Influencers of clinched joints

Jónás Szabolcs¹, Tisza Miklós²
Knorr-Bremse Vasúti Járműrendszerek Hungária Kft., Miskolci Egyetem
Gépészészmérnöki és Informatikai Kar

Abstract

In this article some influencers of clinched joints were studied. A brief comparison of the tests and simulations is presented to prove the results and the usability of the model. The frictional coefficient, the geometry and the description of the material law was taken into consideration to determine the main geometrical sizes of the joint. The presented data can be useful to design clinched connections in the automotive industry without tests between DP600 steel sheets.

Keywords: plasticity, forming, clinch joint, DP600, FEA

INTRODUCTION

These joints are used mostly in automotive, computer and aircraft industries, but for instance according to the standards they are not allowed to be used in food industry [1-3]. The clinch joints are quite new types of joints, the first patent was accepted in 1989. This joint can be created between 2-3 thin sheet plates. The cross section of a joint can be seen in Figure 1, which shows the main geometrical parameters of a joint (2 sheets were joined). The undercut size (C value) and the neck thickness (tN value) are highly affecting the strength of the joints (Figure 1c). In optimal case both of them are as high as possible. The material of the plates can be ferrous or non-ferrous at the same time, so this joint can realize dissimilar joints without any added material (weld material or glue). The joint is made by metal plastic forming by a special tool. After creating the patent the increasing industrial needs of these types of joints drove the researchers to analyse the joints much more deeply. Several studies have been carried out concerning the geometry optimization of the clinching tool to achieve better joints by different optimization methods. Other studies were carried out on the so-called hybrid joints. These joints have an adhesive layer between the sheets. These joints have higher strength but they need much more time because the drying of the adhesive layer is a time-consuming process [4-5].

MATERIAL IDENTIFICATION

The examined material is the DP600 type of steel. The DP600 is an advanced high strength steel, which is a multiphase (ferrite and martensite) steel with excellent combination of strength and formability. The dual phase steels (DP) consist of a soft ferrite matrix with a disperse, hard second phase in the form of islands. They have high strength, high work hardening rates and high strain energy absorption properties. A 100-200-600-1200 μm series of abrasive paper, 3 μm polishing paste
and 3% Nital was used for the specimen preparation. The measurements were done with a Carl Zeiss microscope with an image recognition software module. According to the measurements, this steel contains ~23% of martensite, and its average grain size is 5 µm. For the chemical composition a 60 µm abrasive paper was used. The measurements were performed with an Oxford Instruments Foundry Master Pro spectroscopy machine at the John von Neumann University. The chemical composition can be seen in Table 1.

Table 1 – Chemical composition (in wt%)

<table>
<thead>
<tr>
<th>Material</th>
<th>wt%</th>
<th>Material</th>
<th>wt%</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fe</td>
<td>98.6</td>
<td>Al</td>
<td>0.0553</td>
</tr>
<tr>
<td>C</td>
<td>0.116</td>
<td>Co</td>
<td>0.0125</td>
</tr>
<tr>
<td>Si</td>
<td>0.171</td>
<td>Cu</td>
<td>0.074</td>
</tr>
<tr>
<td>Mn</td>
<td>0.876</td>
<td>Ti</td>
<td>&lt;0.005</td>
</tr>
<tr>
<td>P</td>
<td>&lt;0.005</td>
<td>V</td>
<td>0.0124</td>
</tr>
<tr>
<td>S</td>
<td>&lt;0.005</td>
<td>W</td>
<td>0.0103</td>
</tr>
<tr>
<td>Cr</td>
<td>0.0262</td>
<td>Pb</td>
<td>0.0114</td>
</tr>
<tr>
<td>Mo</td>
<td>0.077</td>
<td>Nb</td>
<td>0.0214</td>
</tr>
</tbody>
</table>

Tensile tests were performed with an MTS electro-hydraulic testing machine at the University of Miskolc to determine the mechanical properties of the steel. The results can be seen in Table 2.

Table 2 – Mechanical properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Dimension</th>
<th>Mean</th>
<th>0°</th>
<th>45°</th>
<th>90°</th>
</tr>
</thead>
<tbody>
<tr>
<td>UTS</td>
<td>MPa</td>
<td>680</td>
<td>669</td>
<td>679</td>
<td>691</td>
</tr>
<tr>
<td>Yield strength</td>
<td>MPa</td>
<td>451</td>
<td>448</td>
<td>451</td>
<td>454</td>
</tr>
<tr>
<td>Fracture elongation</td>
<td>%</td>
<td>19</td>
<td>19</td>
<td>20</td>
<td>18</td>
</tr>
<tr>
<td>Hardening exponent</td>
<td>-</td>
<td>0.14</td>
<td>0.14</td>
<td>0.14</td>
<td>0.14</td>
</tr>
<tr>
<td>Anisotropy</td>
<td>-</td>
<td>0.81</td>
<td>0.71</td>
<td>0.73</td>
<td>0.98</td>
</tr>
</tbody>
</table>

From the tensile tests it can be seen that the value of the planar anisotropy was small. The presumption is that the planar anisotropy has no or negligible effect on the geometry of the clinch joints.

CLINCHING PROCESS

The TOX produced clinching tool was set up in an MTS servo-hydraulic testing machine. The maximum permissible load of the tool is 50kN. The set up can be seen in Figure 1 (where a) shows the assembly and b) shows the specimen). The specimens were pre-drilled for this application. Two holes were drilled which on the one hand centralized the specimens and on the other hand prevented them from moving.
FE PROCESS SIMULATION

FE Model

The FE simulation model was built in ANSYS WB 18.2 [6]. The geometry of the model was built up in ANSYS Design Modeller as a parametric model. A 2D axisymmetric model is presented below (Figure 3). The tools were taken into consideration as linear-elastic materials; the two sheets were simulated with elastic-plastic behaviour with multilinear isotropic hardening rule. The tool has a spring row which was taken into consideration as an elastic body with 57 GPa Young’s modulus. The mesh was built up by 2nd order axisymmetrical quadrilateral and triangular elements (PLANE183). Both of the sheets contain 10 elements in thickness which provides adequate results. For better solution the edges in the contact zones were finer. The contact definition between the parts is Augmented Lagrange formulation with a frictional coefficient $\mu=0.12$ between the parts. Between the punching tool and the simplified spring row the contact definition was bonded with MPC algorithm. The duration of the simulation is 3 time steps. The simulation is performed as a displacement controlled model. According to the final, measured bottom thickness and the measured piston displacement, the vertical movement of the clinching tool was 3.25 mm. The holder was constrained in vertical direction by a compression only support and in the 1st step a force ($F=1000$N) was applied which is constant in the 2nd and 3rd steps. In the 3rd step the tool was removed from the joint. The distribution of the equivalent plastic strain was checked with the unaveraged display option, because if it is not continuous, then the results are unacceptable from the point of view of nonlinear calculations. The unaveraged distribution of the plastic strain can be seen in Figure 3. The high (greater than 2) plastic strains are acceptable, according to the literature [9].
Validation of the model

In this study the formed geometry and the F-d curves were used to validate the FE results. The geometry comparison can be seen in Figure 4. The forming force - displacement results of the FE model can be compared after the correction of the measured values. The difference between the model and the measurement is acceptable. The simulated curve shows a very good agreement with the measured one (Figure 5).

According to [7] the measured curve can be divided into 3 main phases and 5 steps. In Phase I. the testing machine starts to work, the punching tool moves down, the holder moves downward to fix the sheets, the tool comes into contact with the upper sheet (punch side) and the joining process is started (Step I.). The tool punches the sheets and they move together (Step I. - Step II.) and this part of the process
continues until the first bending point (Step II.). The lower sheet (die side) reaches the die; that is why the slope of the curve changes after Phase I. In Phase II. the sheets start to flow around the punching tool and start to flow inside the free space of the die (Step III.). The last part of the process needs more deformation force; the curve rises with the highest slope. In Step IV. the punching tool reaches the end position. In this phase the setting force reaches the maximum as well. After this point the tool starts to move away from the joint with a certain slope depending on the stiffness of the machine. After Step V. the joint is totally released.

Figure 5 Comparison of the measured and simulated forming force and displacement (F-d) curve

**DETERMINATION OF THE INFLUENCE OF DIFFERENT PARAMETERS**

In this section the different influencers of the clinch joints is presented. Several parameter studies were done by FEA. The main geometrical parameters can be seen in Figure 1 c).

*Influence of the frictional coefficient*

To determine the effect of the frictional behaviour on the main parameters of the joint the model was set up a little different from the valid model. The global frictional coefficients were changed at the same time; any other aspects of the model were the same. The difference between the minimal value ($\mu=0.05$) and the maximal value ($\mu=0.2$) is almost 10 kN in force if the consideration is that the displacement is the same (Figure 6).
The undercut is highly responsible for the strength of the joint, so it is a good marker of the joints. In Figure 7 it can be seen that the difference is 0.027 mm, which leads to a lower strength.

In the following bar diagram the parameters can be seen (Figure 8). The protrusion height is slightly decreased, which is a favourable phenomenon, but only in the point of view of appearance. In a previous study [10] it was proved that the thicker bottom thickness leads to a better joint concerning mechanical strength. On the neck thickness the frictional condition between the tools and the sheets has a negligible effect.
Influence of the punching tool geometry

The punching tool in the test equipment can be replaced with another tool. To determine the effect of the different theoretical geometries a row of simulations were performed with the previously presented FE model. The examined parameters were the punch radius (R), the angle of the tip (α) and the angle of the punch cone (β) according to the following schematic figure (Figure 9). The original values were R=0.3, α=5° and β=2° respectively.

![Schematic punching tool geometry](image)

Figure 9 Schematic punching tool geometry

On the figures (Figure 10 a-d) the “0” is the so called original sized tool, the “-” and the “+” are the modified parameters according to the tables. The h (protrusion height) highly depends on the value of β in an inverse way. The α values which are greater than the original case have the most definitive effect on the undercut, but it leads to less favourable mechanical properties.

![Figure 10 Influence of the tool geometry to the geometry of the joint](image)
SUMMARY

A brief instruction of the clinched joints was presented. An experimental test and the validation of the FE model was carried out for DP600 steel sheets. With the valid model several parameters were studied and their effect was determined. Beyond the technological aspects the modelling questions were also examined.

REFERENCES

[6] ANSYS WB 18.2 User’s guide