ISSN 1392-1207. MECHANIKA. Vol. 29, No. 3, 2023: 188–193

Influence of Temperature Variance on the Mechanical Properties of a Welded Bearing Support Flange Under Transient Conditions Using Finite Element Approach

Kishorekanna GUNASEKARAN*, Isaac SOLOMON, Ramūnas SKVIRECKAS***, Gintautas DUNDULIS******
**Kaunas University of Technology, Faculty of Mechanical Engineering and Design, Studentų st. 56, LT-51424 Kaunas, Lithuania, E-mail: kannakishore123@gmail.com*
***Kaunas University of Technology, Faculty of Mechanical Engineering and Design, Studentų st. 56, LT-51424 Kaunas, Lithuania, E-mail: isaac.michael@ktu.edu*
****Kaunas University of Technology, Faculty of Mechanical Engineering and Design, Studentų st. 56, LT-51424, Lithuania, E-mail: ramunas.skvireckas@ktu.lt*
*****Kaunas University of Technology, Faculty of Mechanical Engineering and Design, Studentų st. 56, LT-51424, Lithuania, E-mail: gintautas.dundulis@ktu.lt*

https://doi.org/10.5755/j02.mech.31209

1. Introduction

While performing the groove welding process, the thermal effect plays an important role influencing changes in various mechanical characteristics of the component. This research study is carried out in an intent to provide necessary computational data on the structural behaviour of mild steel flange due to temperature variance during the groove welding process.

This research highlights the methodology to estimate the mechanical response of mild steel components welded together to support heavy loading conditions during their runtime. The existing research on welded steel components is limited to linear weld path, axisymmetric geometry of 2-dimensional models and no major study is done to evaluate the structural response of AISI 1020 mild steel under groove welding process.

In this research study, the stick welding phenomenon for a circular path V-groove is simulated numerically under specified welding parameters for different plate thickness. The computational data from this research study is obtained from a set of simulations using finite element approach enacting the real-world welding conditions through a concept of ‘moving heat source’ integrated with the transient analysis in ANSYS FEA software. The primary objective to perform the simulation using FEA approach is to predict the induced stresses from the temperature variance during welding and the distortion associated with such variance along the path of the weld [1].

The results of this study give an estimation of the induced thermal stresses and deformations during the groove welding process and these parameters are idealised, tabulated, and concluded with an approach to determine the optimised design technique for welded components.

2. Experimental background

Flange supports are seen in piping and other systems because it tends to ease the maintenance criteria of the technical person in charge and also provides feasibility to be repaired and maintained. Flange can be either welded or bolted depending on certain circumstances and needs. Bolted flanges undergo a series of heavy loading during the runtime and hence require periodic testing while operated at higher loading conditions due to their bolt failure [2].

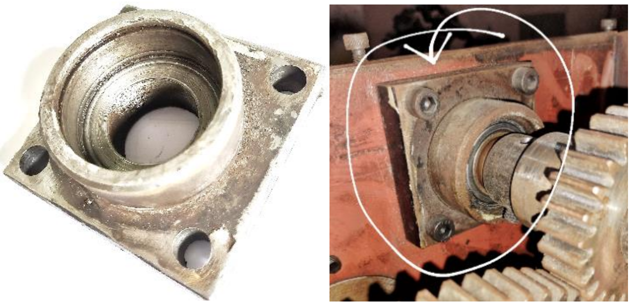
During the welding process, it is important to analyse the structural behaviour of the component due to thermal variation as it may have an impact in the overall performance. The deformation caused during the welding process is of major research consideration since it reiterates the original structural behaviour under loading. Flanges essentially need to be optimised in terms of shape to withstand high thermal stresses during the welding process and at the same time have more durability [3].

The estimation of material properties of components which undergo welding process is of primary importance to determine its life against apparent loadings. Some important material properties which are affected by the temperature variance during welding are thermal conductivity and the coefficient of thermal expansion of the material, yield strength and the elastic modulus. These properties have adverse effect on the material’s structural response under substantial loads [4].

AISI 1020 steels have been used immensely in recent times for their extreme cutting and formability properties. These alloys of steel can also be substituted for AISI 304 which share similar material and structural properties which are suitable for welding and cutting operations. This research study deals with the data and material properties evaluation of welding similar components made of AISI 1020 steel. The process involved is a groove welding process of a circular path containing chamfered edges [5].

The Fig. 1, a show a typical support flange which has a cylindrical pipe welded to the base plate of 20 mm thickness. The plate thickness of hot rolled mild steel plates comes in standard sizes from 3 mm to 25 mm for fabricating industrial machinery. The most commonly used MS plate thickness for commercial machinery is 25, 20, 16 and 10 mm. These bearing support flanges are mostly fabricated using 20 mm plates in order to balance the strength-to-weight ratio.

This type of support flanges are fabricated for supporting a shaft being rotated by gear at 300 rpm. These support flanges are housed in an industrial core cutting machine with a 6004-deep groove ball bearing of 20 mm inside diameter and 47 mm outside diameter housed inside its hollow cylindrical space. These support bearing flanges experiences the forces from the rotation of gear at higher rpm and also the vertical bearing load of the shaft connected to it. These support flanges are welded together using stick welding approach which uses a standard E6013 welding rod to join the two mild steel components. The E6013 is a medium coated rutile based electrode designed for welding structural steel components operating on both AC and DC voltages. The diameter of the electrode selected for welding the flange assembly is of 3.15 mm outer diameter with a length of 350 mm and a standard flux coating throughout the length of the electrode with an electric supply of 100 to 140 amps during the welding operation. The E6013 electrode has a yield strength of 450 MPa and a tensile strength of 510 MPa with an elongation percentage of 2.5 [6].



a b

Fig. 1 a) General view of the fabricated welded flange; b) welded flange of plate thickness 20 mm assembled onto an industrial core cutting machine in the workshop

3. Computational approach

A three-dimensional model of the flange assembly was created in SolidWorks (student version 2021) with the dimensions obtained from revere engineering the original component. In this research, a welded support flange is considered and its structural behaviour due to thermal variance is evaluated, comparing its response to welding scenario with different plate thickness. The computational set up for the simulation plays a major role in this modelling approach as the simulation is performed and evaluated in two distinct phases. The simulation model is then solved in for transient conditions in ANSYS 2021 FEM software.

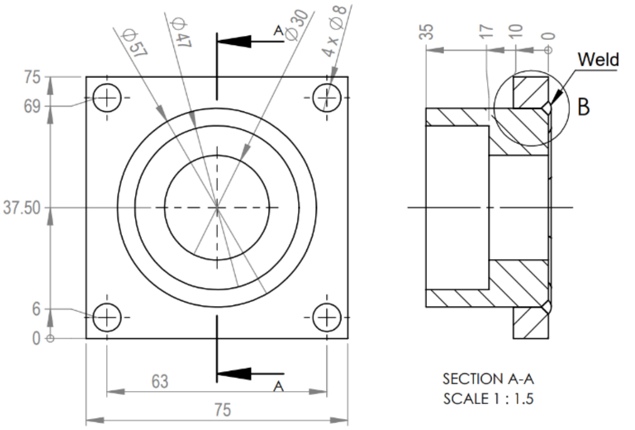
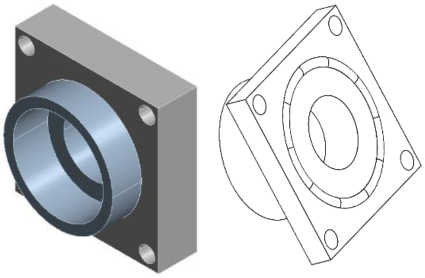


Fig. 2 Technical drawing the welded support flange

The above Fig. 2 shows the technical dimensions and placements of the welded support flange made of AISI 1020 steel alloy.



a b

Fig. 3 a) Isometric view of the computational 3D model; b) the weld split into divisions for transient conditions

The above Fig. 3 is a three-dimensional representation of the computational model welded together at their chamfered cylindrical ends. The weld is split into 10 equal divisions to cater the transient conditions of the welding process in ANSYS.

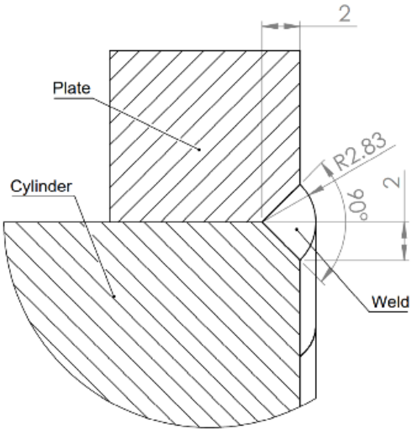


Fig. 4 The weld section of the computational model

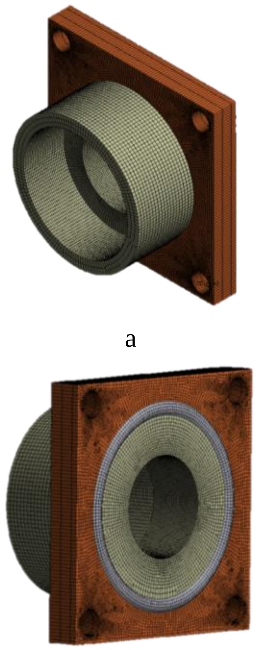
The above Fig. 4 shows the weld region between the two components. The components are chamfered to 2 mm at their ends and the weld is placed on the chamfered end. The thickness of the weld is kept at 4 mm with a groove angle of 90^o between the two components. The weld has an arc radius of 2.83 mm. The model was constructed as an integrated model in three-dimensional space to reduce the computational complexity while simulating transient conditions [7].

3.1. Finite element model

The three-dimensional finite element model was generated in ANSYS workbench using a structured meshing technique for creating quadratic hexahedral elements of 372220 nodes and 79920 elements.

The above Fig. 5 shows the constructed FE elements on the three-dimensional model. Since the computational model is solved under transient conditions with respect to time, the model is constructed with Hex20 and Wed15 elements in ANSYS finite element assembly meshing environment. The Hex20 element is a 3-D higher order hexahedron element of 20 nodes exhibiting quadratic displacement behaviour and is also known as SOLID186.

These SOLID186 elements are generated in the possible mapped region of the model and then transitioned to Wed15 elements in the feasible regions around curvature [8, 9].


a

b

Fig. 5 Structured finite element generation on the welded three-dimensional model assembly in ANSYS workbench a) front: b) rear weld region

Table 1

Finite element quality evaluating parameters

Parameter	Min.	Max.	Avg.	Standard Deviation
Element quality	9.98	0.99	0.63	0.38
Aspect ratio	1.01	60.26	5.87	6.99
Jacobian ratio	1	7.40	1.24	0.36
Orthogonal quality	0.16	0.99	0.84	0.24

This structured meshing technique of generating quadratic hexahedral elements is carried out in various stages and the ANSYS local mesh controls are utilised during the FE model construction. The model undergoes a multizone meshing algorithm where the sweep able regions are constructed with mapped hexahedral elements and the linear path is meshed using edge sizing algorithm which specifies the number of divisions of elements along the length of the path. The weld model is constructed using mapped face meshing with quadratic properties [10].

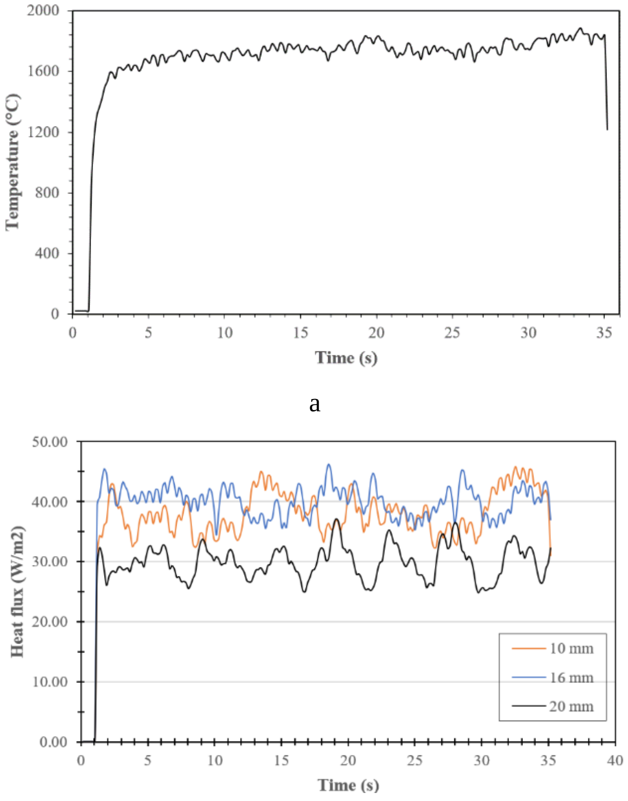
4. Results and discussions

4.1. Simulated three-dimensional thermal response

The phenomenon of "moving heat source" [11] has been adopted to perform transient thermal analysis in which the welded region of the model is subdivided equally into 10 sections and numerically simulated on each surface gradually.

The thermal simulation parameters are taken from the original real-world welding scenario where the components are welded together with a 3.15 × 300 mm E6013 welding electrode with 100 – 140 amps supply from DC +/- AC 50V.

The power of heat flow on the surfaces of the weld is kept at 5000 W with convection values of ‘stagnant air-simplified case’ applied over all outer surfaces of the component. Applying this heat flow through transient thermal analysis, the resultant data gives a maximum temperature of ± 3000°C on the surface of the weld when the room temperature is maintained at 22°C. This preheating temperature of the weld segments is subsequently solved under transient time steps of 10 divisions to evaluate the mechanical behaviour under the preheated temperature [12,13].



a

b

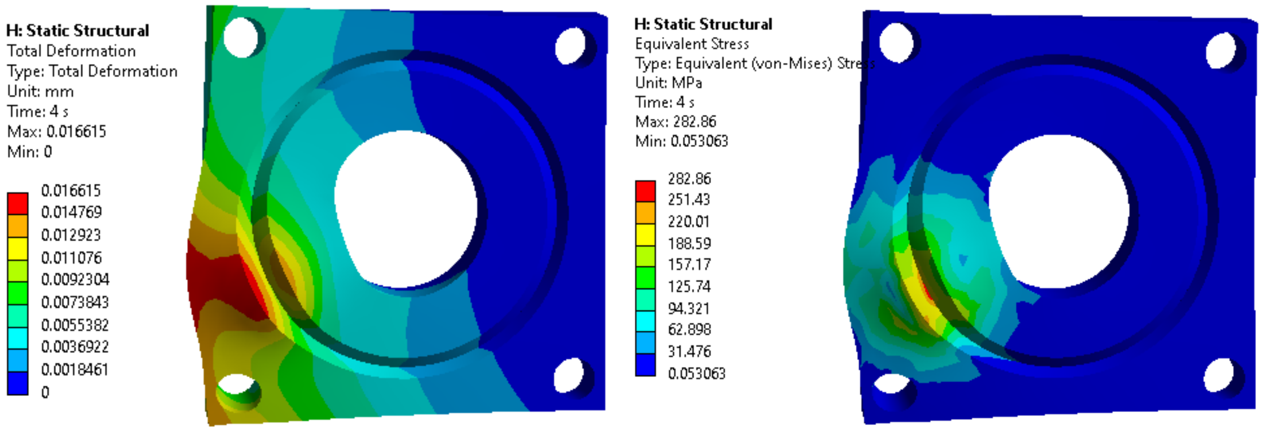
Fig. 6 a) Maximum temperature distribution against time; b) maximum heat flux distribution against time

The above Fig. 6, a shows the plot of distribution of temperature due to heat flow on the subdivided surfaces of the weld with respect to time. This distribution is almost identical on all three flanges of different plate thickness.

The plot in Fig. 6, b shows the maximum heat flux distribution in the three different flanges of different plate thickness *t*. The 16 mm flange tends to have the least distribution at almost all intervals while the 10 mm and 20 mm shift peak points accordingly.

4.2. Simulated three-dimensional structural response to preheated temperature

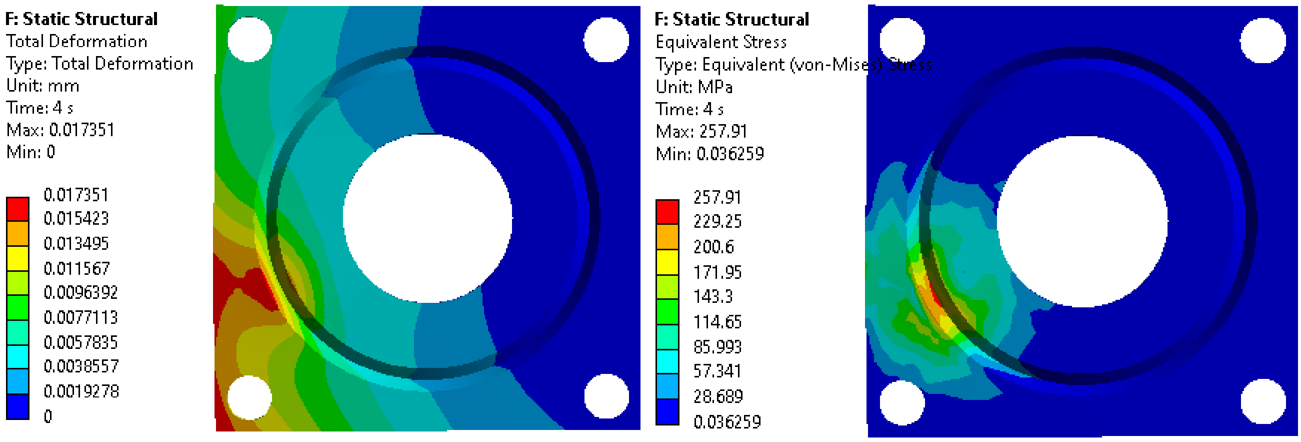
In the second phase, a transient structural analysis with respect to time is performed where the thermal load is imported from the previously performed temperature distribution study with the constraints fixed at the four holes on the four corners of the flange. The model is solved for transient structural conditions with respect to time and the structural response to the thermal loads are predicted by comparing the maximum displacement and stress values of the models.



a

b

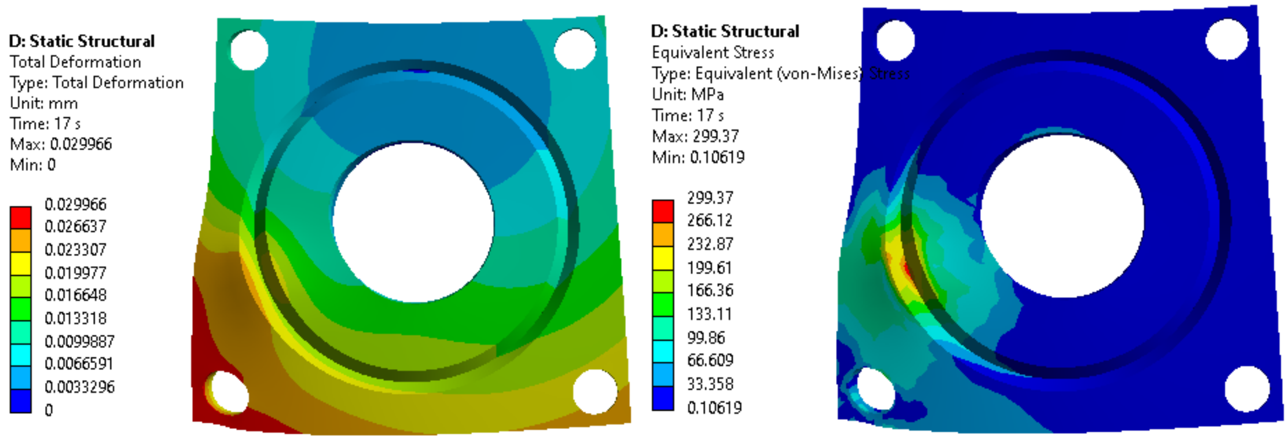
Fig. 7 a) Total deformation; b) max. equivalent von Mises stress distribution contour of the 10 mm plate thickness 3D model



a

b

Fig. 8 a) Total deformation; b) max. equivalent von Mises stress distribution contour of the 16 mm plate thickness 3D model



a

b

Fig. 9 a) Total deformation; b) max. equivalent von Mises stress distribution contour of the 20 mm plate thickness 3D model

In the above Figs. 7 - 9, the time dependent structural responses of the different plate thickness models (10, 16 and 20 mm) are given in the form structural deformation and the von Mises equivalent stress distribution contours under preheated temperature variance under time-based conditions. These contours are derived from the time step 2 which fairly gives an estimation of how the gradual temperature change from one division of weld to and another division induces a structural response around the region of weld causing a drastic change in its structural characteristics due to the effect of plate thickness variation.

Table 2

Results of the numerical study for different plate thickness

Plate thickness, mm	Resultant values	
	Total deformation, mm	Eq. von Mises stress, MPa
10	0.016	282.8
16	0.017	257.9
20	0.029	299.3

The above Table 2 gives the maximum estimates of the deformation and von Mises stress of the varying plate thickness models.

4.3. Comparative estimation of the structural response of the 3D models against time

The Fig. 10 presents the comparison of total deformation of the support flanges with different thickness with respect to their equivalent time steps. The plot explicitly reveals the 20 mm thickness flange has the least deformation while the 10 mm has the highest.

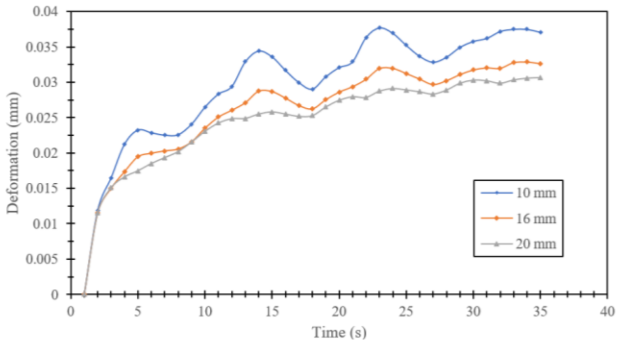


Fig. 10 Comparative deformation plot against time for the 3 different plate thickness models

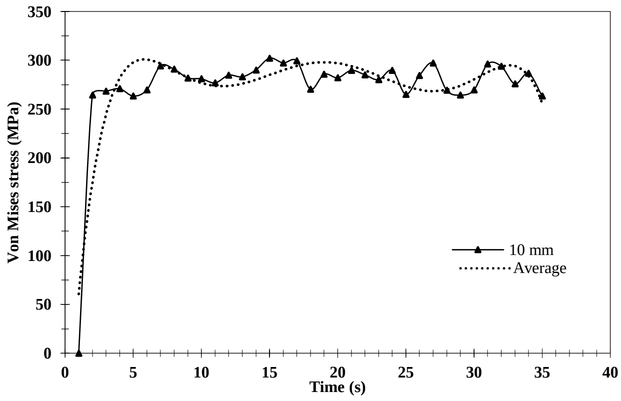


Fig. 11 Plot of maximum stress distribution and average stress distribution of the 10 mm flange against time

The above Fig. 11 displays the distribution of stress in the 10 mm plate component due to thermal variance of weld. The two plots, first with the exact obtained values and the second averaged throughout the simulation, in-order to exhibit clear comparison during the final stage.

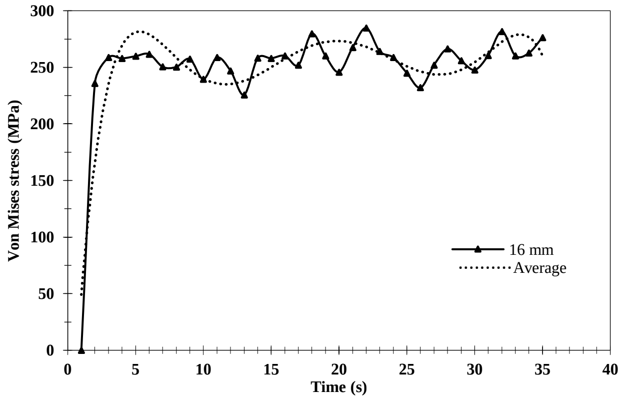


Fig. 12 Plot of maximum stress distribution and average stress distribution of the 16 mm flange against time

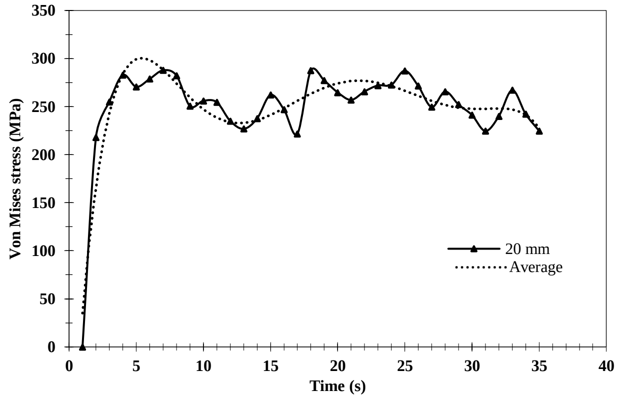


Fig. 13 Plot of maximum stress distribution and average stress distribution of the 20 mm flange against time

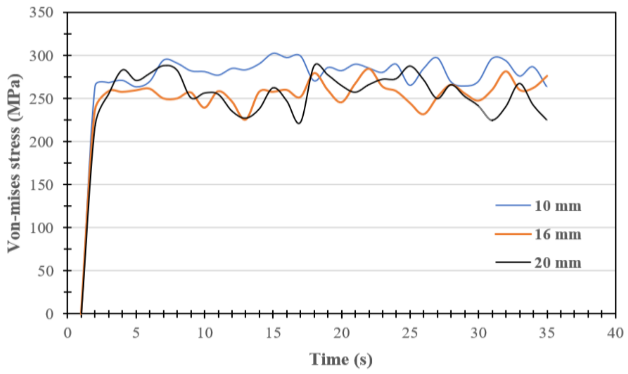


Fig. 14 Comparative plot of maximum stress distribution

The above Fig. 12 and Fig. 13 displays the stress distribution in the 15 mm and 20 mm thickness plate components along with its average stress distribution against time. The primary approach is to give a comparative estimation of the average stress values for the varying plate thickness models with their normal stress values since these obtained values tend to fluctuate rapidly with respect to time.

The above Fig. 14 displays the comparative plot of the maximum stress distribution of the varying plate thickness models with respect to time. From this plot, it can be clearly observed that the 10 mm plate thickness flange has the least stress distribution whereas the 20 mm thickness flange has the highest stress distribution values.

From these stress distribution plots, it is evident that the 10 mm plate thickness model is induced with the maximum thermal stresses due to the temperature variance during the computational welding scenario.

5. Conclusion

The three-dimensional computational model of the welded component was developed and the evaluation of the mechanical behaviour under welding conditions were numerically simulated using the finite element tool ANSYS. The effect of plate thickness which compromises the structural characteristics of welded components based on temperature variance during the welding process is evaluated for a bearing support flange which undergoes a groove welding method of joining the mild steel components.

The circular groove weld of the model was radially split into 10 equal divisions and the concept of moving heat source along the length of the weld was simulated under transient thermal conditions of 10 steps for the varying plate thickness models. The material's structural response towards the predefined thermal loads was evaluated using a combined thermal-stress numerical approach and the resultant contours and graphs were plotted against time.

This research study provides sufficient computational data of mild steel welded components where the mechanical response to thermal loads explicitly affects the deformation characteristics and stress intensities of the entire welded component. The deformations observed in the 16 mm plate to its equivalent von Mises stress parameter is of similar ratio to the 20 mm plate. The 16 mm plate also has a stable stress distribution throughout the 10 steps of the time-based simulation in comparison with the other two plates.

Hence this research proves to be useful in estimating the structural characteristics of mild steel welded components under time dependent predefined thermal loads. Although this research is limited to its numerical capabilities, the experimental estimation under real-world controlled environment could provide a more in-depth understanding of the mechanical behaviour of these industrial components.

References

1. **Antony S.; Deepak R.; Aravinthan G.; Kishore M; Anantha M.** 2018. Simulation and modelling of the effect of welding process parameters of Inconel 625, International Journal of Engineering Research & Technology (IJERT) 6(7). https://www.ijert.org/simulation-and-modeling-of-the-effect-of-welding-process-parameters-of-inconel-625.
2. **Chavan, S. G.** 2017. Stress analysis of a flanged joint using finite element method, International Journal of Science and Research (IJSR).
3. **Sonawane, H. B.; Deore, E. R.** 2014. Finite element model for the effect of heat input & speed on residual stress during weldings, International journal of mechanical engineering and robotics research 2(8).
4. **Singh, M.; Shoor, S.; Kumar, R.** 2018. Effect of welding parameters on temperature distribution using Finite Element Simulation, Journal of Emerging Technologies and Innovative Research 5(11).
5. **Patidar, S.; Soni, D.; Sharma, P.** 2018. Computer aided modeling and analysis of dissimilar metal welded joint using ANSYS software, National Conference on Importance of Inter Disciplinary Studies in Higher Education in the areas of Engineering, Science, Management and Humanities, Feb. 2018.
6. **Suheni; Rosidah, A.; Ramadhan, D.; Agustino, T.; Wiranata, F.** 2021. Effect of Welding Groove and Electrode Variation to the Tensile Strength and Macrostructure on 304 Stainless Steel and AISI 1045 Dissimilar Welding Joint Using SMAW Process, Journal of Physics: Conference Series. http://dx.doi.org/10.1088/1742-6596/2117/1/012018.
7. **Farajpour, M.; Ranjbarnodeh, E.** 2018. Finite Element Simulation of Welding Distortion in Dissimilar Joint by Inherent Deformation Method. Soldagem & Inspeção. http://dx.doi.org/10.1590/0104-9224/SI2301.07.
8. **Kauppilaa, P.; Kouhiab, R.; Ojanpera, J.; Saksala, T.; Sorjonena, T.** 2016. Continuum damage model for creep fracture and fatigue analysis. http://dx.doi.org/10.1016/j.prostr.2016.06.114.
9. **Kalin, A.; Akbas, S. D.;** 2021. Modal analysis of industrial steel stacks with soil-structure interaction.
10. **Mani, P.; Raju, P. M.; Geddada, Y.** 2019. A comprehensive nonlinear finite element modeling and parametric analysis of reinforced concrete beams. http://dx.doi.org/10.1108/WJE-04-2021-0212.
11. **Sagar, P.; Gope, D. K.; Chattopadhyaya, S.** 2018. Thermal analysis of TIG welded Ti-6Al-4V plates using ANSYS. http://dx.doi.org/10.1088/1757-899X/377/1/012113.
12. **Jingnana, P.; Lixin, Y.** 2016. The Mathematical Model Research on MIG Groove Welding Process. http://dx.doi.org/10.1016/j.proeng.2016.08.377.
13. **Wang, A.; Cheng, X.; Meng, G.; Wo, L.; Pang, Z.** 2016. Experimental study on the welding procedure of the central groove of armored face conveyor. http://dx.doi.org/10.1177/1687814016674101.

K. Gunasekaran, I. Solomon, R. Skvireckas, G. Dundulis

INFLUENCE OF TEMPERATURE VARIANCE ON THE MECHANICAL PROPERTIES OF A WELDED BEARING SUPPORT FLANGE UNDER TRANSIENT CONDITIONS USING FINITE ELEMENT APPROACH

Summary

During a welding process, the influence of temperature distribution plays an important role affecting the vulnerable regions of the welded component. This research aims at presenting data on similar cases where a detailed comparative evaluation provides necessary data for designing a component based on its structural behaviour due to induced temperature variance during the groove welding process using finite element approach.

The finite element simulations render a real-world scenario of the groove welding process by integrating the concept of 'Moving heat source' under transient conditions in ANSYS. The numerical comparative study is achieved by simulating a thermal study of the welded components of different bearing support flange plate thickness and evaluating its structural behaviour under induced thermal loads.

The numerical solutions obtained from these simulations provide necessary data which helps in identifying the feasible design approach to increase the life of a welded component under such temperature-dependent conditions.

Keywords: support flange, finite element, weld, structural behaviour, temperature variance.

Received April 19, 2022
Accepted June 14, 2023