Technical Sciences, 2021, 24, 273–281

Yearbook peer-reviewed scientific journal
ISSN 1505-4675
e-ISSN 2083-4527
TECHNICAL SCIENCES
Homepage: https://czasopisma.uwm.edu.pl/index.php/ts/

UNIWERSYTET WARMIŃSKO-MAZURSKI W OLSZTYNIE

DOI: https://doi.org/10.31648/ts.6913

INFLUENCE OF VARIOUS METHODS OF MODELLING THE WELDING PROCESS IN THE CAE ENVIRONMENT ON THE OBTAINED DEFORMATION RESULTS

***Tomasz Zadorożny*[1], *Marcin Kalinowski*[2], *Mirosław Szczepanik*[3]**

[1]ORCID: 0000-0001-7216-1979

[2]ORCID: 0000-0002-2942-6537
Departament of Mechanical Engineering
PhD School, Silesian University of Technology in Gliwice

[3]ORCID: 0000-0001-7983-9463
Institute of Computational Mechanics and Engineering
Faculty of Mechanical Engineering
Silesian University of Technology in Gliwice

Received 06 July 2021, accepted 30 November 2021, available online 30 November 2021.

Keywords: process simulation, FEM, welding, heat deformation, mesh type.

Abstract

By simulating the welding process, potential non-conformities can be detected before serial production is launched, which can significantly reduce operation costs. There are many different possibilities for modeling the process, therefore it is very important to choose a method that will ensure high accuracy of the solution in a relatively short time. The article will present the influence of various methods of modeling the welding process in the CAE environment on the obtained deformation results. For the given geometry and type of weld, the thermal deformations have been simulated based on the Finite Element Method. Several analyzes were carried out using different process modeling approaches (mesh type). Finally, a comparison of the results for the discussed cases is presented to determine the influence of the parameters used on the deformation results obtained.

Correspondence: Tomasz Zadorożny, Katedra Mechaniki i Inżynierii Obliczeniowej, Wydział Mechaniczny Technologiczny, Wspólna Szkoła Doktorska, Politechnika Śląska, ul. Akademicka 2a, 44-100 Gliwice, e-mail: zadorozny.tomasz@gmail.com.

Introduction

Simulations of technological processes are difficult and time-consuming due to many factors influencing their progress. Properly conducted simulation of the technological process can provide many valuable tips. They allow improving the manufacturing process, contributing to a significant reduction in costs related to technological time and potential repairs. The work aims to select an appropriate modeling method to determine the course of thermal deformation in the area of the welded component. The results of this type of simulation can help in the optimal selection of the fixing points of the components to be welded.

During the welding process, the weld area is intensively heated and then cooled. The temperature gradient causes uneven expansion and contraction of the weld and the surrounding area, which results in permanent deformations. The form and size of deformation are influenced by many factors, including mechanical limitations and the amount of heat supplied, which depends on the process parameters (welding current, welding speed, etc.). In most cases, the change of these parameters is limited by the quality requirements for the joint, therefore the method that does not directly affect the quality of the weld is to use appropriate constraints points. Based on the conducted review of the available literature, where the test and simulation results were present, can be clearly stated that ensuring appropriate process parameters (including adjustment of constraints points) significantly affects the form of geometric distortions. In the vast majority of works, 3D elements were used to model the problem, the comparison with real tests confirmed the high accuracy of this methodology. The use of 3D elements allowed to obtain high accuracy of the results of residual stresses and the course of deformation. The simulations and tests concerned small structures where the use of such methodology did not significantly extend the computation time. For these constructions, it was determined that the most effective geometric limitations work when they are as close as possible to the weld line. In industrial practice, it is not always possible to locate mechanical constraints in this way, so it is important to optimize their location.

There are many modeling methods, they assume some simplifications aimed at limiting the size of the numerical model and thus the computing time. Large numerical models (such as a train roof) consisting of 3D elements pose many problems with the calculation time or the visualization itself in the software, therefore a very important aspect is to select the appropriate modeling method at an early stage of the research. In engineering practice, 2D elements are most often used to limit the size of the numerical model. For such complex issues, it must be ensured that the use of 2D elements will ensure the correct response of the structure. One of the approaches is to carry out a series of analyzes on sample models and to compare the results to determine the appropriate modeling method. The following part of the article presents a comparison of three different methods of modeling a welded joint in the simulation of a process aimed at determining thermal deformation. One of the commercial finite element method (FEM) HyperWorks program was used to solve the problem defined.

Description of samples and boundary conditions

The considered welded sample consists of aluminum sheets joined together by a fillet weld (Fig. 1). Three different modeling methods are considered in this article. In order to accurately determine the impact of the modeling method on the time of solving the task, different lengths of the sheets joined together were analyzed for each described below methods (Tab. 1):

– the first model consists of shell elements without additional elements in the weld area (only quad elements);

– the second model is an expansion of the first one with additional surfaces to simulate a fillet weld (only quad elements);

– the third model consists of 3D elements that represent the weld geometry (hexa and penta elements).

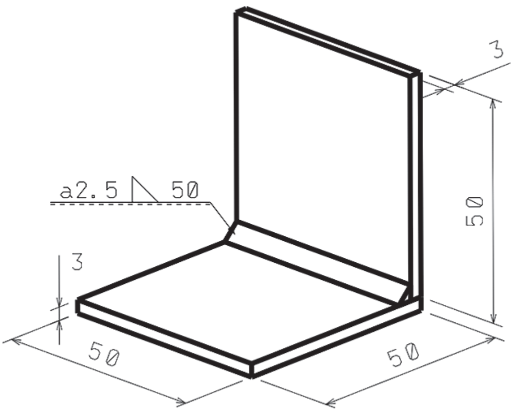

Fig. 1. Dimensions of samples

Table 1

Number of nodes and elements for different types of samples

Length	Sample I		Sample II		Sample III	
	nodes	elements	nodes	elements	nodes	elements
50 mm	2,278	2,178	2,482	2,475	6,902	4,488
200 mm	8,911	8,712	9,709	9,900	26,999	17,952
800 mm	35,443	34,848	38,617	3,960	107,387	71,808
3,200 mm	141,571	139,392	154,249	158,400	428,939	287,232
12,800 mm	566,083	557,568	616,777	633,600	1,715,147	1,148,928

The temperature value adopted in the analysis is 1,230°C, which corresponds to the real value during welding works. The temperature was applied to the point where the sheets were joined along the entire length using the first kind of boundary condition. The values shown were read after the sample had returned to ambient temperature which was set at 20°C. The heat transfer with the environment takes place over the entire surface, additional transfer through mechanical constraints points is not taken into account. Figure 2 shows a comparison of the weld modeling method for each of the samples and the applied constraints. An elastoplastic model of the material was used, Table 2 present the material data.

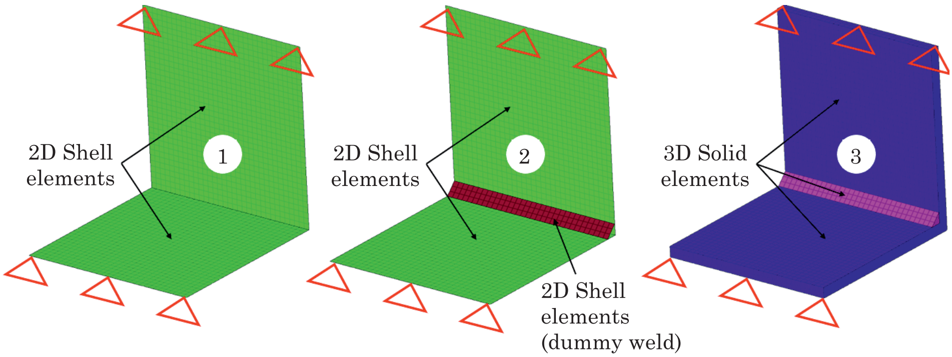

Fig. 2. Visualization of the modeling method and given boundary conditions

Table 2

Material properties

Young's modulus [MPa]	$7{\cdot}10^4$
Density [g/cm^3]	2.7
Poisson ratio	0.3
Thermal conductivity [W/(m·K)]	215
Heat transfer coefficient [W/(m^2·K)]	6.8
Thermal expansion coefficient [10^{-6}/K]	23.1

The Dirichlet boundary condition (the first kind of boundary condition) is based on the assumption that the function that solves a given problem takes predetermined values at the boundary of the domain.

For an ordinary differential equation of the 2nd order, the Dirichlet condition takes the following form:

$$y(a) = y_a, y(b) = y_b \tag{1}$$

where:

y_a, y_b – defined.

A typical and the most common application is its implementation for the Laplace equation, and where for a given area:

$$\Omega \subset \mathbb{R}^n$$

a solution is sought:

$$u: \overline{\Omega} \to \mathbb{R}$$

which is continuous in $\overline{\Omega}$ and fulfill the equation:

$$\Delta u = 0 \tag{2}$$

where:

Δ – a Laplace operator.

And the boundary condition takes the form:

$$u(x) = f(x) \quad \forall x \in \partial\Omega$$

where:

f – a given function defined at the edge of the region,

$f: \partial\Omega \to \mathbb{R}$.

When applying the first kind of condition, two relations should be fulfilled, the resulting system of equations can be written as:

$$\begin{cases} \Delta u = 0 & \text{on } \Omega \\ u = f & \text{in } \partial\Omega \end{cases}$$

This condition is used when describing the phenomenon of heat propagation (transport of internal energy), its adoption means that the temperature at the edge of the object is determined and fully controlled. This means that the area is in contact with the source of thermal energy and the object shows good conductivity properties.

Results

Figure 3 shows the displacement distribution on the deformed model, the adopted scale of deformation is 5:1. Table 3 summarizes the deformation values obtained along with the difference expressed as a percentage in relation to the third sample. The third sample can be taken as representative, the numerical model consisting of 3D elements reflects the real geometry to the greatest extent. This modeling method was widely used in various research works and the numerical simulations of the process were compared with the results of real tests. The test results clearly show that the use of 3D elements guarantees high accuracy in terms of the obtained values of residual stresses and the course of deformation. For this reason, it is justified to consider this sample representative.

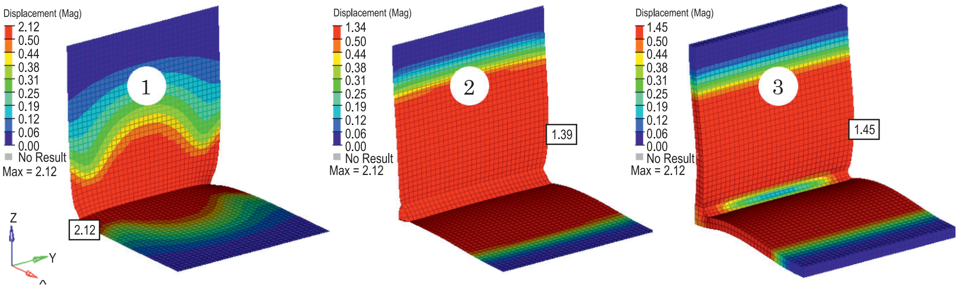

Fig. 3. Distribution of displacements on the deformed model, scale 5:1

Table 3

Maximum displacement values

Sample	Maximum displacement [mm]	Difference [%]
I	2.12	31.6
II	1.39	4.3
III	1.45	0

The course of the deformations obtained for the second and third samples is similar, and the difference in the values of the obtained displacements is 4.3%. For the first sample, the deformation distribution and their maximum value differ significantly from the other two samples. It can therefore be concluded that the geometry of the weld has a big impact on the displacement distribution, which locally increases the stiffness of the structure and directly impacted the response of the system. The deformation distribution for the first sample indicates its unnatural deformation in the connection line where energy in the form of heat has been applied.

Detailed data on the time needed to simulate individual samples are presented in Table 4 and Figure 4. For the length of the joined sheets up to 200 mm, it can be noticed that the modeling method slightly affects the total calculation time. It is caused by a relatively small number of elements, so the time needed to create and solve the stiffness matrix for these cases is very similar. A clear difference in the computational time can be noticed for samples with a length of 3,200 mm, where the solution time of the model consisting of 3D elements (sample III) is significantly longer compared to other samples. With a further increase in the length of the sample, the disproportions increase, clearly indicating the third modeling method is the most time-consuming.

Table 4

Time to solve the task for different scenarios [s]

Sample	50 mm	200 mm	800 mm	3,200 mm	12,800 mm
I	10	12	21	49	186
II	10	12	21	58	227
III	18	18	26	115	545

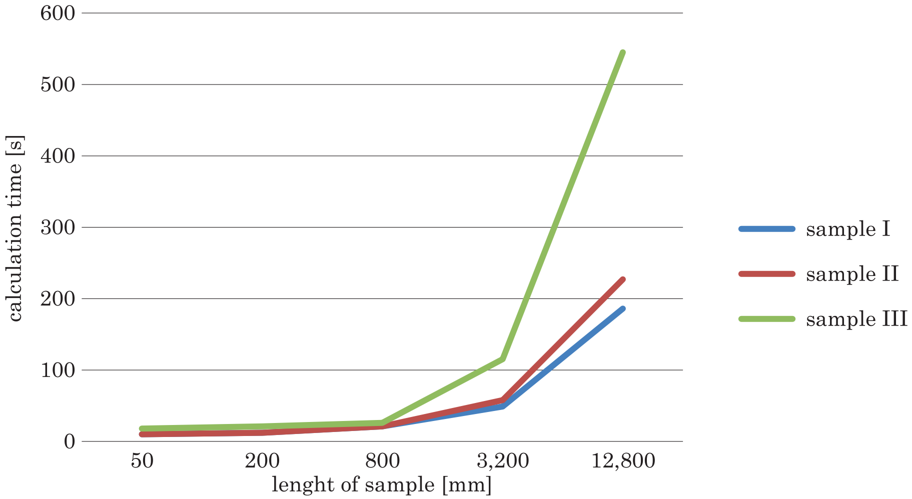

Fig. 4. Comparing the times of solving the task for different scenarios

Conclusions

Thermal deformation of welded structures is a serious problem that entails costly repairs. Numerical calculations allow simulating the deformations with high accuracy. The introduction of this type of simulation into the design process in the case of low-series structures allows avoiding many problems already at the first assembly, where excessive deformations would occur without additional calculations.

Based on the obtained results, it can be seen that the modeling method has a clear impact on the time needed to simulate the process. The differences between the individual computational times increase with the increase in the number of elements of the numerical model, which is a natural phenomenon. Because welded structures are quite big and that simulation requires the creation of a numerical model that consists of a large number of finite elements, the selection of the appropriate modeling method is crucial. As can be observed, the course of deformation in the area of the sample created with the use of shell elements together with the surface simulating the weld is similar to that obtained for a model consisting of 3D elements, the difference is 4.3%. It can therefore be considered that modeling with the use of shell elements is the appropriate approach provided that an additional plane simulating the weld is used. For the vast majority of structures, simulation of the process would not be possible using 3D element modeling, therefore such a simplification seems highly justified. An alternative modeling method using shell elements with additional surface significantly reduces the analysis solution time, which is equivalent to the possibility of simulating the process for large computational models. To optimize the production process, the deformations courses are relevant, and that for the above-mentioned methods are similar.

The deformation distribution obtained for the first sample, where the additional surface in the weld area is not taken into account, differs distinctly from the other results and does not coincide with the deformations obtained in practice. For these reasons, the use of such an approach in modeling seems to be unjustified for this kind of joints.

Acknowledgements

This research was co-financed by the Ministry of Education and Science of Poland under grant No DWD/3/33/2019.

References

Aalami-Aleagha M.E., Eslampanah A. 2012. *Mechanical constraint effect on residual stress and distortion in T-fillet welds by three-dimensional finite element analysis*. Proceedings of the Institution of Mechanical Engineers - Part B, 227(2): 315-323.

Barsoum Z., Lundback A. 2001. *Simplified FE welding simulation of fillet welds – 3D effects on the formation residual stresses*. Engineering Failure Analysis, 16(9): 2281-2289.

Cosimo A., Cardona A., Idelsohn S.R. 2010. *General treatment of essential boundary conditions in reduced order models for non-linear problems*. Advanced Modeling and Simulation in Engineering Sciences, 3(1): 626-636.

Fricke S., Keim E., Schmidt J. 2001. *Numerical weld modeling – a method for calculating weld-induced residual stresses*. Nuclear Engineering and Design, 206(2-3): 139-150.

Gimenez G., Errera M., Baillis D., Smith Y., Pardo F. 2015. *Analysis of Dirichlet-Robin interface condition in transient conjugate heat transfer problems. Application to a flat plate with convection*. European Conference on Turbomachinery Fluid Dynamics and Thermodynamics, Madrid.

Liang W., Murukawa H., Deng D. 2015. *Investigation of welding residual stress distribution in a thick-plate joint with an emphasis on the features near weld end-start*. Materials & Design, 67: 303-312.

Mazumder S. 2016. *Numerical methods for partial differential equations. Finite difference and finite volume methods*. Elsevier, Amsterdam.

ROSCA A.S., ROSCA D. 2006. *About using the Dirichlet boundary conditions in heat transfer equation solved lay finite element method*. International Jurnal of Computers, Communications & Control, 1: 405-409.

SZABO T. 2010. *On the Discretization Time-Step in the Finite Element Theta-Method of the Two-Dimensional Discrete Heat Equation*. Large-Scale Scientific Computing, 591: 626-636.

VENKATKUMAR D., DURAIRAJ R. 2019. *Effect of Boundary Conditions on Residual Stresses and Distortion in 316 Stainless Steel Butt Welded Plate*. High Temperature Materials and Processes, 38: 827-836.

WEN S.W., HILTON P., FARRUGIA D.C.J. 2009. *Finite element modelling of a submerged arc welding process*. Journal of Materials Processing Technology, 119(1-3): 203-209.

ZADOROŻNY T., SZCZEPANIK M. 2020. *Numerical simulation of the welding process, the influence of constraint points locations on thermal deformations*. Scientific-Expert conference on reilways RAILCON 2020, Nis, p. 145-148.

ZIENKIEWICZ O.C., TAYLOR R.L., ZHU J.Z. 2013. *The Finite Element Method: Its Basis and Fundamentals*. Elsevier, Amsterdam.