Tech-Spring Report 23

Finite Element Analysis of Spring Clip Supplied by Metalpol

1. Introduction

The technique of Finite Element Analysis (FEA) is widely used in engineering as a stress analysis and design tool. The aim of this report was to evaluate its effectiveness in analysing a typical component which was a spring, but not a helical coil spring. This clip was in fact a good example to show some of the unexpected features and complexities of a simple looking component.

2. FE background information

The software used was ANSYS Structural v11, which is an industry standard non-linear FEA package. It is of course necessary for the operator to have a certain level of skill, and probably undergo a considerable amount of training. As with all FEA software, there is an obvious temptation for novice users to treat the system as a “magic black box” and trust its output implicitly.

3. Model creation and results

i. Model geometry

The clips were supplied with a drawing defining the dimensions (see Figure 1). In order to create an FE model, these dimensions had to be recreated in an alternative form. A second version was drawn in SmartSketch, which enabled relative co-ordinates of all the dimensions to be found for all the important parts of the section. The points of interest to the manufacturer setting up his tooling are of course different to those needed for FE analysis. This drawing is shown in Figure 2.

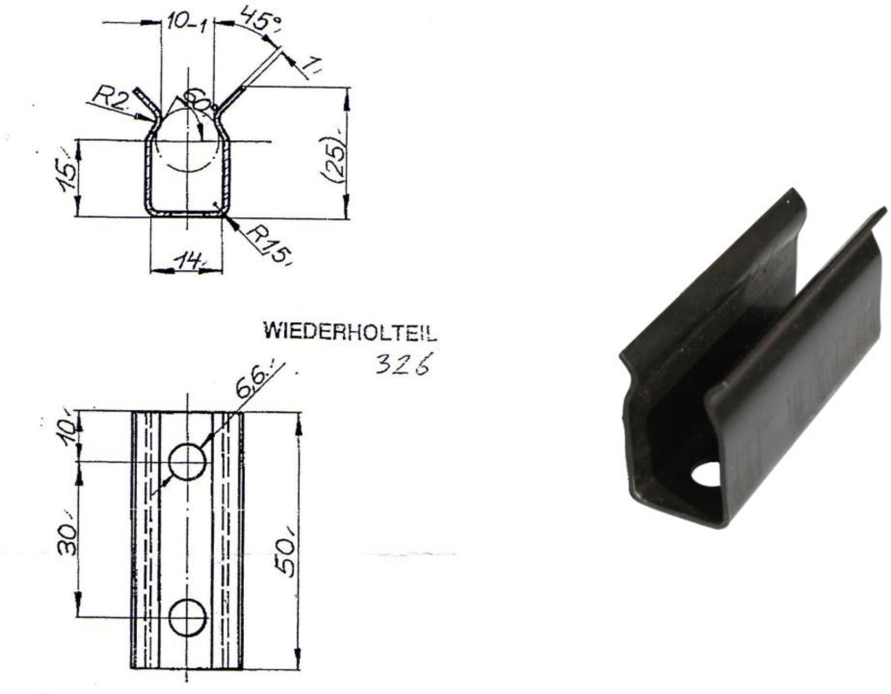


Figure 1 - The original drawing of the component, with a photograph of a supplied clip.

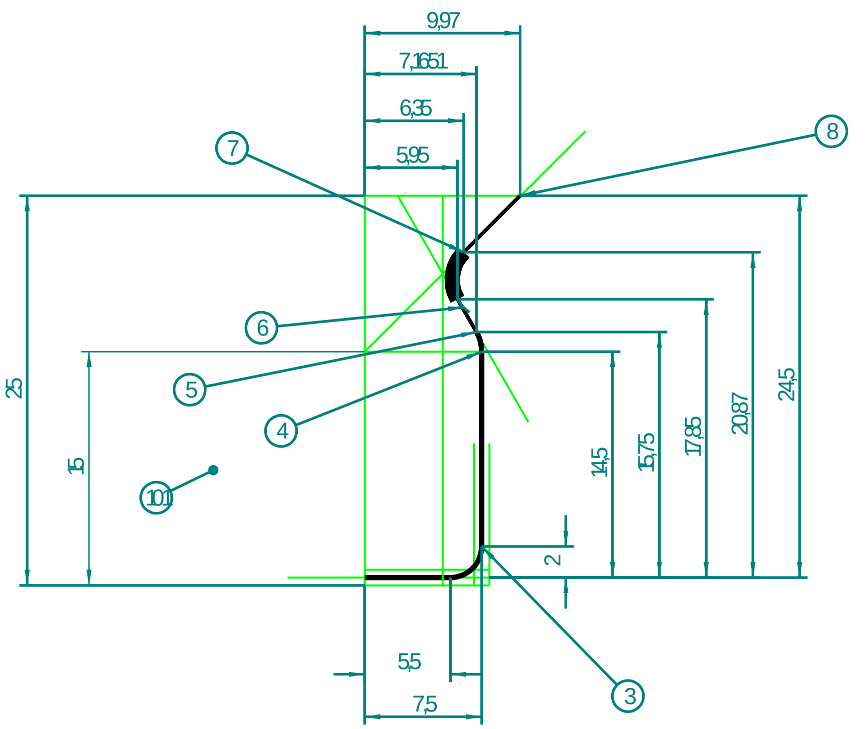


Figure 2 - The SmartSketch drawing used to find co-ordinates of all the important points.

Because of the symmetry, the decision was made to only model half of the clip. In this particular case, computation time was not likely to be too high, but in many cases halving the amount of processing can be a significant saving.

ii. First model

The initial model relied on a very simplified set of boundary conditions. The area providing symmetry was completely fixed in all three dimensions, and a single point in the centre used to provide the sideways deflection, as shown by the red area and arrow in Figure 3 below.

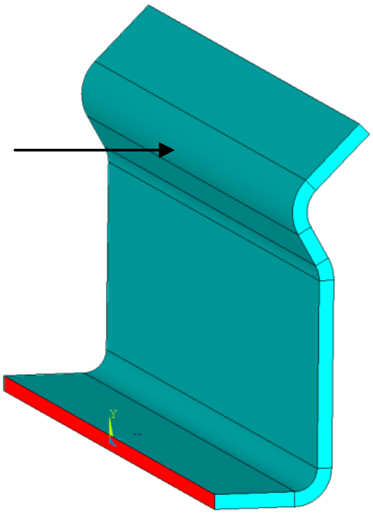


Figure 3 - Boundary conditions used for the first model

This model solved quickly, and gave the resultant deflection and stress plot shown below in Figure 4. This gives a slightly unrealistic deflection, as the bottom corner of the clip deflects below its original level, which would not happen if the clip was bolted down. The predicted maximum stress was just under 800 MPa.

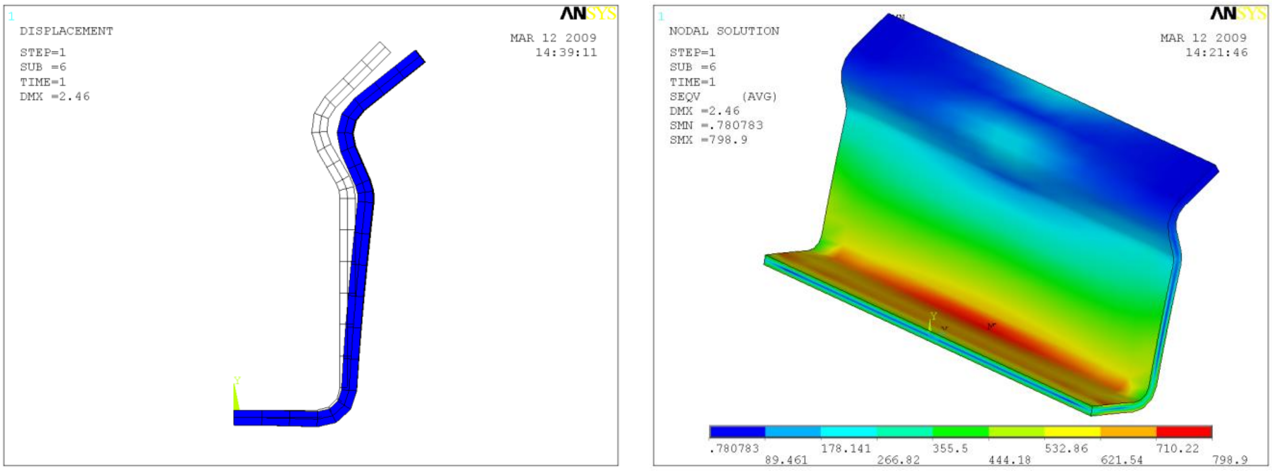
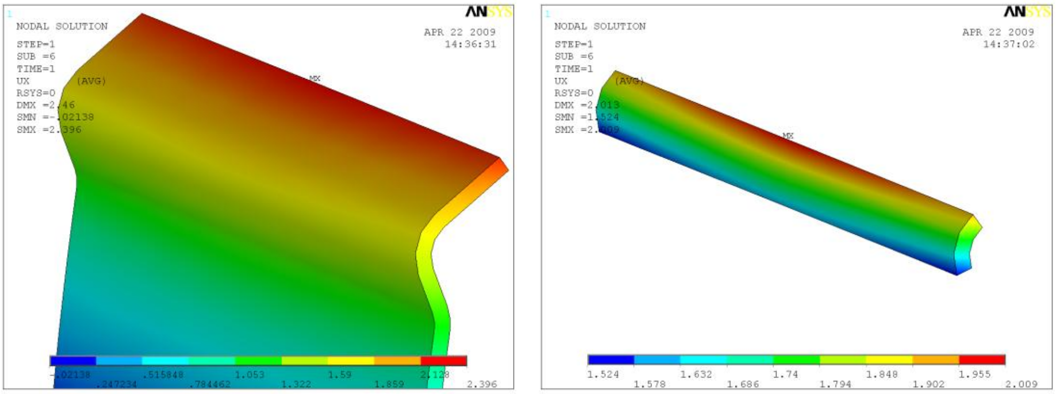


Figure 4 - Deflection and stress results for the first model.

The effect of a single point deflection is not obvious looking at the deformed shape plots, or even a plot of deflection (Figure 5 (a)). Only when the plot is refined to include a small section can the variation of deflection along the length of the clip be see, in Figure 5 (b).



(a) (b)

Figure 5 - Contour plot of deflection in the direction of applied load, for (a) the whole clip, and (b) the bend where the boundary condition was applied.

A slight change was made to restrain the flat bottom surface of the clip. This reduced the active length of material and increased the amount the bottom bend had to “unbend” to achieve the required deflection. This was reflected in the predicted force, which was increased to 567N. The stress in the bottom bend also increased dramatically to over 1400 MPa, as shown in Figure 6 below.

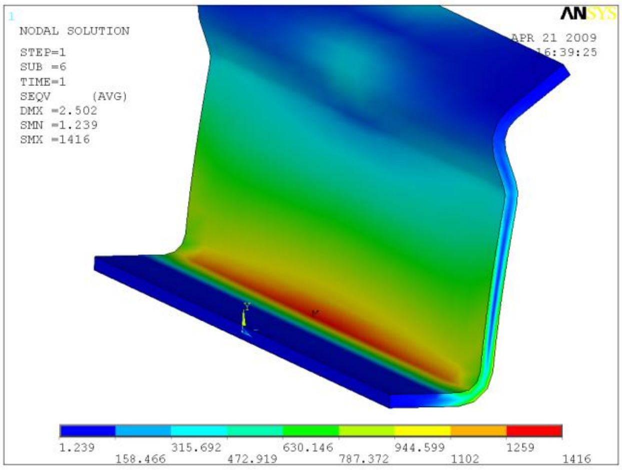


Figure 6 - Large stresses induced by over-defining boundary conditions.

iii. Second model – simulating the bolt holes

For the second model, two bolt holes were created by subtracting cylinders from the main volume. The inside surface of the hole was used to add to the symmetry boundary constraint, as shown by the red areas in Figure 7 below.

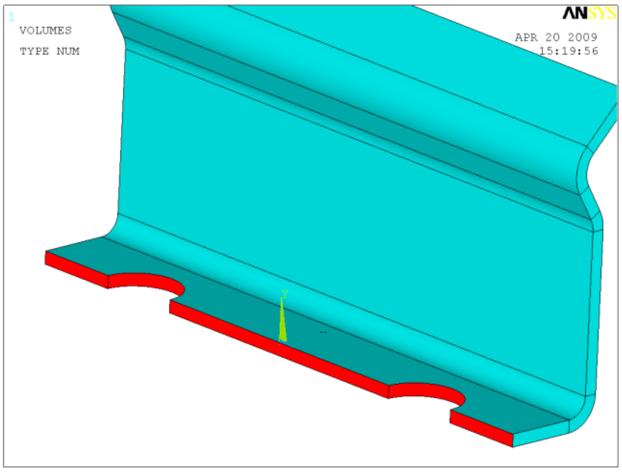


Figure 7 - Boundary conditions on the bolt holes of the second model.

Solving this model shows a typical problem with FEA. These extra restraints create large localised stresses which dominate the stress plots (see Figure 8). The added features were intended to make the model more realistic, but because joints like this are actually quite complex, it can actually detract from the accuracy. It can be seen that the stress on the inside of the bend is similar to the 800 MPa predicted by the initial model, so this model does give comparable results as long as the unrealistic stress concentrations are ignored. The horizontal reaction force was also similar, at 371N.

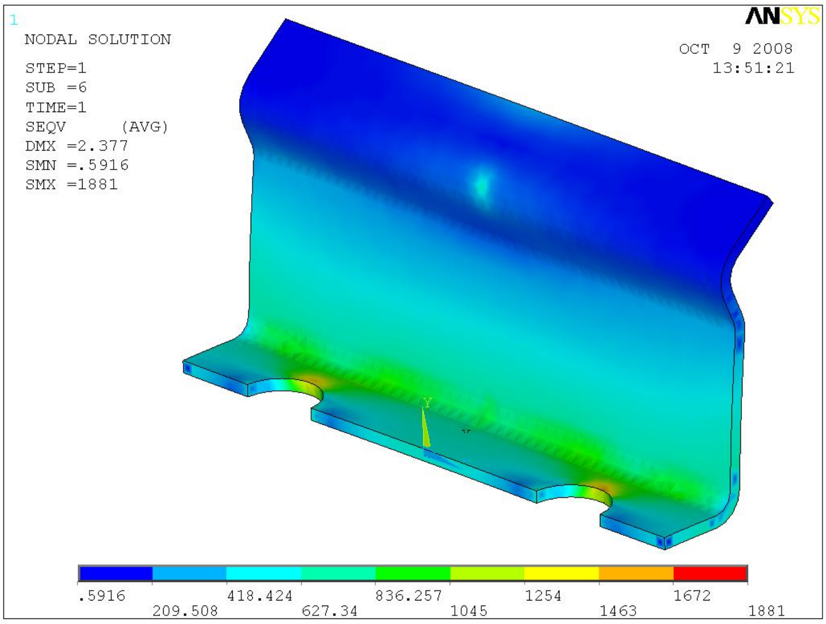


Figure 8 - Large localised stresses created by unrealistic boundary conditions on the bolt holes.

iv. Third model

A third model was created to try to simulate more accurately the effect of bolting through the holes with washers. In this model an extra pair of cylinders was used to split the bottom volume to give an extra cylindrical annulus. The top surfaces of these were then given boundary conditions to simulate a bolted washer (Figure 9). The deflection condition was also modified to be along a line rather than a single point.

Although this model did give a better deformed shape to the flat bottom surface, it did not eliminate the stress concentration around the edge of the holes (Figure 10). The effect of spreading out the deflection can also be seen by the absence of a stress "hot spot". This did however make a significant change to the reaction force, which was increased to a total of 426N spread across the length of the clip. This is not unexpected, as more of the clip needed to be deflected in this model.

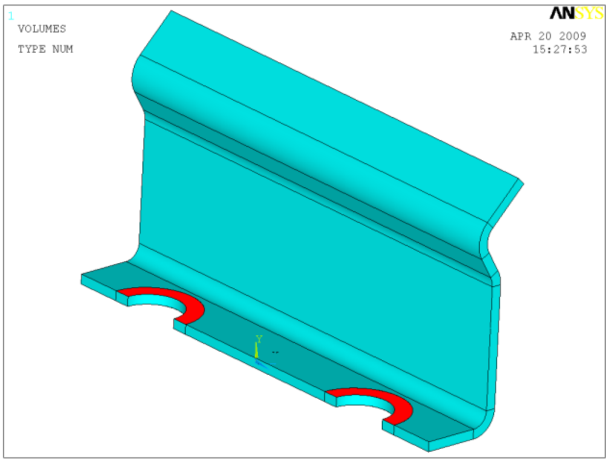


Figure 9 - Boundary conditions used to simulate a bolt and washer.

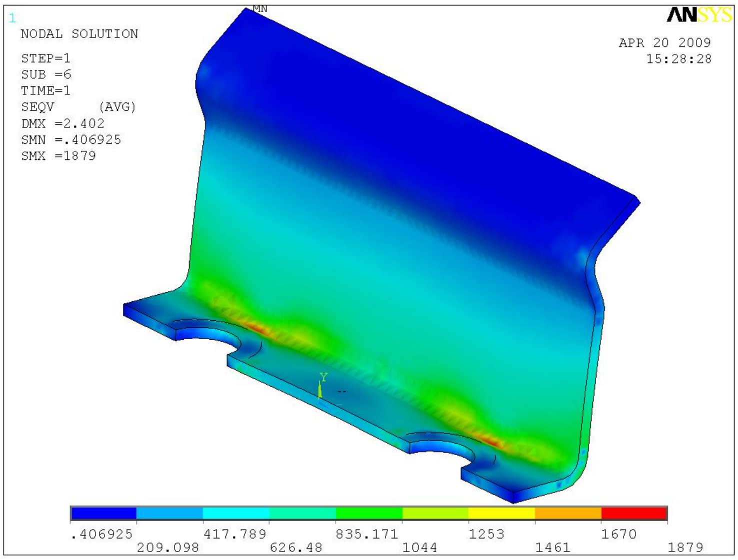


Figure 10 - Stresses near a bolted washer.

v. Fourth model – including plastic deformation

A final model was created to give a more accurate prediction of stresses given that yield was likely. In this model a simple plasticity was used to give a yield point of 1117 MPa (75% of the UTS of 1490 MPa). The Young's modulus then dropped to a very low value. The aim of this was to allow localised yielding, on the assumption that it would only happen over a small area and would not then affect the rest of the model.

The main effect of this plasticity (apart from removing the unreachable high stresses) was to reduce the horizontal load to 264N – the lowest of any model.

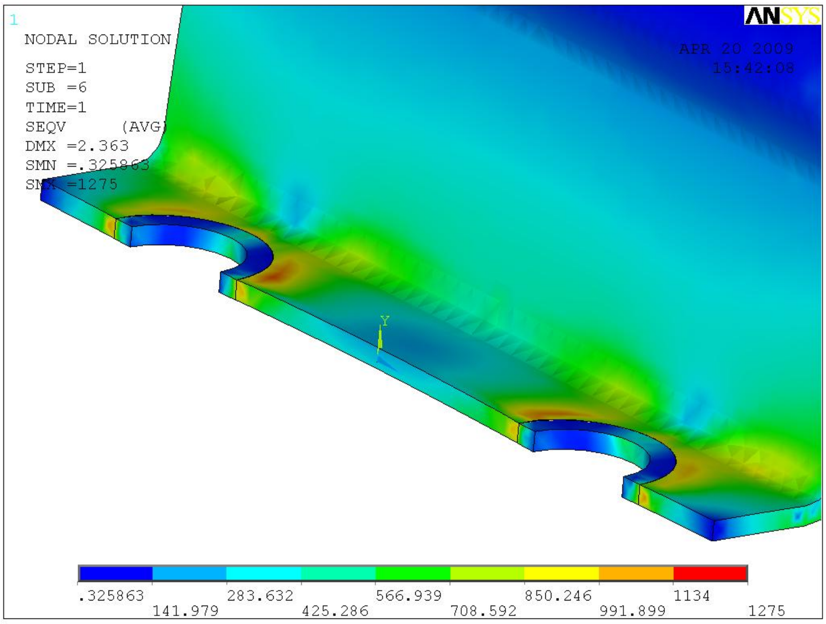


Figure 11 - Stresses around bolt holes for a model with plasticity.

4. Load test results

A simple jig was constructed which clamped a plated to the bottom of the clip through the two bolt holes, to simulate its fixing in operation. The two sides of the clip were deflected apart by 3.5 mm and the load measured. The results below in Table 1 are for three clips without the jig (so the bottom surface was free to deflect), and in Table 2 for three clips with. The average load was 266 N without the bottom surface clamped, and 328.1 N with it clamped.

Sample	Load (N)
1	270.0
2	265.4
3	262.6
Average	**266.0**

Table 1 - Load test results for a deflection of 3.5 mm, without the clamping jig.

Sample	Load (N)
1	335.5
2	315.3
3	333.5
Average	**328.1**

Table 2 - Load test results for a deflection of 3.5 mm, with the clamping jig.

These results must of course be halved when comparing to the FE models, as the FE only dealt with one half of the component. This means that neither of the measured values are close to any of the predicted loads.

5. Conclusions

This seemingly simple component is a good example of the inherent dangers in using complex technology to solve what is in effect a relatively simple problem. A series of load tests was performed in under an hour, whereas the various FE analyses presented here took perhaps two working days all in all. It would be easy to have made one of the assumptions presented in this document about the boundary conditions, and come to an invalid conclusion – armed with impressive looking stress plots which give a high degree of confidence! The software did predict the position of maximum stress accurately, but the magnitude of the loads (and therefore probably the stresses) were completely incorrect.

It is easy to conceive of a situation where a spring maker is confronted by a customer armed with this sort of high-tech analysis, and there are of course structures and springs for which FE analysis is the only feasible way to go. However, for simple parts like this one, the recommendation must be that the spring maker performs a quick and simple load test on a sample component and combines that with a stress calculation based on simplified traditional theory, which in this case would be quite easy to do.