Mechanical Analysis of T-Joint Welding Plate using ANSYS

Abdullah O E M Alrashid[1]; Amer E S E TH Alajmi[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: This study examines the mechanical behaviour and structural integrity of T-joint welds under thermal cycling, representing one of the most important issues from the structural point of view in applications like construction, automotive, and heavy machinery. In fact, because of their geometrical configuration and induced thermal stresses during both the welding and cooling process, T-joints are faced with problems of highly concentrated stresses, deformations, and elastic strains. In this regard, fatigue damage can develop after a certain period of time, as a result of which failure can occur. The objective of this work is to investigate the distribution of stresses and strains in T-joint welds to determine the regions of the weld that are prone to failure, besides proposing optimised welding strategies with the aim of attaining improved durability in the joint. In this paper, ANSYS FEA software is employed in an effort to simulate the effects of thermal cycling on the T-joint weld by studying equivalent stress, total deformation, and elastic strain along the weld bead and HAZ. It follows from the results that peak stresses are concentrated within the HAZ, while the maximum deformation happens around unsupported edges. Optimised cooling rate, structured support configurations, and tailored welding sequences were proposed to mitigate these effects.

Keywords: *Structural Integrity, T-Joint Welds, Thermal Cycling, Mechanical Behaviour, ANSYS FEA Software, Fatigue Damage, Stress, Deformation, Elastic Strain.*

How to Cite: Abdullah O E M Alrashid; Amer E S E TH Alajmi (2025) Mechanical Analysis of T-Joint Welding Plate using ANSYS. *International Journal of Innovative Science and Research Technology*, 10(6), 3226-3236.
https://doi.org/10.38124/ijisrt/25jun784

I. INTRODUCTION

The welding process is the basic art adopted in mechanical engineering for the fabrication of parts used in construction, aircraft, motor vehicles, and other manufactured products. There are several types of welded-joints, but among them, the T-joint is very important in that it allows the linking of two plates at a right angle to each other in order to form the shape of T. This kind of arrangement finds its application in girders, frames, and pressure-vessels. However, mechanical behaviour is complex, particularly under varied loading circumstances. T-joint weld integrity straight influences the performance of the overall structure; hence, mechanical tests of T-joint welds are very significant to forecast their behaviours under diverse stresses. The most powerful of such a tool, which would allow a great amount of detailed simulation concerning stresses, strains, and deformations of welded joints, would be finite element analysis (FEA) software, such as Ansys [1], [2].

In T-joint welds, two metal plates are joined at an angle of a 90-degree, normally by utilising the fillet weld along the junction of the plates. The welds in T-joints are engineered to endure forces exerted in many directions, including tensile, shear, and compressive stresses. Nevertheless, the geometric-discontinuity just at the weld, T-joints are especially prone to the advancement of stress concentration and may fail unless they are well-designed and examined. The quality of the Weld, thickness of the plate, material properties, and the residual stresses as a result of the welding process itself can greatly influence the performance of the joint. However, in practical application, the loading circumstances of a T-joint are usually dynamic with fluctuation or vibration of the forces, which may cause more exacerbation to fatigue and early failure [3].

The primary impetus for research in the welding-field may be said to spring from the need to ensure that weld defects, among other constraints, are tackled as an essential process for joining metals. The mutual unwelcome impacts of putting metals through welding circumstances and cooling subsequently are the material's distortion, and mechanical features change. Deterioration at the area of the weld owing to the induced residual-stresses, among further considerations related to the alterations in the microstructure of the material

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun784

[4] [5]. Residual stresses from welding may have a notable impact on the mechanical behaviour or the structure response. As such, these may raise the possibility of brittle fracture, fatigue, and structural buckling, as well as the stress corrosion cracking of the material [6].

The change in the microstructural of the material is an unavoidable problem in welding as the process inherently requires heating the welding materials above the equilibrium point and melting temperature. There are convective-effects that occur in the liquid pool, which enhance the transfer of the heat. Upon removal of the heat sources, the metal becomes solid as it cools. In the process of cooling, temperature variation in the alloy creates a solid-state transition [7], [8]. The microstructural modifications induce changes in the characteristics of the material as the process advances. The elastoplastic thermal strains taking place around the zone of the welding interact with their resultant stresses to cause permanent -distortions, which in turn impact the dimensional integrity at the joint [9], [10]. The mechanical tests and comprehensive metallographic analysis in microstructural differences of the weld area are necessary for the evaluation of the weld-joint [11], [12], [13], [14].

Prior research facilitated the swift advancement of the welding techniques, thorough examination of thermo-mechanical characteristics of welded-materials, as well as the practical solutions for the defects that associated with the weld, which contributed to the mitigation of essential problems [15], [16]. The associated welding problem gets complex as the base metals or alloys display regional changes in chemical configurations, as observed in dissimilar metals and alloys or various alloys of identical metals, resulting in disparities in thermomechanical characteristics [17], [18].

Consequently, this research further proposes a comprehensive mechanical analysis of T-joint welding plates, focusing attention on the integrity of the structure and the behaviour of the welded-joint under different loading circumstances. Advanced tools will be used to present such an analysis, which is FEA software Ansys since this research aims at simulating the mechanical responses that are produced in the T-joint welds, stresses, strains, and deformations. This study will determine the probable modes of failure, fatigue behaviour, and effects of residual stresses through a detailed mechanical assessment in order to optimise the design and performance of T-joint welds.

II. LITERATURE REVIEW

In mechanical engineering, the fillet welded T-joints have been in great demand for applications, including vehicles, shipbuilding, aircraft, offshore industries, civil engineering, and many other allied fields. These structures are usually welded with the help of filler fusion methods, which produce significant heat. The localised heat produced through welding, followed by quick cooling, results in persistent geometric flaws in structures and the formation of residual welding stresses [19]. These geometrical flaws often lead to serious issues throughout the assembly, while residual-stresses may promote stress corrosion cracking, fragile fractures, or fatigue crack initiations [20]. Mitigating these impacts necessitates supplementary financial expenditures and is sometimes unfeasible owing to the substantial dimensions of the structures. During the last few decades, advances in fast computer technology have stimulated several numerical simulation improvements and applications to forecast the deformations and residual stresses of the welded structures during their initial design stage [21].

However, for the last years, because of their large diffusion in several fields of structural use, the T-joint fillet welded joints have become of great interest to academics. [22] examined the impact of the thickness of the plate on the welding deformation by numerical simulations and experiments. [23], [24]and [25] study the impact of sequences of welding on residual stress distribution and deflections of the plate. [26]Performed a combination of numerical and experimental investigations of various parameter impacts, including the boundary conditions and sequences of welding, on the residual stress fields and the deflections of the plate. While [27] examined the effect of different designs of plates on weld-induced distortions and the utmost strength of plates that have been fillet welded.

The behaviour of T-joint welds is analysed by FEM in [28], through the adoption of the SYSWELD software. In particular, the modelling of distortions of the welding and residual stresses in the simulation of laser and arc beam welding methods are considered. As well in this work, the issue of mesh-sensitivity has highlighted that hybrid hexa-tetrahedral meshes allow getting more effective outcomes in residual-stress and distortion analysis. Both Single and double-pass T-joint configurations were considered to study how such welding processes could influence the natural level of stress concentration and the degree of distortion. The developed simulation methodology in this work is further extended to more complex structures like heat exchanger joints, showing how laser beam welding could reduce the residual stresses and distortion due to its concentrated heat input.

To expedite the simulation process of welding, [29], [30], and [31] introduced innovative welding simulation techniques that integrate shell and three-dimensional finite elements(FE). Meanwhile, [32] and [33] examined the gaps and contact in welded-plates with a T-joint fillet. [34] and [35] examine the impact of simplifications of the property of the material on both temperature and residual stress and the deflections of the plate fillet welded connections in T-joints. [36] created an interface element in order to characterise efficient penetrations, dimensions of fillet weld, and the interaction between the plates and the stiffeners throughout the simulations of welding distortion.

Considering that, in general, the heat generated by the deformation of the material's plasticity may be disregarded in the heat transfer study., in all aforementioned simulations of welding the T-joint fillet, a sequentially-linked nonlinear method of computing is utilised. The computational technique comprises two distinct analyses: thermal and

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun784

mechanical. The transient stress field is computed in the thermal analysis using temperature-dependent material parameters. The acquired Thermal history is subsequently employed in mechanical analysis as a thermal load. The element birth and death approach is used in both thermal and mechanical investigations to model weld filler deposition. This technique first involves deactivating the weld set components in the thermal analysis and adjusting their associated conductivity to a value near zero. Upon the application of heat input, the weld set components are reactivated, restoring their associated conductivity to the initial value. In the mechanical analysis, all weld sets are initially disabled, and their stiffness is adjusted to a value near zero. Upon reactivation of the elements, their stiffness is restored to the initial value.

The birth and death approach of the element is very frequently utilised because it vividly describes the physical characteristics of a welding process. Conversely, it is well known that large displacements are induced by enormous strains; hence, using this method can lead to a significant issue that arises during the mechanical analysis of the welding [37].

A hybrid method, as presented in [38], was proposed to overcome these problems. This hybrid method focused on performing an accurate simulation using Finite Element Analysis (FEA) for the welding process of T-joint fillets. This study addressed some of the related challenges that resulted from the large-displacements and stresses in conventional birth-and-death techniques that are widely utilised to simulate the welding process. The hybrid method makes the simulation processes simpler, especially when performing thermo-mechanical analysis and customising simulation thermal inputs. The validity of the simulated and thermo-mechanical analytical model is verified by comparing the simulation results (deflections of the plate and residual stresses) with the values of the actual practical application of the experiment. The outcomes of the comparison between both procedures confirmed the high agreement between the simulation and the actual experiments. Thus, the effectiveness of applying the hybrid method to study the thermo-mechanical effects of T-joint welding is very high. This could greatly improve welding procedures while maintaining the structural integrity of the structure.

III. METHODOLOGY

The mechanical integrity of welded structures is a very important feature. Especially for practical applications, structures may be subjected to thermal and mechanical loads periodically. This paper provides an analysis of a T-joint under a thermal cycle. This is because T-joint is a basic form of welded joints. It is also widely used in most structural frameworks.

Besides that, the T-joints is that they provide high structural support, and they are capable of high load bearing. Thus, these joints are susceptible to stress concentration that may lead to fatigue or failure, especially with the repeated heating and cooling processes during the welding process. The analysis in this paper will rely on the finite element analysis (FEA) in ANSYS. This is to accurately model the mechanical behaviour of the T-joint welds under thermal cycles. This analysis will provide sufficient details on the mechanical flexibility of the joint and possible sites of failure.

A. Overview of T-Joint Weld Simulation

ANSYS simulations are performed in this paper to evaluate the stress, strain, and deformation that T-joint welds undergo during the welding process. The high temperatures that the joint is exposed to are estimated to be around 400 °C. The T-joint is then gradually cooled to room temperature (25 °C). This is a thermal cycling scenario, which represents the operational conditions of the real-application of the welding process. Then, during the welding, the T-joint is exposed to large temperature fluctuations, which can lead to thermal expansion and contraction within the joint's material. This event can lead to residual stresses and potential deformation in the T-joint. Therefore, the joint may experience structural stability over time.

The applied simulation involves several steps and procedures. Starting with the sequential creation of a thermo-mechanical model of the T-joint in ANSYS. The created model allows the temperature distribution across the T-joint to be represented. It also allows the mechanical response of the joint to be analysed later, providing a detailed mechanical analysis of T-joint welds under thermal cycling. The outcome of this simulation will enable the study of how temperature affects the mechanical properties- stress, strain, and deformation- of the T-joint. After that, It will be possible to detect the potential weak points inside the joint. Based on these results, some improvements can be suggested to enhance the T-joint design.

B. Ansys's Modelling Approach for the T-Joint

Step-by-step procedures were used to achieve the study's objectives. This is carried out to ensure that the temperature and mechanical behaviours of the T-joint are accurately modelled. At this stage, the joint was created in ANSYS through the following steps:

- *Step 1: Create a Geometry Model of the T-Joint in SpaceClaim*

In this step, the T-joint geometry is created using the ANSYS SpaceClaim tool. This tool prepares and modifies CAD models specifically designed for FEA. The T-joint model was created using two metal plates intersecting at a 90-degree angle. The geometry of the constructed model is (0.15m*0.16m*0.1m). The weld bead was modelled as a triangular section along the intersecting edge of the plates. The following Figure shows the created T-joint geometry:

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun784

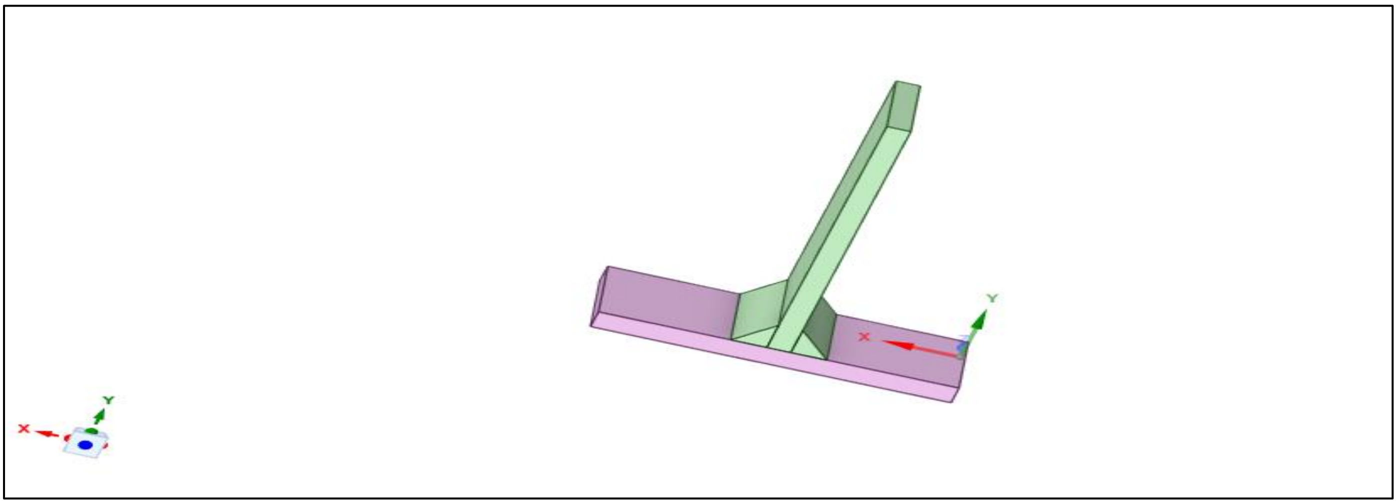

Fig 1 T-Joint Welding Plate Designed on Space Claim

The beads represent the fusion zone between the joint's plates. This configuration accurately simulated the weld bead, allowing for accurate simulation of the heat-affected zone (HAZ) and stress concentration points.

➢ *Step 2: Importing the Created Geometry into the ANSYS Mechanical Module*

After the T-joint, 3D geometry is created in SpaceClaim and saved in a separate IGES file. In the current step, it will be imported into the ANSYS Mechanical module as '.scdoc' file. The properties of the imported model are a volume of $3.22 * 10^{-4}\ m^3$ and an estimated mass of $2.5268\ kg$. After the geometry is loaded in the mechanical module, it can be analysed to simulate a real-world system. In this module, the direct measurements, applied loads, and boundary conditions can be set.

➢ *Step 3: Setting up the Initial Load and Meshing*

At this stage, the approved material properties will be assigned, which is Structural steel, S355J. The properties of this material which were included in this simulation were the material temperature, thermal conductivity, specific heat, and others. Some of the main properties are shown in the Figure below:

Table 1 The Properties of S355J Steel Material

Density	7835 kg m^-3
Tensile Yield Strength	3.834e+008 Pa
Tensile Ultimate Strength	5.653e+008 Pa
Specific Heat	469.9 J kg^-1 C^-1
Resistivity	2.232e-007 ohm m

To obtain an accurate description of how stress and strain are distributed across the T-joint, meshing of the model is a very important step. It provides the ability to analyse the areas close to the weld bead and HAZ. The meshed model is shown in the subsequent Figure:

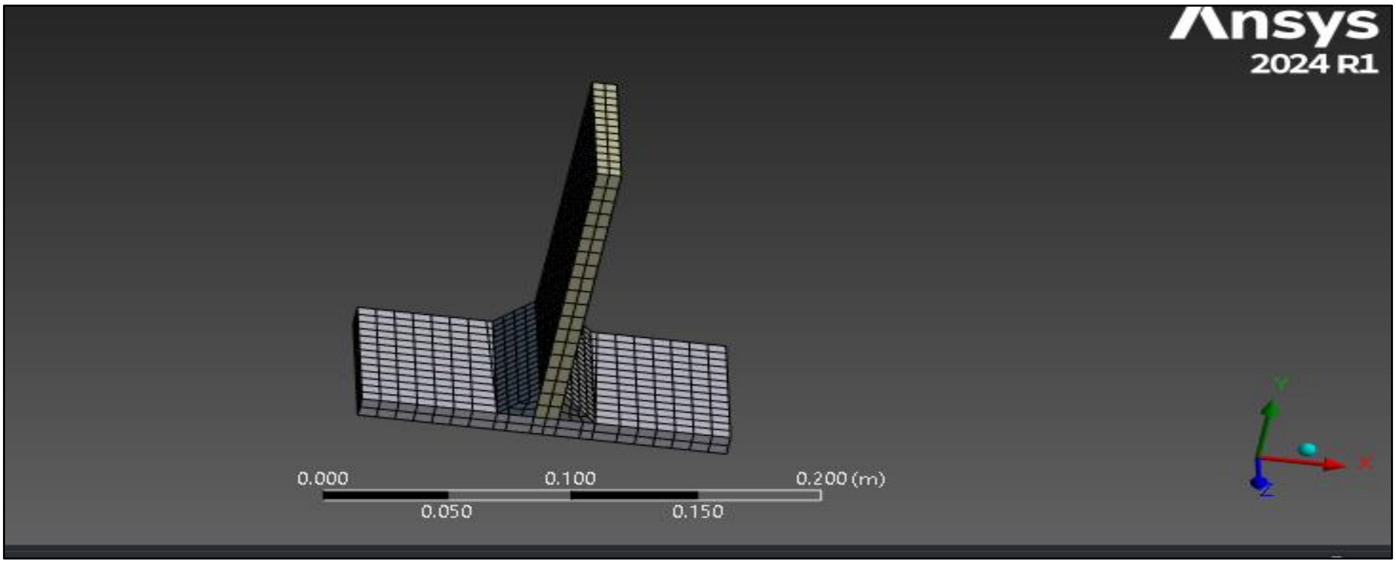

Fig 2 Mesh Applied on T-Joint Welding Plate on ANSYS

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun784

After that, the detailed meshes were created, and the boundary conditions were set. This module n supports a wide range of material and load configurations. Four bodies were used to represent T-joints (two plates and two triangular beads, therefore, after the meshing step, 7719 nodes and 1269 elements were identified.

In this step, the loads will be determined, and the initial constraints will be determined in preparation for the thermal and mechanical analyses in the subsequent steps.

➢ *Step 4: Setting the Support Constraints*

In the current step, the fixed supports will be applied in certain locations on the T-joint structure, which are the top surface and the two-end surface. This step encounters the realistic boundary conditions in the simulation, which can prevent unwanted displacements of the joint throughout the thermal cycle simulation. Therefore, setting these constraints helps in the stabilisation of the model during the thermal cycle. The following Figure presents the location in the T-joint model:

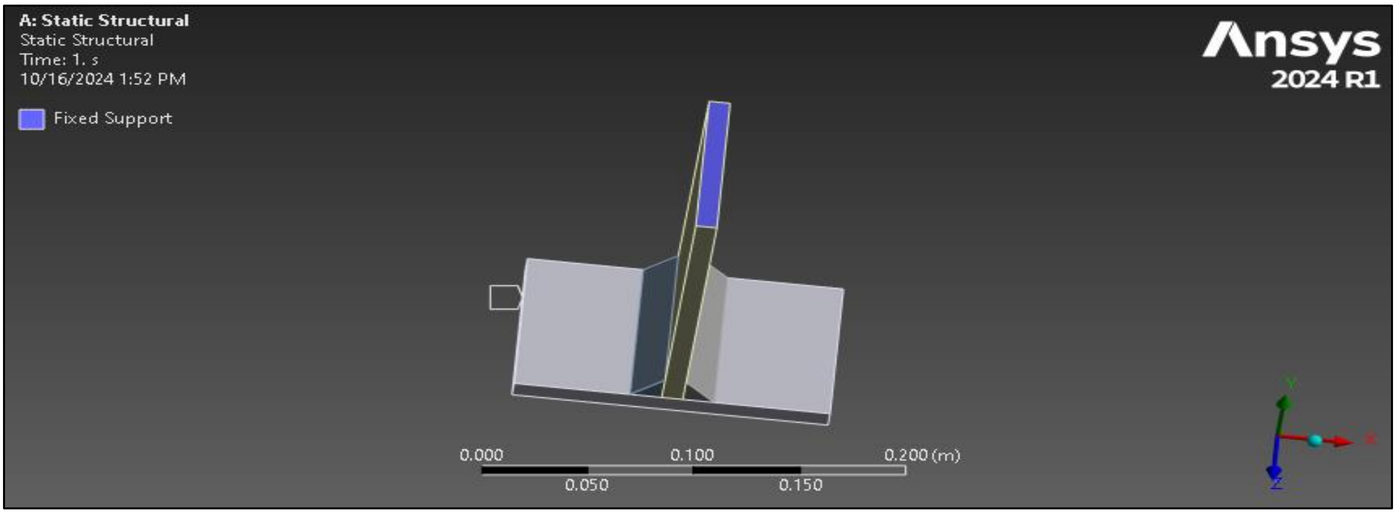

Fig 3 Application of Fixed Support on Modelled T-Joint.

These constraints will provide thermal loading stability in the structure of the model. The real deformation that normally happens and is bound near the regions of interest around the weld is ensured here. Next, the thermal boundary conditions for defining will be done in the next step.

C. Thermal Simulation

Thermal loading and boundary conditions are indispensable in simulating the realistic behaviour of the T-joint weld during thermal cycling. In this stage, the modelling of a joint being exposed to high temperature and subsequently cooled will be done. Stress and deformation increase from there due to thermal expansion/contraction, which are then extracted and presented in the Results section.

➢ *Step 5: Setting Thermal Boundary Conditions*

The boundary conditions were applied in two sub-steps. This is to apply a realistic simulation of the thermal cycle. This was to simulate the real thermal exposure of the joint during welding operations. The details of the two steps are as follows:

- High-temperature initial load at 400°C. In this sub-step, the entire body of the T-joint weld was subjected to a uniform temperature load of 400°C. This step simulates the exposure of the weld to extreme operating temperatures. The image below shows the thermal condition under the initial condition of the structure:

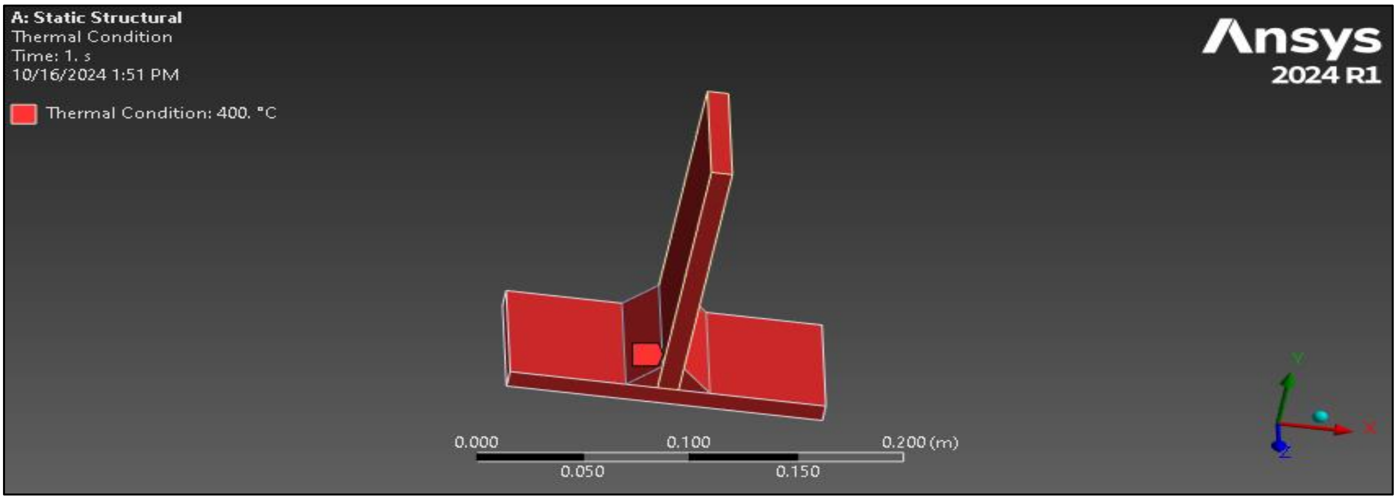

Fig 4 Thermal Condition Applied to the T-Joint Weld Plate on ANSYS.

- The second stage is the gradual cooling stage to room temperature, which is 25 °C. In this step, the thermal load will be gradually reduced from 400 °C to 25 °C, which represents the cooling stage. The cooling is applied uniformly across the T-joint structure. This is to capture residual stresses within the T-joint weld.

D. Mechanical Analysis Residual Stress/Deformation

In this step, the mechanical analysis of the T-joint structure during the thermal cycle from 400 °C to 25 °C will be performed. The analysis includes residual stress distribution, deformation, and strain behaviour of the structure. This analysis is crucial for understanding the response of the T-joint structure after being exposed to high temperatures and subsequent cooling. Static Structural (A5) properties are set to perform the analysis, as Table 2 shows.

Table 2 Analysis Definition and Settings

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

Object Name	***Analysis Settings***
State	Fully Defined
Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	On
Define By	Substeps
Initial Substeps	20.
Minimum Substeps	10.
Maximum Substeps	20.

➤ *The Following Subsection Provides the in-Details Evaluation Steps:*

- *Residual Stress Evaluation*

Since the residual stress is one of the most prominent aspects of the mechanical behaviour of T-joints, it represents the effects of the recurrent expansion and contraction in the joint structure due to temperature changes. This analysis is performed by mapping the distribution of residual stress using the von Mises criterion across the T-joint. In this map, the region of high residual stresses in the structure can be identified. These regions act as a stress concentration area, which will be a potential weak point under cyclic thermal loading. High values of residual stresses in these areas mean that the area is more susceptible to material stress.

- *Deformation Analysis*

In this analysis, the recurrent joint expansions and contractions will deform its structure. This may affect the dimensional stability of the structure and lead to joint misalignment during the thermal cycle. Full deformation of the entire T-joint is measured in this analysis. Areas with a high deformation value mean that there is a large displacement within these areas. This means that the specified areas might experience failure later under operational loads.

- *Normal Elastic Stress Distribution*

In this analysis, the elastic distribution of the stress in a T-joint weld will be measured. Since the elastic stress distribution demonstrates how the material resists the load without yielding. This analysis gives an idea about the zones which can fail due to fatigue. Elastic stress analysis normally relies on mapping the elastic distribution of stress throughout the T-joint structure. Correspondingly, after elastic stress mapping, areas with severe elastic deformation can be located according to the highest value of elastic stress.

IV. RESULTS AND DISCUSSION

This section outlines the outcomes of the ANSYS simulation for the T-joint weld plate under thermal cycling. The three critical mechanical responses from the simulation are; The equivalent (Von-Mises) Stress, Total Deformation, and the normal Elastic Strain, which enable the understanding of the regions of high stresses, deformation modes, and the elastic strain distribution along the T-joint.

A. Equivalent (Von-Mises) Stress Distribution

The outcomes for the equivalent (Von-Mises) Stress Distribution develop critical insight into the thermal cycling-induced stress behaviour of the T-joint weld. This type of analysis is considered very important for the identification of highly stressed regions, which act as possible crack initiation

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun784

and fatigue failure sites in high-temperature applications. The distribution and intensity of von Mises stress around the T-joint weld would better explain the action of thermal expansion and contraction on the structural integrity of the weld. Figure 2 shows a pattern of distribution of the stress, pointing to the main area where stress accumulation can compromise the joint durability under cyclic loading conditions.

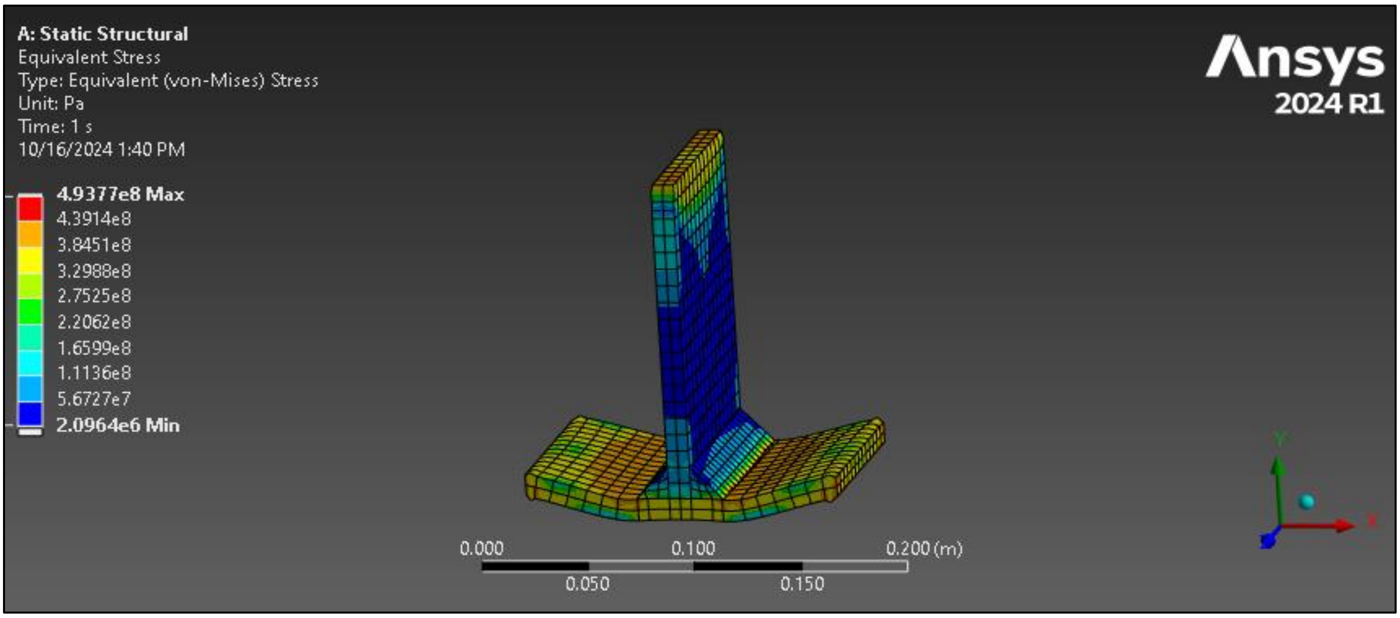

Fig 5 Equivalent (Von-Mises) Stress of T-Joint Welding Plate Resulted from the Thermal Condition on ANSYS.

Figure 5 displays the equivalent (Von-Mises) Stress distribution in the T-joint welding plate under thermal circumstances. It can be seen from the gradient colour that the low stresses are blue- colour, while the high stresses are red-colour. It is noticed that the highest value of stresses, with a maximum magnitude of about 4.9377×10^8 Pa, is concentrated within the HAZ near the weld bead and also at the intersection between the vertical and horizontal plates. This area of high-stress concentration indicates portions within the welds that may most easily fail under the stress.

B. Total Deformation Analysis

The Total Deformation experienced by the T-joint welding plate as a result of thermal cycling from 400°C to room temperature (25°C) are examined. This analysis is an important aspect in order to find out the influence of thermal expansion and contraction on the structural integrity of the weld, especially in the case of an unsupported area around the weld. The deformation pattern allows the identification of the regions that are most vulnerable with respect to displacement. Problems in alignment or affecting the stability of the whole structure may appear when there is repeated thermal loading. Figure 3 displays the Distribution of deformation - maximum and minimum displacements across the T-joint weld.

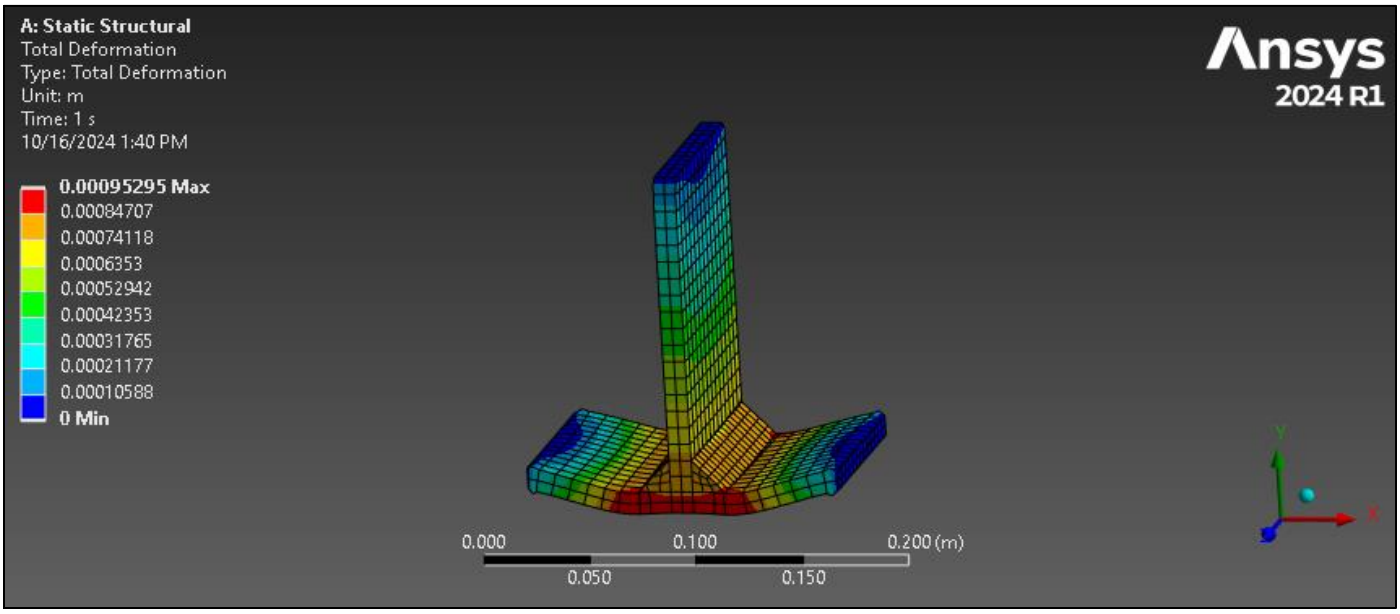

Fig 6 Total Deformation of T-Joint Welding Plate Resulted from the Thermal Condition on ANSYS.

The total Deformation of the T-joint welding plate is represented in Figure 3 above. The minimum value of deformation is $6.5878 \times 10^{-7} m$, while the maximum value of deformation is $9.5295 \times 10^{-4} m$. Maximum deformation is located around the free edges of the vertical plate. Minimum displacements are located in the fixed support regions. It therefore follows that the influence of thermal expansion and contraction is most pronounced in those areas of the T-joint left unsecured, while it becomes less pronounced near the constrained areas themselves.

C. Normal Elastic Strain Distribution

The distribution of the normal elastic strain imparts information on the deformation that occurs within the T-joint weld plate due to thermal cycling without material yielding. As a matter of fact, this parameter is of primary importance while assessing the material resilience against the repetitive action of thermal stresses, since it shows to what extent the material will be elastically stretched before switching to plastic deformations. The elastic strain in the weld is investigated, particularly in the HAZ and in regions around the weld bead, to identify where fatigue processes are likely to occur. With this study, the long-term performance of the weld in-service loading conditions could be identified. Figure 4 displays the normal elastic strain contours that highlight areas which could potentially be problematic for the accrual of fatigue damage over time.

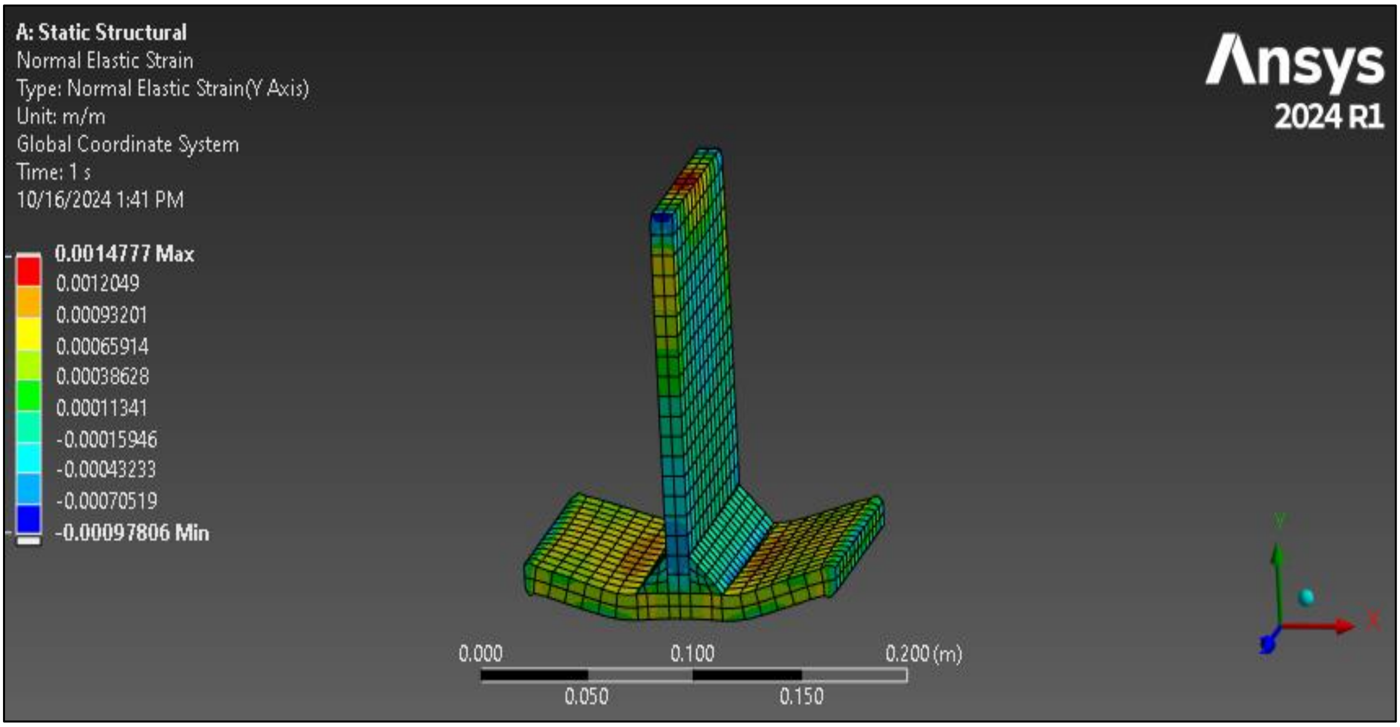

Fig 7 Normal Elastic Strain of T-Joint Welding Plate Resulted from the Thermal Condition on ANSYS.

Figure 7 displays the Normal Elastic Strain distribution for the T-joint weld plate along the Y-axis. The minimum and maximum values were $-9.7806 \times 10^{-4}\ m/m$ and $1.4177 \times 10^{-3}\ m/m$, respectively. High elastic strain concentrations are reflected in the strain distribution around the weld bead and along the edges of the plates adjacent to the weld. These areas are subjected to very high elastic stretching due to thermal loading; they show the probable regions of material fatigue in cases of long-term exposure to thermal cycling.

D. Discussion

This part discusses the implication of findings from simulation, with a focus on how stress distribution, deformation, as well as strain observed can influence the structural soundness and the fatigue resistance of the T-joint weld, besides relating them to the results from existing literature on T-joint weld behaviour under thermal cycling. Comparing our findings with prior studies enables drawing more specific inferences related to how the distributions of stress, deformation, and strain impact structural integrity and fatigue resistance in T-joint welds.

➢ *Analysis of Equivalent (Von-Mises) Stress Distribution*

Figure 2 shows that high values of the stress concentration are developed around HAZ and at the toe of the weld. These are locations where stress-induced failures associated with crack initiation and fatigue fracture are most likely to occur. This agrees with observations reported by [12]., in which residual stresses tend to concentrate around the HAZ because of geometric discontinuity and localised heating and cooling during welding. These sites represent critical crack initiation and possible fatigue fractures under cyclic loading conditions.

In our simulation, the peak of the stress was around 4.9377×10^{8} Pa . which shows a good agreement with the work done by [12]. Came to the conclusion that the thermal gradient and the welding sequence have substantial influences on the residual stresses. The longitudinal stresses can vary according to the welding passes in thicker plates, as derived from Chen's case study. In turn, our findings also show that control of the cooling process and optimisation in weld design can mitigate residual stresses, which would

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun784

benefit fatigue life and structural stability. However, our results show that these HAZ geometric discontinuities constitute a residual source of stress, even for optimum cooling, which is in agreement with the observation by [20] . that there is no way to avoid stress concentration around T-joints but it can be minimised through an appropriate design and selection of the materials.

- *Analysis of Total Deformation*

The deformation result is displayed in Figure 3, where high magnitudes of displacement are seen in the unsupported regions, especially around the free edges of the vertical plate of the T-joint. From this mode of deformation, thermal expansion and contraction due to heating and cooling, respectively, are most effective in those regions of the structure where there is no support; thus, misalignment may be expected in welded assemblies. The optimisation of the support configuration and cooling strategies will contribute toward minimising the permanent distortions, hence improving the dimensional stability of T-joint welds under cyclic thermal loading. Gradual cooling rates can further help control the deformation magnitude to ensure that residual distortions remain within acceptable limits for the application intended. This finding is in good agreement with the remarks by [21] and [28], who pointed out that deformations in T-joints are mostly concentrated in unsupported regions, especially in the neighbourhood of the weld bead. The rather high values of deformations that have been found in the vicinity of the free edges in our studies confirm that lack of support worsens structural misalignment and makes it problematic for applications requiring high accuracy.

Moreover, the study by [25] showed that the sequence of welding is very important in controlling deformation and that starting from the centre of the joint towards the outside reduces the angular distortion. Though our study is not explicitly related to the sequence of the welding passes, deformation patterns indicate that controlled cooling rates and structured support can minimise permanent distortions. Our findings support the suggestions of [30] that multi-pass and controlled cooling approaches can manage deformation; hence, these are the indispensable methods that shall be considered in their improvement of dimensional stability in the T-joint welds.

- *Analysis of Normal Elastic Strain Distribution*

The normal elastic strain distribution displayed in Figure 4 highlights the high elastic stretching around the weld bead and along the HAZ. These regions correspond to a maximum accumulation of elastic strain, and high residual stress correlates with high strain values, which may indicate a possible transition from elastic to plastic deformation under repeated thermal cycling. Such a transformation can be led to strain hardening, which over a period of time will reduce the fatigue resistance of the joint and increase the likelihood of material failure. Therefore, the optimisation of weld geometry and control of thermal gradient during cooling can maintain the strain within the elastic limit and thus reduce the risk of fatigue. Those are in agreement with the study of [32], which reported high elastic strain within HAZ as one main cause of fatigue damage since these regions often jump from purely elastic into plastic deformation when thermal loading is repeated. According to Barsoum, such a transition may result in strain hardening and fatigue failure, also as insinuated in our study. The high elastic strain level in our results indicates the need for optimised thermal gradients in a way that decreases strain accumulation in order to prevent fatigue damage.

On the other hand, [29]showed that certain simplifications for welding simulation, such as using shell elements instead of solid elements, tend to sometimes underestimate strain accumulation in the HAZ. In this investigation, structured meshing was utilised, which is likely to give a better prediction of strain accumulation. This also supports the procedures of [28] to analyse complicated welds for distributions of stress and strain, further justifying our approach to get realistic strain behaviour in the T-joint weld under thermal cycling.

Additionally, the authors, of [30] [24] further emphasised that multi-pass welding configurations present a concern about strain gradients, which are required to be controlled in order not to get an accelerated strain accumulation. Our work upholds this view since strain gradients identified in this investigation enforce the need for gradual cooling and optimised welding sequences as one of the means of maintaining strains within elastic limits and reducing the risk of fatigue failure.

V. CONCLUSION

This paper discusses the mechanical behaviour and structural integrity of T-joint welds under thermal cycling using ANSYS finite element analysis. It also includes stress concentration, deformation, and elastic strain within the joint. The T-joints are a critical component in several structural frameworks, including girders and frames, which are prone to develop complex distributions of stresses and strains because of geometrical configuration and thermal stresses induced during welding and subsequent cooling. The results of the simulation show that a high concentration of stress in the HAZ and its surroundings from the weld bead face makes them susceptible to fatigue damage and crack initiation from repeated thermal loading. From the deformation analysis, it can be noticed that unsupported areas around free edges are most vulnerable to displacement and, hence require controlled support. It also manifests in elastic strain distribution with high accumulation in HAZ, which indicates an elastic-plastic deformation transition and, thus, a possibility of poor fatigue resistance.

The research work proffers enhancing the cooling rate, conceptual supportive configurations, and welding sequences to reduce negative effects on joint durability. The multi-pass welding sequence has demonstrated a reduced level of residual stresses and deformation, especially the medium cooling and alternate passes; these can balance the thermal gradient across the joint. The current research gives hands-on recommendations for improving dimensional stability, fatigue life, and integrity of T-joint welds against residual stresses, deformation, and strain. This simulation-based

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun784

approach forms a basic foundation for further studies and applications, mainly optimisation of the welding process to enhance lifespan and reliability in real applications of T-joints.

REFERENCES

- [1]. K. H. Chang, C. H. Lee, G. C. Jang and H. C. Park, " Behaviour of stresses in T-joint fillet welds under superimposed mechanical loading. ," Int J Steel struct, vol. 7, pp. 311-7, 2007.
- [2]. M. A. Thirugnanam, " 'Analysis of Stress in Welded Joint in Bending And in Torsion Using 'Ansys," Middle-East Journal of Scientific Research, vol. 20, 2014.
- [3]. Y. e. a. Mori, "Fatigue strength and failure mechanism of T-joint fillet welds under cyclic loading," Journal of Constructional Steel Research., 2012.
- [4]. S. M. Azimi, D. Britz, M. Engstler, M. Fritz and F. Mücklich, "Advanced steel microstructural classification by deep learning methods.," Sci Rep , 2018.
- [5]. J. L. Chukwuneke, M. I. Ndefo, J. E. Sinebe and S. M. . Ofochebe, "Numerical and Thermal Analysis of Mechanical Properties Degradations and Distortions in Steel Weld Joint.," Cleaner Materials, vol. 6, p. Cleaner Materials, 2022.
- [6]. J. S. Dias, T. C. Chuvas and M. D. P. C. Fonseca, "Evaluation of residual stresses and mechanical properties of IF steel welded joints by laser and plasma processes ," Materials Research, vol. 19, no. 3, pp. 721-727, 2016.
- [7]. A. Chiocca, F. Frendo and L. Bertini, " Evaluation of heat sources for the simulation of the temperature distribution in gas metal arc welded joints," Metals,, vol. 9, no. 11, p. 1142, 2019.
- [8]. Y. Lu, S. Zhu, Z. Zhao, T. Chen and J. Zeng, " Numerical simulation of residual stresses in aluminum alloy welded joints," Journal of Manufacturing Processes, vol. 50, pp. 380-393, 2020.
- [9]. A. Ayjwat and Z. hatti, " Barsoum., and Hidekazu, Murakawa., and Imad, Barsoum.,"Influence of thermo-mechanical material properties of different steel grades on welding residual stresses and angular distortion," Materials and Design, vol. 65, pp. 878-889, 2015.
- [10]. K. Hemmesi, M. Farajian and M. Boin, " Numerical studies of welding residual stresses in tubular joints and experimental validations by means of x-ray and neutron diffraction analysis," Materials & Design, vol. 126, pp. 339-350, 2017.
- [11]. K. MI, K. ML, E. Biro and Y. Zhou, " Microstructure and mechanical properties of resistance spot welded advanced high strength steels," Materials Transactions, vol. 49, no. 7, pp. 1629-1637, 2008.
- [12]. J. Hu, L. X. Du, J. J. Wang and C. R. Gao, "Effect of welding heat input on microstructures and toughness in simulated CGHAZ of V–N high strength steel," Materials Science and Engineering;A577, pp. 161-168, 2013.
- [13]. A. Grajcar, M. Różański, M. Kamińska and B. Grzegorczyk, " Study on non-metallic inclusions in laser-welded TRIP-aided Nb-microalloyed steel," Archives of Metallurgy and Materials, vol. 59, no. 3, pp. 1163-1169, 2014.
- [14]. A. Chiocca, F. Frendo and L. Bertini, "Evaluation of residual stresses in a pipe-to-plate welded joint by means of uncoupled thermal-structural simulation and experimental tests," International Journal of Mechanical Sciences, vol. 199, p. 106401, 2021.
- [15]. A. Trivedi and P. Chauhan, ". Modeling of Welding Heat Source for Laser Spot Welding Process," In National Conference on Recent Trends in Engineering & Technology, BVM Engineering College, VV Nagar, Gujarat, India., (2011, May).
- [16]. G. Salerno, C. J. Bennett, W. Sun and A. A. Becker, " Residual stress analysis and finite element modelling of repair-welded titanium sheets," Welding in the World, , vol. 61, pp. 1211-1223, 20107.
- [17]. J. Zhu, M. Khurshid and Z. Barsoum, "Accuracy of computational welding mechanics methods for estimation of angular distortion and residual stresses," Welding in the World, vol. 63, pp. 1391-1405, 2019.
- [18]. H. Li and L. Li, "Distributions of temperature and stress fields on penetration assembly during multi-pass welding," IOP Conference Series: Materials Science and Engineering , vol. 10, no. 1, p. 012137, 2010.
- [19]. D. Wang, H. Zhang, B. Gong and C. Deng, " Residual stress effects on fatigue behaviour of welded T-joint: a finite fracture mechanics approach," Materials & Design, vol. 91, pp. 211-217.
- [20]. S. H. Oh, T. Y. Ryu, S. H. Park, M. G. Won, S. J. Kang, K. S. .. Lee and J. B. Choi, "Evaluation of J-groove weld residual stress and crack growth rate of PWSCC in reactor pressure vessel closure head," Journal of Mechanical Science and Technology, vol. 29, pp. 1225-1230, 2015.
- [21]. T. K. S. Aburuga, A. S. Sedmak and Z. J. Radakovic, " Numerical aspects for efficient welding computational mechanics," Thermal Science, vol. 18, no. 1, pp. 139-148, 2014.
- [22]. D. Deng, W. Liang and H. Murakawa, " Determination of welding deformation in fillet-welded joint by means of numerical simulation and comparison with experimental measurements," Journal of Materials Processing Technology, vol. 183, no. 2-3, pp. 219-225, 2007.
- [23]. L. Gannon, Y. Liu, N. Pegg and M. Smith, " Effect of welding sequence on residual stress and distortion in flat-bar stiffened plates," Marine Structures, vol. 23, no. 3, pp. 385-404.
- [24]. R. Lostado, R. F. Martinez, B. J. Mac Donald and P. M. Villanueva, "Combining soft computing techniques and the finite element method to design and optimise complex welded products," Integrated Computer-Aided Engineering, vol. 22, no. 2, pp. 153-170, 2015.
- [25]. R. Konar, M. Patek and M. Sventek, "Numerical Simulation of Residual Stresses and Distortions of T-Joint Welding for Bridge Construction Application,"

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun784

Communications-Scientific Letters of the University of Zilina,, vol. 18, no. 1A, pp. 75-80, 2016.
[26]. G. L. Fu, D. M. M. I. and S. F. Estefen, " Effect of boundary conditions on residual stress and distortion in T-joint welds," Journal of Constructional Steel Research, vol. 102, pp. 121-135, 2014.
[27]. G. L. Fu, D. M. M. I. and S. F. Estefen, " Effect of boundary conditions on residual stress and distortion in T-joint welds," Journal of Constructional Steel Research, vol. 102, pp. 121-135, 2014.
[28]. H. M. E. Ramos, S. M. O. Tavares and P. M. S. T. de Castro, "Numerical modelling of welded T-joint configurations using SYSWELD," Science and Technology of Materials, vol. 30, pp. 6-15, 2018.
[29]. J. Shen and Z. Chen, " Welding simulation of fillet-welded joint using shell elements with section integration," Journal of Materials Processing Technology, vol. 214, no. 11, pp. 2529-2536, 2014.
[30]. M. Perić, Z. Tonković, A. Rodić, M. Surjak, I. Garašić, I. Boras and S. Švaić, "Numerical analysis and experimental investigation of welding residual stresses and distortions in a T-joint fillet weld," Materials & Design, vol. 53, pp. 1052-1063, 2014.
[31]. Y. Rong, G. Zhang and Y. Huang, "Study of welding distortion and residual stress considering nonlinear yield stress curves and multi-constraint equations," Journal of Materials Engineering and Performance, vol. 25, pp. 4484-4494., 2016.
[32]. Z. Barsoum and A. Lundbäck, " Simplified FE welding simulation of fillet welds–3D effects on the formation residual stresses," Engineering Failure Analysis, vol. 16, no. 7, pp. 2281-2289, 2009.
[33]. C. Wang, Y. R. Kim and J. W. Kim, "Comparison of FE models to predict the welding distortion in T-joint gas metal arc welding process," International journal of precision engineering and manufacturing, vol. 15, pp. 1631-1637, 2014.
[34]. A. A. Bhatti, Z. Barsoum, H. Murakawa and I. Barsoum, "Influence of thermo-mechanical material properties of different steel grades on welding residual stresses and angular distortion," Materials & Design (1980-2015), vol. 65, pp. 878-889, 2015.
[35]. M. Perić, Z. Tonković, I. Garašić and T. Vuherer, " An engineering approach for a T-joint fillet welding simulation using simplified material properties," Ocean engineering, vol. 128, pp. 13-21, 2016.
[36]. Y. Li, K. Wang, Y. Jin, M. Xu and H. Lu, " Prediction of welding deformation in stiffened structure by introducing thermo-mechanical interface element," Journal of Materials Processing Technology, vol. 216, pp. 440-446.
[37]. A. R. Safari, M. R. Forouzan and M. Shamanian, "Modeling of fusion zone in FE analysis of the welding process," Journal of Advanced Research in Mechanical Engineering, vol. 2, no. 1.
[38]. M. Perić, Z. Tonkovic, I. Karšaj and D. Stamenković, " A simplified engineering method for a T-joint welding simulation.," Thermal Science, vol. 22, pp. 867-873, 2018.