Društvo za tehniku zavarivanja Slavonski Brod

12. Međunarodno znanstveno-stručno savjetovanje SBZ 2023

„STROJARSKE TEHNOLOGIJE U IZRADI ZAVARENIH KONSTRUKCIJA I PROIZVODA, SBZ 2023."

Slavonski Brod, 26. i 27. 04. 2023. i Požega 28. 04. 2023.

IMPLEMENTATION OF BIRTH AND DEATH TECHNIQUE IN ANSYS SOFTWARE FOR WELDING DISTORTION SIMULATION

P. Poljak[1,*], M. Pavković[1], P. Konjatić[1], I. Gelo[1]

[1]Mechanical Engineering Faculty in Slavonski Brod, University of Slavonski Brod, Croatia

[*] Corresponding Author. E-mail: ppoljak@unisb.hr

Abstract

This paper presents numerical simulation of welding procedure using finite element method implemented in Ansys software. Element birth and death tehnique is applied for purposes of this simulation, which alters the Young's Modulus of the weld sections reducing it to approximately zero value in order to simulate weld generation. This tehnique enables generation of the weld in desired number of steps, thus simulating heat input to the welded parts which can cause distorsion of welded components. Numerical analysis of welded joint on steel structures is performed on a stainless steel manhole cover example for determination of deformation values caused by the welding procedure.

Keywords: FEM simulation, welding distortion, element birth and death, welding simulation

1. Introduction

Manhole covers and gully tops are an inevitable part of every industrial building and their surrounding areas where they are used as a closing lid over openings for large maintenance holes and inspection chambers. They vary in size and shape and usually consist of lid and frame that tends to be fixed into concrete slab, while the lid must be removable with or without opening mechanism to allow inspecting or service personnel to enter the chamber below the opening. Since the frame has rigid connection to the slab, lid should be the primary focus during the design process. EN 124-1:2015 [4] and EN 124-3:2015 [5] are used for design of gully tops and manhole covers, classification, performance requirements and test methods. In this paper, a cover with lid dimensions of 870x870 mm was used. It consists of steel plates with 3 and 4 mm thickness. Thicker one is used for the lid itself, while the thinner ones are laser cut to a predefined dimension and used as stiffening plates, also referred to as ribs, to provide bearing capacity of the cover while maintaining low weight. TIG welding process is used with 130 A welding current. T-joint fillet weld is selected with weld dimension a2. The ribs and the lid are welded with approximate weld length intervals of 50 mm which represents 50 mm weld on one side of the fillet weld, then 50 mm on the other side, etc. Since the heat input during welding is significant and the weld length is very long with most of the welds being applied to the lid, plate distortion is inevitable. Since the covers usually have to be waterproof in order to prevent the rainwater getting in and keep the chamber dry, a sealing gasket is added at the edge of the lid. For the sealing functionality to be preserved, the lid must not be distorted beyond the gasket thickness, therefore, displacements must be measured and controlled. Even though experienced welders can avoid significant distortions, due to large number of welding points, there is always a certain amount of displacement present. In this paper a finite element method (FEM) supported approach has been used to predict distorted shape of the lid and displacement values. 3D geometry model was created and implemented into Ansys software, where element birth and death technique is applied. This technique allows the simulation of incremental heat input, thus applying thermal load in more realistic manner. Material used for cover production is stainless steel 316L with welding filler material of the same type as the steel plates. This material is used for all parts of the lid, as well as the frame. As mentioned above, the lid is a primary focus from the design standpoint and will be used in this paper as well. Frame will not be analyzed in this paper.

2. Calculation example geometry

The lid is made of rectangular shaped 4 mm thick stainless steel plate with dimensions 870x870 mm, and stiffened with 3 mm thick rib plates. Lid dimensions and rib arrangement are presented on Figure 1, where the dimensions on the ribs represent the middle surface of the steel plate.

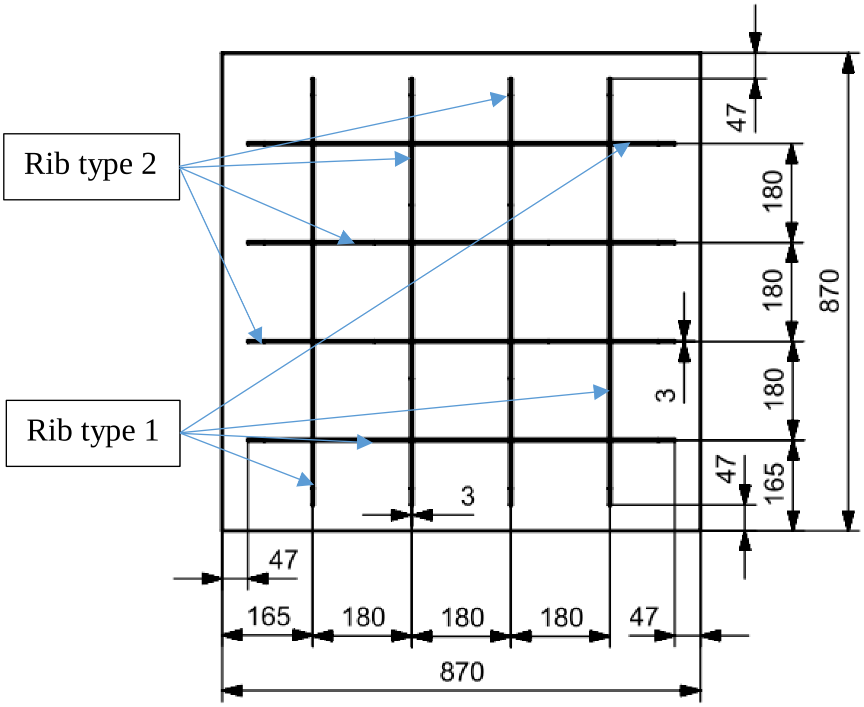

Figure 1. Lid dimensions and rib arrangement, *mm*

To provide enough bearing capacity, ribs are defined in two types, as presented on Figures 1 and 2. While the thickness and the length of both rib types are identical, the profile is different. Higher ribs are used for inner part of the cover since the edges of the lid plate are supported by the frame.

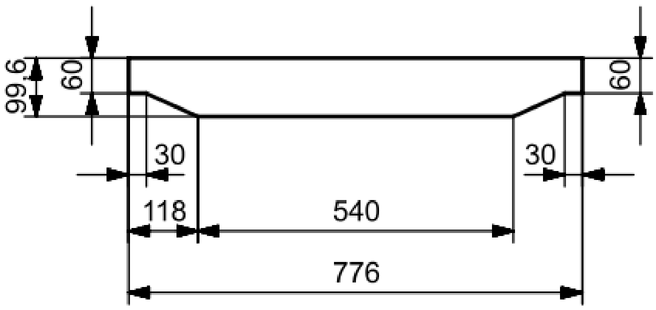

Figure 2. Rib type 1 dimensions, *mm*

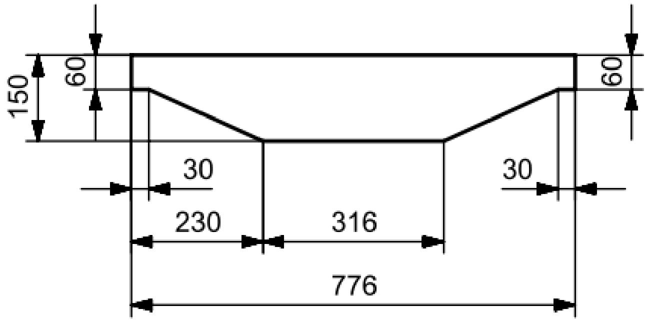

Figure 3. Rib type 2 dimensions, *mm*

Beside FEM calculation, this cover is produced according to the above-mentioned dimensions and other input data. Figure 4 presents the realistic cover lid with ribs welded to it.

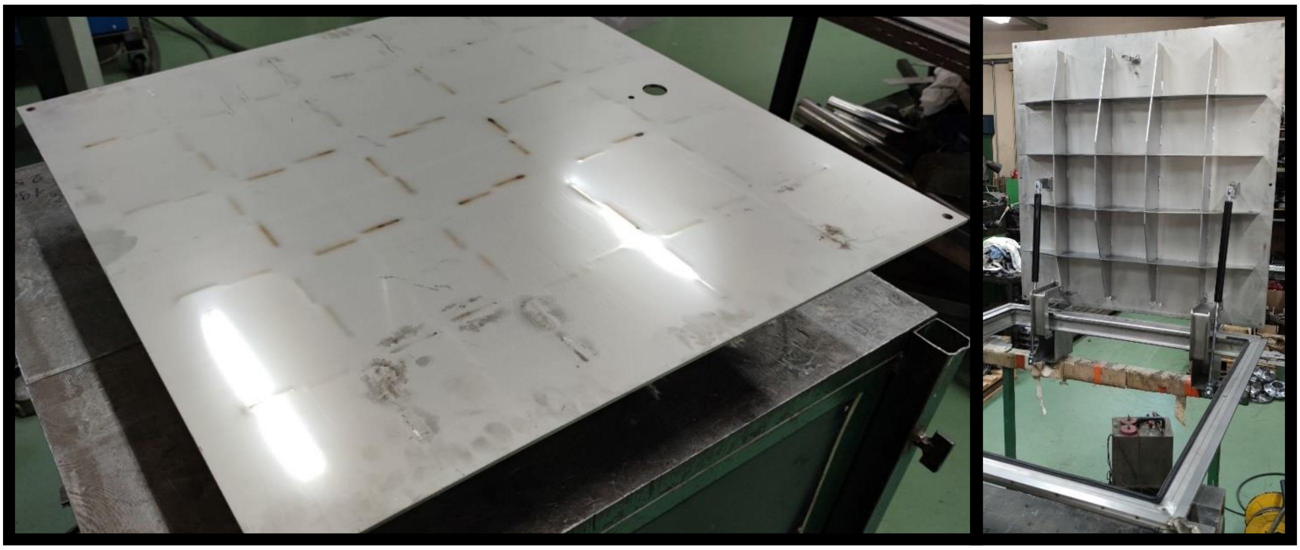

Figure 4. Manhole cover

3. Finite element model

The 3D modeling program used in this paper is Siemens NX 12. Due to limited memory and computer disk space and the simplicity of calculations, only a quarter of the model was generated. This was possible because the model is symmetrical in two planes. Generated model ready to be imported into the Ansys software is presented on Figure 5.

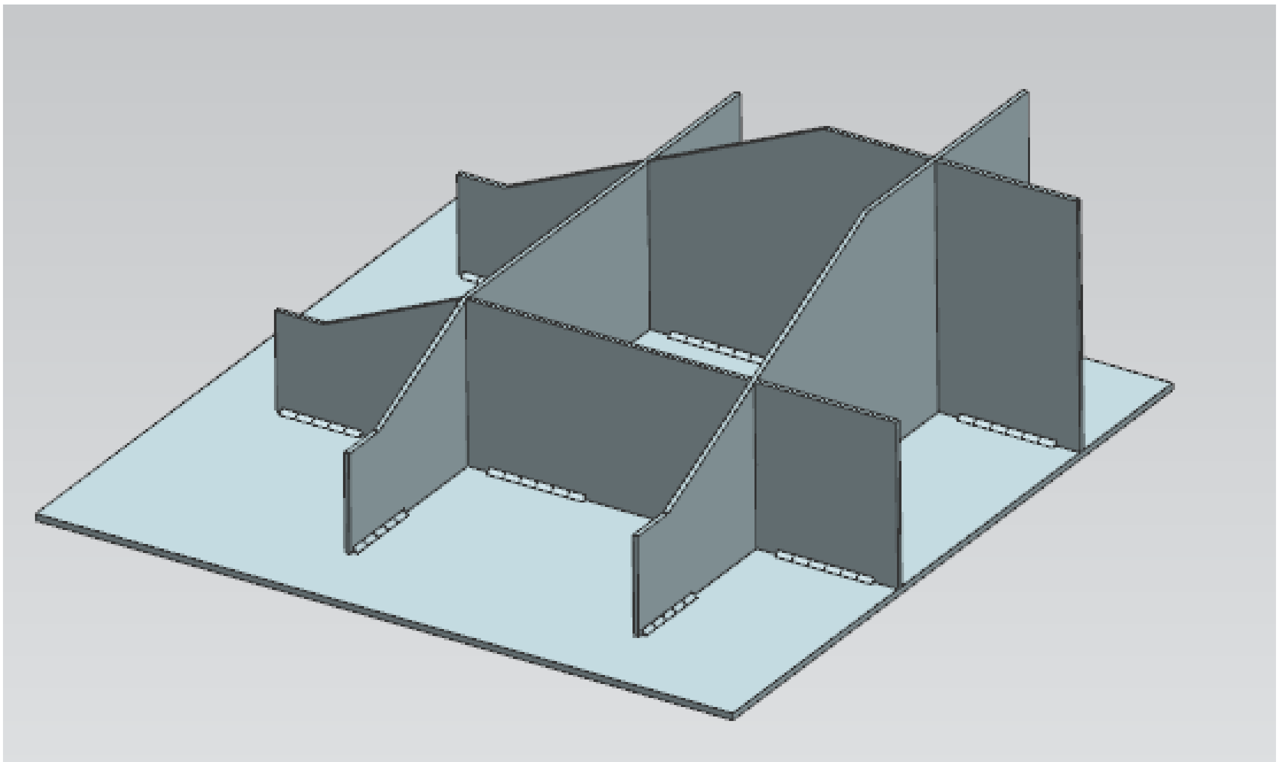

Figure 5. Geometry modeled with NX 12 software

For discretisized model the finite element size used is 12 mm on the cover and the ribs, while the weld mesh is refined with 3 mm elements. The method used for meshing elements is hex dominant. A sphere of influence with a radius of 25 mm and element size of 3 mm was also placed in local areas around the weld. This mesh containts 71693 elements with 205871 nodes. Figure 6 represents the mesh placed over the entire model, while figure 7 is a detailed view of a denser mesh in the vicinity of the weld.

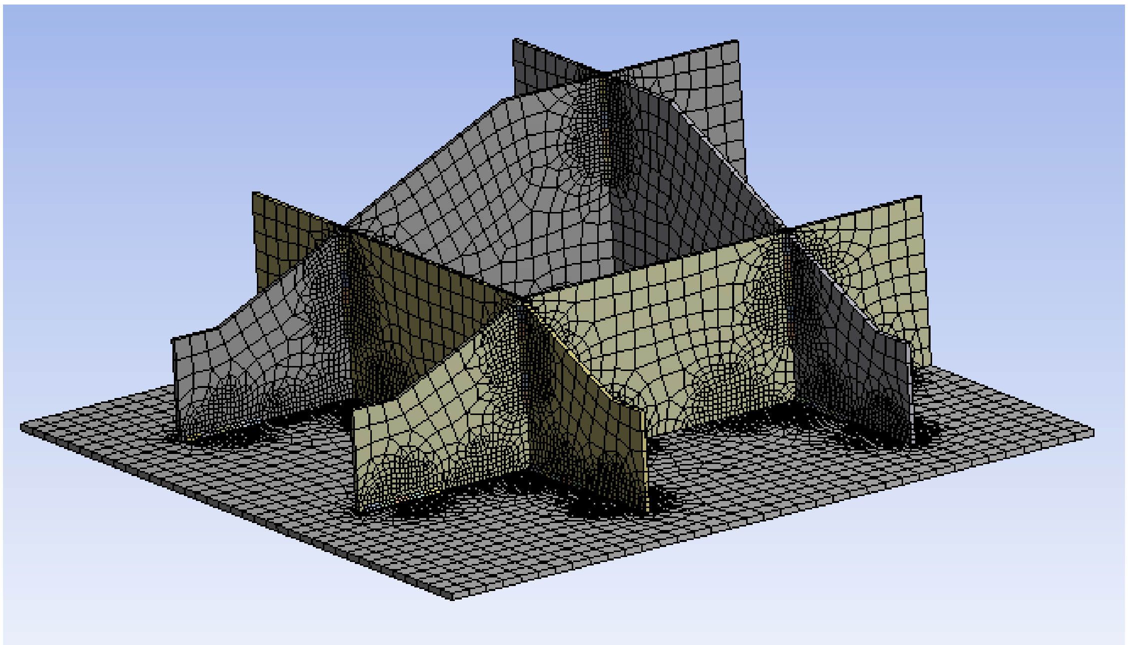

Figure 6. Finite element mesh of the entire model

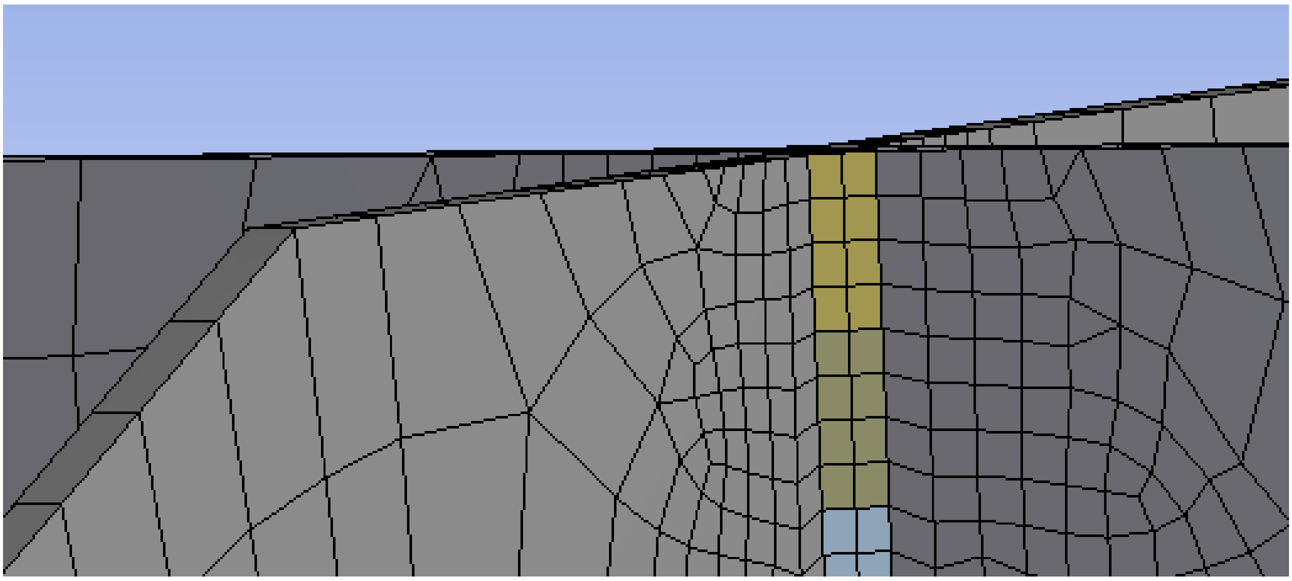

Figure 7. Detail of finite element mesh in the vicinity of the weld

For each segment, it is necessary to set the internally generated heat (Internal Heat Generation) as shown in Figure 8. In order to simulate the birth and death of the elements it is necessary to set in the first second that the first element contains the value of internally generated heat, for which the value of 43 W/mm^3 is selected, while in the other elements this value equals 0. In the following second it is necessary that the value of internally generated heat for the second element is 43 W/mm^3, while in the others elements, that value is equal 0. The same logic must be used on all other elements. After heat has been generated in all weld segments it is necessary to set Convection on all elements of the model. Figure 8 shows Internal Heat Generation in third element of weld.

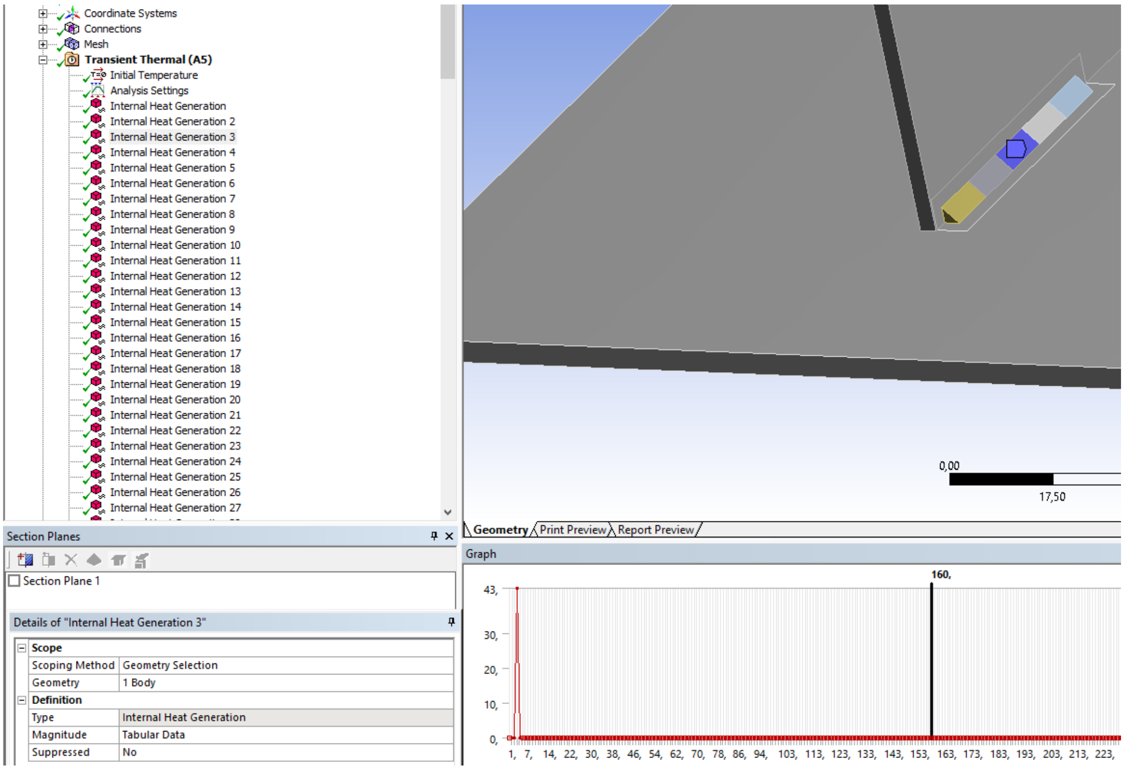

Figure 8. Internal Heat Generation

This method is very time and computer resource consuming, so it is recommended to use a workstation in order to reduce the time necessary for the calculation. These calculations were made on desktop computers predicted for educational purposes, and time required for solving this problem was approximately 7 hours.

4. Results

The temperature results obtained by numerical calculations in Ansys software are shown on Figure 9 (in the 2nd second of the simulation). It can be seen from the obtained diagram that during welding process the temperature is the highest with values up to approximately 3400 °C.

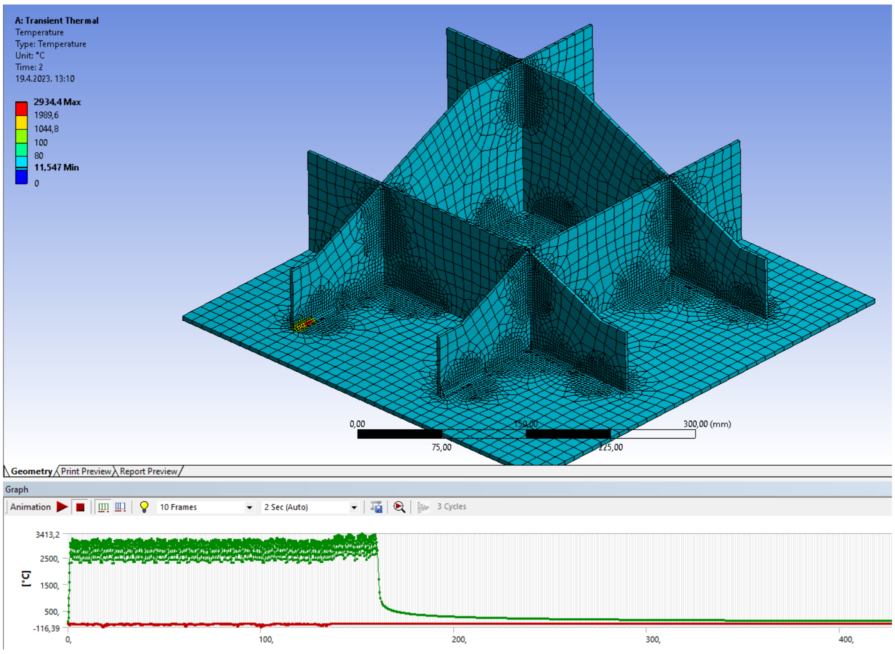

Figure 9. Temperature after two seconds

After the welding stops, the temperature decreases (Figure 10) until it reaches the ambient temperature, which is set to 20 °C. Negative values are noted during the welding process which occur in front of the segment that is under heat input.

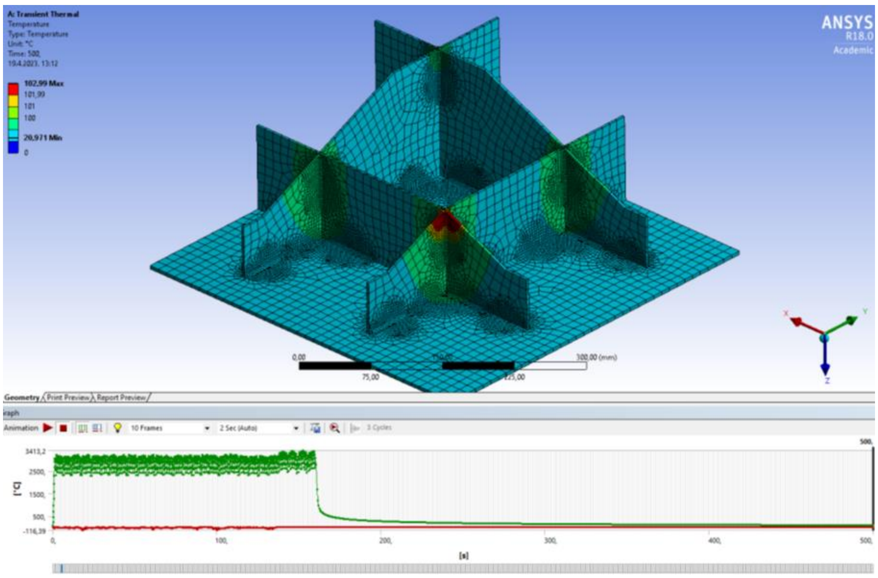

Figure 10. Temperature after 500 seconds

Generated heat causes deformation of the cover, and Ansys software also offers the calculation of their values with respect to the heat input. The results for total deformation after 500 seconds obtained by numerical calculations are presented on Figure 11. Maximal value of displacement is approximately 0.72 mm, and it is located at the edge point of the cover. The cover takes distorted shape after welding in a way that the lid plate is curved with the middle being the lowest point and the edge points being the highest points of the plate. Since this cover was produced in the same dimensions out of the same material used in the calculation, numerical calculation results can be compared to the real ones. The distorted shape of the realistic cover is identical to simulated one, but the displacement values are higher. Maximal value of the displacement is approximately 8 mm.

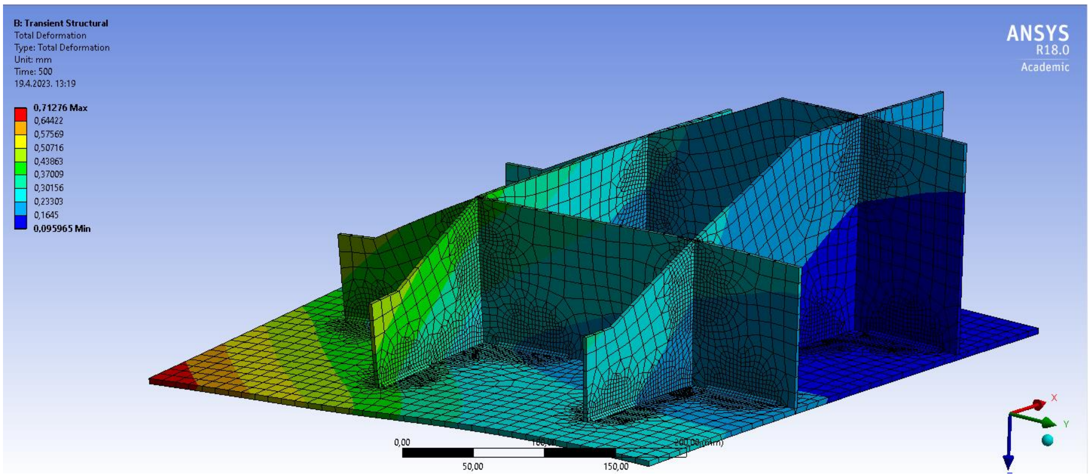

Figure 11. Total deformation after 500 seconds, *mm*

5. Conclusions

Time-temperature diagram displays realistic heat input which causes rapid temperature increase at the start of welding process, maintaining the temperature during welding and slowly decrease after the welding is completed.

When simulating welding in Ansys, a negative temperature appears at the point of current heat input due to an error in the numerical method, and this error would be minimized by mesh refinement, which requires more computer resources than those available for this research. The results obtained by numerical calculations for displacements are approximately 0.72 mm, while in reality displacements are approximately 8 mm. Although the difference in displacements is significant, the distribution of displacements is exactly as it occurs in reality, which indicates a well-fitted model. The difference in the displacement value is expected since the weld size on realistic model varies, and is mostly higher than 2 mm, which leads to higher heat input, which also leads to higher deformation values.

As a continuation of this research, it would be beneficial to use a computer suited for this type of calculations. This allows, of course, to observe the entire model in order to get a realistic picture of the process itself. Also a very important part is the mesh itself, which should be smaller and more uniform in order to obtain good simulation results and to be able to consider longer simulation times (to reach the ambient temperature). In order to obtain results closer to those in practice, it would be necessary to set the model to be more realistic like those in reality, for example follow the actual sequence of welding in practice, pauses between two welds, etc.

6. Acknowledgements

Authors would like to thank Acrom d.o.o. for supporting the research with manhole cover geometry model, information on design, production, testing and general know-how in the field of lid production.

7. References

[1] ANSYS Release 17.0, Ansys Workbench Help. (2015). Ansys Documentation. Canonsburg , ANSYS Inc.

[2] Capriccioli A.; Frosi P. (2009). Multipurpose ANSYS FE procedure for welding processes simulation, Fusion Engineering and Design, Vol. 84, 2009, pp 546-553

[3] Kovačev Z., Konjatić P., Damjanović D., Turk D. (2019). Welding Simulation Using Element Birth And Death Technique In Ansys Software And Residual Stress Comparison. 10. *International scientific-professional conference SBW 2019 „Engineering technologies in manufacturing of welded constructions and products, SBW 2019"*, Samardžić, Ivan; Duspara, Miroslav; Despotović, Božo (ur.). Slavonski Brod: University of Slavonski Brod, Mechanical Engineering Faculty in Slavonski Brod, 2021. str. 437-444

[4] EN 124-1:2015. (2015). Gully tops and manhole tops for vehicular and pedestrian areas -- Part 1: Definitions, classification, general principles of design, performance requirements and test methods

[5] EN 124-3:2015: (2015). Gully tops and manhole tops for vehicular and pedestrian areas -- Part 3: Gully tops and manhole tops made of steel or aluminium alloys.

[6] Ramakrishna, Ch. Siva; Rao, B.N. Dhananjaya. (2015). Characteristics of Residual Stresses on Welded Tabular T-Joints. *International Journal of Mechanical Engineering*. Vol. 4, Issue 1, Jan 2015, 37-46

[7] Van Puymbroecka, Evy; Nagya, Wim; Fanga, Heng; De Backer, Hans. (2017). Determination of residual weld stresses with the incremental hole-drilling method in tubular steel bridge joints. *International Conference on Fatigue Design*, Prezime, Ime; Prezime, Ime. Senlis: CETIM Technical Centre for Mechanical Industries, 2017. 651-661.