Advanced Simulation Software is Essential for Fastener Manufacturing

by:
Nick Biba
Micas Simulations Ltd.
Oxford, UK
www.qform3d.co.uk

Sergie Stebunov
QuantorForm Ltd.
Moscow, Russia
www.QForm3D.com

Tom Ellinghausen
Forge Technology, Inc.
Woodstock, IL, USA
www.ForgeTechnology.com

Cost-effective numerical simulation is an important process for ensuring a fastener manufacturer's competitiveness.

Numerical simulation is one of the most effective ways to reduce the cost and development time for forming technology and tool design, and it is a vital tool for competitive fastener production. Usually, simulation models of forming processes are based on Finite-Element Method (FEM). QForm 7 is the first metalforming simulation software that is based on a hybrid approach that combines the advantages of Voronoi Cells (VC) and Finite-Element Method (FEM).

The solution obtained by Voronoi Cells has much less dependence on the mesh distortion that inevitably happens when simulating large deformations using the Lagrange approach. It also allows for significantly faster simulations with more accurate results than any other purely FEM code. QForm software models the forming processes and accurately predicts the actual material flow and temperature distribution as well as contact traction and die deformation even for the most complicated cases. QForm works on PCs or multi-CPU workstations and allows for the optimization of material flow and die stress state for fastener forming technology development and optimization without the need for expensive preproduction trials.

The program has a clear and self-explanatory graphical interface that allows a user to set up a new simulation in less than a minute. Advanced users have access to sophisticated facilities to specify any kind of material models, local boundary conditions and controlled motion of any number of tools and any number of deformable bodies (including bimetallic, multiple layers billets or composites). Users can easily program User Defined Functions that allow for the inclusion of user-defined criteria into the analysis of fracture prediction, product properties and to extend the tool life. The program can simulate mechanically and thermally coupled problems with thermo-elastic-plastic deformation of the tools. Thus complex assembled pre-stressed tooling sets can be included in a simulation. The deformation of even such a complex tool set is coupled with the material flow to achieve a realistic simulated shape of the product.

The speed of simulation is quite fast. Typical simulation time for most 2D cases is less than a minute and most 3D simulations typical for fasteners can be completed in just a few minutes. A unique automatic and adaptive mesh generator works effectively without the need for user intervention for even the most complex shapes typical for metal forming including cases of self-contact areas in folds and laps, very thin flash, tiny burrs and even chip formation. Since the mesh generation is automatic, even novice users are guaranteed very high quality results.

Simulation in Fastener Manufacturing

Fastener manufacturers produce a wide variety of products and generally produce them in huge quantities. Manufacturers must be able to develop and deliver new product very quickly to satisfy the urgent demands of customers. In addition to typical bolts and nuts, there is an increasing demand for specialized new products such as self-installed nuts and bolts or different kinds of blind rivets that provide easier installation and better performance when installed. Simulation of the forming process is an essential tool to quickly meet these market demands and to make the development of such products faster. The simulation should also be implemented not only to the manufacturing process, but to fastener installation and analysis of performance in pull or torque tests.

During installation, both the installed piece and the base material are subject to deformation that should provide secure fixing of the product. The deformation of the installed fastener must be strictly controlled to avoid excessive shape deterioration. For this purpose, simulation software like QForm 7 provides indispensable assistance to determine the proper shape of the fastener and selecting material of the product as well as the installation tool.

Let us illustrate this simulation approach of forging and subsequent installation of a blind nut. This nut is produced in several operations and the ribs are formed in the finish operation as shown in **Figure 1**. The simulation shows the formation of the ribs as well as any possible laps.

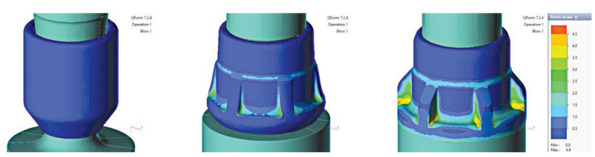

Fig. 1 — Forging of a blind nut.

The simulation of the installation operation shows the deformation of both the sheet of metal and the nut and their final configuration (**Figure 2**). The purpose of this analysis is to be sure that the nut is properly fixed and can carry the required pull load and torque. It is also very important to be sure that the nut will not be subject to excessive deformation that may deteriorate its shape especially on the thread surface.

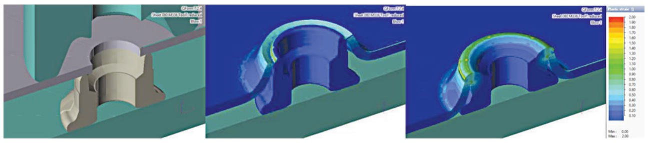

Fig. 2 — Installation of a blind nut with center position.

With the help of simulation, it is also possible to see how the sheet deforms and if it has any folds, laps or buckling (**Figure 3**).

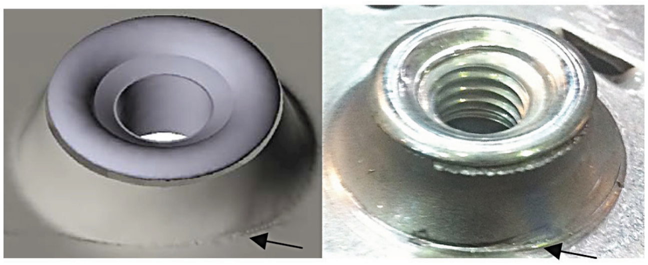

Fig. 3 — Installed blind nut: the simulation (a) and actual installation (b). The arrows show the lap in both cases.

Tool Life in Fastener Manufacturing

Figure 4 shows an example of a cold heading operation of an aluminum rivet. When produced in a solid die block, the highest equivalent stress exceeds the yield stress of the tool material in the fillet of the upper die and plastic deformation occurs (**Figure 4** left, point 1). The evident solution is to make an assembly die with an insert to reduce the stress concentration in this point as shown in **Figure 4** right.

The stress on die contact surface is still very high in the points 1, 2 and the efficiency of the modification is not clear. Thus in both variants the dies are subject to low cycle fatigue and it is necessary to estimate the expected amount of forging blows before the die failure.

The comparison was done for the points with maximum value of mean stress (points 1, 2 on **Figure 5**). The distribution of the expected number of cycles (**Figure 5**) shows considerable influence of the tensile stress state on tool life (see **Figure 5** left). In the other area there is a compressed stress state (see **Figure 5** right). Thus the program identified these two critical points and they were taken into consideration. The table in **Figure 6** contains the summary of the parameters required for the expression (2) and the results of calculation.

Even though the maximum equivalent stress reduction is not that big (about 13%) the strain and the stress distributions have a greater influence. Particularly we observe that the mean stress changes from tensile to compressive (see stress indicator) and there is a considerable reduction of the plastic strain. According to expression (2) it results in increasing of the expected number of die cycles by more than five times from 1756 to 9692.

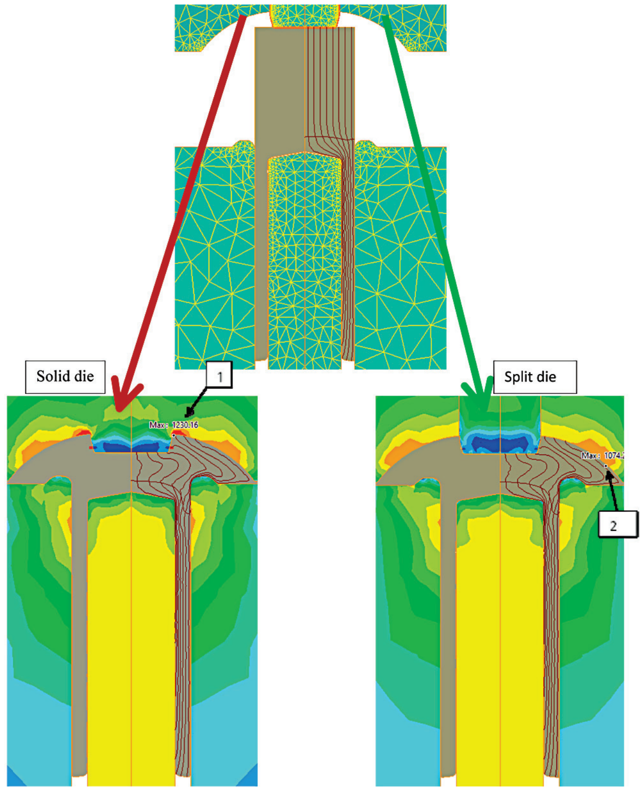

Fig. 4 — Effective stress distribution in the upper solid die (1) and assembly die (2). Initial positions of the dies (upper), finish position of the die (below). On the left – solid upper die, on the right – spit upper die.

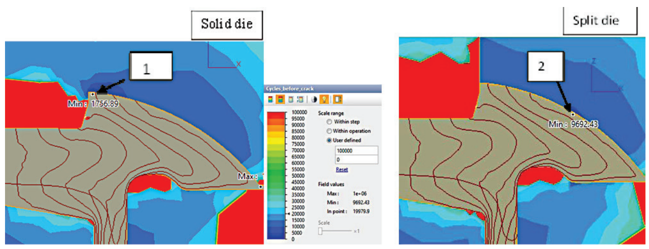

Fig. 5 — Expected number of cycles in the upper solid die 1756 (left) and in the assembly die 9392 (right).

Point	Maximum equivalent stress, MPa	Maximum plastic strain $\Delta\varepsilon$	Stress indicator	Critical strain ε_{cr}	Power index a	Expected amount of cycles
1	1230	0.01	+0.23	3.1	1.85	1756
2	1074	0.004	-0.43	5.9	1.77	9692

Fig. 6 — The number of forging cycles: solid die - 1756: split die - 9692.

Damage Analysis

In order to carry out qualitative tool damage analysis, the tool components must likewise be considered as elastoplastic components. Using a qualitative damage analysis, critical zones in forging dies may be identified, even if no material-specific data is available. If required, macromechanical damage models must be implemented into the FE system to be used. Macromechanical damage models include the deformation history in their analysis in order to determine local material damage independent of material parameters. This can be achieved by the integration of a stress function over the equivalent plastic strain. According to this approach, a damaging plastic work is calculated which is introduced into the material. The most well-known approaches are the damage criteria according to Latham & Cockcroft and Ayada. Below is the Latham & Cockcroft criterion:

$$D_{L\&C} = \int \frac{\sigma_1}{\bar{\sigma}_f} d\varepsilon_p$$

Where $D_{L\&C}$ - macromechanical damage value; σ_1 - 1st principal stress; $\bar{\sigma}_f$ effective stress; ε_p – plastic strain increment.

The macromechanical damage value is bigger than zero in areas of the loaded tool where the plastic zones will appear. It is typical loading for cold and warm forging when we have very high values of contact stress from the forged material.

In the case study below the reasons of the tool failure during cold forging of the guide bush from the aluminum alloy AA2014 was investigated. The forging was performed in an assembly die in a hydraulic press (**Figure 7**).

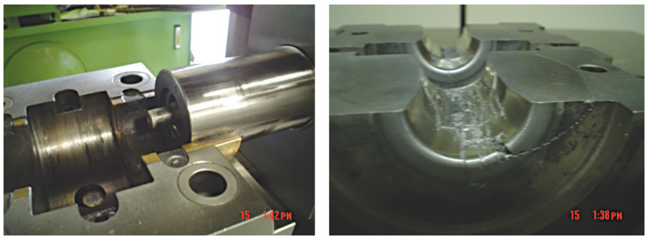

Fig. 7 — Failure of the steel insert of an assembly die during cold forging of a guide bush from Al alloy AA2014.

Figure 8 show the results of simulation of the failure of an insert. Latham & Cockcroft criterion shows a dangerous zone in the tool where plastic zones appear due to loading. The yield strength of the tool steel BOHLER W360 is 1200 Mpa after heat treatment to HRC39. Simulation of the forging and tool loading has indicated that the maximum stress reached is about 1300 MPa and is higher than the yield strength. The danger value of Latham & Cockcroft criterion can be determined in experimental studies of tensile or upsetting loading samples of tool steels.

Assembled Die Simulation & Improvement

With the help of QForm, it is possible to analyze complicated assembly die designs when the tool set consists of many pieces. For example, in **Figure 9** we can see two different designs of the die insert with and without a horizontal split. The results of the simulation of a cold aluminum cap forging when the die insert consists of single piece are shown in **Figure 9a**. The concentration of the effective stress is observed in the area of the fillet. Practical experience has shown us that premature die fracture will happen in this area and that variation of the interference in the shrink fitting assembly does not help to reduce this stress concentration.

This time the solution was found by using a split insert that allows "release" of the axial tension component in it.

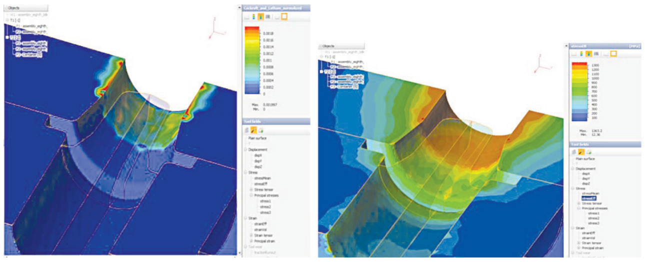

Fig. 8 — Results of simulation of the failure of an insert. On the left is the Latham & Cockcroft criterion in an assembled die. On the right - effective stress distribution.

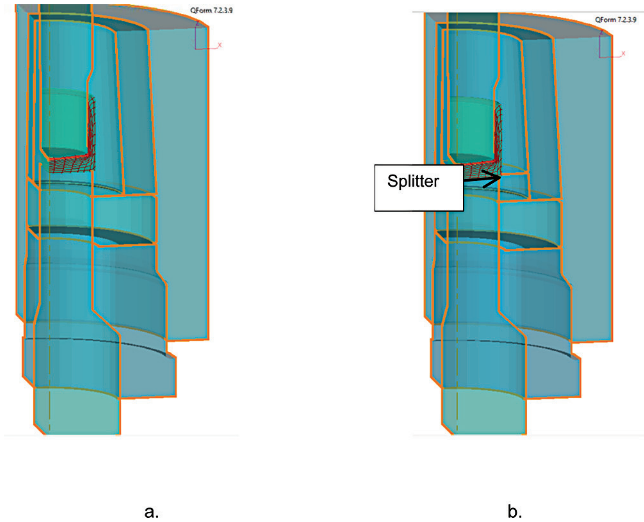

Fig. 9 — Forging of a cap using two variants of the insert design: single piece design (a), two piece insert with splitter(b).

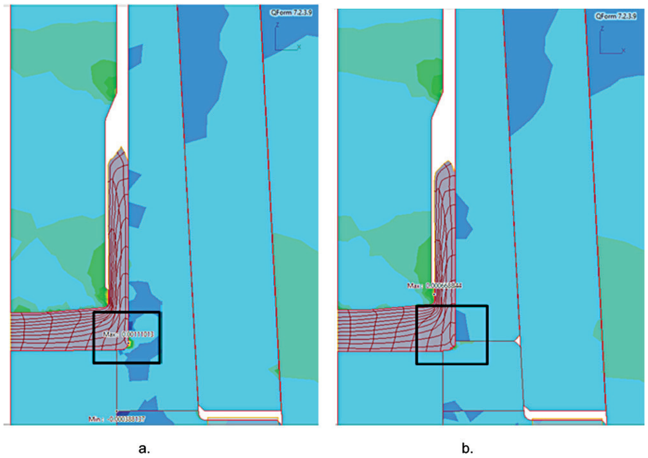

Fig. 10 — Removing the critical zone with the maximum value of Latham & Cockcroft criteria in a single piece insert (max L&C= 0.0011) (a) by introducing two piece insert (b). Critical zone disappears in other places that are not dangerous due to compression stress state.

The Latham & Cockcroft criteria distribution is shown in **Figure 10a,b**. Critical zone disappears in other places that are not dangerous due to compression stress state in the bearing zone of the punch as shown in **Figure 10b**.

www.qform3d.co.uk / www.QForm3D.com
www.ForgeTechnology.com

FTI