Simulation of stress-strain state of resistance spot welded joints in NX environment

A L Fedorov, S V Mkrtychev, S Yu Mamushkina

Togliatti State University, 14 Belorusskaya St., Togliatti, 445020, Russia

E-mail: sm5006@yandex.ru

Abstract. The paper considers the study of the condition of resistance spot welded joints using their digital prototyping and the Finite Element Method (FEM). Numerical simulation of welded joints and the study of their stress-strain state were performed in the NX software environment. The obtained results fully coincide with the data on the destruction of real welded joints. The proposed simulation method and calculation technique can be recommended to design engineers for practical purposes.

1. Introduction
Resistance spot welding is a highly productive process. Therefore it is most widely used in mass production when joining car body panels. However, unlike welded joints made by arc welding, spot-welded joints are characterized by a strong-unequal stress distribution. Literature review shows, that under alternating loads and for materials with low plasticity the stress concentration the welded structure performs is significantly limited.

Thus, investigating the stress-strain state of welded joints made by resistance spot welding is of undoubted interest to practice [1,2]. One of the ways to solve this problem is simulation study of a digital prototype of the spot-welded joint [3,4].

2. Background
Figure 1a shows an example of a welded joint.

Here, the following variants of destruction are possible: along the weld spot (Figure 1b) and along the base metal (a weld spot pull-out, Figure 1c).

It is possible to make a cut of the weld spot if its diameter (d) is less than that recommended by GOST (Russian State Standard) [5] or is calculated by the formulas:

\[ d = 1.25 \delta + 4 \text{ mm}, \text{ if } \delta \leq 3 \text{ mm}; \]  
\[ d = 1.55 \delta + 5 \text{ mm}, \text{ if } \delta > 3 \text{ mm}, \]

where \( \delta \) is the thickness.
Figure 1. Variants of destruction of welded joints made by resistance spot welding (F - force applied to the sample, d – weld spot diameter).

There are slight discrepancies between the recommendations for the weld spot diameter in GOST and the diameters obtained from the calculated dependencies (1) and (2). So, for the thickness of the parts connected by resistance spot welding of 3 mm, the diameter of the weld spot according to the results of the calculation according to the formula (1) will be 7.75 mm. If the thickness of the parts to be joined is 5 mm, the weld spot diameter according to (2) will be 12.75 mm. According to GOST the diameter of the weld spot for a thickness of 3 mm to be joined is required at least 9 mm. The minimum overlap is required at least 21 mm. For a thickness of 5 mm, these parameters are 14 mm and 34 mm, respectively.

A weld spot pull-out occurs if its diameter exceeds the recommended one. Failure of the welded joint in the base metal is typical for joints with a relatively small width. It is true provided that the weld spot diameter is equal to the one recommended by GOST.

3. Digital prototyping spot-welded joint

The object of simulation is a single-spot lap joint.

NX environment (Siemens PLM Software) is utilized for simulation [6].

Simulation is performed according to technique presented below.

We build 4 models of a welded joint.

In the first model (Figure 2), with a plate thickness of 3 mm, we set the cylinder diameter to 5 mm.
In the second model (Figure 3) these parameters are 3 mm and 13 mm, respectively.
In the third model (Figure 4) - 5 mm and 10 mm, respectively.
And in the fourth model (Figure 5) - 5 mm and 17 mm, respectively.

When performing Boolean join operations, the cylinder joins the plates in the same way as a weld spot made by resistance spot welding.
4. Utilizing FEM module
Let us consider the process of calculating the developed digital prototypes of welded joints using FEM module [7].
Since the calculation of digital prototypes is performed using the finite element method, at the first stage we will split these models of joints into finite elements. We used for this a mesh of volumetric
finite elements - tetrahedrons. When calculating, it is important to set the size of the finite element. The smaller the size of the finite element, the higher the accuracy, but the greater the amount of computer time required to carry out calculations. The conducted studies of computer models showed that for the thicknesses of welded plates of 3 and 5 mm, the size of the final element of 1 mm provides acceptable results in terms of accuracy.

Hence, the size of the finite element when conducting research in the FEM module should be no more than 30% of the thickness of the investigated part. Steel 08kp, widely used in the automotive industry for the manufacturing of body panels, was taken as a material. The value of the yield point of this steel is 175 MPa, therefore, the load value for models of welded joints with a thickness of 3 mm was taken 20 kN and for models of welded joints with a thickness of 5 mm, 40 kN, respectively. In this case, the stress in the plates themselves was 130 MPa and 160 MPa, respectively, which is less than the yield point.

The load application diagram is shown in Figure 4.

Figure 5 shows the stress distribution for a plate thickness of 3 mm and weld spot diameter of 5 mm after performing the calculations. By clicking the “Edit Post View” button we can obtain various options for cutting the analyzed 3D object. It should be noted that figures 5-8 were obtained with the cutting plane along the plate (parallel to the ZOX plane) in the middle of the weld spot. The analysis shows that the stress concentration reaches a maximum along the perimeter of the weld spot. Hence, destruction will occur along the weld spot.

Figure 4. The load application diagram.
Figure 5. Simulation of stress distribution in the joint of plates with a thickness of 3 mm with a weld spot diameter of 5 mm.

The stress distribution for a plate thickness of 3 mm with a weld spot diameter of 13 mm is shown in Figure 6.

If in the previous figure the stresses are concentrated along the perimeter of the weld spot, then here the stress concentration spreads into the base metal, which is consistent with the fracture scenario by the weld spot pull-out.
The results of calculating the stress distribution for a plate with a thickness of 5 mm, with a weld spot diameter of 10 mm, are identical to those shown in Figure 5 for a plate with a thickness of 3 mm. Here, too, a characteristic point is the concentration of stresses along the perimeter of the weld spot without much penetration into the plate being welded.

For a plate with a thickness of 5 mm, with a weld spot diameter of 17 mm, the stress distribution is identical to that shown in Figure 6. As well as for a thickness of 3 mm, with a weld spot diameter exceeding the calculation recommendations and GOST, the stress concentration spreads into the depth of the joined plates.

Another characteristic feature of the operation of multi-spot joints made by resistance spot welding when the weld spots are arranged in a row parallel to the acting force is the low value of stresses at the middle weld spots.

To study this phenomenon of multi-spot joints, when digital prototyping, the size of the overlap was increased, while the cross-section of the plates remained unchanged and the distance between the spots was taken according to the recommendations of GOST. A joint containing 4 spots was simulated. The plate thickness was assumed to be 5 mm at a load of 40 kN.

The calculation results are shown in Figure 7. At the extreme points, there is a significant jump in stresses, while the average ones practically do not bear the load. When testing for rupture of such multi-spot joints, destruction begins at the extreme points. A detailed analysis of the stress distribution at the extreme points shows that the highest stress values are achieved at the outer diameter of the spot, Figure 8.

Figure 7. Simulation of stress distribution at multi-spot joint.
5. Conclusion
The paper presents an example of numerical simulation of a welded joint made by resistance spot welding in the NX environment. A welded joint model is built by digital prototyping a cylinder and combining it with 3D plate models using a Boolean operation. As a result of the study of such simulations by the FEM method, it was found out that an acceptable accuracy of calculations is achieved when the size of the finite element is not more than 30% of the thickness of the investigated part.

The obtained calculation results coincide with those obtained during the destruction of real samples in tensile studies. This allows us to conclude that this method of simulation and calculation of spot welded joints is applicable in practice of design engineers.

References
[1] Kuang J H, Liu A H A Study of the Stress Concentration Factor on Spot Welds 1990 Welding J 69(12) 468–474
[2] Daunys M, Dundulis R, Kiliņevičius S et al. 2020 Investigation of the Stress-Strain State in a Welded Joint with a Hard Single-V Butt Weld Strength Mater 52 440–450
[3] Nielsen C V, Zhang W, Perret W, Martins P A, Bay N 2015 Three-dimensional simulations of resistance spot welding Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering 229(7) 885–897
[4] Fedorov A L, Mkrtychev S V and Enik O A 2020 Utilizing NX CAD extension components to model welded joints J. Phys.: Conf. Ser. 1661 012204
[5] GOST 15878-79. Resistance welding. Welded joints. Design elements and dimensions Available at: https://docs.cntd.ru/document/1200006488
[6] NX for Design. Available at: https://www.plm.automation.siemens.com/global/ru/products/nx/nx-for-design.html
[7] Lei Z Z, Kang H T, Wang J H, Ren L C and Liu Y G 2012. Finite Element Analysis for Mechanical Characteristics of Resistance Spot Welding Process with Three Sheets Assemblies Applied Mechanics and Materials 233 369–373.