ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun783

Weld Distortion Modelling in Ansys Workbench Mechanical

Amer E S E Th Alajmi[1]; Abdullah O E M Alrashidi[2]

[1]Specialized Trainer (B) at the Public Authority for Applied Education and Training, Vocational Training Institute.
[2]Specialized Trainer (B) at the Public Authority for Applied Education and Training, Vocational Training Institute.

Publication Date: 2025/07/17

Abstract: Significant progress has been made in the development of computational tools to facilitate the modelling of the welding process. These tools have made it possible to evaluate very complicated phenomena, such as welding distortion along with stress. When welding, the most modern instruments take into consideration the thermo-mechanical and thermo-metallurgical modifications that occur. These changes occur throughout the welding process. It is feasible to build reliable simulations of the methods of welding by taking into consideration these modifications as well as the characteristics of the materials at various temperature levels. For this sector, T-joint welded topologies are studied. This work deals with the simulation of welding distortions on a T-joint plate using ANSYS Workbench, considering thermal and structural dynamics in the HAZ in detail. Thermal stress, strain, and deformation responses during different thermal loading conditions have been investigated by using static structural and steady-state thermal analyses. Precise meshing techniques and application of boundary conditions based on strategic relevance have been used to sufficiently replicate practical scenarios that could be presented by such welding. The results of the simulations support the importance of appropriate thermal management, giving quantifiable predictions of deformation patterns across the welded joint. This work has contributed to the improvement of predictive accuracy in welding simulations to further optimisation of welding processes, reduce post-weld rework, and enhance structural integrity for industrial fields.

Keywords: *ANSYS Workbench, Welding Distortion, Structural Simulation, Thermal Analysis, FEA, HAZ, Deformation Predictions, Thermal Stress, Simulation Optimisation, and Welding Applications.*

How to Cite: Amer E S E Th Alajmi; Abdullah O E M Alrashidi; (2025) Weld Distortion Modelling in Ansys Workbench Mechanical. *International Journal of Innovative Science and Research Technology,* 10(6), 3213-3225. https://doi.org/10.38124/ijisrt/25jun783

I. INTRODUCTION

In the field of large-scale metal structures, such as shipbuilding, welding is widely used as a high-productivity method for joining multiple components of the structure. The structure made by welding is characterised by flexibility in design and low weight. Welding also contributes to saving the manufacturing costs of these structures when compared with other manufacturing methods (Churiaque, et al., 2018).

However, welding processes lead to some issues, such as deformation, which results from repeated heating and cooling of the metal. This irregular expansion and contraction of metal cause a distortion (Lee, et al., 2023). This distortion has an unacceptable appearance ,and it may negatively affect the structural integrity of the structure (Kim et al., 2015). Therefore, welding-distortion often requires treatment or rework. They can be mitigated through additional procedures such as component straightening or alignment. These additional procedures may cause delays in the delivery time of the structures, and also cause increased costs (Kim, et al., 2015). After welding, the distortion can be mitigated by rework and component alignment, but the increased manufacturing time and material costs cannot be avoided. Therefore, weld distortion should be considered in the structure planning process before starting to implement it.

Predicting the amount of welding deformation in advance greatly supports the manufacturing process effectively (Seong et al., 2020). Implementing advanced steps to maintain the quality of the structure and designing a work schedule based on marginal deformation considerations reduces potential delays in work execution and the possibility of rework (Vladulescu, 2018).

Welding deformation modelling gave designers the ability to predict and reduce its drawbacks. Weld distortion left unmanaged or uncorrected can lead to major structural problems, which cannot be avoided when the structure is in operation. The most prominent of these problems are misalignment, weak strength, and poor quality of the whole structure (Domański, et al., 2022).

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun783

In the construction of large structures, distortion prediction methods have focused on the deformation of the plate in particular, ignoring the deformation of the stiffener as less important (Kim, et al., 2015). However, real-world experiments of structures have demonstrated the need to take both component's distortion into account. Therefore, modern weld distortion prediction methods predict angular deformations in end joints. They also model the inherent stress distribution of welding on these joints. They take into account complex crack joints in modelling and simulation (Kim, et al., 2015). These advanced methods enable the prediction and visualisation of welding deformations. Thus, some measures and considerations can be implemented during and before design to mitigate them more effectively. Modern welding-distortion prediction methods have proven their ability in improving welding processes and structural results (Vladulescu, 2018).

Distortion modelling helps to improve the dimensional accuracy and structural integrity of welded components, especially large and complex structures, where accuracy is of utmost importance (Azad, et al., 2020). Modelling tools allow engineers to apply different welding parameters and use different sequences and designs to improve results and reduce defects using experimental methods (Vladulescu, 2018).

One of the most prominent distortion modelling tools is ANSYS Workbench. This program allows engineers to simulate welding effects and deformations before they really occur. Thus, this tool allows engineers to better plan the welding process and modify designs to achieve minimal distortion and better structural quality. The use of ANSYS adds predictive capacity to the design and reduces the need for post-welding corrections (Granell, et al., 2021)

ANSYS is considered a powerful tool to predict and mitigate welding deformations, it is able to model the complex model effects. It allows automated simulation settings using thermo-mechanical coupled analyses. ANSYS has achieved a predictive accuracy ranging from 80% to 98% in some studies (Granell, et al., 2021).

ANSYS software has given a great advantage in controlling deformation patterns and improving manufacturing accuracy (Azad, et al., 2020). Moreover, simplified methods, such as the inherent stress method, have been developed to efficiently predict deformation while reducing computational costs, especially when applied to large structures (Wu, et al., 2021).

This research aims to design a model for the welding process of structural joints using ANSYS Workbench. This model can accurately predict the welding deformation in structures, which will be analysed later. By utilising ANSYS and its advanced thermomechanical analysis and modelling capabilities, this study will provide a simulation of complex thermal cycles and material behaviours during the welding process. ANSYS's modelling and simulation capabilities will be exploited to reduce welding deformation, and thus, it will strive to improve welding and reduce post-weld rework in industrial applications.

II. LITERATURE REVIEW

There are many studies that have addressed the topic of welding-distortion, how to manage and mitigate its effects and how to simulate and predict it. In this study this chapter will address the most prominent previous studies that worked on simulating welding and predicting its distortion using an ANSYS workbench. Different studies implemented different software and methods to present their results and conclusions. However, this study focuses only on studies that exploit ANSYS capabilities. This study will be a complement to previous related efforts and use them to enrich its results.

In a recent study, Nurzhanova et al. (2023) presented a welding model using ANSYS Workbench 19.2. in this model, the distortion of a toothed shaft was investigated. The authors applied subsequent restoration processes to this shaft. They simulated the hardening process of the tooth surface and analysed the thermo-mechanical behaviour of this layer. Model parameters and temperature distribution were determined to find the range of the residual stresses and the range of welding deformations. These findings were compared with real practical investigation data, and a good agreement was reached between them. This study demonstrated that finite element analysis (FEA) can predict welding-distortion very well using simulation tools. FEA correctly simulate temperature distribution, residual-stresses and distortions. The results of this study also confirmed the importance of using simulation tools and computational models to predict welding-distortion. Which in turn help the engineer to accomplish the best welding quality of the structure. These results confirmed the modelling ability to predict structural distortion in the welded zone (Nurzhanova et al., 2023).

On the other hand, predicting weld distortions requires accurate temperature and material data, as well as good experience in simulation programs. This good experience gave the engineer the ability to define parameters (e.g. materials properties, mesh, thermos-mechanical settings) clearly. Granell et al. presented a study based on using ANSYS to correctly simulate weld distortions. The researchers used Python scripts in Ansys to set modelling settings with the least possible number of inputs, which added a kind of flexibility in using Ansys to model different geometric shapes and sizes of structures. In this study, the researchers applied thermo-mechanical analysis in combination with the birth-and-death technique to predict weld distortions. They automate all settings to make accurate predictions of weld distortions. The results of the study confirmed a high prediction accuracy ranging between 80 and 98% (Granell, et al., 2021).

In 2019, several studies were presented that predict welding-distortions using ANSYS software and relying on the FEA. Matuszewski presented a study based on 3D modelling of the welding between the ends of two aluminium sheets. Through simulation, the researcher used the elliptical heat source model, and inter the thermo-mechanical properties of aluminium into ANSYS. They used several ANSYS modules (Design Modeler, Mechanical, etc.) to

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun783

complete the 3D simulation process. They applied cube-type elements and increased the mesh density in the heat-affected zone. Then, they analysed the distributions of temperature in the cross-sections of the welded joints. They verified the simulation results by comparing them with experimental results. The comparison confirmed a satisfactory agreement between them (Matuszewski, 2019).

In another study conducted by Vemanaboina et al. (2020), the researchers introduced all the thermo-mechanical properties of the structure material (SS316LN) into ANSYS. They also used the heat equation and heat flow inputs. They performed a thermos-structural analysis of the material and modelled the thermal distribution to expect the structural deformation of the material. The researchers conducted 9 experiments using different laser power levels to weld the structure. They also studied different speed levels of welding and shield gas flow rates. The experimental results found the optimum levels for each of the laser power levels (2750 W), welding speed (2500 mm/min) and gas flow (10 L/min). The comparison between the experimental and simulation results was only 8% (Vemanaboina et al., 2019). In the same context, Azad et al. presented another research paper based on the simulation of welding sequences using ANSYS software. The researchers used the FEA method to analyse and explore the effect of welding-sequences on the shape and magnitude of the deformation. The study used a number of flat fixing bars with a steel plate. The researchers conducted 3D thermos-structural analysis simulations in combination with the birth-and-death technique. The results of the simulation and experiments were to verify the effect of 8 different welding-sequences on the deformation. The study concluded that the results of both simulation and experiments were in agreement, it also confirmed the differences in the deformation characteristics of shape and size based on the applied welding-sequences (Azad, et al., 2020).

In a paper by Venkateswarlu et al. (2018), they presented a single-elliptic Goldak model for heat-flow simulation in a dissimilar butt joint. The researchers used ANSYS software to simulate the model thermally and structurally. They entered all the thermal inputs for all elements, including the temperature distribution, spatial coordinates, and time of the structures during welding. The fusion and thermal affected regions were identified, and the results of the thermos-mechanical analysis were performed to identify potential welding-distortion of the panel and to identify the residual-stress patterns of the plate (Venkateswarlu, et al., 2018).

Vladulescu (2018) presented another study that summarises the simulation of the laser-welding method using ANSYS. The main steps of welding simulation were followed, starting with developing the geometric model of the structure and mesh, entering all parameters and settings, determining the loads, and finally performing the thermos-structural analysis to simulate the welding stresses and deformations. This study showed that the complication of the thermos-structural analysis of the model, was due to the mixing of the time and location analysis of the varying thermal loading along the weld path. The research results confirmed the importance of welding modelling, as it provides an accurate prediction of the welding effect and the resulting residual stresses (Vladulescu, 2018).

Bajpei et al. (2017) conducted an experimental investigation of the welding process to study the temperature distribution, residual-stresses, and deformations in the joining of two dissimilar thin aluminium alloy plates. At the same time, the results were verified by numerical simulation using ANSYS. A FEA of thermo-mechanical model was used for 3D modelling of the structure. Heat dissipation, convection, and radiation were also studied. The researchers tested the welded samples using X-ray diffusion, and they conducted a comparison between the simulation and experimental results. This comparison showed good agreement, especially in residual-stress values (Bajpei, et al., 2017).

III. METHODOLOGY

A. Design Overview

The aim of this paper is to investigate the welding distortion at the T-joint welding plate. Distortions in welding plates are typically predicted by means of applying finite element analysis. For this purpose, Ansys software is adopted to mimics the physical occurrence taking place on the welding plate in terms of static structural based on thermal condition load followed by a steady-state thermal analysis based on connection and radiation model. This chapter is composed of the phases followed to analyse the welding plate distortions, beginning from the geometry modelling and reaching the analysis of the results.

B. Geometry Modelling

Abaqus software, particularly the part modelling, was adopted to design the T-joint welding plate due to its unique features. A feature-based depiction applies to a component that was developed using Abaqus/CAE. In addition to being a significant component of the design, a feature offers the engineer a straightforward and uncomplicated approach to the construction and modification of a component. A structured collection of properties and the values of parameters that govern the shape of every characteristic are used to produce the parts that are created in Abaqus/Computational Engineering (CAE). Figure 1 shows the 2D dimensions of the plate base (10 mm length x 150 mm width).

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun783

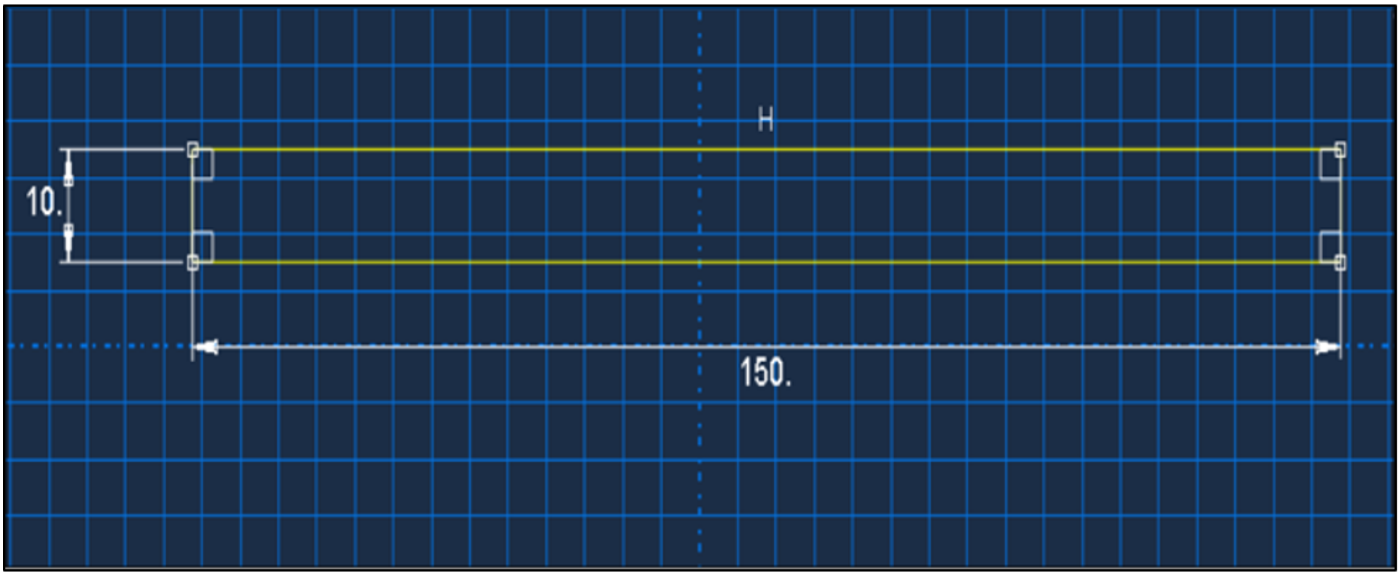

Fig 1 A capture screen from Abaqus illustrating the plate base dimensions.

Then, the plotted 2D plate base was extruded by a depth of 100 mm to build a 3D model of the plate base, as shown in Figure 2.

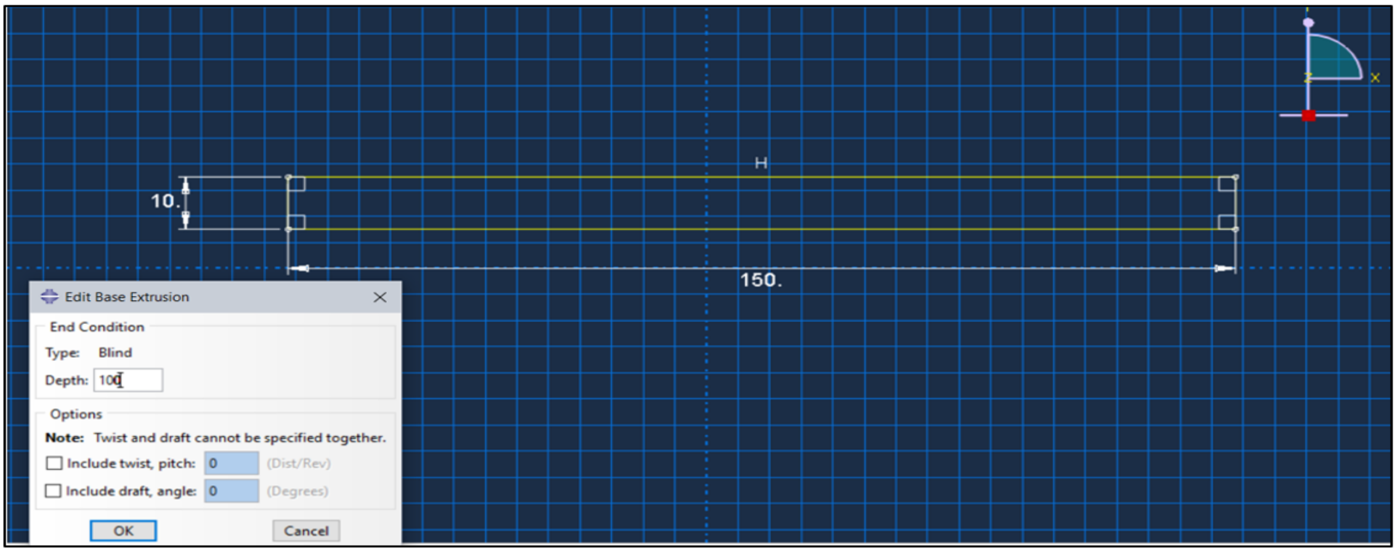

Fig 2 A capture screen from Abaqus illustrating the base extrusions using the edit base extrusions feature.

Figure 3 shows the 3D model of the extruded base by 100 mm.

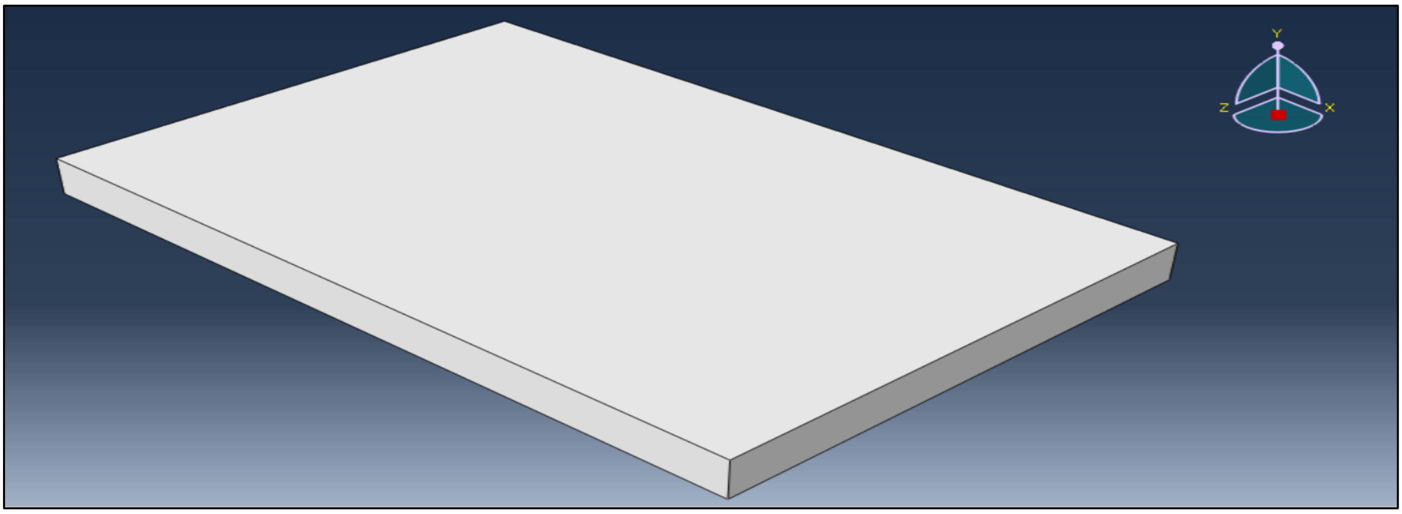

Fig 3 A capture screen from Abaqus illustrating the extruded base.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun783

At that point, the extruded base part was replicated and rotated by 90 degrees to be perpendicular to the modelled base part at first. Then, the rotated part was positioned at the centre of the horizontal directed-based part modelled, initially forming the T-joint shape. After that, the Heat-Affected Zone HAZ was modelled using the chamfer feature in Abaqus, by means of selecting the edges at the intersection points between the two-base part, as shown in Figure 4.

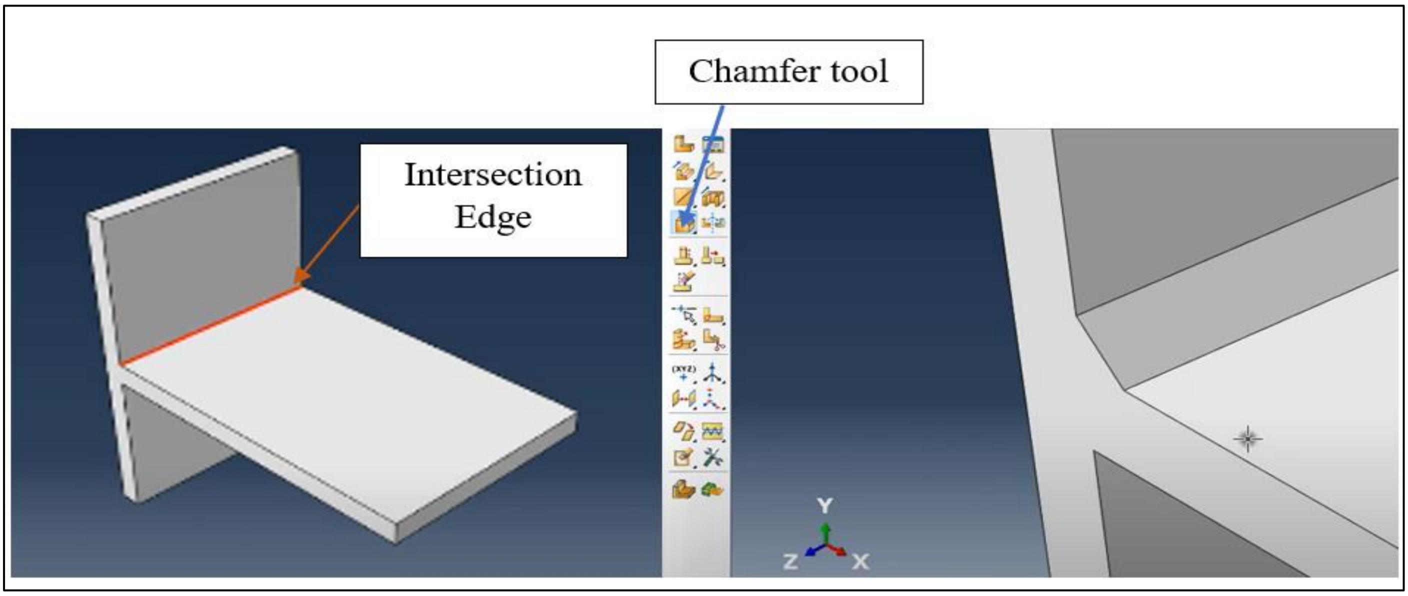

Fig 4 A capture screen from Abaqus illustrating the modelling of the HAZ using the chamfer feature.

Figure 5 shows the final results of Abaqus. At this point, the geometry modelling using Abaqus software is done. The next phases were performed using Ansys software.

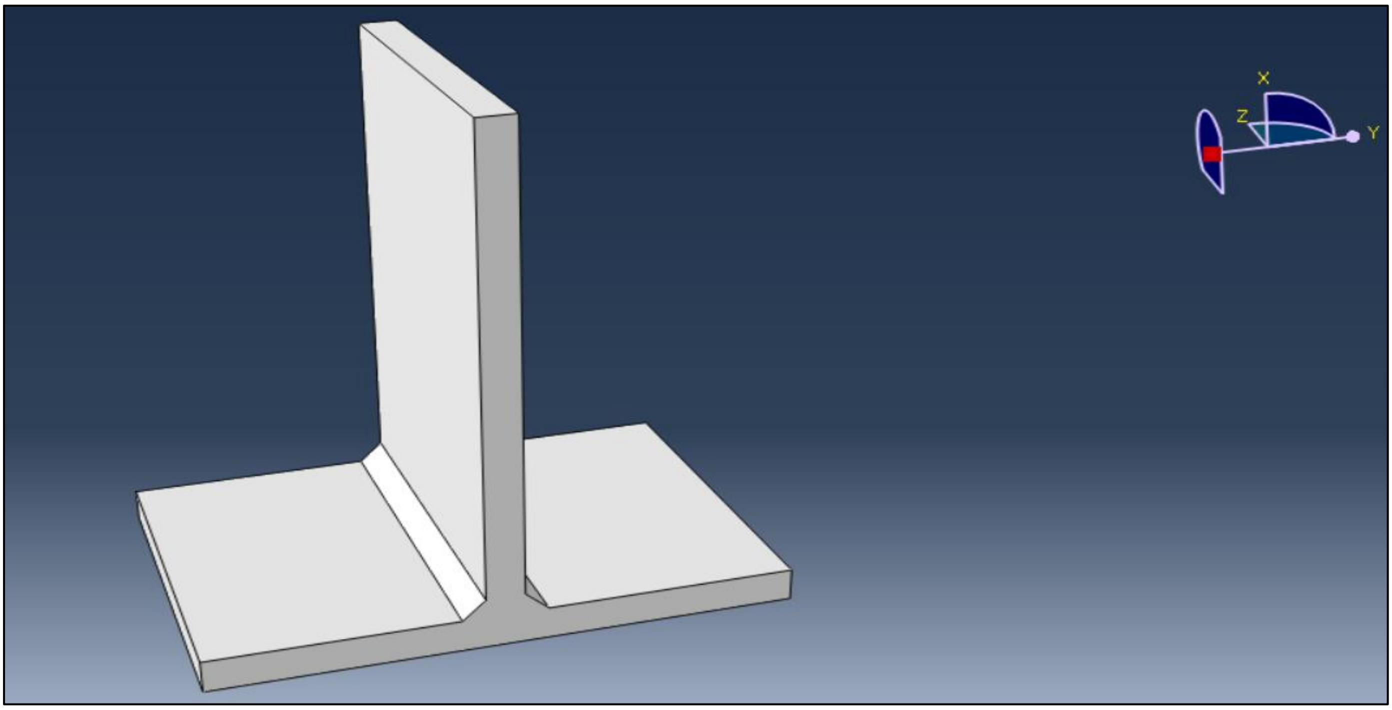

Fig 5 A capture screen from Abaqus illustrating the final T-joint welding plate geometry.

C. Meshing and Boundary Conditions

In order to apply the meshing process; a key step prior to applying the boundary conditions, the built geometry was exported from Abaqus as part file, then imported into two customised tools in Ansys workbench: the “Static Structural” and the “Steady-state Thermal”, since the study is concerned with both mechanical and thermal analysis. Figure 6 shows a capture screen from the Ansys design modeller illustrating the imported part. Figure 7 shows a capture screen from Ansys mechanical model illustrating the imported part.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun783

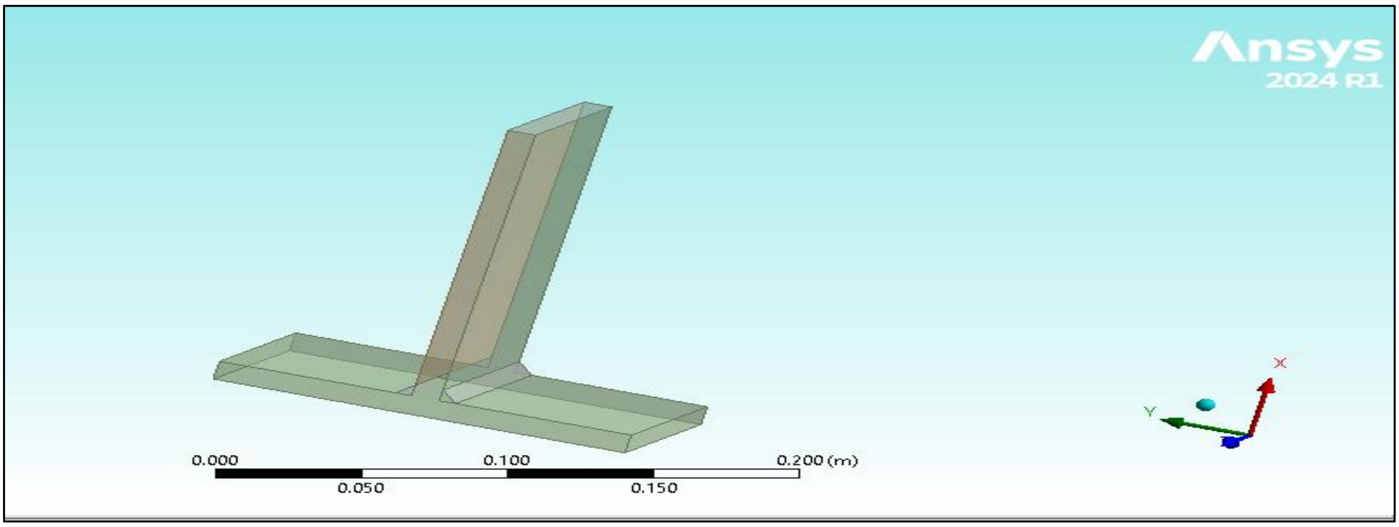

Fig 6 A capture screen from Ansys design modeler c the imported part.

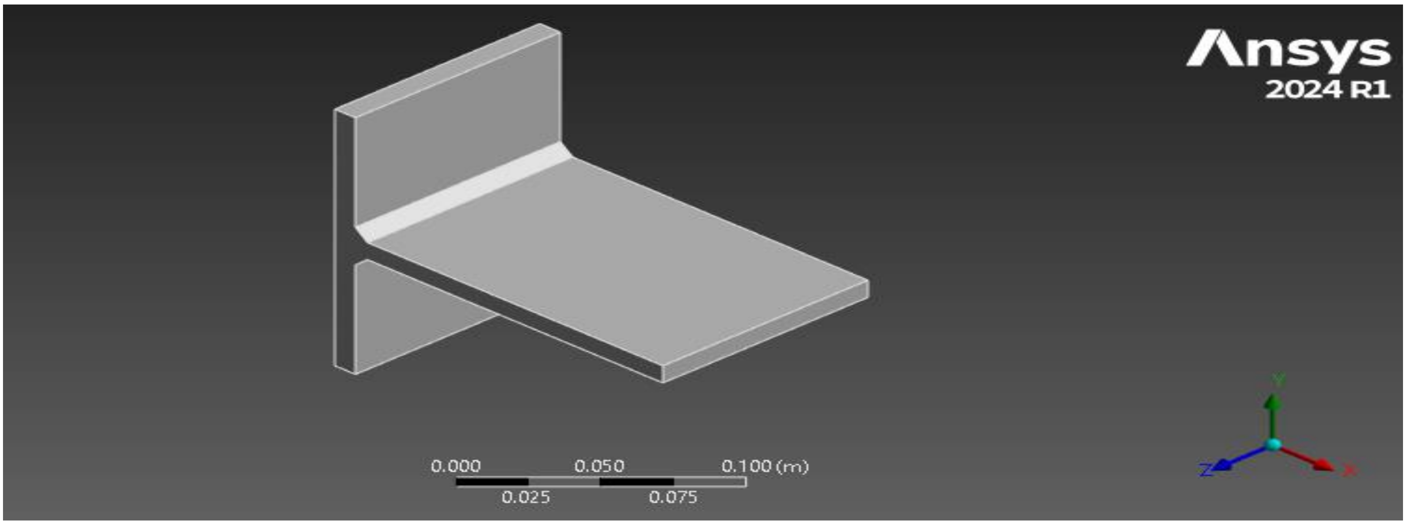

Fig 7 A capture screen from Ansys illustrating the modelled T_joint welding plate.

Table 1 shows the connections information in both the “Static Structural” and the “Steady-state Thermal” tools, showing there are 5 contacts with five active contacts.

Table 1 Model (B4) > Connections.

Object Name	***Connections***
State	Fully Defined
Auto Detection	
Generate Automatic Connection on Refresh	Yes
Transparency	
Enabled	Yes
Statistics	
Contacts	5
Active Contacts	5
Joints	0
Active Joints	0
Body Interactions	0
Active Body Interactions	0

In the mechanical model, in both the “Static Structural” and the “Steady-state Thermal” tools, the imported geometry has meshed using the default meshing settings with Tetrahedrons Inflation Element Type and 1.2 growth rate shown in Figure 8, resulting in 13552 nodes and 12753 elements. The unit system is based Metric system (m, kg, N, s, V, A) Degrees rad/s Celsius.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun783

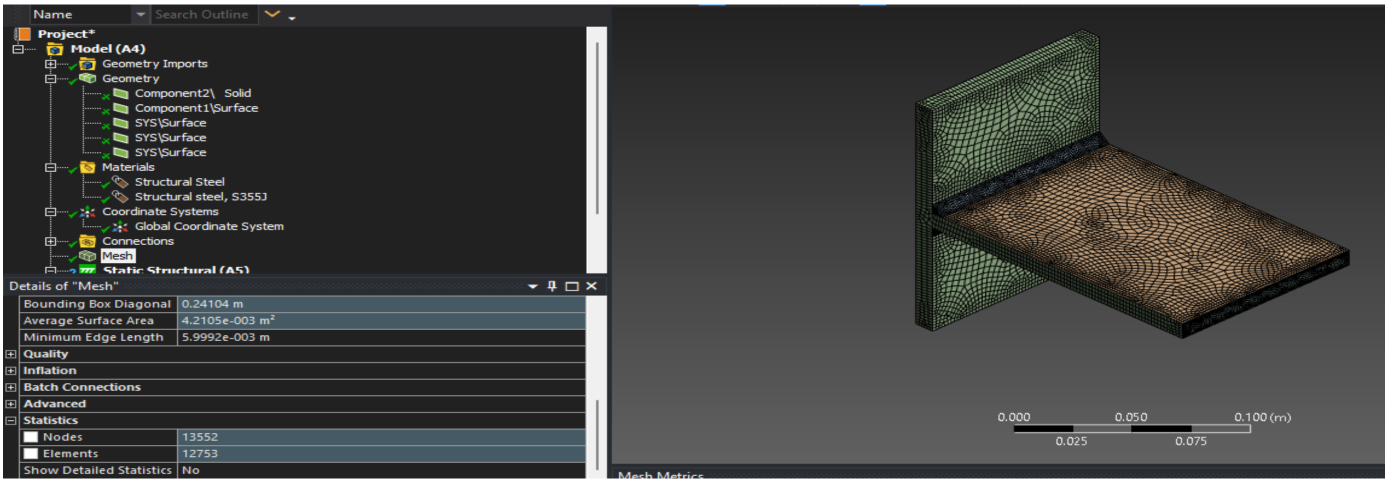

Fig 8 The meshed geometry of the T_joint welding plate in Ansys Static structural- Mechanical with Mesh details.

The analysis in the "Static Structural" model is Structural physics type using the Mechanical APDL solver target, as well as the analysis in the "Steady-State thermal" model is thermal physics type using the Mechanical APDL solver target.

Table 2 Object Name *Static Structural (AS)*

Object Name	***Static Structural (A5)***
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22. °C
Generate Input Only	No
Object Name	*Steady-State Thermal (B5)*
State	Solved
Definition	
Physics Type	Thermal
Analysis Type	Steady-State
Solver Target	Mechanical APDL
Options	
Generate Input Only	No

As shown in Figure 9, two steps were applied in the Analysis setting of the Ansys mechanical solver.

Worksheet

Analysis Settings

Properties	Step 1	Step 2
Step Controls		
Step End Time	1.	2.
Auto Time Stepping	Program Controlled	On
Define By	N/A	Substeps
Carry Over Time Step	N/A	Off
Initial Substeps	N/A	20
Minimum Substeps	N/A	10
Maximum Substeps	N/A	20
Nonlinear Controls		
Force Convergence	Program Controlled	Program Controlled
Moment Convergence	Program Controlled	Program Controlled
Displacement Convergence	Program Controlled	Program Controlled
Rotation Convergence	Program Controlled	Program Controlled
Line Search	Program Controlled	Program Controlled
Stabilization	Program Controlled	Program Controlled
Output Controls		
Stress	Yes	Yes
Back Stress	No	No

Fig 9 A capture screen from Ansys illustrating the details of the analysis settings.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun783

Next, before solving the model in both the "Static Structural" and the "Steady-state Thermal" boundary conditions were applied. For the "Static Structural", a displacement was inserted at three points, particularly at the edges of the plate base, with component vectors of (x=0, y=free, and z=0), which means the model was fixed at and x and z directions while remaining free at y. Moreover, a thermal condition with 400 Celius was loaded on the HAZ solids. Figure 10 shows a capture screen from Ansys mechanical model illustrating the applied boundary conditions in the "Static Structural" tool.

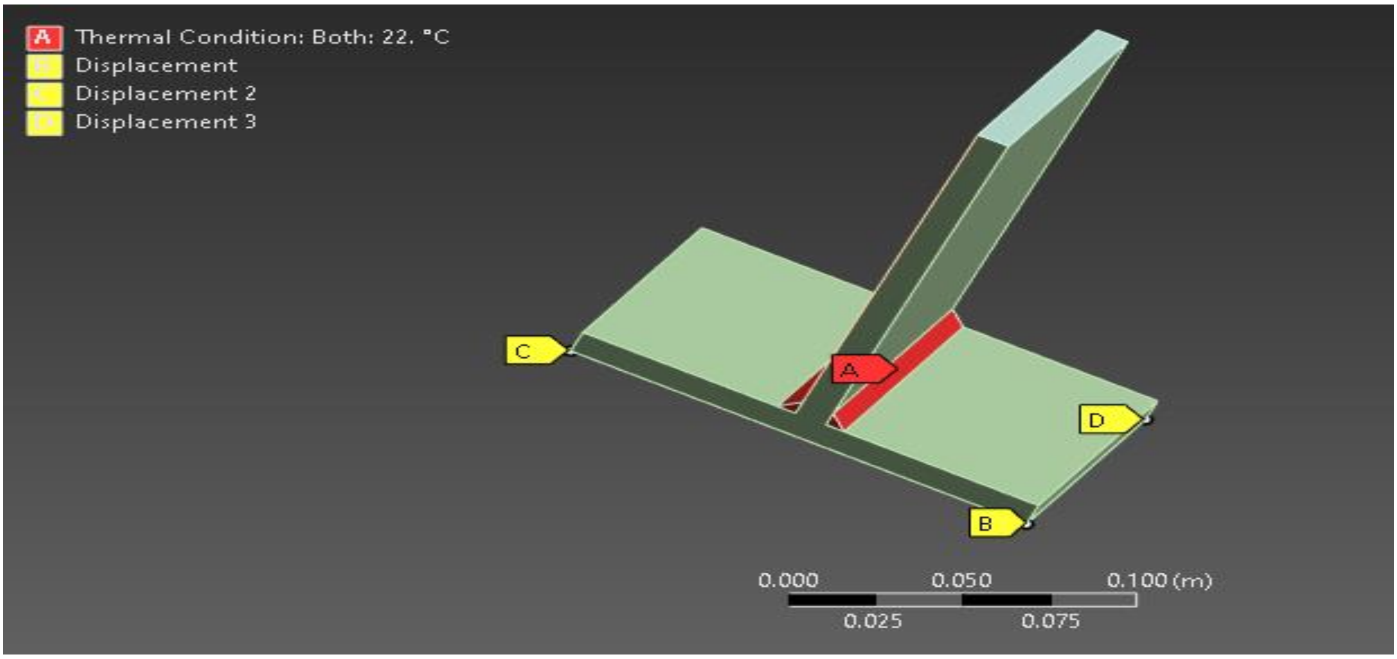

Fig 10 Boundary conditions: Displacement and thermal loading conditions.

For the "Steady-state thermal", a fixed support was inserted at three faces, particularly at the top of the upper plate, along with the right and left sides of the plate base. Moreover, radiation with 22 Celius (ambient temperature) was loaded on the HAZ solids, and at the side faces of the top part, along with convection with 400 Celius, was loaded on the HAZ solids. Figure 11 shows a capture screen from Ansys mechanical model illustrating the applied boundary conditions in the "Steady-state thermal" tool.

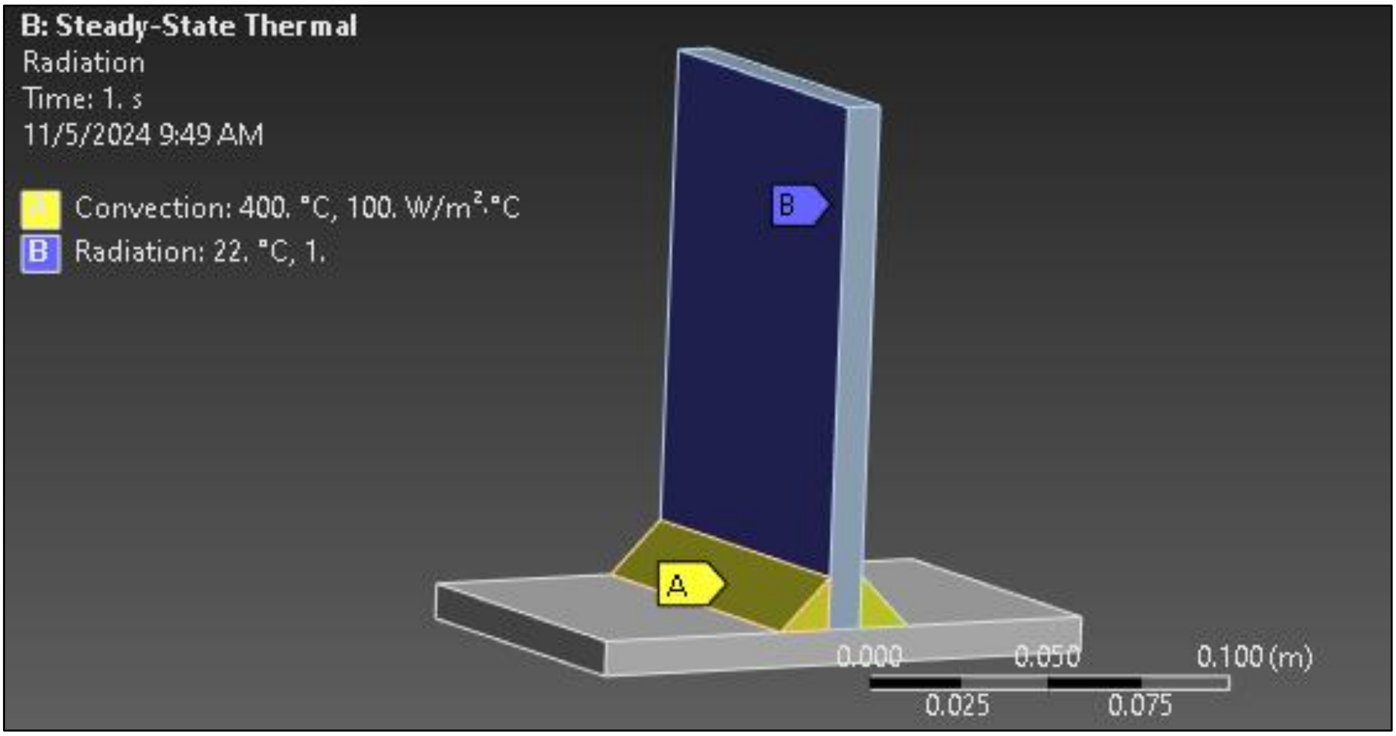

Fig 11 Boundary conditions: Thermal conditions: Convection and radiation loading conditions.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun783

Table 3 and Table 4 show the Static Structural and the Steady-State Thermal Results information, respectively.

Table 3 Static Structural Results information.

Object Name	*Equivalent Stress*	*Total Deformation*	*Normal Elastic Strain*
State	Solved		
Scope			
Scoping Method	Geometry Selection		
Geometry	All Bodies		
Definition			
Type	Equivalent (von-Mises) Stress	Total Deformation	Normal Elastic Strain
Orientation			Y-Axis

Table 4 Steady-State Thermal Results information.

Object Name	*Total Heat Flux*	*Directional Heat Flux*	*Temperature*
State	Solved		
Scope			
Scoping Method	Geometry Selection		
Geometry	All Bodies		
Definition			
Type	Total Heat Flux	Directional Heat Flux	Temperature
Orientation		X-Axis	
Coordinate System		Global Coordinate System	

IV. RESULTS AND DISCUSSION

This chapter presents the results with its corresponding analysis after applying both the "Static Structural" and the "Steady-state Thermal" conditions, in terms of capture screens taken from Ansys mechanical at results.

A. Static Structural Analysis

This section shows the effect of applying thermal conditions on the mechanical properties of the HAZ of the T-joint welding plate to provide the ability to analyse the phenomena that arise because of the welding process. Therefore, three key measures were obtained: total deformation, equivalent stress and Normal elastic strain.

As mentioned before, the thermal conditions of 400 Celsius were loaded onto the HAZ regions concerning the excepted material expansion. Typically, the material expansion is non-uniform because of the temperature gradient observed through the HAZ regions, leading to variance expansion and contraction throughout and afterward the welding process. As expected, the inclusion of thermal conditions (heat) within the HAZ regions initiates thermal stress since the material subjected to a heat load tries to handle this loading by expanding across the cooler regions, such as in the farmost regions of the plate base, to perform expansion resistance. In accordance with the arise phenomena, the high-stress concentration is observed at the HAZ regions. Figure 12 shows a capture screen from Ansys Static structural- Mechanical illustrating the resulting total deformation under thermal conditions.

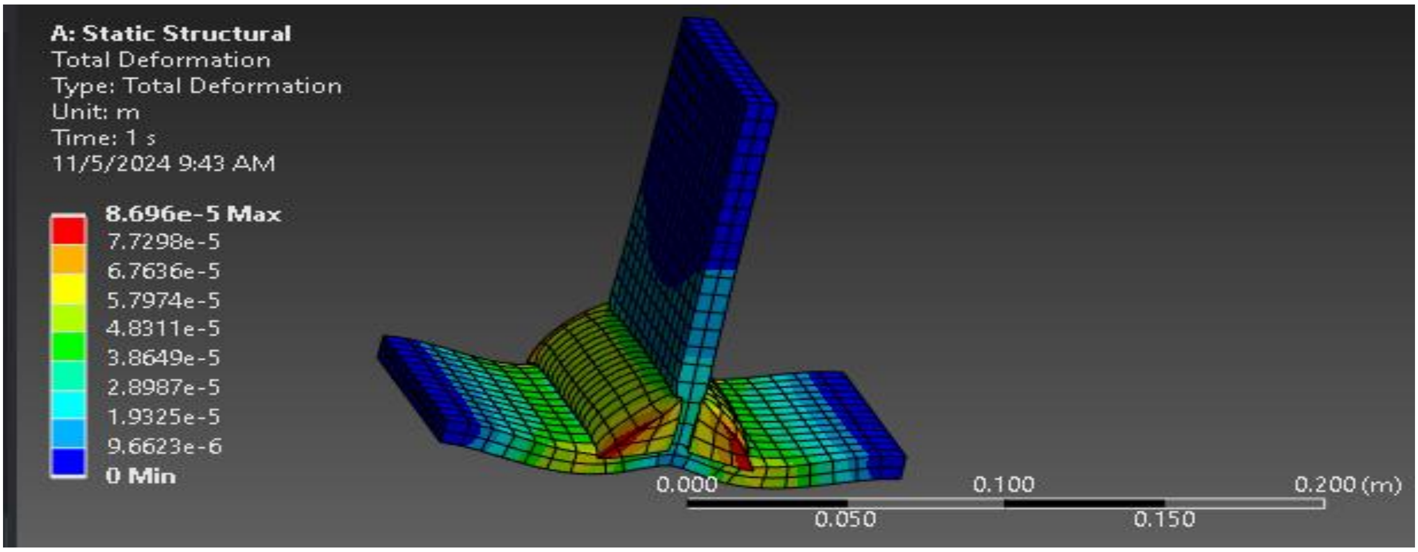

Fig 12 A Capture Screen from Ansys Static Structural- Mechanical Illustrating the Resulting total Deformation under Thermal Conditions.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun783

Based on the observation that the HAZ exhibits the greatest amount of deformation with 86.96 micrometres at the edge of the HAZ regions, while the top of the upper plate, along with the right and left sides of the plate base, were not influenced by thermal condition since these surfaces were fixed supported.

It can be inferred that the thermal expansion that occurs as a result of the heat generated by welding would be enough. As a consequence of the fact that the temperatures attained during the process are higher than the yield temperatures of the material, non-uniform heating and cooling have the potential to create plastic stress and strain, as stated by the non-uniform heating and cooling. Therefore, large local distortions in the HAZ lead to distortions of the joints, such as bending or buckling, which may bring about the destruction of the dimensional correctness and structural integrity of the welded assembly.

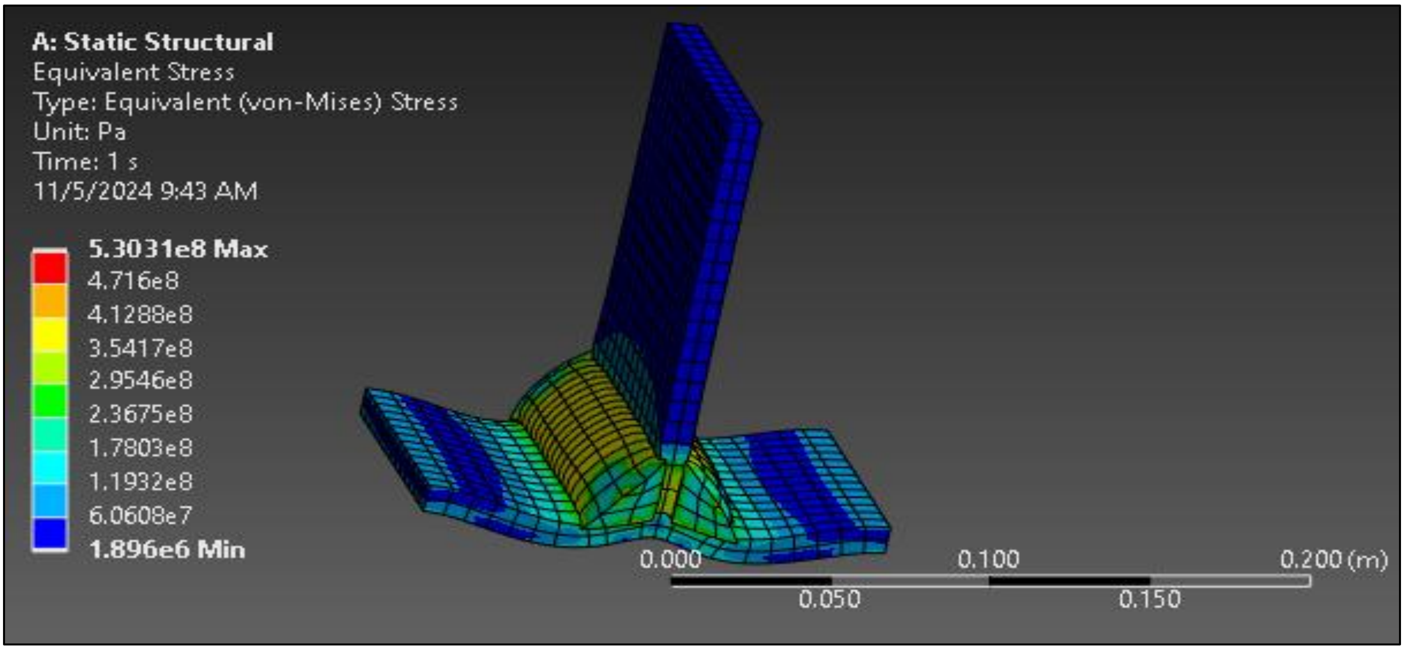

Fig 13 A capture screen from Ansys Static Structural- Mechanical Illustrating the Resulting Equivalent stress under thermal Conditions.

Based on the observation, the HAZ exhibits the greatest amount of stress, with 530.3Mpa at the surface of the HAZ regions, which aligns with the total deformation. Figure 14 shows an Ansys Static structural- Mechanical capture screen illustrating the resulting normal elastic strain under thermal conditions.

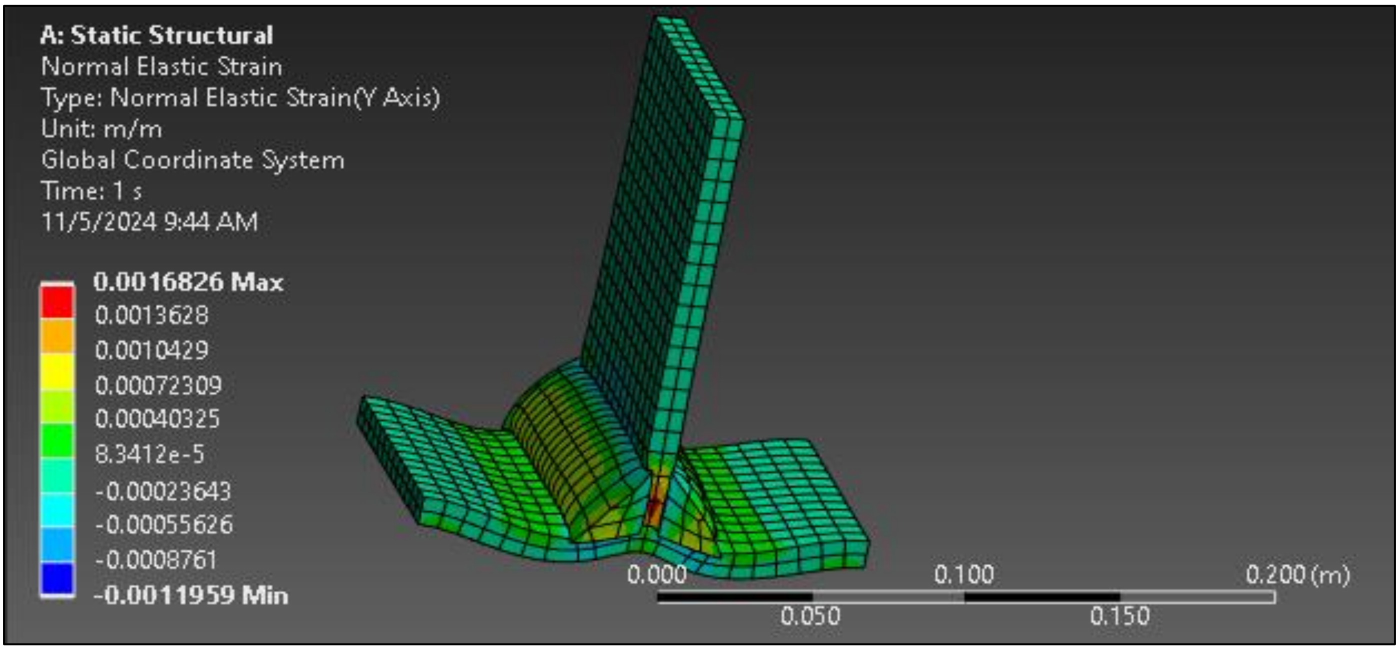

Fig 14 A capture screen from Ansys Static Structural- Mechanical illustrating the Resulting Normal elastic strain under Thermal Conditions.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun783

B. Steady-State Thermal Analysis

This section shows the effect of applying thermal conditions on the thermal properties of the HAZ of the T-joint welding plate to provide the ability to analyse the phenomena that arise because of the welding process. Therefore, three key measures were obtained: total heat flux, directional flux and temperature distribution. Figure 15 shows a capture screen of Ansys Steady-state thermal Mechanical illustrating the resulting total heat flux under radiation and convection conditions.

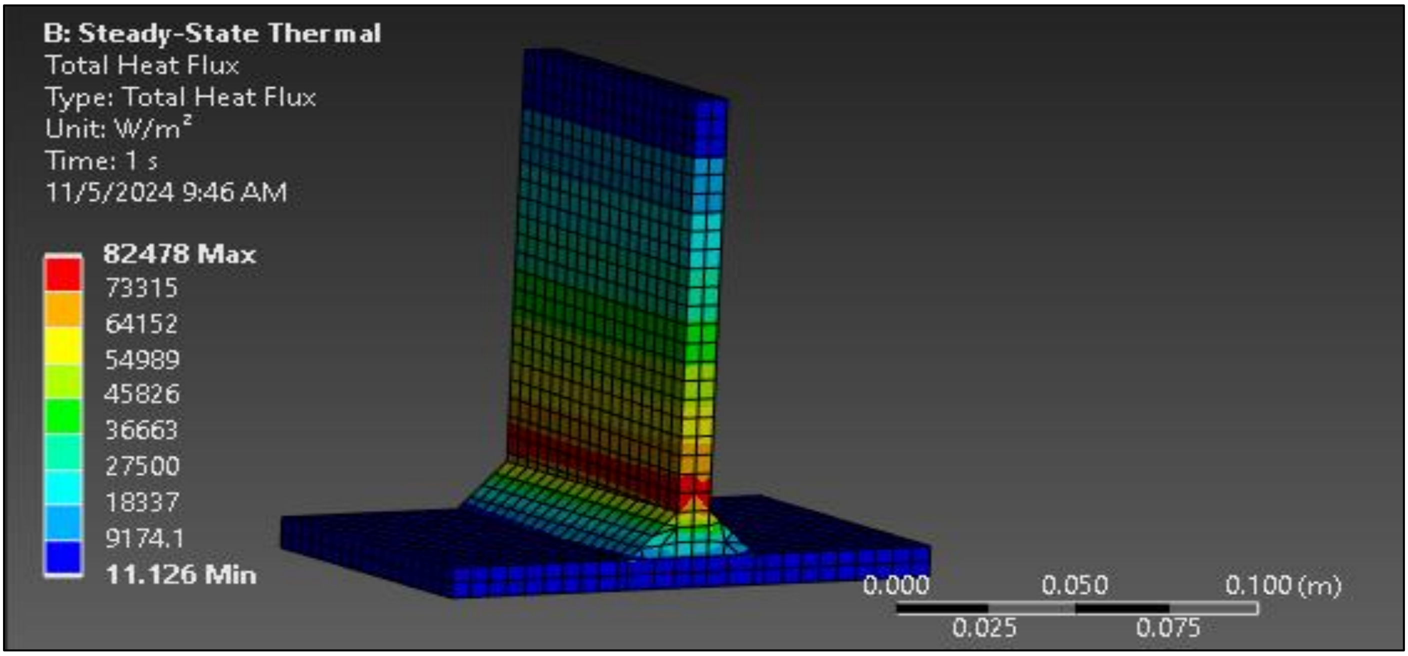

Fig 15 A Capture Screen from Ansys Steady-state Thermal Mechanical Illustrating the Resulting total heat flux Under Radiation and Convection Conditions.

The total heat flux shown above describes the overall overheating region that is centred nearly totally at the weld joint (HAZ regions). As observed, it has the greatest heat flux of 82,478 W/m2, suggesting that this specific location has experienced a significant amount of overheating during the process of welding. This excessive heating indicates that huge rates of heat are being introduced into the welding material, and large rates of heat are being introduced into the surrounding regions. This is extremely significant for the qualities of the material and, ultimately, for the quality of the weld. The surrounding area becomes cold as the weld zone is overheated to extreme temperatures, resulting in local thermal stresses and deformation of the material.

Figure 16 shows a capture screen of Ansys Steady-state thermal Mechanical illustrating the resulting directional heat flux under radiation and convection conditions.

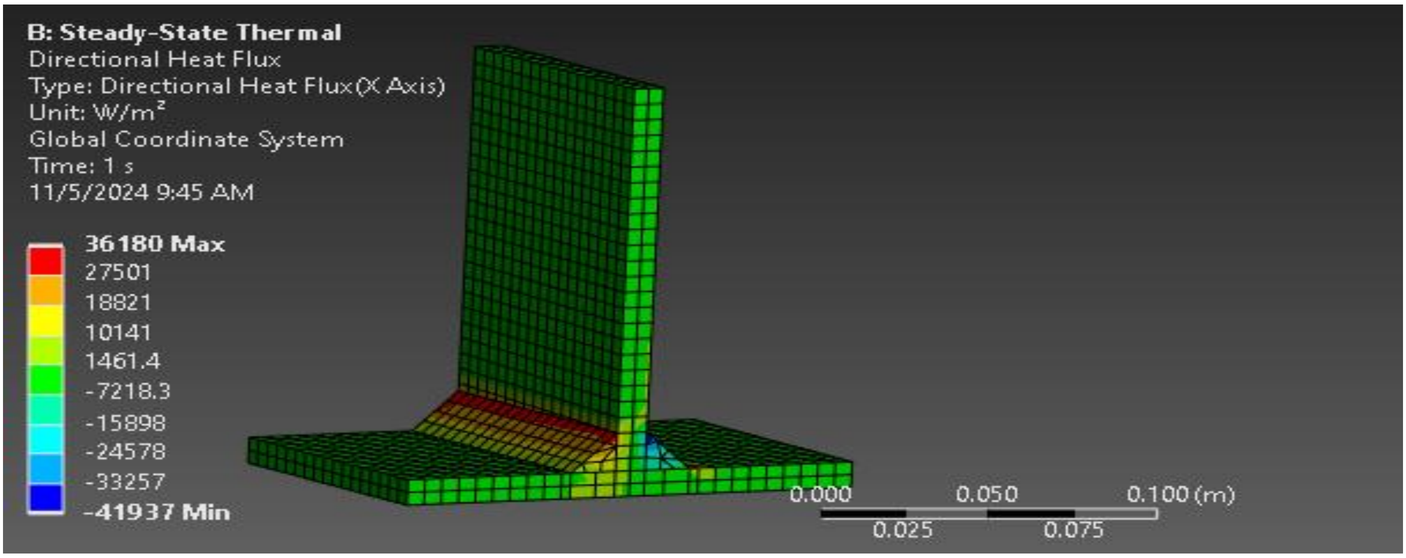

Fig 16 A capture screen from Ansys Steady-state thermal Mechanical illustrating the resulting directional heat flux under radiation and convection conditions.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun783

As for the total heat flux, the directed heat flux reaches its peak of 36,180 W/m2, as mentioned above. This measure shows the information on the thrust angle and dominating force of the melt, which is considered to have been blasted from the weld hole. Convection and radiation to the outside surface are applied loads of the various paths in which heat is transported outward, and negative zones of heat flow represent some of these directions. Figure 17: shows a capture screen of Ansys Steady-state thermal Mechanical illustrating the resulting temperature distribution under radiation and convection conditions.

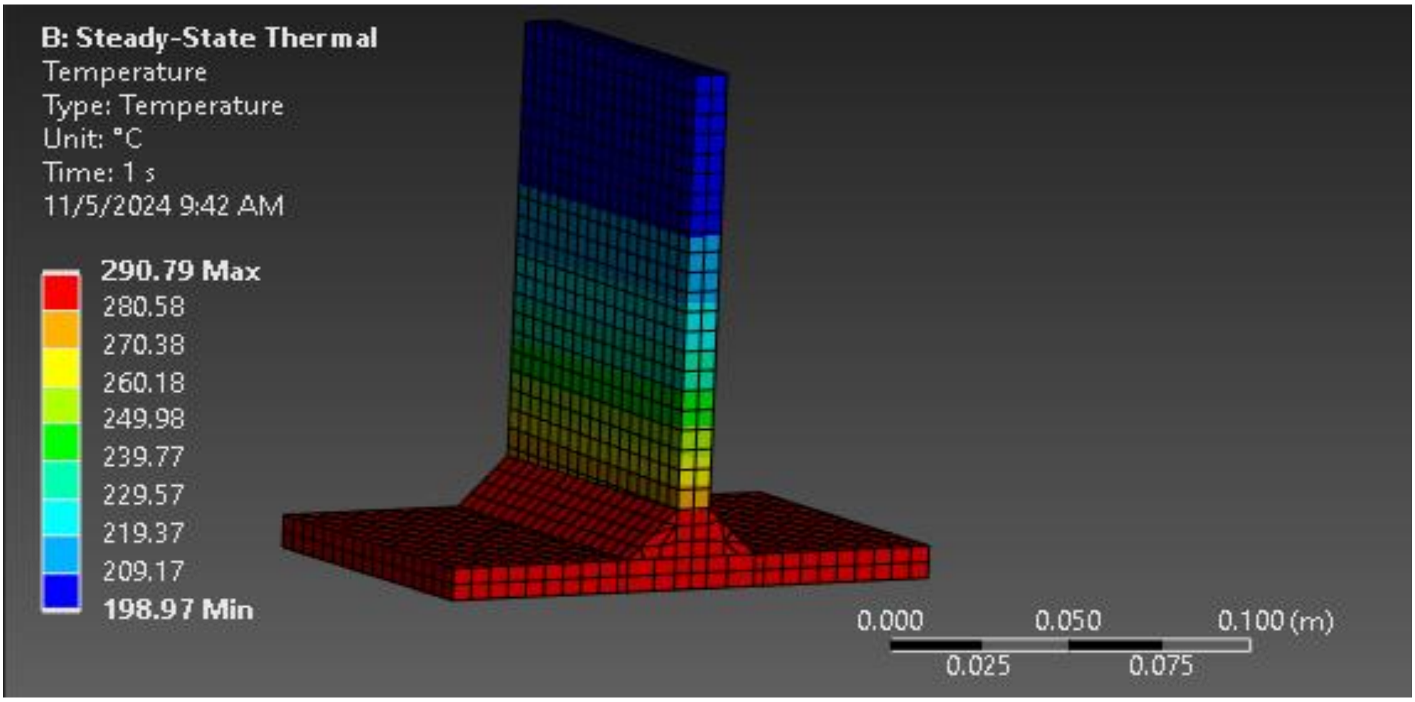

Fig 17 An Ansys Steady-state Thermal Mechanical capture screen Illustrating the Resulting Temperature Distribution under Radiation and Convection Conditions.

According to the results shown from the distribution of the highest temperature, a maximum of 290.79 0C is realised. This temperature was measured above the weld joint and decreased gradually on the borders of the surface of the plate. To be precise, this effect is considered to emulate the welding process that is normally applied, resulting in the localised heating of the area. Starting from its origin, the heated zone expands in all directions while sustaining consistently high temperatures for a time period during which the surrounding regions are experiencing decidedly lower temperatures. Given such an analysis, consideration would be given that some tension and deformation would result in the regions of joints or corners.

V. CONCLUSION

In recent times, the formation of a heat-affected zone (HAZ), which may lead to a considerably lower strength in comparison to the base metal, represents one of the biggest serious issues that arises during the welding process of aluminium. When welding, the zone immediately close to the welding joint is where the majority of the changes in the characteristics of aluminium occur. For the purpose of solving such an issue, the computationally linked analysis of procedures for welding, which enables the identification of distortions and precise stress fields, is made feasible by the simulation instruments for the welding process, which are now considered to be state-of-the-art. Temperature transient analysis serves as the foundation for these models, which take into consideration metallurgical and mechanical processes, among other considerations. It is also necessary to have material characteristics that are a function of temperature in order to conduct an appropriate simulation; however, these properties are uncommon and difficult to locate in either the published literature or the material information databases. The work here, using ANSYS Workbench, has significantly enhanced the study of weld distortions within the context of a T-joint welding plate. An integrated approach that makes use of static structural and steady-state thermal analyses has served well in disclosing complicated relationships between applied thermal loads and resultant material behaviours within the HAZ.

The study was rather rigorous, documenting how changes in thermal loads affect the mechanical properties of the material with a view to yielding important information about the development of thermal stresses and strains responsible for structural deformations. The analysis of the presented results confirms the significance of correctly defining simulation parameters, mesh, boundary parameters, and thermal load in realistically imitating the thermal and structural behaviour of a welding process. The simulations accomplished included representative images of stress fields and temperature fields and, therefore, were useful in predicting the regions most affected by distortions. These results are very important in relating the behaviour of materials during welding processes, therefore permitting the design of better welding patterns that help reduce distortions.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun783

The scans received in regard to the predicted models further contribute to knowledge because they make it possible to predict what the welding parameters would be effective in accomplishing long before actual welding is done. Such an ability not only assists in perfecting the welding parameters to alleviate undesirable distortions but also assists in improving the quality and strength of the welded joints in the structures. Conclusively, the paper underlines the role and contribution of modern analytical tools such as ANSYS Workbench in designing and optimising welded joints with the aim to decrease production costs, rework, and improve structural integrity and safety.

REFERENCES

[1]. Azad, N., Iranmanesh, M. & Darvazi, A., 2020. A study on the effect of welding sequence on welding distortion in ship deck structure. Ships and Offshore Structures, Volume 15, pp. 355-367.

[2]. Bajpei, T., Chelladurai, H. & Ansari, M. Z., 2017. Experimental investigation and numerical analyses of residual stresses and distortions in GMA welding of thin dissimilar AA5052-AA6061 plates. Journal of Manufacturing Processes, Volume 25, p. 340–350.

[3]. Churiaque, C. C. M. et al., 2018. Laser Hybrid Butt Welding of Large Thickness Naval Steel. Metals, 9(1).

[4]. Domański, T. et al., 2022. Numerical Prediction of Strength of Socket Welded Pipes Taking Computer Simulated Welding Stresses and Deformations into Account. Materials, 15(9), p. 3243.

[5]. Granell, I., Ramos, A. & Carnicero, A., 2021. A Geometry-Based Welding Distortion Prediction Tool. Materials, 14(17), p. 4789.

[6]. Kim, M., Kang, M. & Chung, H., 2015. Simplified welding distortion analysis for fillet welding using composite shell elements. International Journal of Naval Architecture and Ocean Engineering, 7(3), pp. 452-465.

[7]. Lee, S., Jeon, H. & Park, J., 2023. Reductions in the Laser Welding Deformation of STS304 Cylindrical Structure Using the Pre-Stress Method. Metals.. Metals , 13(4), p. 798.

[8]. Matuszewski, M., 2019. Modeling of 3D temperature field in butt welded joint of 6060 alloy sheets using the ANSYS program. In IOP Conference Series: Materials Science and Engineering, 659(1), p. 012034.

[9]. Nurzhanova, O. et al., 2023. Simulation of the distribution of temperature, stresses and deformations during splined shafts hardfacing. Journal of Applied Engineering Science, 21(3), pp. 837-845.

[10]. Seong, W., Yoo, J. & Yun, K., 2020. Deformation Prediction Algorithm for Smart Manufacturing of Large Welded Structures. Journal of Welding and Joining, 38(1), pp. 98-104.

[11]. Vemanaboina, H., Akella, S., Buddu, R. K. & Gundabattini, E., 2019. Distortion Validation of Laser Beam Welded SS316LN Steel Plates. IEEE, pp. 1-5.

[12]. Venkateswarlu, K., Kumar, P. N. & RaviKumar, P. S., 2018. Finite Element Simulation of Temperature Distribution, Distortion and Residual Stresses of Dissimilar Welded Joints. Materials Today: Proceedings, 5(5), p. 11933–11940.

[13]. Vladulescu, F., 2018. Structural-thermal analysis of welded joints using the ANSYS workbench platform. Advanced Materials Research, Volume 1146, pp. 17-21.

[14]. Vladulescu, F., 2018. Structural-Thermal Analysis of Welded Joints Using the ANSYS Workbench Platform. Advanced Materials Research, Volume 1146, pp. 17-21.

[15]. Wu, C., Wang, C. & Kim, J., 2021. Welding Distortion Prediction for Multi-Seam Welded Pipe Structures using Equivalent Thermal Strain Method. Journal of Welding and Joining, 39(4), pp. 435-444.