RECENT APPLICATIONS OF PROCESS SIMULATION IN FORGING AND FORMING

Taylan Altan, Adam Groseclose, Soumya Subramonian and Xi Yang

Center for Precision Forming
The Ohio State University, Columbus, Ohio USA

Summary

Process Simulation using FEA is routinely used in industry for optimizing forming processes and designing dies. Researchers expand the usual application of FE simulation to investigate new and relatively complex processes in forging as well as sheet metal forming. This paper discusses recent advances using case studies on die wear in warm forging, estimation of fracture in hot forging, distortion in rolled and heat treated rings, and improvement of tool life in blanking. The results of these selected case studies illustrate the capabilities as well as limitations of process simulation in metal forming.

1. INTRODUCTION

The use of Finite Element (FE) based simulation in metal forming industry has become a widely accepted integral part of the forming process design to analyze and optimize the metal flow and conduct die stress analysis before conducting trials.

The main goal of simulation in process design is to reduce part development time and cost while increasing quality and productivity. For instance, in cold forging, process simulation can be used to develop the die design and establish process parameters by a) predicting metal flow and final dimensions of the part, b) preventing flow induced defects such as laps and c) predicting temperatures (warm forging operations) so that part properties, friction conditions, and die life can be controlled. Furthermore, process simulation can be very beneficial in predicting and improving grain flow and microstructure, reducing scrap, optimizing product design and increasing die life. With asymmetrical complicated parts, the use of three-dimensional FE simulations is a must. During recent years, 3-D simulation has also gained widespread acceptance in industry. Improvements in computer algorithm efficiency and user interface development have resulted in simulation becoming both robust and practical in a wide range of applications [1].

FE simulations are generally used in sheet metal forming for formability and springback analysis during die development or troubleshooting an existing process. Process parameters are optimized by (i) predicting metal flow, temperature and final dimensions of the part for given die design, and (ii) predicting and preventing defects such as excessive thinning and wrinkling. Despite recent improvements to enhance the application of FE simulation for process modeling, the successful application of FEA depends mainly on the reliability of the required input parameters such as material properties and friction and, the user's ability to transform the physical problem to the FE problem and interpret the simulation results.

2. APPLICATION OF PROCESS SIMULATION

The accuracy of FE simulation depends on the reliable input data namely, (a) CAD data of the die and billet or blank geometry, (b) in case of sheet forming force applied on the sheet metal by cushions/springs/pads, etc during the forming process, (c) flow stress of the deforming material as a function of strain, strain rate and temperature in the range relevant to the process being analyzed, and d) friction characteristics at the interface between the deforming material and the dies.

For cold and hot forging simulation it is necessary to have data on flow stress, i.e. true stress/true strain, in function of temperatures and strain rates that exist in the actual process. The cylinder compression test is predominantly used for this purpose [2]. In sheet metal forming, to determine the flow stress, the industry standard uniaxial tensile test and the biaxial hydraulic bulge and dome tests are commonly used. The bulge and dome tests require advanced testing and analysis methods. However, these tests allow to obtain the flow stress data at larger strains (up to 0.5 or 0.7) than the tensile test (up to 0.2-0.3). Nevertheless, the tensile test is still the only practical test to obtain the anisotropy of sheet materials, a parameter that is essential in conducting reliable simulations for certain materials, for example Ti alloys [3].

In sheet metal forming, (1) the stamping lubricant tester (SLT), (2) modified Limiting Dome Height (MLDH) test, (3) ironing test and (4) deep drawing test are used to estimate the friction coefficient (μ) of the lubricant used, and to evaluate commercially available lubricants for use in production. The deep drawing process itself can be used as a test to evaluate the performance of the lubricant. This test emulates the interface conditions that exist in production, both in the flange and in the punch. In this test, circular cups are drawn from a blank to a fixed draw depth but with different blank holder forces. The higher is the achievable blank holder force, the better is the performance of the lubricant in the test. Test measurements, namely punch force, draw-in of material in the flange; cup wall thinning can be used along with FE simulation of the test to quantitatively estimate the coefficient of friction for the lubricant used in the test [3].

In determining a) the coefficient of friction and b) comparing the performance of various lubricants, the reliable and practical tests in cold forging are double cup extrusion test in cold forging and the ring test in hot forging. In conducting these tests, however, it is necessary to emulate the process conditions (die temperature, billet temperature, ram speed and contact time) that exist in real forging operations [2,3].

3. DEFORMATION AND WEAR IN WARM FORGING DIES

3.1 Methodology

In warm forging, die life is affected by abrasive wear and plastic deformation and may be shortened considerably due to thermal softening of the die surface caused by forging temperature and pressure. In a recent study, a methodology was developed for estimating abrasive die wear and plastic deformation in warm extrusion of a transmission shaft, using a tempering parameter [4]. This methodology for estimating plastic deformation and abrasive wear in forging dies consists of:

a) Conducting finite element analysis of the steady state temperature distribution of the dies, using multiple operations using DEFORM-2D, a commercial FE software package

b) Estimating the plastic deformation from the results of the steady state temperature distribution of the dies

c) Extracting the wear profile form the measured die surface profile (obtained by CMM) by separating the plastic deformation from abrasive wear

d) Determining the abrasive wear parameters (K and a), in order to predict abrasive wear on the die surface

In order to predict the abrasive wear and plastic deformation, subroutine programs for the commercial code DEFORM-2D were developed to a) calculate the tempering parameter from the calculated steady state temperature distribution of the die, b) modify the flow stress data based on the calculated tempering parameter, and c) compute the amount of wear with the advanced Archard's wear model. Finally, the wear parameters (K and a) are mathematically optimized (to give the minimum difference between the predicted wear and actual wear profiles), and then verified by comparing the predictions with the experimental results.

3.2 Results

Using the results of simulations, a predicted wear profile is determined and compared with the measured profile, after compensating for plastic deformation (Figure 1). The prediction matches the actual wear very well, in location as well as in magnitude. This study helped to develop a useful methodology for the application of FEM to analyze and predict abrasive wear and plastic deformation in forging dies. It was found that abrasive wear is not the only form of die failure acting upon the die at elevated temperatures. Plastic deformation also occurs due to high temperatures and pressures during extrusion. Therefore, it is impractical to predict wear without first predicting and accounting for the plastic deformation. After the wear and plastic deformation are estimated, FEM can then be used to optimize the forming process. This could be through changes in material (workpiece or die), press speed, operating temperature (of the workpiece or die), or die shape. As a result, costly experimental work and die tryouts can be minimized.

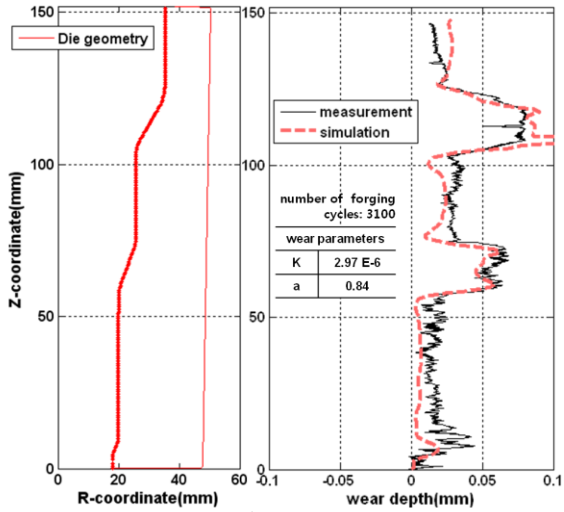

Figure 1: Predicted wear from FE simulation versus CMM measured wear compensating for the plastic deformation for Extrusion Insert (3100 Cycles)

4. Fracture in Hot Forging

In hot forging high corrosion resistant materials surface or internal fracture may occur. FE simulation, along with an appropriate damage or fracture criterion may help to predict and avoid such failures.

4.1 Fracture in forging engine valves

The precision hot forging of engine valves consists of a) extrusion of a heated billet to a preform, and b) coining of the valve head. In forging valves from a Cr based Ni alloy, fracture was observed at the coining stage, Figure 2. Experiments and FE simulations indicated that fracture (or cracks) could be eliminated by increasing the diameter of the billet in the head of extruded preform. In the simulations the Cockcroft and Latham (C&L) fracture criterion, originally developed for cold forging, was used to calculate the "damage factor". The valves of damage factors, calculated during the coining stage, are given in Figure 3. This study indicated that even the relatively simple C&L damage or facture criterion can be useful in investigating fracture in hot forging.

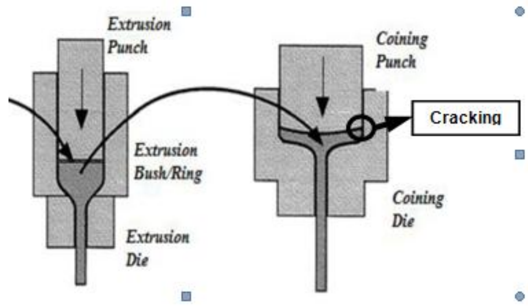

Figure 2: Extrusion of a preform (left) and coining of the valve head (right) in forging engine valves

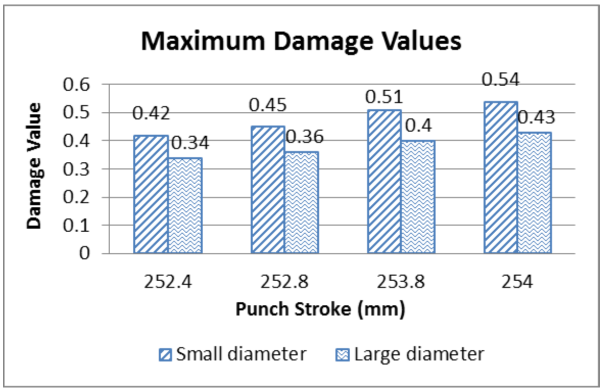

Figure 3: Maximum damage values, estimated for coining preforms with different head diameters

4.2 Fracture in open die forging of round billets

A study is in progress for estimating surface fracture in hot forging of billets form Ti-6 Al-4V. Experimental data and simulations indicate that in the present case metal flow is mainly determined by ram speed and contact time between hot billet (815C) and cold dies (Room temperature) [5]. FEM allows the prediction of local strains, strain rates and temperatures for various forging conditions (press speed, friction, billet and die temperatures) although the effect of microstructure and its variation during forging can only be estimated indirectly, by determining the change of the flow stress with strain, strain rate and temperature. The estimation of fracture in forging is a challenging problem and will require future research.

5. Distortion in Rolled and Heat Treated Rings

After being rolled at forging temperature (1200C), most alloy steel (AISI 4140) rings are heat treated as follows: a) after rolling the rings are arranged individually or in stacks of four to six units and normalized at 925C for two hours,

b) air cooled, c) austenitized at 950C with the same stack used during normalizing, d) submerged into the quench tank where the cooling rate is fast enough to generate a martensitic microstructure and hardness distribution in the ring material. The ring dimensions, considered for FE simulations were: Outer Diameter = 1,296 mm, Linear Diameter = 1,164 mm and Height = 163 mm [6].

Convection, conduction and radiation are the heat transfer mechanisms during air cooling. Experiments and FE simulations indicate that, although ring stacking will cause non-uniform cooling, the observed distortion is not significant due to slow cooling rate. However, quenching a stack of rings in a tank with agitated solution results in non-uniform heat transfer at various locations of each ring. Non uniform cooling along with the volume changes due to microstructure evolution results in geometrical distortion of the rings, Figure 4. The progress made in this study includes:

- Different steps of heat treatment (up to quenching) have been simulated in a commercial FE code in order to predict ring distortion and distribution of residual stresses.
- According to FEA results, air cooling will not create any significant distortion (ovality).
- Heat-transfer variation during quenching as a function of temperature, tank and stack location, and quenchant agitation is the key factor in calculating distortion, hence the importance of correctly modeling the heat-transfer coefficient.
- Through FEA, distortion and residual-stress distribution have been predicted assuming certain quenching conditions.

The results of the study indicate that it is very difficult if not impossible to eliminate the ring distortion in quenching. Thus, the mechanical straightening of distorted rings, already used by several companies, may be the most practical and inexpensive method to eliminate distortion. However, a physics-based methodology that will optimize the mechanical straightening procedure could be developed to save time and trial and error in obtaining acceptable tolerances in concentricity of heat treated rings.

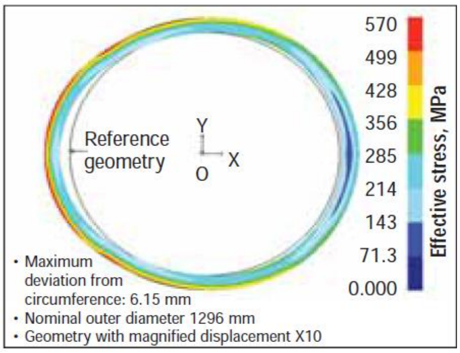

Figure 4: Geometrical distortion and residual stresses after FE simulation of heat treatment

6. Improvement of Tool Life in Blanking

FE simulation is very useful to study the influence of process and die parameters (punch force geometry, punch corner radius, punch/die clearance, punch velocity and stripper (blank holder) design and force) on (1) blanked edge quality, (2) blanking force, and (3) tool stresses.

Blanking is a very high deformation forming process with a significant temperature increase in the sheet especially when thick and high strength materials are blanked. In the deformation zone of the sheet the strains may reach values as high as 2-3. This also translates to high strain rate which can be anywhere between 10^3 sec^{-1} to 10^5 sec^{-1} depending on the blanking velocity. The temperature in the sheet can also go up to 300°C depending on the thermal conductivity, strength and thickness of the material. Figure 5 shows an FE model with the details of the deformation generated using the software DEFORM 2D.

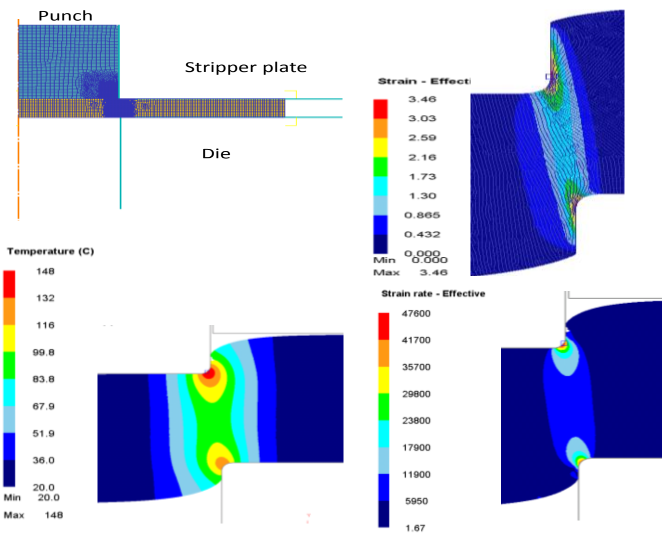

Figure 5: (from top left clockwise) FE model setup; strain distribution; strain rate distribution; temperature distribution in the sheet during blanking of 0.2mm thick C51100 sheet

Flow stress curves for materials to be input into simulations are generally obtained by conducting uniaxial tensile tests or biaxial dome test or bulge test. These methods give the flow stress of the material for strain values less than 1 in most cases. Very high strain rates of 10^4 - 10^5 are not very commonly achieved in these tests either. Hence, the common practice is to extrapolate the curves to higher strains by fitting an equation, assuming that the material continues to follow the stress-strain relation. In this study, blanking is investigated as a potential test to obtain flow stress data for large strains and strain rates simultaneously and a methodology is proposed [7]. A combination of blanking experiments and simulations is used to determine flow stress and is explained in the following steps.

- Obtain blanking force-stroke curve from experiments conducted at low speed (quasi-static condition)
- Conduct blanking simulation using extrapolated flow stress curve obtained using uniaxial or biaxial test.
- Obtain the force F_s, average stress σ_s, average strain ε_s, average strain rate $\acute{\varepsilon}_s$ and average temperature τ_s in the sheet at small intervals 'i' of stroke.
- Compare the experimental ($_{Fe}$) and simulated force ($_{Fs}$) at each step 'i' of stroke
- At each step 'i', if $F_s \neq F_e$ → $\sigma_{s\ new} = \sigma_s*(F_e/F_s)$
- Average (of all steps 'i') strain rate for the process $\acute{\varepsilon}_{speed}$ is calculated. Flow stress curve for $\acute{\varepsilon}_{speed}$ is obtained.
- The procedure is repeated with different blanking speeds corresponding to different strain rates to obtain flow stress data for different higher strain rates.

6.1 Effect of punch-die clearance upon punch force and stresses

The results of FE simulation, using actual die/punch geometries, indicated that as expected the punch corner stresses decrease with increasing punch/die clearance. Thus, it is possible to decrease the probability of punch corner chipping by appropriate punch/die design and clearance [7].

6.2 Effect of punch/die clearance on tool life

Experiments were conducted by Högman [8] to study the influence of punch-die clearance on the punch wear along the corner radius of a rectangular punch shown in Figure 6.

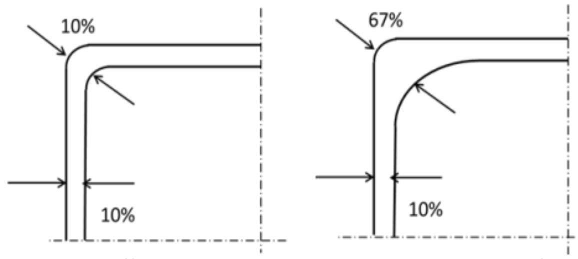

Figure 6: Different clearances around the corner radius of a rectangular punch used in [8]

The sheet material used in this study is Docol 800DP, 1mm thick and the tool material used was Vanadis 4 with a hardness of 60 HRC. The tests were conducted with 10% and 67% punch-die clearance. The corner radius of the punch chipped after 45000 strokes in the case of 10% clearance. The punch used in the 67% clearance test lasted 200000 strokes without chipping. FEA simulations of the experiments showed that the maximum punch stress increased beyond the ultimate compressive stress of the material of ~2200 MPa in the case of 10% clearance, while it remained below 2200 MPa in the case of 67% clearance.

This study shows that using variable punch-die clearance gives a significantly less and more uniform wear on the punch compared to using a uniform punch-die clearance. It also shows that correlating punch stresses from simulations to punch wear in experiments is a good approach to select optimum punch-die clearance.

7. Summary and Conclusions

This paper reviews briefly some of the recent applications in forging simulation, conducted at Center for Precision Forming (CPF – www.cpforming.org) and Engineering Research Center for Net Shape Manufacturing (ERC/NSM – www.ercnsm.org). The studies discussed include: a) plastic deformation and abrasive wear in warm forging, b) fracture in hot forging, c) distortion in rolled and heat treated rings, and d) improvement and tool life in blanking. These studies illustrated several challenges to and limitations of FE simulation. These challenges include a) obtaining mechanical properties of forging dies at high temperature and in function of heat treatment, b) statistical variations in die wear, observed in tests, c) selection of a fracture criterion in hot forging, d) estimation of heat transfer coefficient during quenching of large rolled rings, e) estimation of flow stress values at very large strain and strain rates. Despite the difficulties associated with FE simulation, results are found to be useful to assist in making process improvements.

References

[1] Ngaile, G., Altan, T., “Computer Aided Engineering in Forging”, Proceedings of the 3rd JSTP International Seminar on Precision Forging, Nagoya, Japan, March 14-18 (2004)

[2] Altan, T., Ngaile, G., Shen, G., “Cold and Hot Forging – Fundamentals and Applications”, ASM International (2005) ISBN 0-87170-8051

[3] Altan, T., Tekkaya, A.E., “Sheet Metal Forming – Fundamentals, Processes and Applications”, (2 Volumes), ASM International (2012), ISBN 978-161503-844-2

[4] Changhyok C., Groseclose, A., Altan, T., “Estimation of plastic deformation and abrasive wear in warm forging dies”, J.M.P.T., v. 212 (2012), pp. 1742-1752

[5] Semiatin, S.L., Groetz, R.L., Shell, E.B., Seetharaman, V., Ghosh, A.K., “Cavitation and Failure during Hot Forging of Ti-6Al-4V”, Metallurgical and Materials Transactions A, v. 30A, (1999), p. 1411

[6] Gonzalez-mendez, J., Duarte daSilva, A., Jiang, X., Altan, T., “Distortion in Rolled and Heat Treated Rings”, Forging Magazine, February (2012), p. 12

[7] Subramonian,S., Altan,T., Campbell,C., Ciocirlan,B., “Determination of forces in high speed blanking using FEM and experiments”, Submitted to CIRP (2013).

[8] Högman,B., “Punching tests of ehs- and uhs- steel sheet. Recent Advances in Manufacture & Use of Tools & Dies”, October 5-6, (2004), Olofström, Sweden.