CASE 3: Analysis of tooling failure

Product: Valve spring retainer

Product Material: 34Cr4

Tool Type: Rigid for the plastic analysis / elastic for the punch analysis

Process Type: 2D Axi-symmetric, Isothermal, Multistep

Press Type: Eccentric press

Software Used: eesy-2-form

Company: Kinnings Marlow, UK

1. Introduction

In this analysis an existing process was analysed to find the reason for the premature failure of the tooling.
The process design was based on empirical rules and unexpectedly the punch in the last operation failed after production of a few thousand pieces only.

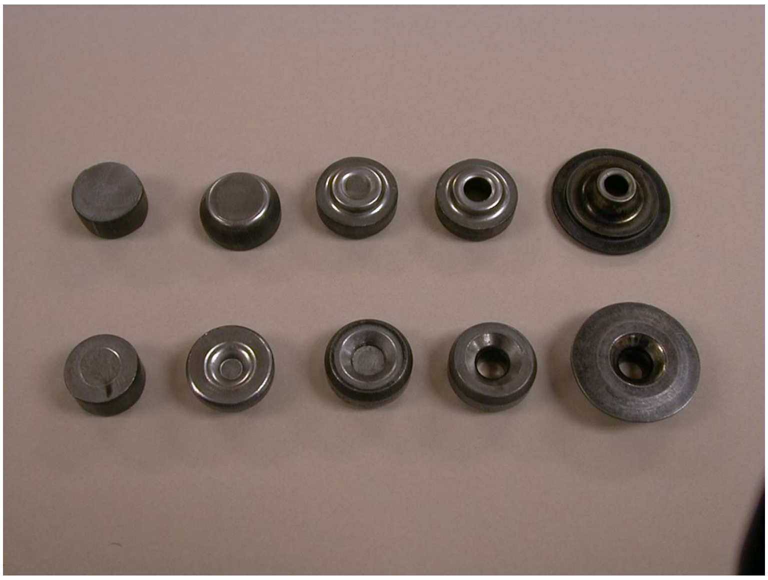

Figure 1: Progression used

After cutting of from the wire the cut-off is set in a first operation to make it more symmetric. Then it is preformed in two steps. After that there is a piercing and thereafter the final forming of the retainer. In this final forging operation the punch fails prematurely.

Figure 2: Punch failing

All forming operations are axi-symmetric. The initial wire is phosphated and pre-drawn.
The work piece material is 34Cr4.

2. Key Points of Finite Element Model

Plastic simulation

The process was modelled in five subsequent model files. The first simulation model to simulate was the upsetting of the material in operation one. In the definition of the cut-off the pre-drawing was considered as well. The user gave the conditions of the pre-drawing like % of reduction and drawing angle and the system provided the resulting pre-straining information in the cut-off. An average temperature of 80℃ was assumed to represen t the typical isothermal condition of the process. 80℃ was assumed to be less than the real temperature so the simulation results were expected to show higher stresses then the real ones. The second, third etc. simulation files included the simulation models for operation two, three etc.. The resulting piece was transferred from Model one to model two including, geometry, pre-deformation, temperature etc.. In the same way the piece was moved into operation three, four and five.

The following data had to be put in during the modelling:

- Geometry of the cut-off (Volume of the piece, diameter of the wire)
- Angle of the wire drawing die and percentage of reduction
- Material characteristics of 34Cr4 (provided from the system´s database)
- Properties of the press to be used in means of stroke and number of strokes per minute
- Friction coefficients for the used combined Coulomb / shear law.
- All tooling dimensions

Any geometry could be created within the software’s interface or imported via .iges files.

A crucial input to the results and quality of the simulation was the elements size. This was detected by the geometry of the part to be simulated. The mesh must be fine enough to fill the “smallest” areas with a reasonable number of elements. Simple rules allow the user to determine this.

The mesh generation, the re-meshing options, the stepping (increments) of the simulation etc. were set by the system automatically.

With this information the plastic analysis of the process was performed. The simulation time was some minutes per operation on modern Pentium 4 processors.

A speciality was the simulation of the piercing in operation four. The principle is to simulate the movement of the piercing pin till a specific shear stress is reached in front of the pin so that the material will start to crack. Then there is a special feature that will remove the cracked part.

The results of the plastic simulation showed a perfectly formed part as expected.

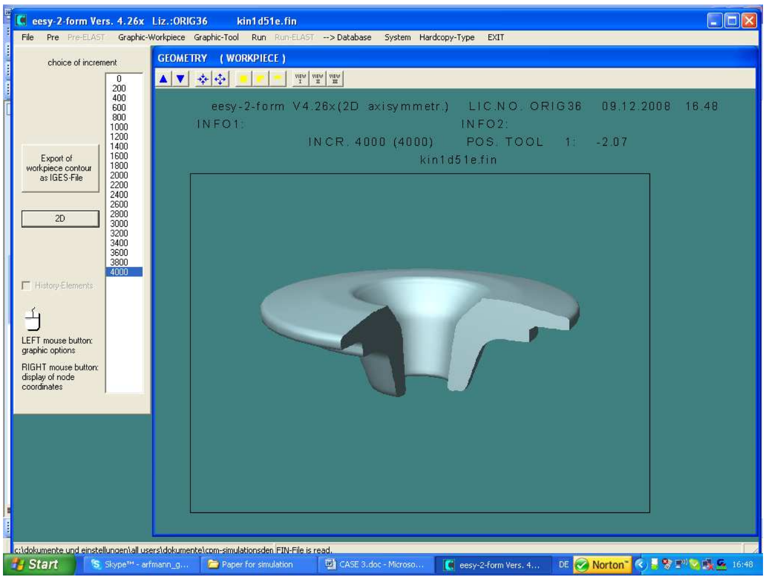

Figure 3: Final shape of the retainer

Elastic simulation

After the plastic simulation an elastic analysis of the pin was performed.

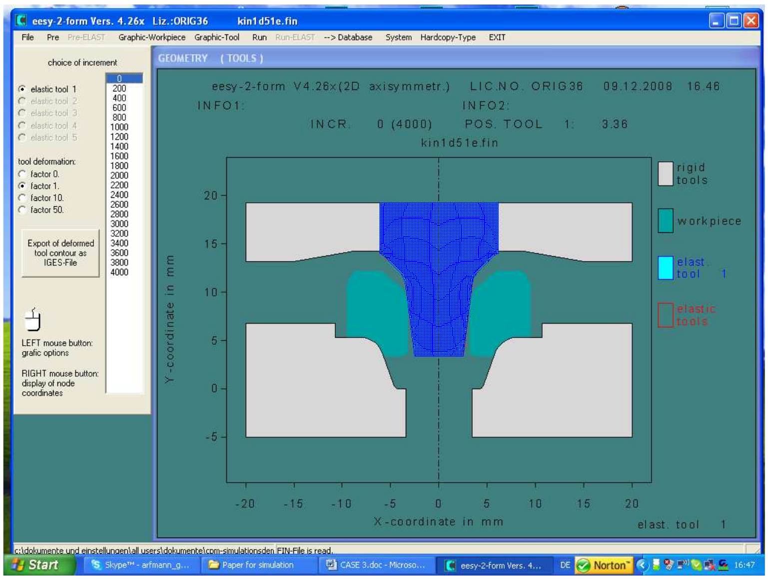

Figure 4: Pin modelled elastic

The focus was to find critical stresses in the pin.

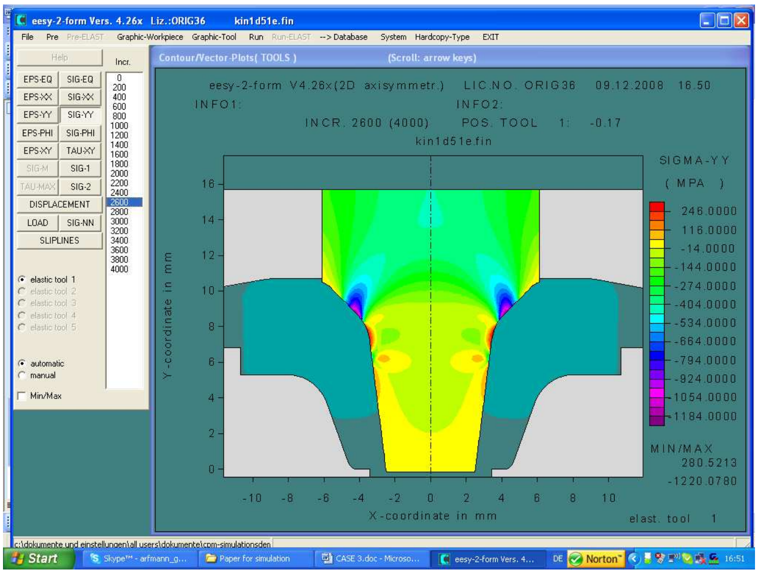

Figure 5: The simulation showed tensile stresses in axial direction

Positive stresses were found in the area of the radius exactly in the position where the pin failed.

Then the normal pressure on the pin was studied during the process.

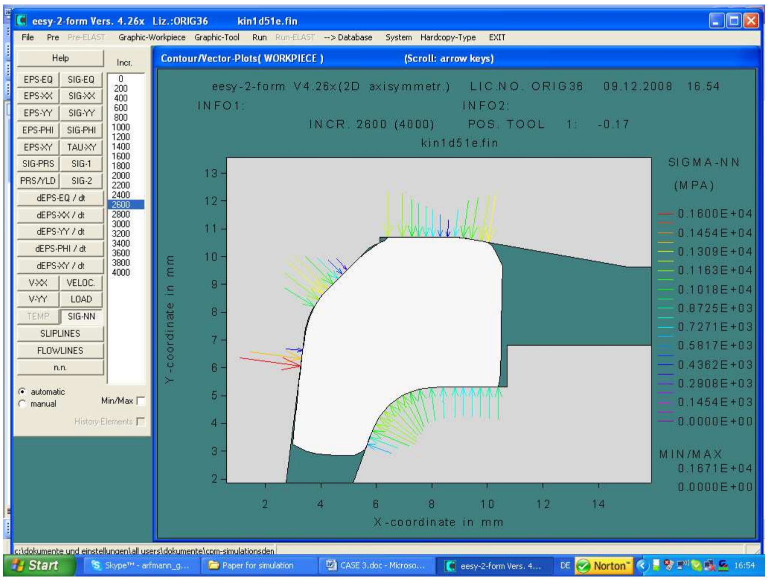

Figure 6: Normal pressure distribution during the operation

Obviously the gap in the pressure distribution on the pin caused the positive stress in the pin.

In a later punch position this “pressure gap” disappeared due to radial contact in the die.

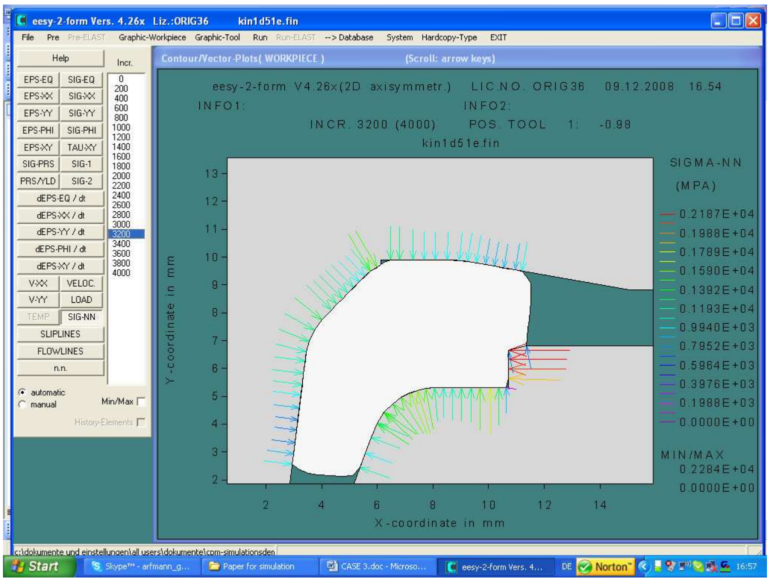

Figure 7: Normal pressure distributions during the operation

From this observation the engineer got the information how to change the process. The aim was to change the preform so that the material gets in radial contact with the die at an earlier position. Then the "pressure gap" could be avoided.

The reason for the breakage was not the absolute amount of positive stress in the pin. The pin faces dynamic loading and unloading during the process. Due to the process this caused an alternating stress situation during the forging which takes place only at the very end of the stroke of the machine. So there was a short period of slightly positive stress followed by pressure at a small time period of the stroke of the machine. The pin is "stress free" most of the process time and then faces a very sudden alternating stress situation. This is causing fatigue in the material and therefore caused the premature failure.

By changing the progression and getting the material in contact to the pin during the forging this alternating stress situation could be changed and tool live could be increased by more than ten times.

Conclusion

The FEM could not only show the reason of the tool breakage. It showed as well why that positive stress occurred. It showed as well how such stress could be avoided. All this knowledge could only be got by use of simulation. Working the empirical way would have brought an improvement just be chance. Using FEM enables a systematic analysis in short time and enables therefore to optimize a process and to save tooling costs.