

# Comparison of Residual Stresses in Butt-Welded Plates Using Software Packages Abaqus and Ansys

Mato Perić<sup>1)</sup>  
Dragi Stamenković<sup>2)</sup>  
Vili Milković<sup>3)</sup>

Nowadays software packages Abaqus and Ansys are very often used for numerical simulation of welding process. The procedure of weld modeling by these two software packages is slightly different, which can result in differences in obtained results. In order to compare the results obtained by using these two software packages, a parallel analysis of residual stresses in the case of butt-weld of two plates has been carried out. The residual stresses have been calculated in cases of a shell/3D modeling and a full 3D modeling.

*Key words:* welded structure, butt-welding, plate, residual stresses, process modeling, finite element method, software realization, parallel analysis.

## Introduction

**M**ETALURGICAL joints made by welding are extensively used in fabrication industry, offshore structures and pressure vessels. Due to localised heating by the welding process and subsequent rapid cooling, residual stress and distortions can occur near the weld. Such stresses and deformations may cause serious problems in welded constructions. Combination of strong tensile residual stresses and stresses due to workload in the area of a welded joint could accelerate the initiation of a fatigue crack, brittle fracture or stress corrosion cracking. In order to estimate residual stresses, in the last 30 years, the finite element method has successfully been used in order to substitute a rather expensive experiment [1-5]. As presented in literature, accurate predictions require the use of three-dimensional models that are often very complex and computationally expensive.

In the present investigation the shell/3D modelling capability is used to increase the efficiency and accuracy of the numerical analysis of welding processes. Residual stresses in the butt-weld of two plates were considered. The main idea of the shell/3D modelling technique is to perform shell-3D transition. Therein, three-dimensional finite elements are connected to shell elements by using a shell-to-solid coupling technique. Thus, the technique combines the accuracy of the full three-dimensional solution with the computational efficiency of a shell finite element model.

## Computational Approach

The simulations performed in this work use the sequential thermal-stress solution procedure in which the transient heat transfer analysis is followed by the thermal

stress analysis. Temperatures predicted by the heat transfer analysis are used as loading for the thermal stress analysis. To solve the problem here, shell and solid elements are applied. Solid elements are used close to the weld where the temperature gradients are high. On the other hand, the area outside the welding zone where temperature gradients are not prominent is modeled with shell elements. The transition between the shell and solid elements is achieved by tie constraint in case of thermal analysis; while in the stress analysis transition is achieved by using the shell-to-solid coupling option in references [6, 7].

## Butt weld of two plates

For the verification of the shell/3D and full 3D modeling, examples of butt weld of two plates made of S15C carbon steel [1] are chosen. In both cases, it is assumed that the base metal and the weld metal have the same thermal and mechanical properties.

An elastic-perfectly plastic material behavior model is assumed and its mechanical as well as thermal properties are shown in Figs.2 and 3. The welding parameters chosen for this analysis are as follows: tungsten inert gas (TIG) welding, welding current  $I=300\text{A}$ , welding voltage  $U=21\text{V}$  and welding speed  $v=10\text{ mm/s}$ . Since the data for the convective heat transfer coefficient  $k$ , the efficiency of heat input  $E_{ff}$  and the emissivity  $\varepsilon$  are not provided in Ref. [1], the following values are assumed:  $k=15\text{ W/m}^2\text{K}$ ,  $E_{ff}=70\%$  and the  $\varepsilon=0.77$ .

The butt weld of two plates is modeled in a single pass. Owing to symmetry, just one side of the plate is modeled. Both analyses, the thermal and the stress one, are carried out using a standard method for weld modeling.

<sup>1)</sup> Bureau of Energetics and Mechanical Engineering Ltd., Petrovaradinska 7, 10000 Zagreb, CROATIA

<sup>2)</sup> Termoelektrona Ltd. Uralska 9, 11060 Belgrade, SERBIA

<sup>3)</sup> Croatian Railways Ltd. Strojarska 17, 10000 Zagreb, CROATIA

Shell/3D model

Fig.1 shows the computational model of the butt weld of two plates. The length of each plate is 200 mm, the width is 100 mm and the thickness is 6 mm (Fig.1).

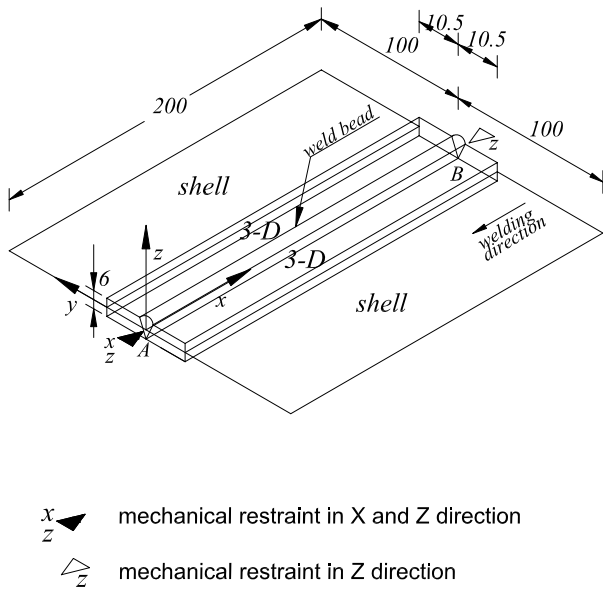


Figure 1. Geometry of welded plates

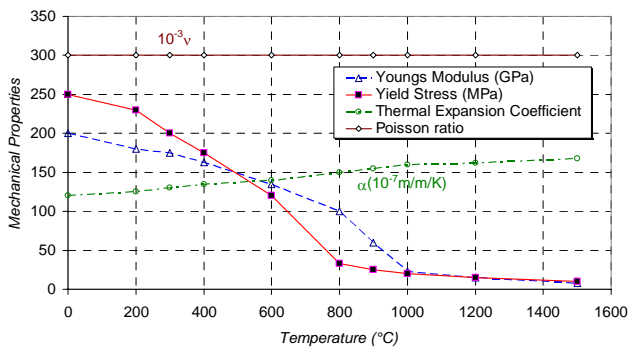


Figure 2. Mechanical properties of S15C steel

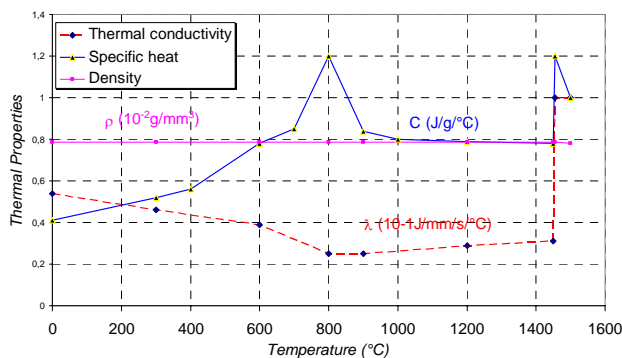


Figure 3. Thermal properties of S15C steel

In the case of shell/3D model, the mesh consisted of 960 3D elements and 240 shell elements. 3D elements were used on the part of the plate close to the root of the welded joint.

By using Abaqus, in the thermal analysis DC3D8 solid elements and DS4 shell elements were used, while C3D8 solid elements and S4 elements were applied for the mechanical analysis.

On the other hand, by using Ansys, in thermal analysis SOLID 70 elements and SHELL 57 shell elements were

used, while SOLID 45 elements and SHELL 43 elements were applied for the mechanical analysis.

On the part of the plate modeled with solid elements, the size of the smallest elements was 5 x 1,5 x 1 mm and they gradually became larger as the distance from the root of the welded joint increased.

The same mesh of elements was used in the case of modeling with both Abaqus and Ansys.

The temperature fields of 10s after the start of welding by using Ansys and Abaqus are shown in Fig.4 and 5.

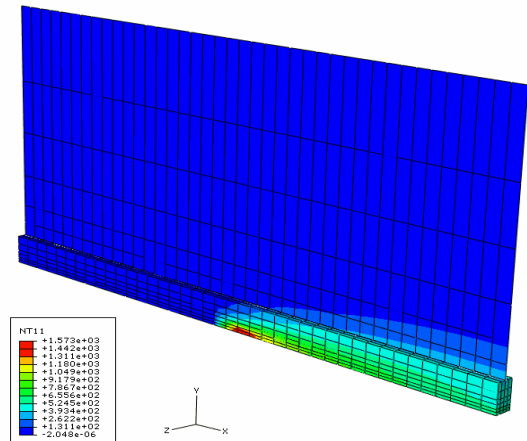


Figure 4. Temperature field of 10s after the start of welding in the case of modeling of welded plates by 3D/shell elements, Abaqus

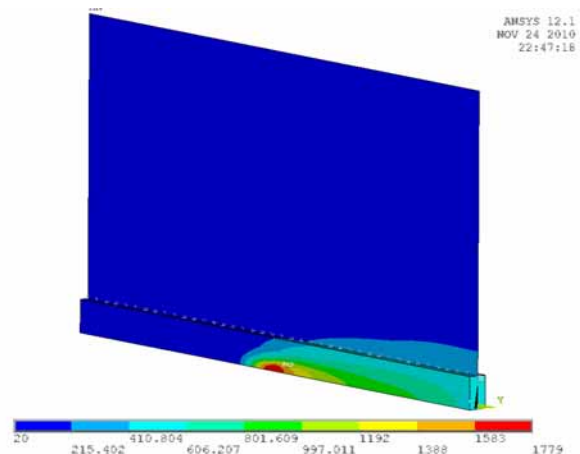


Figure 5. Temperature field of 10s after the start of welding in the case of modeling of welded plates by 3D/shell elements, Ansys

Figs.6 and 7 show the comparison between the temperature profiles in the case of 3D and shell/3D modeling in the middle of the plate (x=100 mm), 10 seconds after the beginning of welding process.

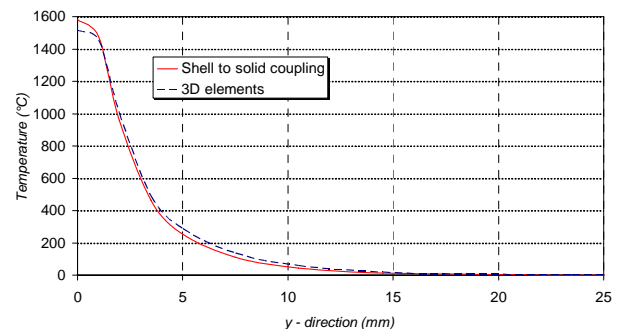


Figure 6. Comparison of temperature profiles in the case of modeling with 3D elements and shell to solid technique in the middle of the plate, x=100 mm, Abaqus

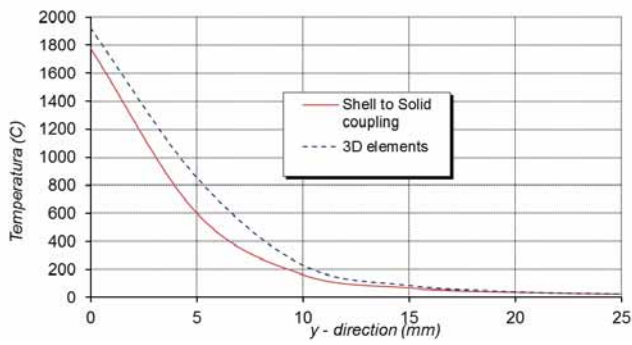


Figure 7. Comparison of temperature profiles in the case of modeling with 3D elements and shel/3D in the middle of the plate, x=100 mm, Ansys

Fig.8 shows the residual stress in the x direction in the middle of the plate, x=100 mm, in Abaqus and Ansys.

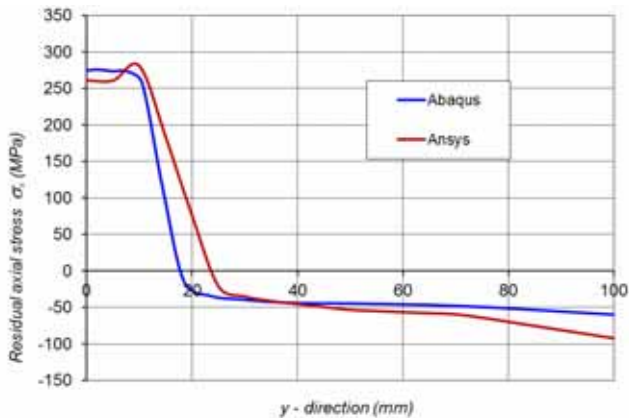


Figure 8. Residual stress in the x direction in the middle of the plate, x=100 mm, Abaqus/Ansys

Fig.9 shows the residual stress in the x direction in the middle of the plate, x=100 mm, in Abaqus and Ansys.

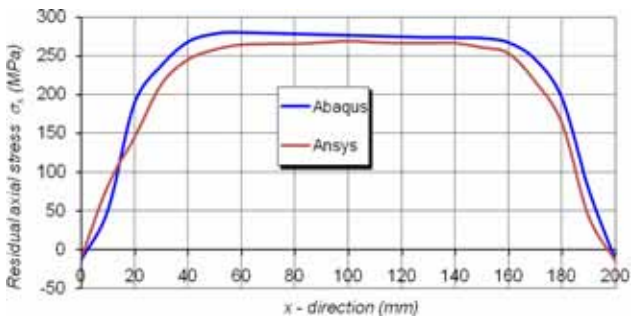


Figure 9. Residual stress in the x direction in the plate at mid span along the weld, Abaqus/Ansys

Full 3D model

The butt weld of two plates is modeled in a single pass. Owing to symmetry, just one side of the plate is modeled.

The three-dimensional mesh consisted of 2240 elements. The same mesh is used for both the thermal and mechanical analysis. In the case of the thermal model, DC3D8 elements were used while C3D8 elements were used for the mechanical model (Abaqus).

In the case of the thermal model, SOLID 70 elements were used while for the mechanical model SOLID 45 elements were used (Ansys).

The size of the smallest elements, used at the root of the welded joint, is 5 x 1.5 x 1 mm. The size of elements enlarged as the distance from the root of the welded joint to the edge of

the plate increased. The same mesh of elements was used in the case of modeling with both Abaqus and Ansys.

The temperature fields of 10s after the start of welding using Ansys and Abaqus are shown in Fig.10 and 11.

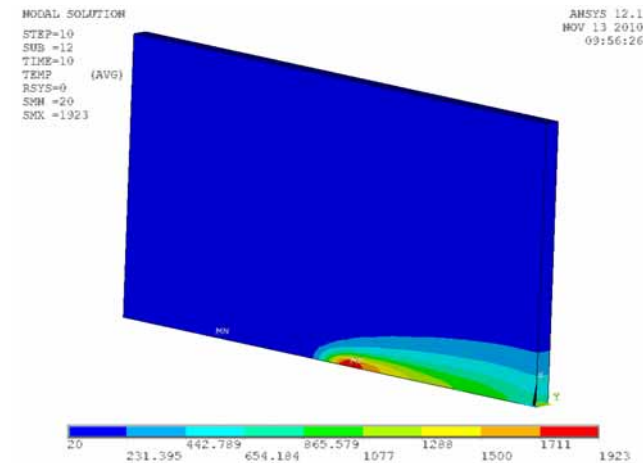


Figure 10. Temperature field of 10s after the start of welding in the case of modeling of welded plates by 3D elements, Ansys

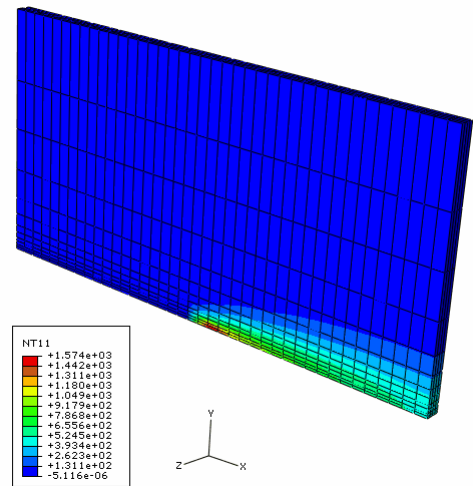


Figure 11. Temperature field of 10s after the start of welding in the case of modeling of welded plates by 3D elements, Abaqus

Figs.12 and 13 show the residual stress distribution using Abaqus and Ansys after the welding process in two butt welded plates.

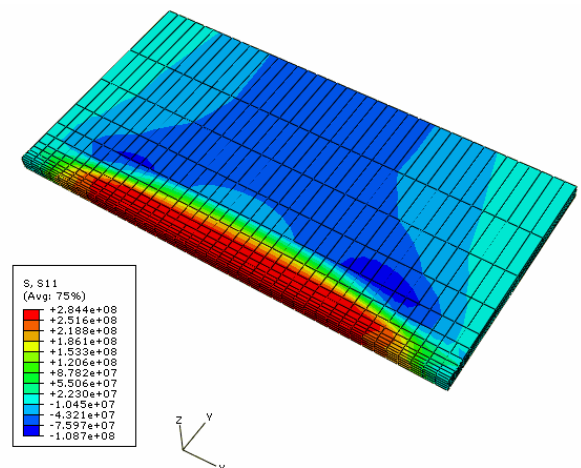


Figure 12. Stress field after welding in the case of modeling of welded plates by 3D elements, Abaqus

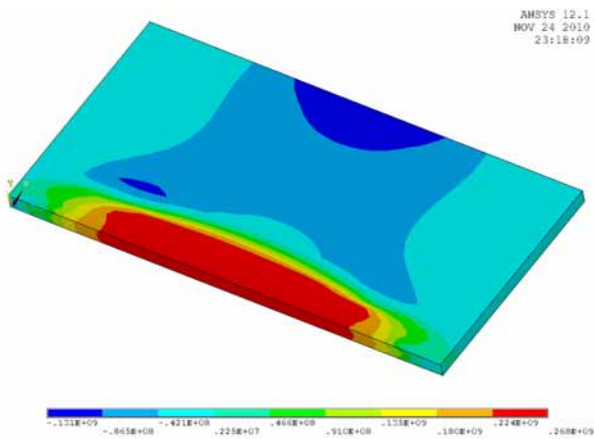


Figure 13. Stress field after welding in the case of modeling of welded plates by 3D elements, Ansys

Fig.14 presents the residual stresses in the direction perpendicular to the direction of welding in the middle of the plate. Fig.15 presents the residual stresses in the direction of welding at mid span.

The obtained residual stress values were calculated using both Abaqus and Ansys.

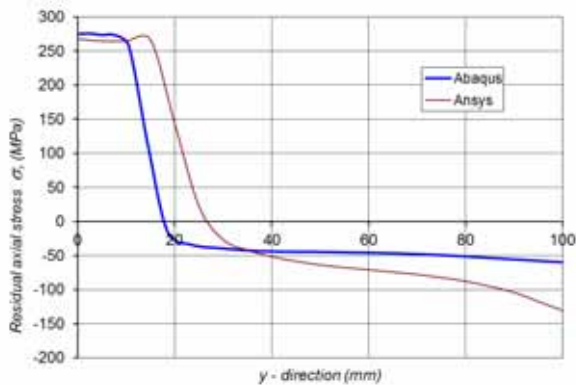


Figure 14. Residual stress in the  $x$  direction in the middle of the plate,  $x=100$  mm

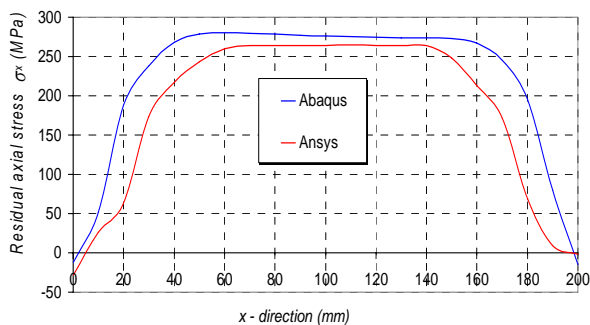


Figure 15. Residual stress in the  $x$  direction plate at mid span along the weld, Abaqus/Ansys

### Discussion and conclusion

Modeling of the welding process with the application of a three-dimensional model frequently requires considerable computer resources. On the other hand, with shell modeling it is not possible to obtain a more detailed distribution of temperature and stress alongside the wall. For this reason, a shell/3D modeling technique appears to be a possible solution, where a particularly interesting part of the model is modeled with a finer 3D mesh, and the rest of the model with shell elements.

The presented analysis included separate simulations of modeling of butt weld of two steel plates with 3D elements

and the shell/3D modeling technique. In the described process, we have shown the comparison of the maximum temperatures of welding (Figs.6 and 7).

From the results it could be deduced that the distribution of the temperature profiles for the case of the modeling of weld with 3D elements corresponds well with the modeling of weld by the shell/3D modelling technique.

The comparisons of temperature fields, modeled by shell/3D elements in Abaqus and Ansys (Figs.4, 5, 6 and 7), show that Ansys gives a slightly higher temperature in the root of the weld. This difference quickly vanishes by moving away from the weld pool towards the plate edges.

The comparisons of residual stresses in the  $x$  direction, (Fig.8) in the middle of the plate,  $x = 100$  mm, modeled by shell/3D elements in Abaqus and Ansys show that stresses  $\sigma_x$  in the root of weld are approximately the same while the difference is visible on the outer edge of the plate.

The comparison of residual stresses in the  $x$  direction along the weld modeled by shell/3D elements in Abaqus and Ansys (Fig.9), at mid span shows that there are no significant result deviations.

From the above mentioned analyses we can conclude that the shell/3D modelling, based on the combination of the three-dimensional and the shell model could be applied very efficiently for a detailed consideration of welding processes.

It is concluded that the shell/3D modeling technique, based on the transition from the shell model to the 3D model discretization with a finer mesh of elements, is very efficient for this purpose.

The comparisons of temperature fields modeled by the shell/3D elements in Abaqus and Ansys (Figs 6, 7, 10 and 11) show that Ansys gives a slightly higher temperature in the root of the weld. This difference quickly vanishes by moving away from the root of the weld towards the plate edges.

The obtained residual stress values in the  $x$  direction (Figs.12, 13 and 14) in the middle of the plate,  $x=100$  mm, were calculated using both Abaqus and Ansys. The maximum values of residual stresses near the weld bead have approximately the same values. However, residual stresses at the ends of the plates are considerably different.

The comparisons of residual stresses in the  $x$  direction along the weld modeled by full 3D elements in Abaqus and Ansys (Figs.12, 13, and 15), in the plate at mid span show that there are no significant deviations in the obtained results.

### References

- [1] DENG, D.: *FEM prediction of welding residual stress and distortion in carbon steel considering phase transformation effects*, Materials and Design 30 (2009) 359-366
- [2] AKBARI, M., MIRESMAEILI, R.: 2008. *Experimental and numerical analyses of residual stress distributions in TIG welding process for 304L stainless steel*, J. Materials Processing Technology, in press
- [3] ZENG, Z., WANG, L., WANG, Y., ZHANG, Z.: *Numerical and experimental investigation on temperature distribution of the discontinuous welding*, Computational Materials Science, in press
- [4] PERIĆ, M., TONKOVIĆ, Z., KARSAJ, I.: *Numerical analysis of residual stresses using a shell/3D modeling technique*, Proceedings of the International Conference on Advances in Welding Science and Technology for Construction, Energy and Transportation AWST-2010, pp.75-80.
- [5] STAMENKOVIĆ, D., VASOVIĆ, I.: *Finite Element Analysis of Residual Stresses in Butt Welding Two Similar Plates* Scientific Technical Review, ISSN 1820-0206, 2009, Vol. LIX, No.1, pp.57-60.
- [6] ABAQUS/Standard, *User's guide and theoretical manual*, Version 6.8, Hibbit, Karlsson & Serensen, Inc. (2008).
- [7] *Theory Reference for ANSYS and ANSYS Workbench*, ANSYS Inc. Release 12.1

## **Poređenje zaostalih napona kod sučeono zavarenih ploča korišćenjem softverskih paketa Abaqus i Ansys**

U današnje vreme za numeričku simulaciju procesa zavarivanja vrlo često se koriste softverski paketi Abaqus i Ansys. Postupak modelovanja zavarenog spoja kod ovih softverskih paketa neznatno se razlikuje što može rezultirati razlikama u dobijenim rezultatima. U cilju poređenja rezultata dobijenim korišćenjem ovih softverskih paketa urađena je uporedna analiza zaostalih napona za slučaj sučeono zavarenih ploča. Zaostali naponi izračunati su za slučajeve shell/3D modelovanja kao i za slučaj potpunog modelovanja sa 3D elementima.

*Ključne reči:* zavarena konstrukcija, sučeono zavarivanje, ploča, zaostali naponi, modelovanje procesa, metoda konačnih elemenata, softverska realizacija, uporedna analiza.

## **Сравнивание остаточных напряжений у встык сварённых плит при пользовании коммерческих пакетов программного обеспечения компьютера Abaqus и Ansys**

В наше время для цифрового моделирования процесса сварки очень часто пользуются коммерческими пакетами программного обеспечения компьютера Abaqus и Ansys. Методика проведения моделирования сварного соединения у этих пакетов программного обеспечения слегка различна, что в итоге может привести и к различиям в полученных результатах. В целях сравнения результатов полученных при пользовании этими коммерческими пакетами программного обеспечения сделан аналогичный анализ остаточных напряжений у встык сварённых плит. Остаточные напряжения рассчитаны на случаи shell/3D моделирования, а в том числе и на случаи моделирования 3D элементами.

*Ключевые слова:* сварённая конструкция, сварка встык, плита, остаточные напряжения, моделирование процесса, метод конечных элементов, реализация программного обеспечения, аналогичный анализ.

## **La comparaison des tensions résiduelles chez les plaques soudées en bout via progiciels Abaqus et Ansys**

Les progiciels Abacus et Ansys sont très utilisés de nos jours pour la simulation numérique du processus du soudage . Le procédé du soudage de la jointe soudée chez ces progiciels se différencie peu ce qui peut causer les différences dans les résultats obtenus. Dans le but de comparer les résultats obtenus par l'emploi des progiciels cités on a fait l'analyse comparée des tensions résiduelles pour le cas des plaques soudées en bout. Les tensions résiduelles sont calculées pour les cas de modélisation shell / 3D ainsi que pour le cas de la modélisation complète avec les éléments 3D.

*Mots clés:* construction soudée, soudage en bout, plaque, tensions résiduelles, modélisation du processus, méthode des éléments finis, réalisation logiciel, analyse comparée.