

SOME RECENT SIMULATION CASES OF COLD FORMING WITH DEFORMTM-3D

B. K. Chun, J. Fluhner, M. Foster, G. Li and W.-T. Wu
Scientific Forming Technologies Corporation, Columbus, Ohio, U.S.A.

Abstract

Tremendous progress in numerical analyses of metal forming processes has been achieved in the recent years and the finite element simulation has become a powerful tool in many industries. Trends of development of simulation technology are summarized. Some recent applications of the DEFORMTM-3D system to a wide range of cold metal forming and associated processes are presented, including mesh-to-mesh contact cases and a thread rolling example.

Introduction

In the last decade, industrial acceptance of metal forming modeling with the finite element method (FEM) has increased so rapidly that its industrial applications have become routine as the technology continues to spread.

The advent of simulation technology in metal forming could not have come at a better time when computer hardware has experienced dramatic price reduction and speed increases. PCs have become so powerful as well as affordable that their popularity has surpassed that of workstations. The dream to carry a laptop with a simulation running on it to the plant, conference room or onboard a plane has come to fruition.

The finite element formulations for various material models have become mature [1, 2]. The updated Lagrangian approach still dominates most of the forming applications, but the ALE approach also finds increasing applications. Tetrahedral elements using a mixed formulation and hexahedral elements are found most suitable for large plastic deformation. The wider use of tetrahedral elements is supported by mesh generators, which now can successfully create tetrahedral meshes with great complexity and with large contrast in element sizes to model a variety of processes in the forming industries. All of these have improved the ability to analyze complex three-dimensional forming processes accurately and efficiently and have opened up additional avenues for the application of simulation. The commercial programs integrate the finite element technology into comprehensive software systems, complemented by improved graphical user interfaces, thus making themselves very user-friendly and flexible. It is not surprising that many forming engineers and researchers find such tools essential.

Mature applications of metal forming simulation include the prediction of the material flow, die fill and loading condition [3]. The information thus obtained can be used for process design, defect prediction or analysis, and cost analysis. The scope of simulation continues to broaden as the recent trends are discussed as follows.

The combination of modeling individual processes have evolved into the development of progressions and tool and die design [4]. Optimizing the progression design using process simulation is superior to trial-and-error on the shop floor. In the development of metal forming progressions, the designer balances 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 behavior of the workpiece. In conjunction with the progression development, a multiple operation template can be designed to integrate a series of individual operations into a complete job to facilitate the simulation management, so the modeling work can be carried out as an automated processes.

New finite element applications to more sophisticated and complicated forming processes are continuously being explored. These continuous challenges have enriched the repertoire of the simulation technology.

In addition, process simulation capabilities have been expanded to some associated areas beyond forming modeling. Die stress analysis has been shown to be a very cost-effective use of simulation. Die costs have been estimated at 5 to 15% of the cost of sales. It is noted that the dies are typically subjected to a severe operating environment due to the high interface pressure experienced in the manufacture of cold-formed parts. In warm and hot forming, these effects are compounded by extreme temperatures. Similarly, die wear analysis can also be conducted using the FEM results.

In order to achieve a desirable combination of microstructure, mechanical properties, residual stresses and dimensional accuracy in the final product, a heat treatment process that involves several heating and cooling cycles may be employed. Each cycle could involve complex thermal boundary conditions, e.g., air/fan cooling, oil/water quenching, and furnace/induction heating. The material responds to the complex coupling of stress, microstructural and chemical (carburizing and nitriding) conditions over a wide range of temperatures. Designing a heat treatment process sequence is complex and has generally been done based on experience, as was the case with forming prior to process simulation. With the advent of heat treatment simulation, it is now possible to detect, understand and correct potential heat treatment problems early in the manufacturing process cycle through the use of simulation.

Another active field of finite element research is the modeling of machining processes. This includes the numerical analysis of chip formation and the part distortion after material removal.

In the following sections, the use of DEFORM™ as a design tool for selected cold forming applications is presented.

Mesh-to-Mesh Contact

The contact problem is most challenging in the finite element analysis of forming processes. If contact phenomena cannot be appropriately modeled the accuracy of the simulation results will be compromised. As a mesh in the finite element analysis represents a deforming workpiece, the contact algorithm should:

- Enforce the normal constraint to a contact node so as to prevent it from penetrating into the other side it is in contact with;
- Apply the tangential friction to the contact node in the direction opposite to the relative movement;
and
- Release the node from the contact status once it is detected of being pulled by the other side.

There are two types of contact in the finite element technology. Mesh-to-die contact deals with the contact of a workpiece with a non-deforming rigid die. Mesh-to-mesh contact, on the other hand, is

concerned with the contact between two meshed parts of the same object or two different objects. Of the two contact types, the mesh-to-die contact is well established and therefore will not be discussed here.

The mesh-to-mesh contact models the interaction between the two meshed parts in the contact area. It in turn can be subdivided into: (1) contact between two deforming workpieces, and (2) contact between two parts of a single workpiece (self-contact) and (3) coupling for rotational (or cyclical) symmetry.

In a static structural analysis of simple geometry, if we can create a mesh system in such a way that there are always node-to-node contact pairs connecting both sides in the contact area, their coupling can be readily implemented without much difficulty. However, in the metal forming applications, when the part geometry is complicated, it is quite difficult to generate a mesh that meets this requirement. Even if such a mesh is generated, the two contacting parts are prone to slide along each other when both sides are subject to large deformation. Naturally, the node-to-node coupling cannot be maintained. Generally, the node-to-segment or segment-to-segment coupling has to be established. Here a segment is a surface element edge in 2D cases or a surface triangle or quadrilateral in 3D cases.

Being able to handle the mesh-to-mesh contact, DEFORM™-3D has been used to analyze many multi-deforming object cases, self-contact cases and cases with rotational symmetry. Three cold forming examples are presented here to illustrate the contact treatment.

Self-Contact

Fig. 1 is a heat sink in an electronics product made of Al 5052. A shallow rectangular recess 1.5 mm deep is required on a plate part 2.35 mm thick. The original manufacturing method was to machine the recess, which is quite inefficient, so stamping is used instead. If the recess is formed from a solid part, to remove the excessive material requires considerable force and the part will experience large distortion. An improved design is to punch an elliptical hole in the center together with blanking. Then the recess is coined, while the hole is closing up. To justify this idea and to determine the optimal size and shape of the initial hole, the simulation was conducted during the process design.

Fig. 2 shows the meshes used at the beginning and ending of the recess coining. Due to the symmetry, only a half of the part is actually modeled. The two small round indentions are omitted for simplicity.

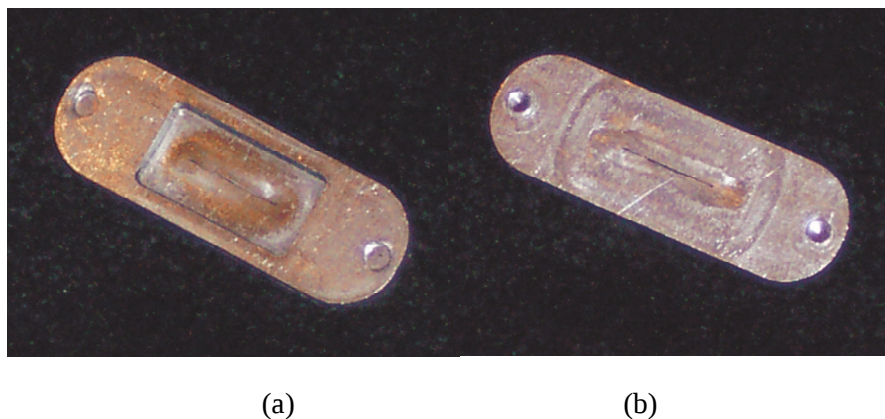


Fig. 1 Heat sink: (a) top view and (b) bottom view.

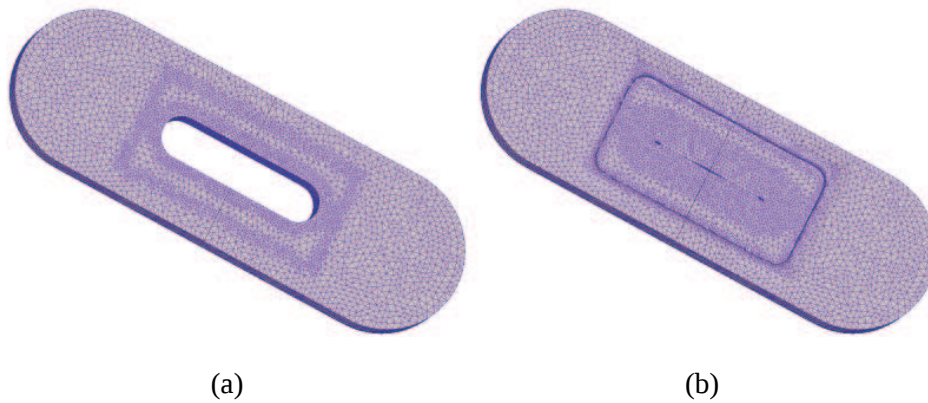


Fig. 2 FEM meshes: (a) beginning and (b) near ending.

Fig. 3 shows the evolution of the shrinking hole. The upper series is the flownet, through which the material flow can be observed. The lower series are the top view of the part at different stages. From a certain stage in the process, the sidewall of the hole starts touching itself. It is at this stage where the self contact capability is tested. Eventually the contact length grows until the hole becomes a seam. It should be mentioned that without a powerful mesh generator, this simulation could not have been completed.

It is interesting to note from the strain distribution in Fig. 4 that large strains are accumulated at both ends of the seam. Actually the two ends of the seam is closing up more slowly and there are two tiny holes left in the part even the whole seam is tightly formed, which can be seen on the real part in Fig 1. The simulation has correctly reproduced the reality in detail.

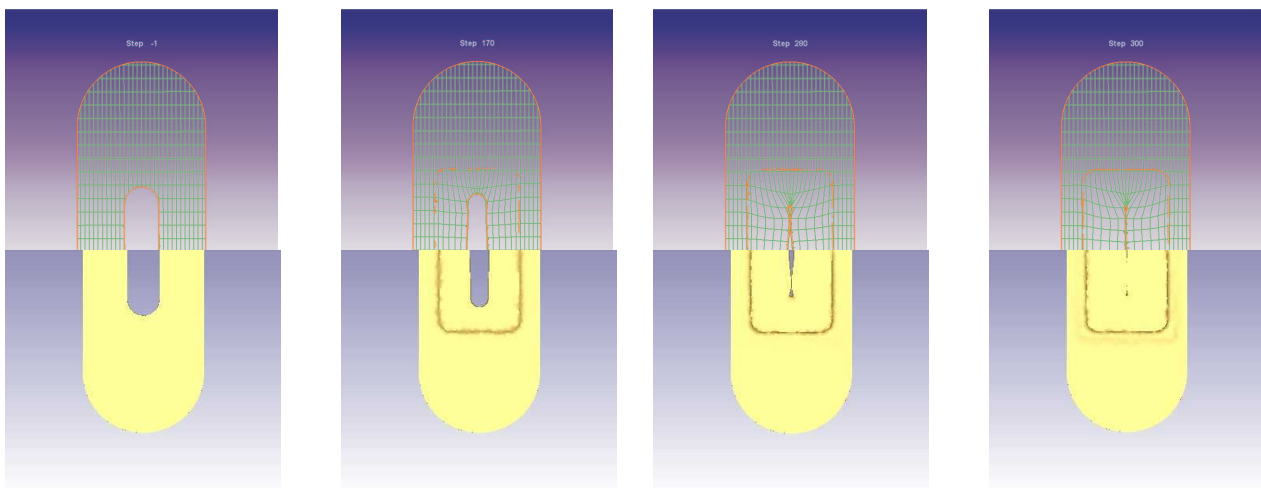


Fig. 3 The evolution of the central hole to a closed-up seam in simulation:
upper half -- flownet; lower half – shaded view.

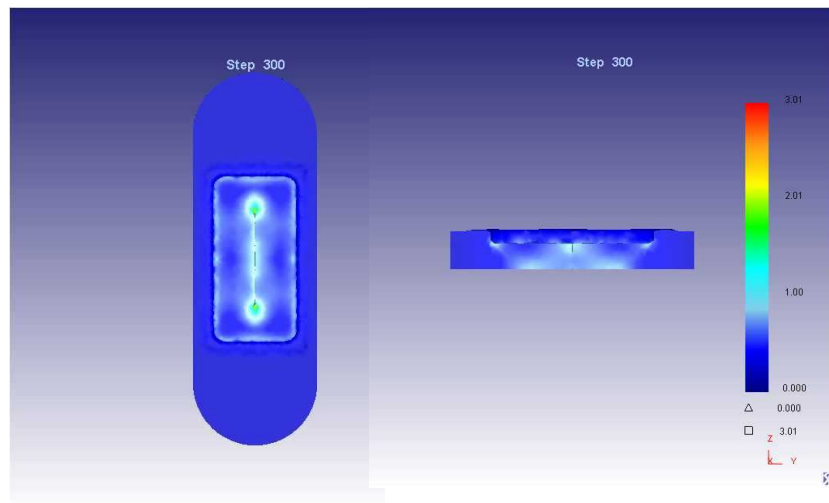
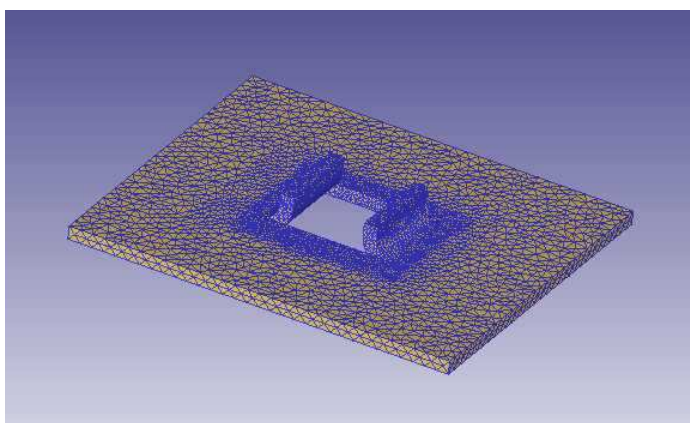


Fig. 4 The strain distribution: top view and on the central cross-section.

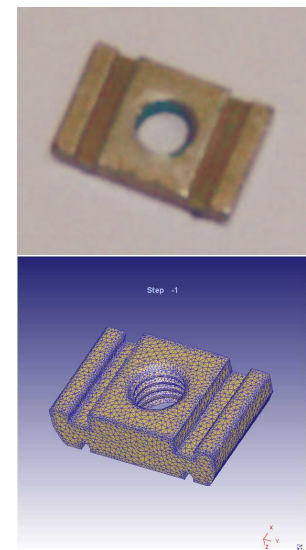
Multiple Deforming Objects

The second example is a nut assembly (Fig. 5), in which two stamped parts are to be joined together in an assembling die. The original nut and plate are shown. Fig. 6 illustrates how the two meshed objects are put together at the beginning of simulation.

When the stamping begins, the punch pushed the nut downward, thus upsetting the two upturned tongues on the plate to fill up the slots on the nut until the two parts form a tight joint.



(a)



(b)

Fig. 5 FEM meshes of the initial parts to be joined together: (a) the plate and (b) the nut (top - picture of the real part; bottom – FEM mesh)

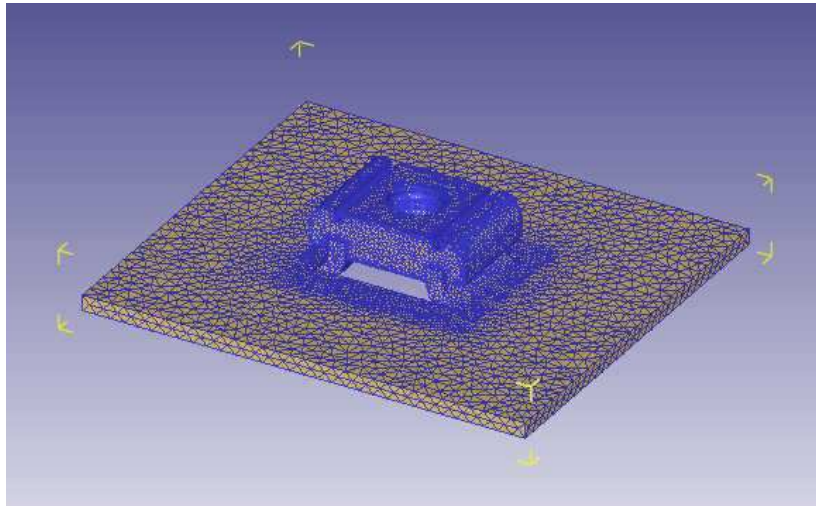
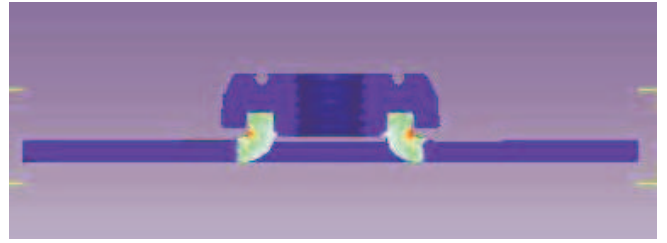


Fig. 6 Initial FEM meshes of the two parts put together.

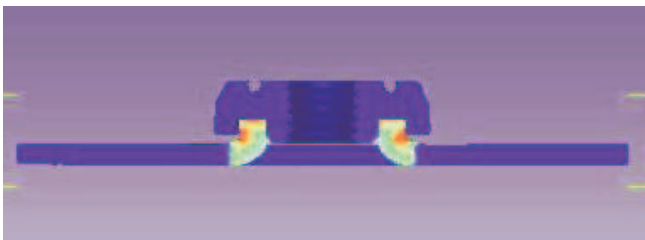
Fig. 7 is the effective strain pattern in both parts at different stages of assembling. It is visible in Fig. 7-a that at the beginning, there are gaps between the tongues of the plate and the slots on the nut. When the nut is pushed down by the punch, the tongues first begin to bend downwards. At Step 30



(a)



(b)



(c)



(d)

Fig. 7 Strain pattern at various stages:
(a) Step 20, (b) Step 30, (c) Step 35 and (d) Step 42.

(Fig. 7-b), the tongues have fully filled the slots. The localized deformation of the two tongues (Fig. 7-c and -d) is desirable, as it makes them expand sideways and thus a tight joint with the nut can be possible. It is clear that this simulation would not have been possible without the mesh-to-mesh contact treatment.

Rotational Symmetry

Rotational symmetry is a special case of self-contact. When a part repeats itself cyclically, we can model a small “pie” segment of it to reduce the complexity of the simulation, or to increase the accuracy by using finer elements. This is achieved by coupling the two sides of the segment model, i.e., any corresponding two points on both sides of the pie should have the same velocity with the respect to its central line.

The helical gear in Fig. 8 is a good example for rotational symmetry. Historically helical gears were machined at a high cost from round stocks. A cold-formed process to develop net shape helical gear components was developed by the Yamanaka Engineering Corporation in Japan. The gear with eight teeth (Fig. 8-b) is extruded from an AISI 1035 round stock of 28 mm OD with a fillet at the bottom edge (Fig. 8-a). The regular full model is simulated for comparison.

The rotational symmetry takes only one-eighth of the material (Fig. 9-a) with two slicing planes 45 degrees apart. As the material is squeezed into the extrusion die, the slicing planes automatically become twisted surfaces, following the spiral curve. The deformation at different stages is shown in Fig. 10. The quality and stability of the coupling is important in simulation. This can be examined by graphically assembling the segments into a whole part (Fig. 11). On the mating surfaces, there should be no gaps or overlaps. Good agreement between the strain patterns obtained from the one-eighth and full models at the final stage of deformation is presented in Figs. 12 and 13, respectively.

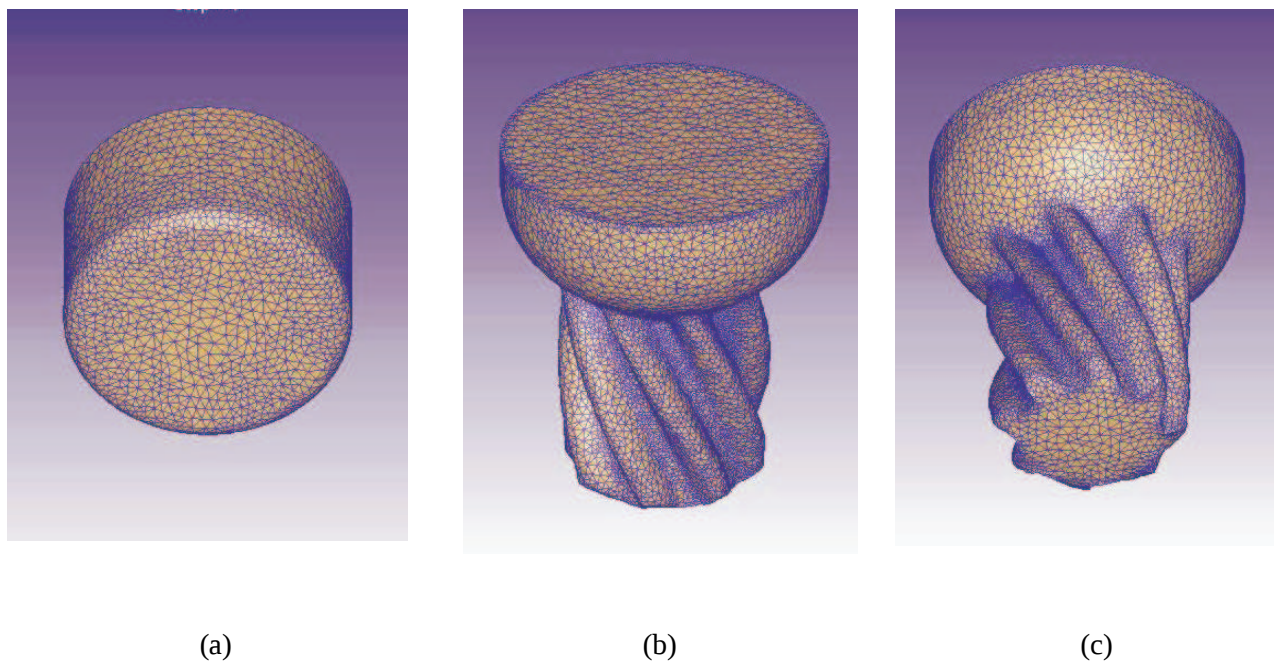


Fig. 9 The extrusion of a helical gear – full model:
(a) initial billet; (b) and (c), final stage.

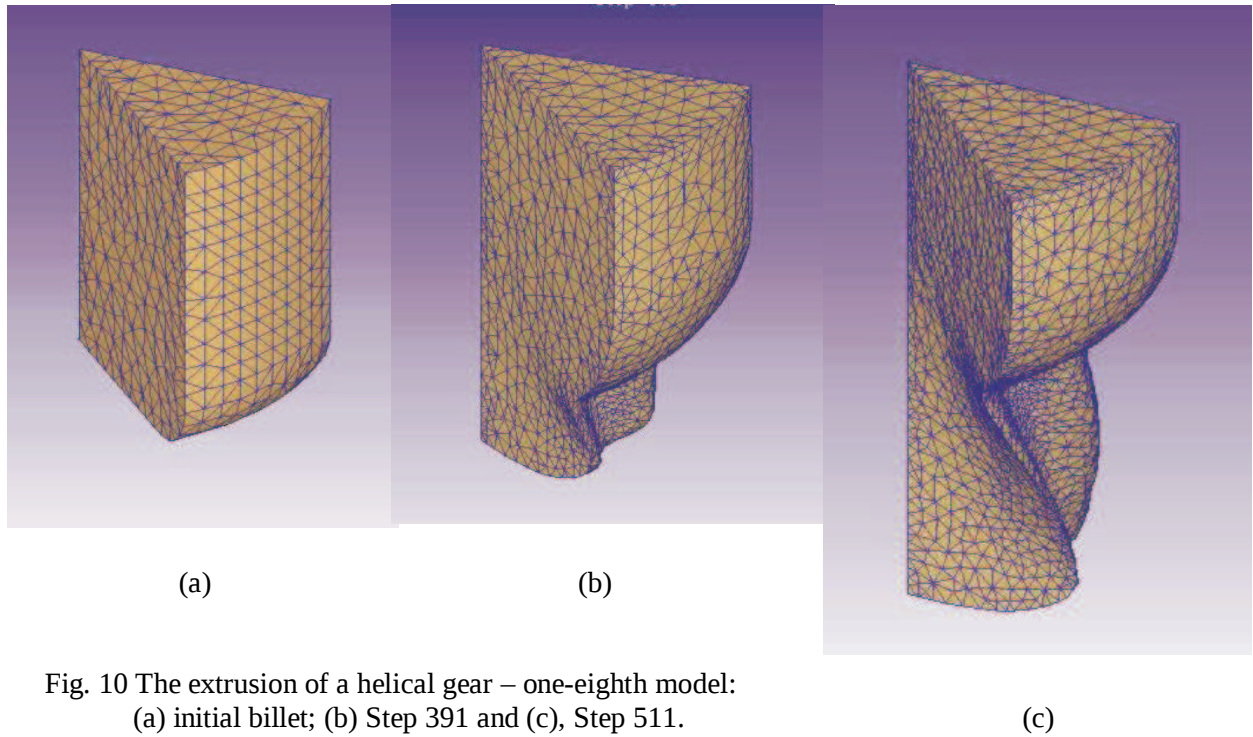


Fig. 10 The extrusion of a helical gear – one-eighth model:
 (a) initial billet; (b) Step 391 and (c), Step 511.

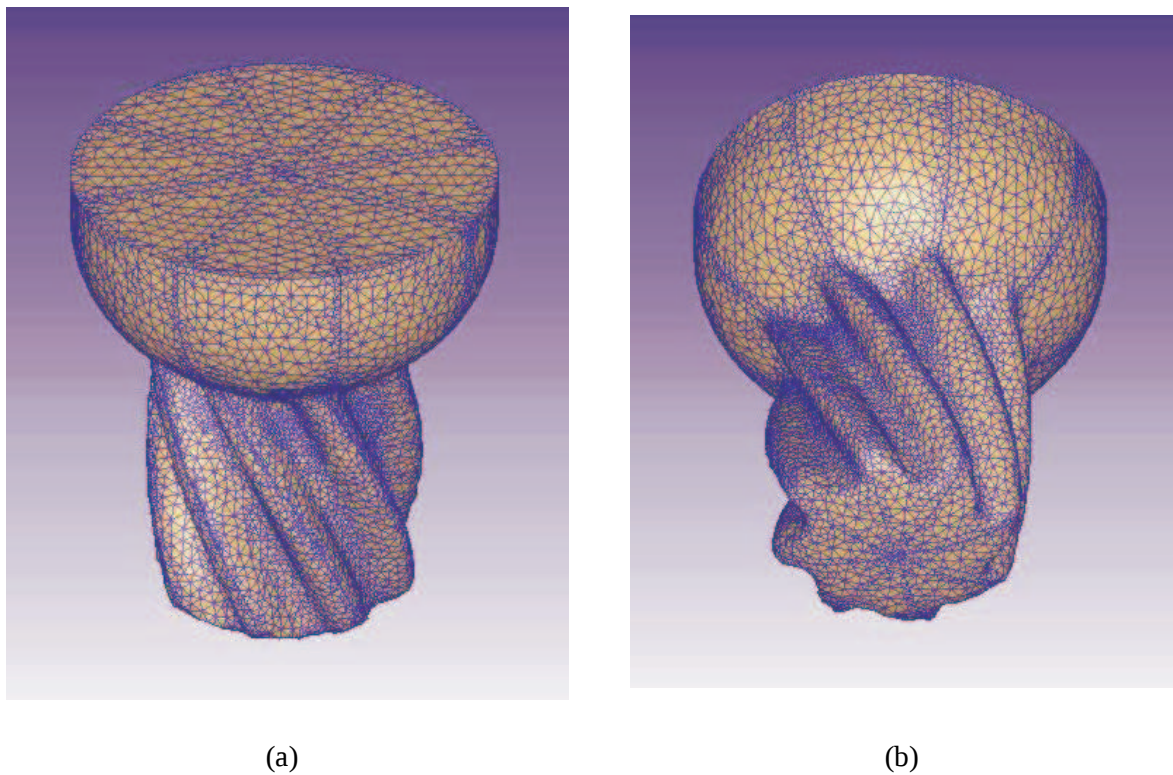
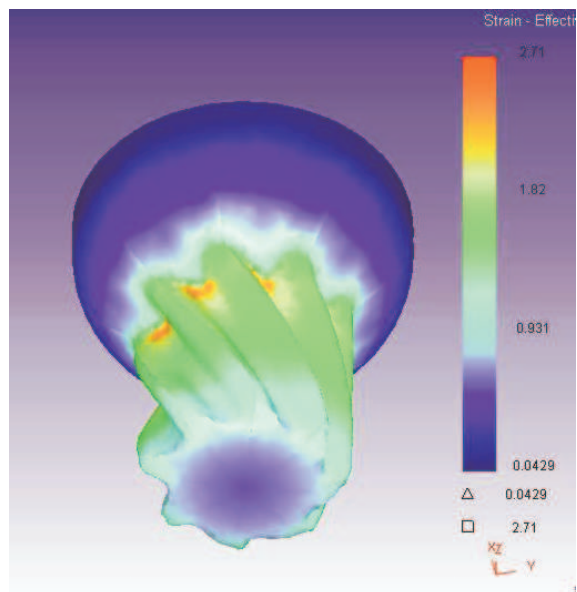


Fig. 11 The assembled image of the repeated one-eighth model at the final stage



(a)

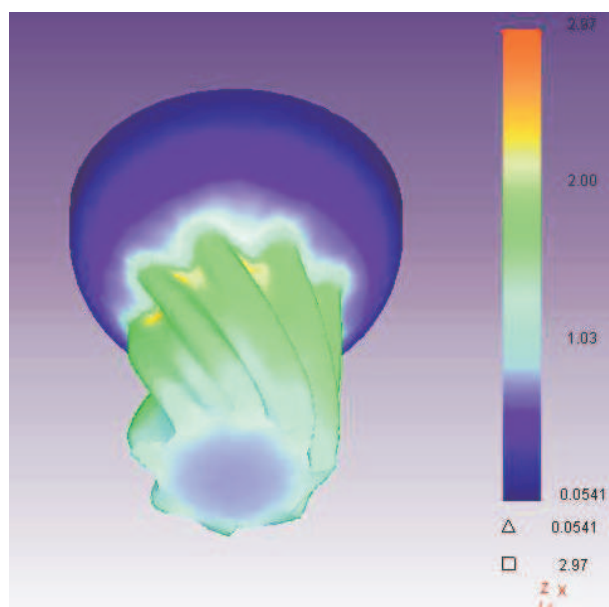


(b)

Fig. 12 The strain distribution, repeated image of one-eighth model at the final stage.



(a)



(b)

Fig. 13 The strain distribution, full model, at the final stage.

Thread Rolling Simulation and Stress Analysis of the Die

Threaded fasteners are used in most mechanical assemblies. Machining or rolling generally forms the threads. Turning or grinding processes can also be used to produce machined threads. Machined threads are cut into the screw or bolt by removing the material. On the other hand, threads can be cold formed on the blank using hardened steel dies.

Thread rolling has several major advantages over thread machining:

- The deformation involved in the rolling process work hardens the threads, resulting in increased strength.
- Rolled threads have improved fatigue resistance. The rolling process puts the surface in a state of compression, making it more difficult for crack formation and propagation to occur. The grain structure in a rolled thread is continuous, as opposed to the cut grains found in a machined product.
- Rolled threads typically have superior surface finish and a lower cost relative to machined threads.

The major challenges for such a simulation are simulation speed, meshing and handling of the rotational movement. The considerable contrast in mesh densities over different parts is necessary, as the threads need to be constructed of very fine elements. This increases the total problem size and computing time. The total turn around time is multiplied by the many revolutions of the bolt needed to complete the thread formation. As a result of continuous development, the thread rolling process now can be modeled in DEFORMTM-3D, and can be used to investigate thread formation, underfill and stress in the threading dies.

The images shown are from a thread rolling simulation that was run on a desktop PC in approximately one day. The simulation was set up based on a translational threading rolling machine. The blank, coming from the header, is placed between the two threading dies, and then one of the dies moves in translation while the other die remains stationary. The friction between the blank and the dies causes the blank to spin. As the blank is rolled, the threads are formed (Fig. 14).

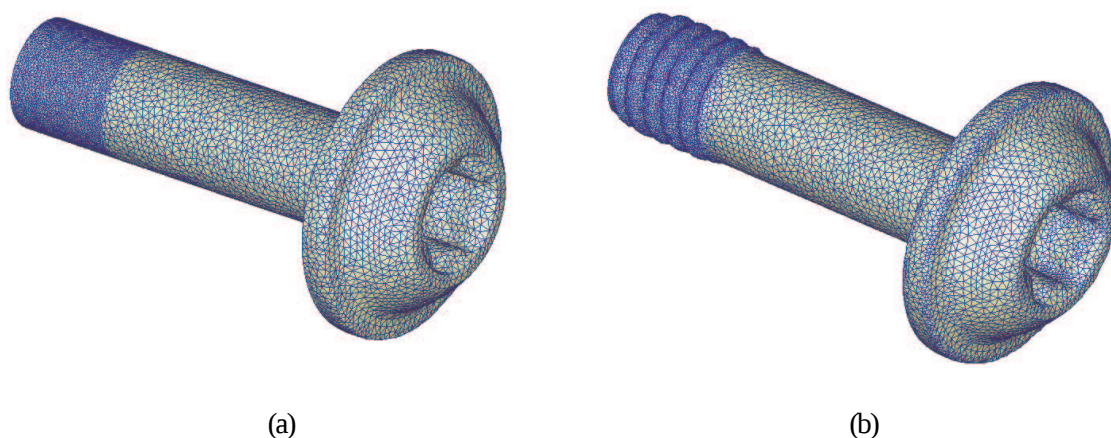


Fig. 13 The FEM meshes are shown (a) at the beginning and (b) at the end of the thread rolling simulation.

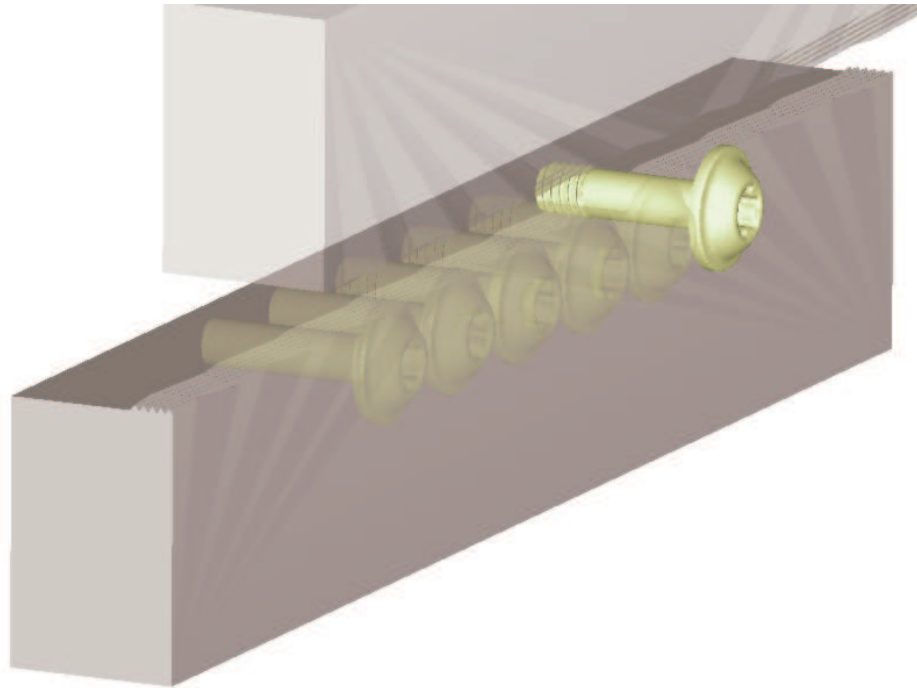


Fig. 14 The process of thread rolling.

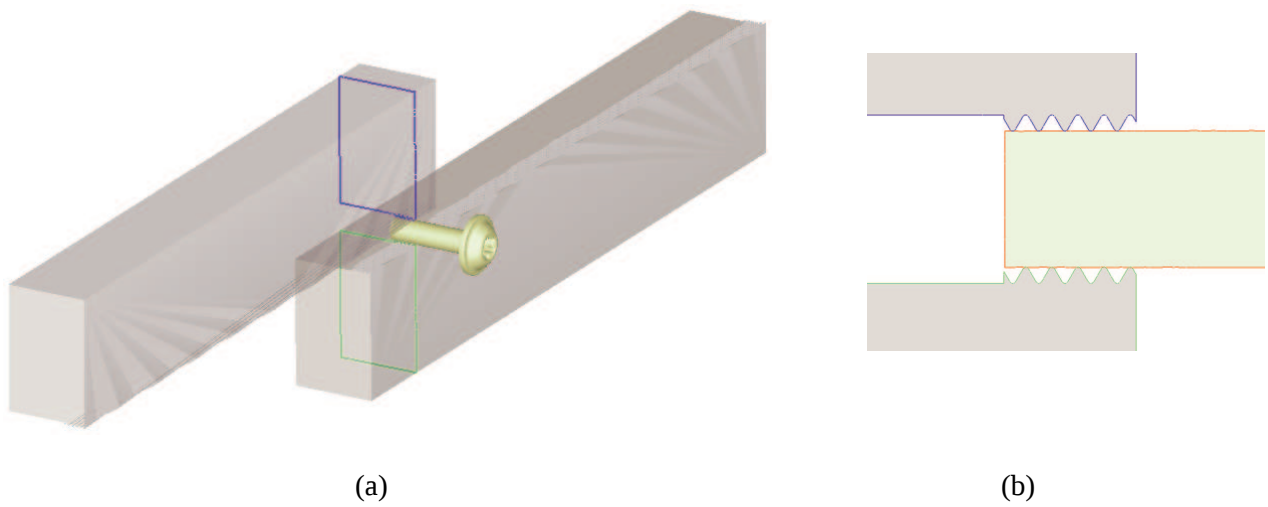


Fig. 15 Die matching before the simulation starts:
(a) die position; and (b) examination on a slicing plane.

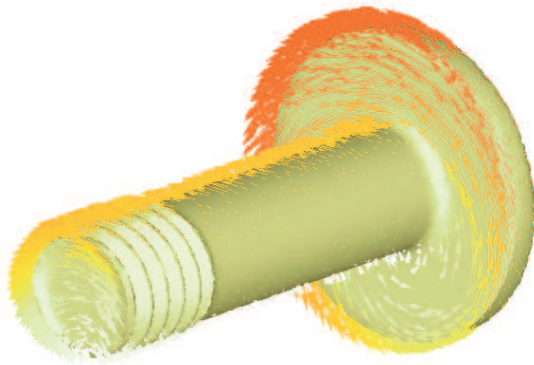


Fig. 16 The velocity vector plotting showing the part movement.

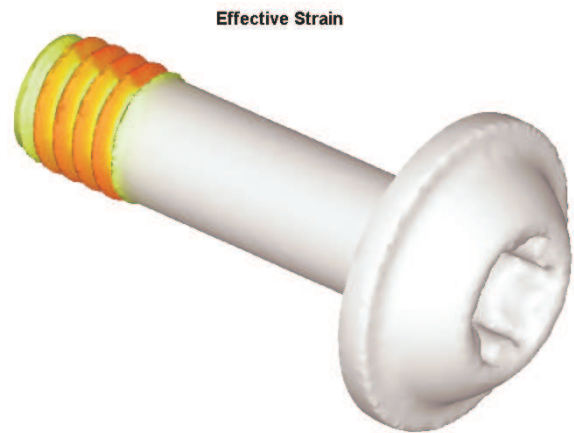


Fig. 17 The effective strain distribution.

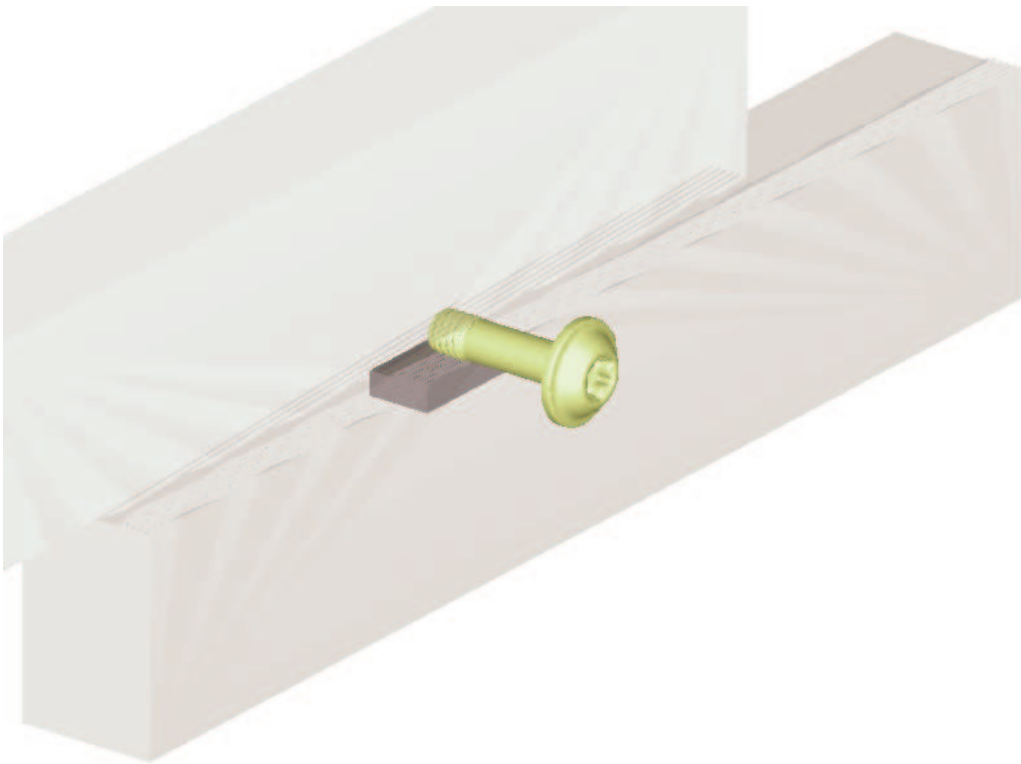


Fig. 18 A subdomain is used in die stress analysis.

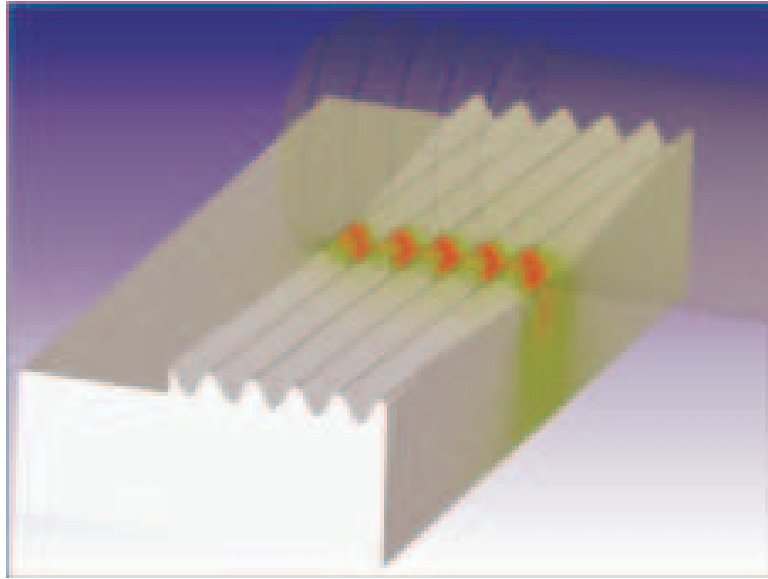


Fig. 19 The effective stress (red is higher) is shown on the thread rolling die near the end of the forming process.

Before a thread rolling simulation can be run, the dies need to be matched. Position the dies in the starting position, and then slice them down the center of the bolt. Thread root (top die) needs to correspond to thread tip (bottom die). The velocity plotting is shown in Fig. 16 and the strain distribution is shown in Fig. 17.

In addition to the thread filling study, the elastic stress analysis is often needed for the threading dies. The threading dies used in the rolling simulations are quite large compared to the bolt itself. When performing stress analysis on the dies, a subdomain of the original die geometry was used (Fig. 18). The effective stress of one of the dies is shown in Fig. 19.

Summary

Recently, the finite element-based software has become an integral part of process and progression design and research in the forming industries to analyze and optimize the metal flow and to minimize the die stress and tool wear. The industrial acceptance of this computer-aided engineering system could not have achieved without the significant progress in the finite element theory and practice and the rapid advances in computer technology.

In this paper, a few selected cold forming simulations by using DEFORMTM-3D are presented. These examples demonstrate the important part of finite element contact technology for the multiple deformation bodies problems, and capability to handle large rotational problem such as thread rolling. Although the FEM has proven itself to the forming industry, the development of finite element technology will always be challenged by the newer and wider scope: the entire manufacturing processes.

References

- [1] Kobayshi, S; Oh, S; Altan, T: Metal forming and finite element method, Oxford Univ. Press, 1989.
- [2] Wagoner, RH; Chenot, JL: Metal forming analysis, Cambridge Univ. Press, 2001.
- [3] Altan, T; Ngaile, G; Shirgaokar, M: State of cold forging technology in global competition, in Siegert, K, ed.: New Developments in forging technology, Fellbach, Germany, 2003.
- [4] Walters, J; Wu, WT; Hermann, M: The simulation of bulk forming processes with DEFORMTM, ibid.
- [5] Wu, WT; Jinn, JT; Yang, JB; Oh, JY; Li, G: The finite element method and manufacturing processes, Proceedings of the 6th ESAFORM Conference on Material Forming, Salerno, Italy, 2003.