CASE 6: Extrusion

Die design for difficult extrusion operation

Product: Fastener

Product Material: 10B21

Tool Type: Rigid for the plastic analysis, elastic for the die analysis

Process Type: 2D, axi-symmetric, isothermal

Press Type: Eccentric horizontal press

Software Used: eesy-2-form, eesy-DieOpt

Company: German fastener producer

Introduction

A relative extreme extrusion of close to 80% should be performed. The initial diameter was 12.93mm and should be reduced to 5.96 mm.

The process should be designed for a machine with a die holder diameter of only 70mm.

For empirical knowledge this die design is very difficult if not impossible. Therefore the tool design was done using FEM and a special system to allow for pre-stressing layout.

Typical tooling failure

Typical tool design failures lead to tool breakage caused by stresses in the inserts.

Figure 1 and 2 show such failures caused by positive axial stress respectively by too high inner pressure resulting in positive tangential stress.

In case of axial stresses the tooling has to be split by construction at the location of the positive stress while in case of tangential stress the insert has to be pre-strained by providing pre-straining rings.

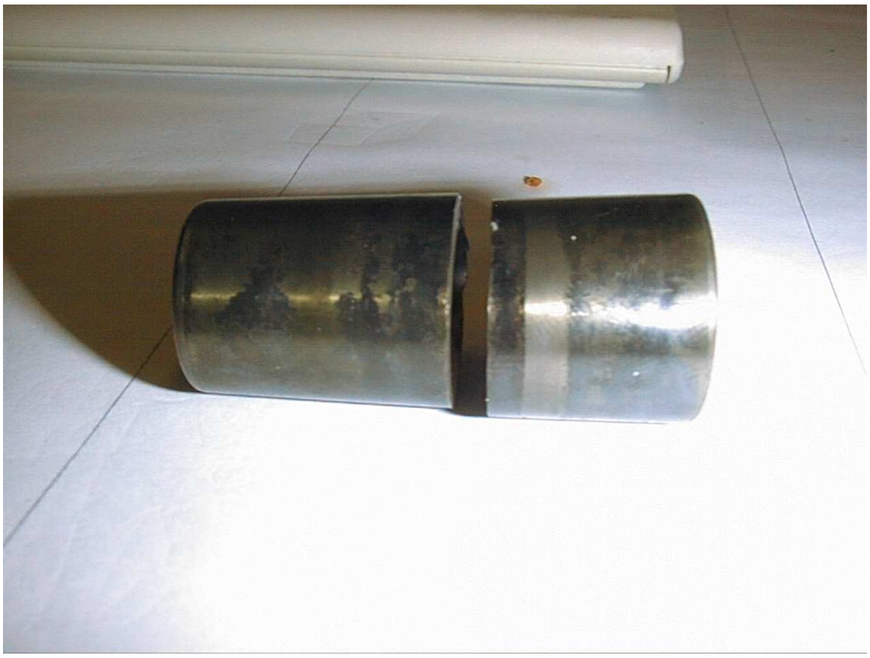

Figure 1: Die failing due to positive axial stress

Figure 2: Die failing due to positive tangential stress

Key Points of the Modelling

Plastic analysis

For modelling the FEM code eesy-2-from was used to perform a plastic analysis of the process.

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 10B21 (provided from the systems database)
- Properties of the press to be used in means of stroke and number of strokes per minute
- Friction coefficients for the used combine Coulomb / shear law.
- All tooling dimensions

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

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.

Figures 3 and 4 show the initial and final situation for the extrusion process.

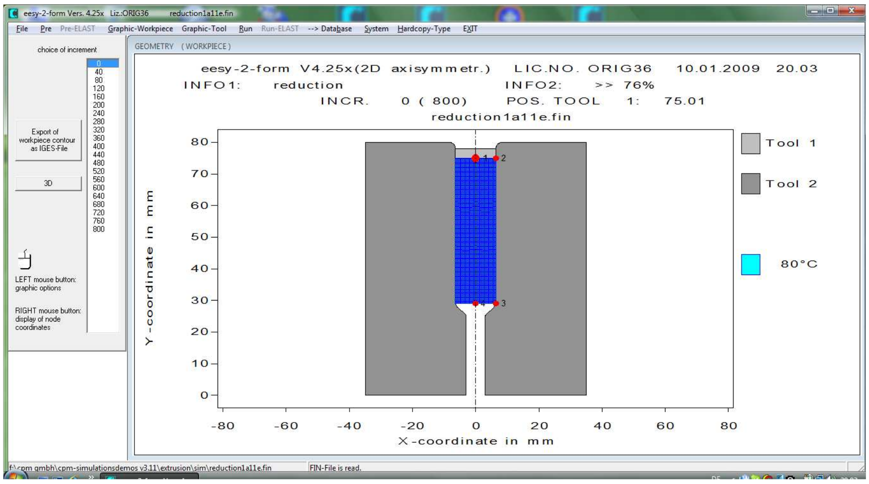

Figure 3: Initial model of the extrusion process

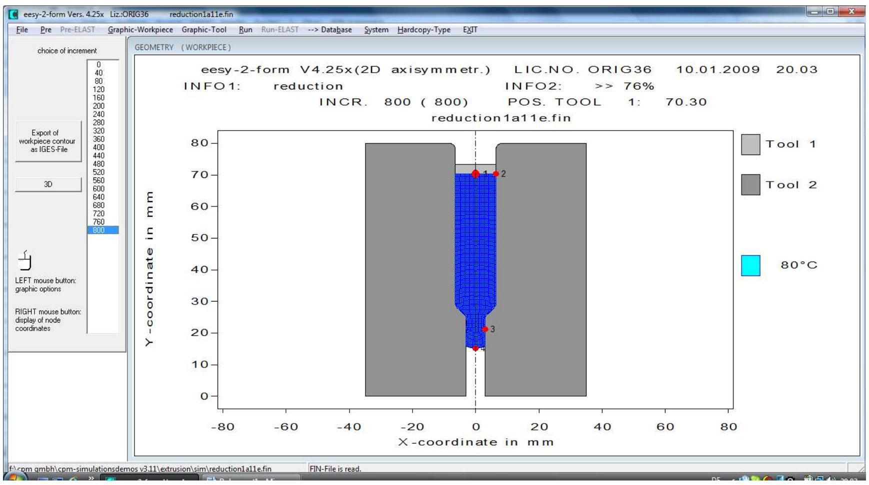

Figure 4: Result of the plastic simulation

Elastic analysis of a typical die insert

After the plastic simulation provided the internal pressure of the die a typical die insert was chosen from experience and a first elastic analysis of that insert was performed. The aim was to learn about the axial and tangential stresses in an insert in this process.

For this analysis the insert is not pre-stressed. Figure 5 shows the insert with the inner pressure only.

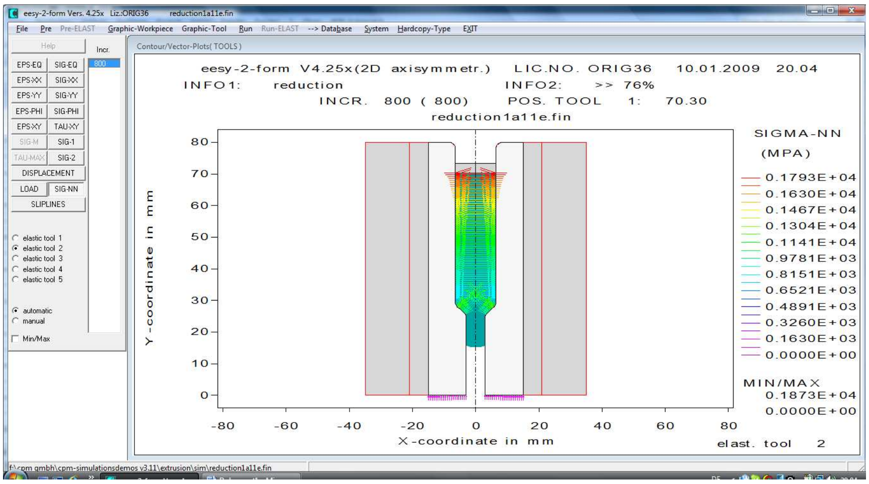

Figure 5: Insert with empirical dimension and inner pressure only.

This analysis shows us the position where the insert later has to be split due to axial stress.

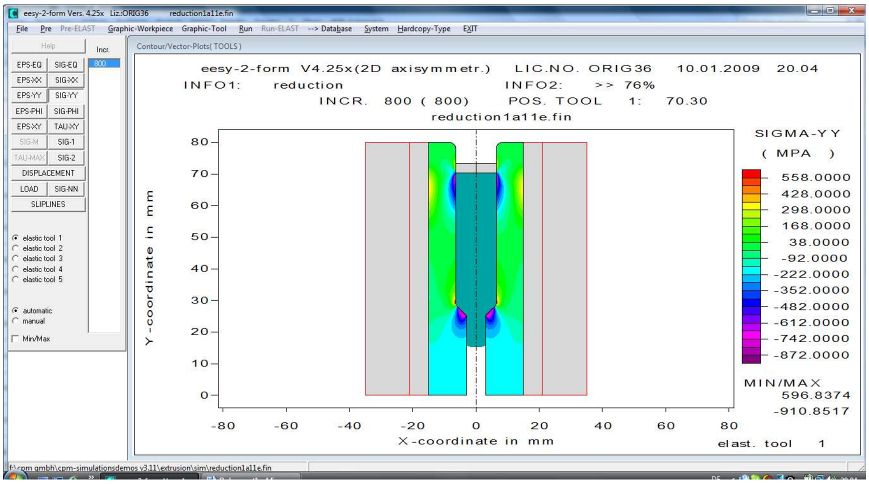

Figure 6: Distribution of axial stress (red area shows positive stress)

The distribution of tangential stress shows high values on the inner surface of the insert (figure 7). This tangential stress had to be compensated by pre-stressing. Therefore some elastic simulations with a variation of pre-stress values were performed. Figures 8 and 9 show an analysis with a pre-stressing of about 900 MPa resulting in compressive tangential stress in the insert.

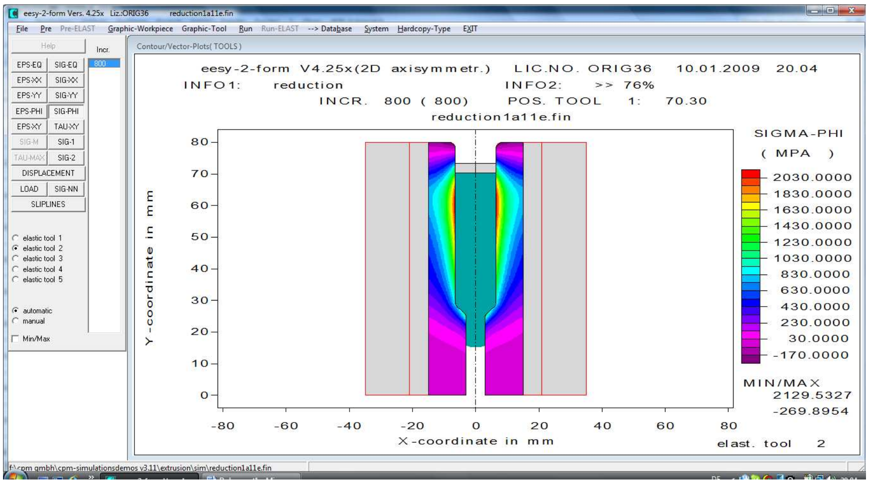

Figure 7: Distribution of tangential stress (no pre-stressing)

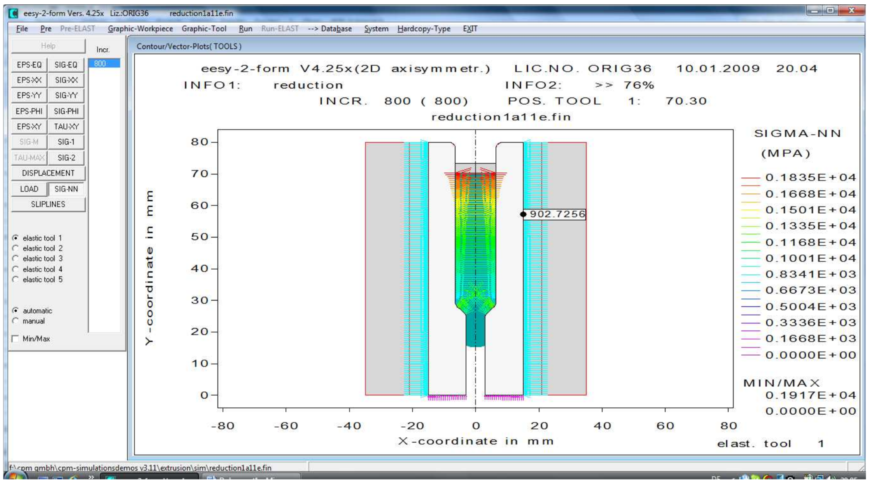

Figure 8: Pre-stressing with about 900 MPa

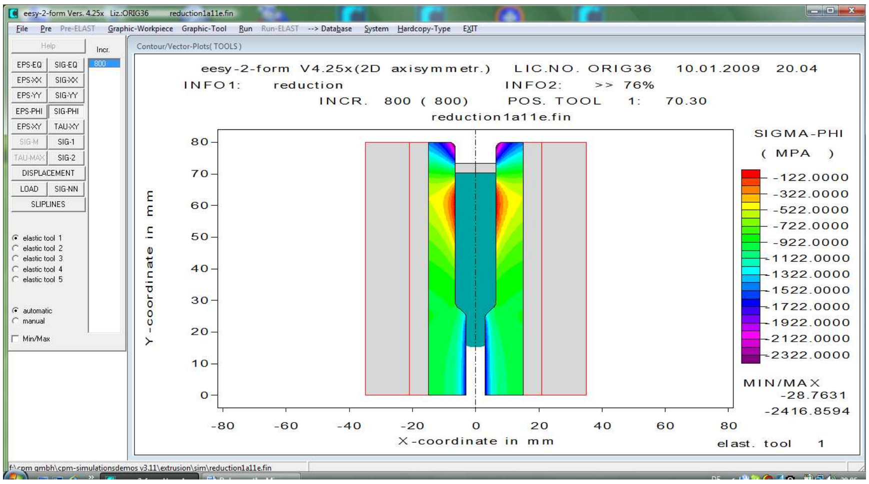

Figure 9: Distribution of tangential stress (900 MPa pre-stressing)

Using the die layout system eesy-DieOpt

The elastic simulation of the chosen insert showed the amount of pre-stress that had to be generated by the die construction.

eesy-DieOpt allows to calculated multi-ring pre-straining systems for tubes. In reality the insert is no tube but the other components are very similar. This is why this system can be used to design a die system to provide a pre-stress of about 900 MPa on an insert. This data had to be transferred in to FEM subsequently and a precise analysis of the real insert and the other die components had to be performed to finally check for the stresses.

eesy-DieOpt allows the user to input the inner diameter of the insert and the diameter of the die holder. The user has to decide for the layout (number of rings) and has to choose the material and their properties (HRC) for the components.

The system calculates the optimum dimensions and interferences and gives information about the amount of pre-stressing on the insert.

Figure 10 shows the system and the data defined or chosen by the user

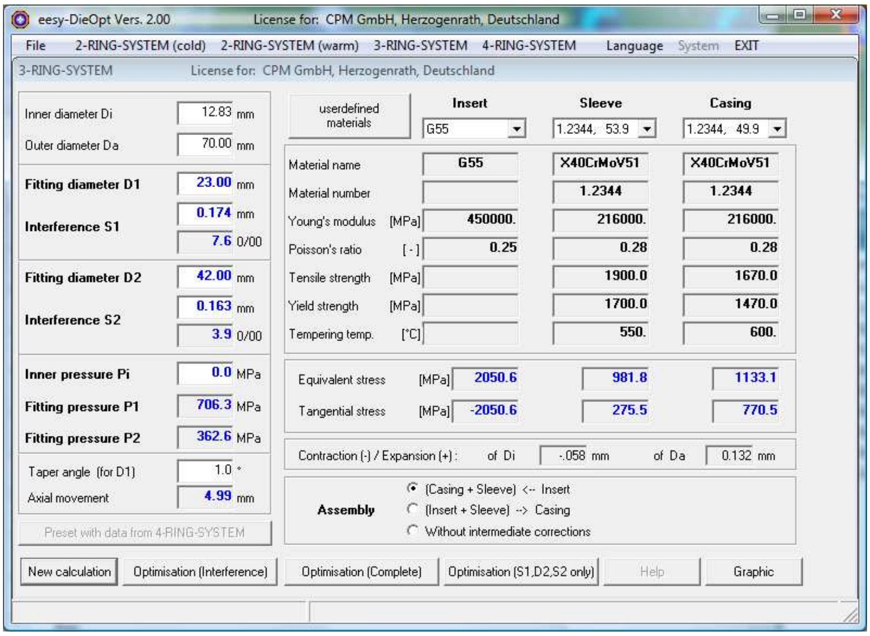

Figure 10: eesy-DieOpt (input data before optimization)

The user chose then the optimization method (complete) and set some restrictions (figure 11).

These restrictions limit the optimization algorithm. Such restrictions are “no tangential stress in the insert” or limits of the equivalent stress in the rings (typically set as percentage of the yield strength given by the hardening of the material.

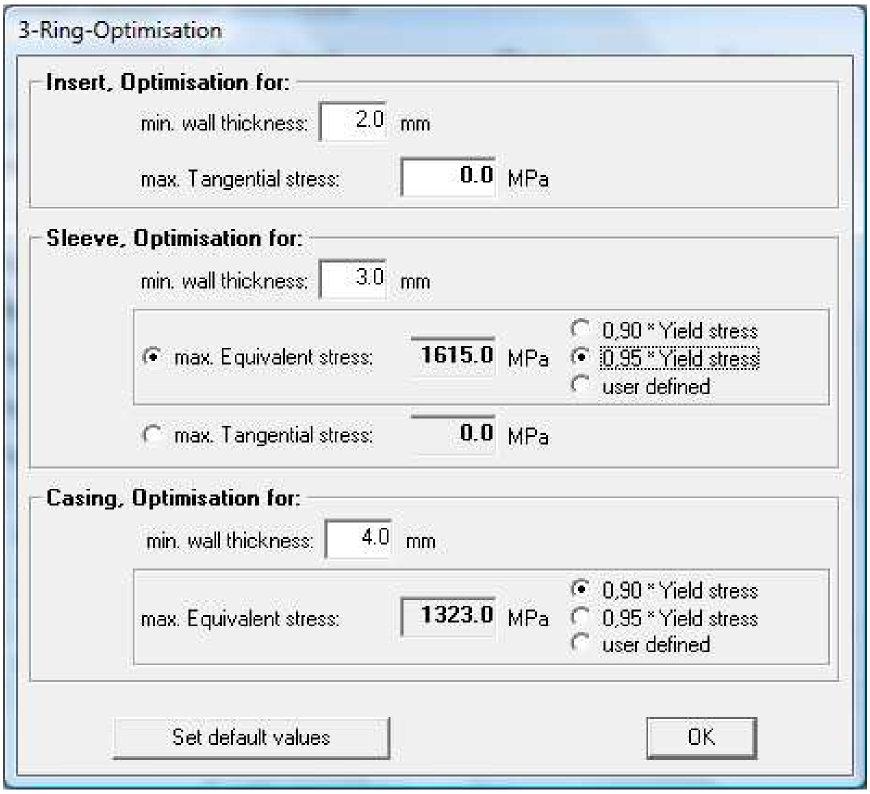

Figure 11: eesy-DieOpt - restrictions

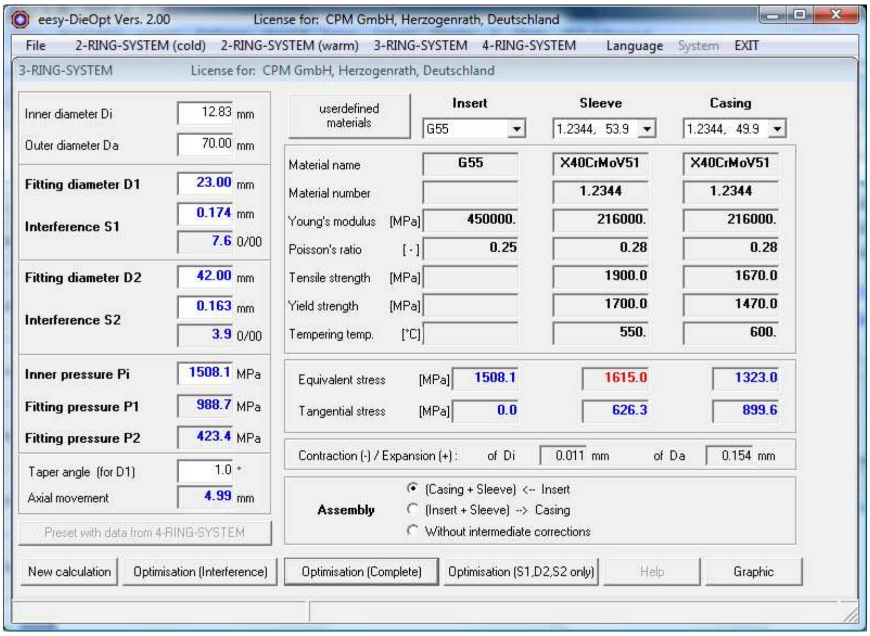

Figure 12: eesy-DieOpt – results

eesy-DieOpt gave a solution that may provide up to 980 MPa pressure on the insert (Fitting pressure P1). The insert dimension would be 23 mm while the interference was given as 0.174 mm.

The intermediate ring had an out diameter of 42 mm and an interference of 0.163 mm.

The material chosen for the ring and the body was H13 with a hardening of 54 HRC and 50 HRC.

Elastic analysis of the optimized design

eesy-DieOpt and eesy-2-form provide an interface to each other. So the data could easily exchange between eesy-DieOpt and eesy-2-form.

Then a final elastic simulation was performed for all the parts of the die. The insert was checked for tangential stress while the ring and the body were checked against their yield stress.

Figures 13 to 15 show these results.

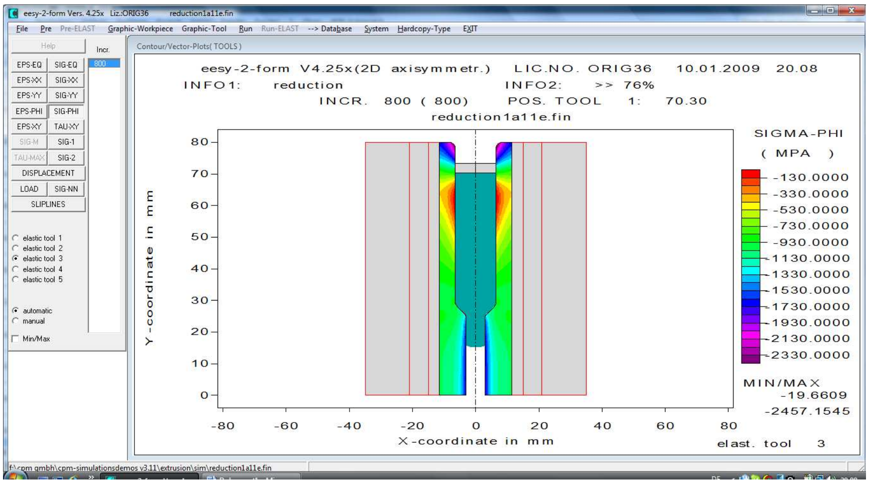

Figure 13: Tangential stresses in the insert under forging load (all compressive)

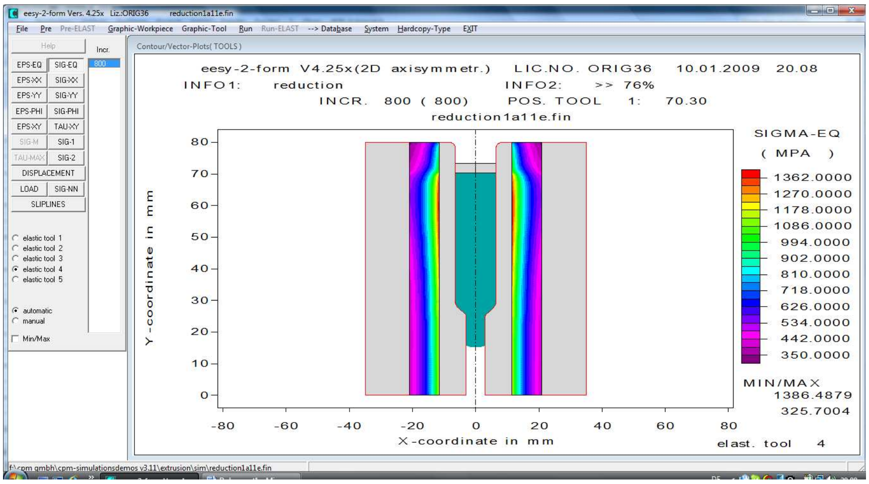

Figure 14: Equivalent stress in the ring
(max of 1386 MPa is below the yield stress of 1700 MPa (HRC 54)

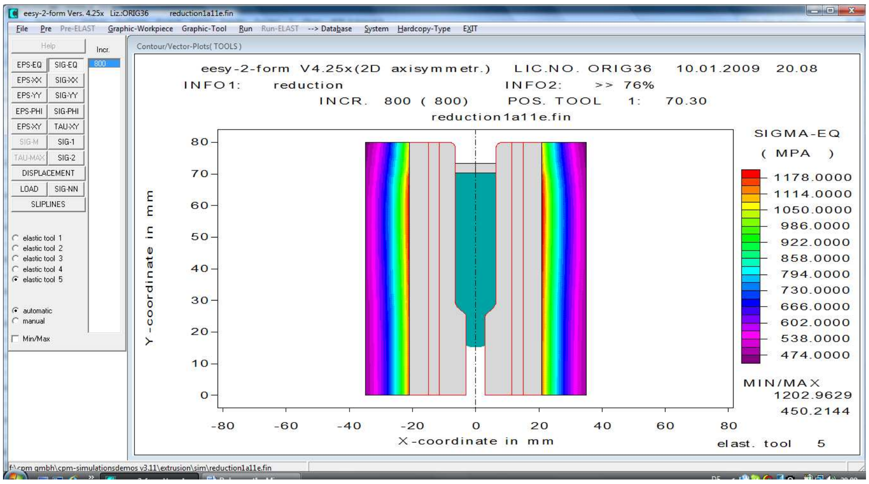

Figure 15: Equivalent stress in the ring
(max of 1202 MPa is below the yield stress of 1470 MPa (HRC 50)

As result all stresses were found to be not critical so the final tool design was found and the die could be made. The practical test showed that this tool was wearing and not breaking.

Conclusion

eesy-2-form in connection with eesy-DieOpt are a good solution to make a systematic approach to die design.

The data input is simple and the computing time is short. The plastic simulation takes some minutes while the elastic simulations take less then a minute each.

And they are easy to use.

Therefore they are ideal for the daily design work even in small companies without very skilled engineers in computing.