash thickness is 1.3 mm (0.05 in.) (upper part only). The predicted shape from the simulation at a stroke of 35.8 mm (1.41 in.) is shown in Fig. 78(b). The areas that need to be modified are shown in Fig. 78(c). Finally, the modified preform shape is shown in Fig. 78(d). In this example, an additional 0.6% volume was added to make the modified preform shape.

To validate the modified preform, another FEM simulation was performed. The new simulation result is shown in Fig. 79(a). From this figure, it is clear that the underfill defect is removed and the flash is more uniform compared to the initial design. The outer boundaries of the final product without flash, the predicted forging using the initial preform, and the predicted forging using the modified preform are shown in Fig. 79(b).

![Fig. 68](image1.png)

**Fig. 68** Beginning of the pull test (top). Below, unbending of the shackle (failure mode)

![Fig. 69](image2.png)

**Fig. 69** Load results for various pull tests

![Fig. 70](image3.png)

**Fig. 70** Inertia welding of two disk-shaped objects showing the initial geometry and the temperatures at the completion of weld. Note the flash expulsion at both the inside and outside diameters. This is a two-dimensional model shown in a three-dimensional view for easy visualization. Source: Ref 33

![Fig. 71](image4.png)

**Fig. 71** Cutaway view of a typical inertia weld of aircraft engine disks and a close-up view of the welded region. Source: Ref 33

![Fig. 72](image5.png)

**Fig. 72** Measured and predicted upsets as a function of time for a typical inertia welding process. Note the good agreement between the predicted and the measured upset rates. Source: Ref 33

![Fig. 73](image6.png)

**Fig. 73** Comparison of the predicted temperature (left) at the completion of the weld with a macrograph (right) showing the heat-affected zone. Source: Ref 33
Conclusion

Growing together with computing power, FEM has evolved rapidly and is making great contributions to conventional metalworking technologies. The thousand-years-old technology has quickly evolved in the last 20 years in such a way that the FEM method is now routinely and successfully used in a wide range of bulk forming applications, including material flow, defect prediction, microstructure/property predictions, heat transfer, forming equipment response to the workpiece, die stress and deflection, and so on. With continuous improvement in computing resources and demand from industry, it is expected that future developments will likely be in the following areas:

- **Optimization:** Currently, FEM users play an important role in solving a specific process design by performing sensitivity analysis of multiple FEM models. Convergence speed is highly dependent on the user’s understanding of the FEM model and the actual process. It is believed that optimization techniques will be extremely helpful in finding an optimal solution systematically and quickly on the computer. Currently, optimization has been used with success in conjunction with FEM to determine not only the heat-transfer coefficient but also the material parameters and friction factor, the interface heat-transfer coefficient between two contact bodies (Ref 41) and in the preform design to remove the underfill flow defect in 2-D application (Ref 9). In order to carry out the 3-D optimal shape design, a closer integration of computer-aided design/computer-aided engineering (FEM) is necessary and is expected in the near future. For a true optimal product and process design, the costs that are associated with the production process, such as tooling, material, equipment, heating (when necessary), inspection, and inventory, should also be characterized and considered in the model.

- **Computer modeling of microstructural features:** Much work has been carried out to predict microstructural features, such as grain size and phase transformation, in the past by using phenomenological approaches. Texture is another important microstructural feature that will affect mechanical properties. Crystal-plasticity modeling has made excellent progress in predicting texture (Ref 42). As the computing environment continues to
improve, the crystal-plasticity method coupled with the FEM code (CPFEM) to describe the texture/anisotropy evolution during the forming process will become realistic and practical. The texture model can be further integrated with the microstructural model so that transformation, grain-size evolution, and grain-size effects can be taken into account throughout the various stages of thermal-mechanical processes. The framework established for CPFEM and the predicted crystallographic texture will also facilitate further investigations of mechanical properties, fracture, and life prediction.

With the continuously improving understanding of material response to thermal-mechanical changes and proven success in metalforming applications, it is believed that FEM will make significant contributions to the postforming operations, such as heat treatment and machining. In due time, it should be possible to analyze the entire manufacturing process, from the cast ingot, open-die forging, closed-die forging, heat treatment, and machining. This will allow designers to include residual stress and grain flow in their product design and application analysis. The benefits from this should include reduced product life-cycle cost and increased safety margins on critical service components.

ACKNOWLEDGMENTS

In the past 20 years, many of our colleagues have worked on this subject and made process simulation successful not only for academic research but also for practical industrial use. It is not possible to properly thank everyone who has made significant contributions to the FEM methodology for metalforming applications. The authors would like to thank and acknowledge Professor T. Altan at The Ohio State University and Drs. S.I. Semiatin at the U.S. Air Force Research Laboratory, S. Srivatsa at General Electric Aircraft Engine, and K. Sawamiphakdi at The Timken Company. We thank them for their work and continuous guidance, support, and encouragement. The authors would also like to thank the entire staff from Scientific Forming Technologies Corporation for their enthusiasm and diligent work to meet all the technical challenges throughout the years. The authors would also like to thank Mr. Jeffrey Fluhrer for editing this manuscript.

REFERENCES

18. F. Ikuta, K. Arimoto, and T. Inoue, in Proc. Conf. Quenching and the Control of Distortion, 4–7 Nov 1996 (Cleveland, OH)


41. J. Yang, DEFORM User’s Group Meeting, Fall 2002