May – June 2020
ISSN: 0193-4120 Page No. 8062 - 8068

Finite Element Analysis of T-Joint in Arc Welding Process

K.Sravanthi[1] P.Satyakrishna[2], Ch. Sridevi[3],
[1]Associate Professor, Department of Mechanical Engineering,
MLRITM-Marrilaxman reddy institute of technology-Dundigal.
kundhurusravanthi@gmail.com,
[2]Assistant Professor, Department of Mechanical Engineering,
MLRITM-Marrilaxman reddy institute of technology-Dundigal.
satyakrishna.palla@gmail.com,
[3]Assistant Professor, Department of Mechanical Engineering,
MLRITM-Marrilaxman reddy institute of technology-Dundigal
chagantipatisridevi@gmail.com

Article Info
Volume 83
Page Number:8062 - 8068
Publication Issue:
May-June 2020

Article History
Article Received: *19 November 2019*
Revised: *27 January 2020*
Accepted: *24 February 2020*
Publication: *18 May 2020*

Abstract:
Welding process is used in joining similar or dissimilar metals pieces which is either by creating a strong metallurgical bond between them or by heating or application of pressure or both. Automobile manufacturing, construction sites & power plants are some of the areas in which Arc welded structures are commonly used. weldment failure is caused by design defect and overload therefore analyzing maximum stresses in the weldment is essential.

The effort is made to optimize the fillet angle and gap between the parent materials of arc welded T joint with plain carbon steel material. Here finite element analysis (ANSYS19.0) is carried out on welded T-joint under tensile load which determines the von-misses stress induced in the weldment.Distribution of stress is evaluated along the weld size & throat thickness and later compared with analysis results.

Keywords- Arc welded Structures, Finite Element Analysis, T-joint weld, Weldment, metallurgical bond.

I.INTRODUCTION

The American Welding Society [AWS] defines “welding”, as a process in which similar or dissimilar metal pieces are joined by establishing a strong metallurgical bond between which is by either heating or by applying pressure or by both. Welding is one of the earliest machining processes in forming most reliable and strong joints. This process is different when compared with the other form of interlocking mechanical processes such as riveting or bolting formed by friction.

Metals of varied shapes and sizes can be fused with the help of welding. The joints produced with this process are air and water tight hence used in oil storage tanks, ships etc. When compared with the structures made from other form of joining process such as riveting, Welded structures are stronger and ideal resulting in less Stress concentration effect with no constraint on the joints

May – June 2020
ISSN: 0193-4120 Page No. 8062 - 8068

To carry out welding a skilled manpower is required to perform the operation as well as inspection which is one of the disadvantages of it. Apart from this, to detect the defects in welds a effective evaluation is needed through non-destructive inspection. The location and environment is also one of the hurdles in this fusion process. Due to fatigue loading the Welded joints are subjected large residual stresses, distortions resulting in cracking.

Arc welding is a metal joining process used in automobiles and construction applications etc. In bridge construction T- joints and butt joints are mostly used. The failure of T-joint weldment is mainly due to its defect in design & overload; hence it is necessary to analyze the maximum stresses and breaking stress in the T-welded joint. Finite element analysis (ANSYS) used to predict this causes and to gives a applicable accurate weldment.

Two plates of uniform dimensions of (100 mm X 50mm X 5mm) thick are welded with weld size of 4 mm with a gap in between them to facilitate the positional error caused during welding or manufacturing. Later the T joint weldment are subjected to FE analysis

A gap of 0.6 mm and 1.0 mm is maintained between the parent plated for positional error. It is also felt necessary to take into account the effect of chamfer normally provided on vertical plate and hence chamfer angle of 30º, 45º & 60º are considered during the analysis. In this work static analysis, determination of breaking stress of weldment using Ansys is carried out.

This research work aims in finding out of the applicable accuracy fillet angle and gap between parent material, weld size, changing the gap between parent plates to predict the static & fatigue failure for automobile and construction application.

The design model is created by using catia software package as per the given dimensions, analyzed in ANSYS and the problems have been also solved theoretically. The experimental investigation of the same will be carried out in future.

2 MATERIAL SELECTION

Plain carbon steels are taken as to be a weld material. Plain carbon steel is important classes of engineering materials that have been used extensively in structural and automobile engineering applications. These are the one class of carbon steels. The iron alloys in which carbon content is less than 1%,manganese not even exceeding 1.65% and less than 0.6% of copper and silicon the plain Carbon steels with exceptional weld ability having 0.15 to 0.20% carbon.

They seldom require anything beyond standard welding procedures, and these can be welded with all types of mild steel electrode. Copper is alloy taken as a filler material. Copper has a cubic crystal structure with face centered. They have good formability and malleability. They have welding factors of high electrical and thermal conductivities and certain elevated copper alloys with noticeable effect of weld ability.

Due to high thermal conductivities possessed by Copper results in conducting welding heat quickly into the base resulting in short of fusion in weldment. To ensure good fusion it is important to preheat the alloy which in turn reduces the weld heat. For this reason plain carbon steel and copper is selected as base and filler material.

May – June 2020
ISSN: 0193-4120 Page No. 8062 - 8068

3. NUMERICAL CALCULATION

Weldment under Tensile Load

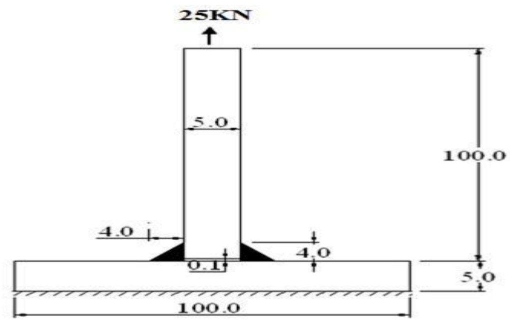

Fig. 1 Arc weld of T-joint subjected to tensile load

The above fig 6.1 represents the T-joint fillet weld sections with dimensions (100x50x5mm) with fillet angle 45º. Here 25 KN tensile load acting upward on the top of the section. The base weld material is plain carbon steel and filler material is copper (Cu).

T-joint fillet weld dimensions are specified are as follows.

F -Tensile load on vertical plate (N) w -Leg length of weld (mm) h -Throat of fillet weld (mm)	= 25KN = 4.0 mm = w X [cos45º]
l -Length of weld or size of weld (mm)	= 50mm
b- breath of the weld(mm)	= 50mm
l_T- length of top load section	= 5mm

The material properties are specified are as follows,

Modulus of elasticity of parent plate material (E)	= 2.1 x 10^5 MPa
Poisson's ratio of parent plate material (μ) Modulus of elasticity of weld material (E)	= 0.3 = 1.1 x 10^5 MPa
Poisson's ratio of weld material (μ)	= 0.37

Breaking stress = (Breaking Load /Throat Area) X Stress Concentration Factor
Stress concentration factor (k) parent material given by Karl Heinz Frank is the range of 3.5 to 4.

4 NUMERICAL RESULTS

The values of Breaking Stress from Numerical Calculation are tabulated as follows:

Considering the gap variation of 0.6 & 1.0 mm between the parent plates at throat thickness of T-joint weldment and providing the chamfer a angle of 30º, 45º & 60º to the vertical plat e, the Maximum breaking stress are calculated and tabulated as follows in Table 7.1

May – June 2020
ISSN: 0193-4120 Page No. 8062 - 8068

Gap between parent plates (mm)	Breaking stress for 25KN		
	30º	45º	60º
0.6	130.34	188.50	223.41
1.0	130.60	189.90	224.60

Table4.1 Maximum breaking stress results with varied gap and diverse chamfer angle are calculated and Numerical results are tabulated.

4.1FINITE ELEMENT ANALYSIS

The strength and behavior of engineering structures are usually analyzed and calculated by using one of the popular numerical computer-based techniques called Finite Element Analysis (FEA). This method is also employed in calculating deflections of large or medium scales when the objects are subjected to loading or displacement, stress, vibration, buckling behavior etc. The numerical technique employed in the element analysis is FEM, Finite Element Method, in which the finite elements are used to represent the actual continuum.

To carry out the work in this project, a finite element analysis is performed using FEA software ANSYS, involving various unknowns such as displacements in structural analysis where other quantities are derivative from the nodal displacements such as strains, stresses, and other reaction forces.

4.2 Design Model

Below Fig8.1 represents the design model of T-joint welded section with (100×50×5mm) and the fillet angle 30 degree. This design model element is created by using CATIA software

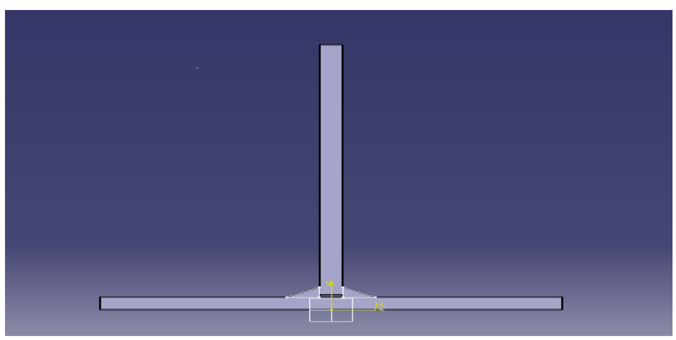

Fig.1 T-joint of design model

4.3 Element Meshing

Meshing is the discretization of the element into number of small components. It used to increase accuracy result of the element. For this analysis course mesh type is used. This below Fig.8.2 represents the meshed element of T- joint weld section by using ANSYS 19,0 workbench software.

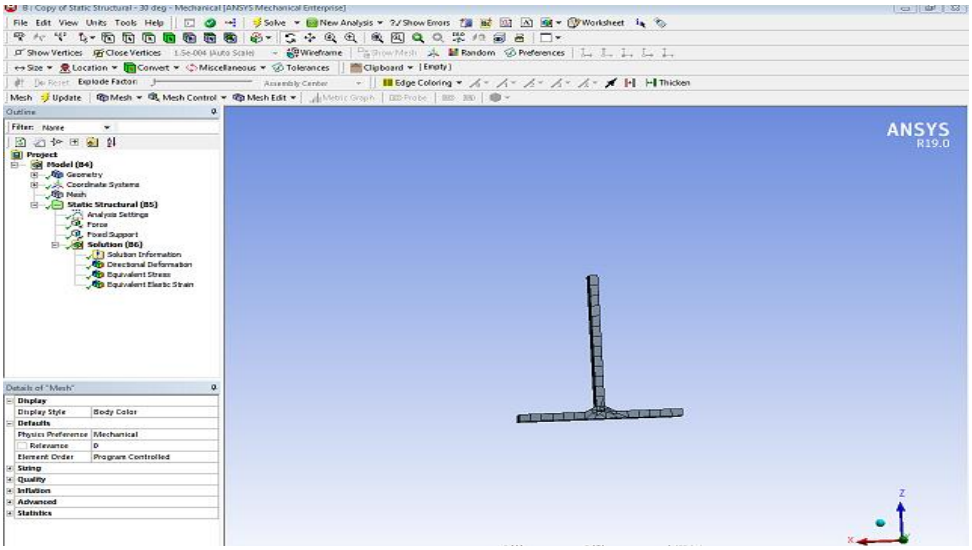

Fig.2 Meshed Element

4.4 Boundary conditions

The below Fig8.3 represents the tensile load 25KN applied on top of plate and bottom surface of the plate fixed by using ANSYS workbench software.

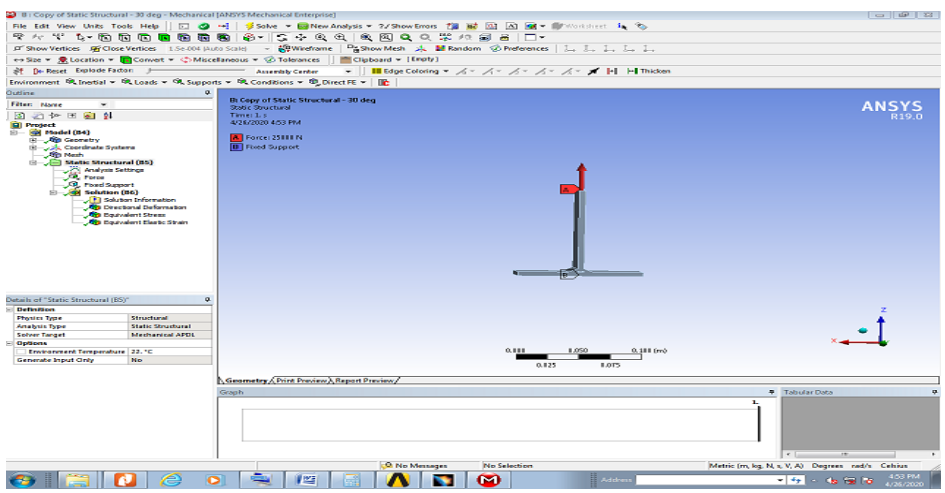

Fig.3 Applied tensile orce on top of the plate**Equivalent (Von-Misses) Stress**

Most of the designers make use of Von misses stress, as this helps in knowing if the design is able to resist the load in given condition. When the von misses stress induced is more than strength of the material, it is measured as the factor of safety by design engineers. This theory is mainly applicable to ductile materials.

Considering 25KN load, θ = 45º, 30 º, 60 º and 1 mm gap

May – June 2020
ISSN: 0193-4120 Page No. 8062 - 8068

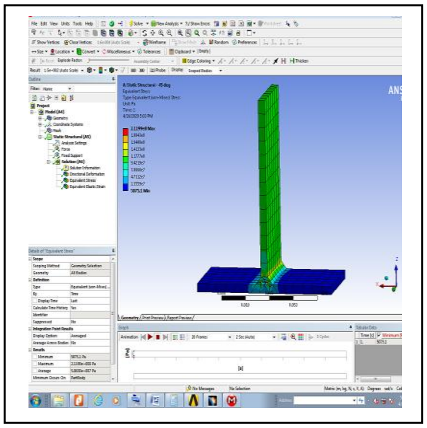
Fig .4

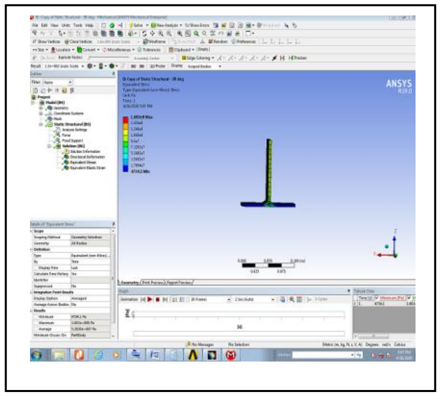
Fig .5

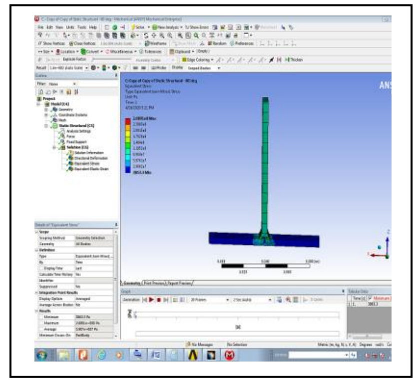
Fig 6

The above Figures. represents the equivalent von misses stress value 211.99Mpa for 45º, 160.20Mpa for 30º, and 268.91Mpa for 60º where parent materials are given a gap of 1mm.

With 25KN load, θ = 45º, 30 º, 60 º and 0.6mm gap

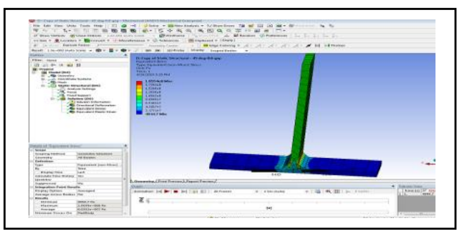
Fig .7

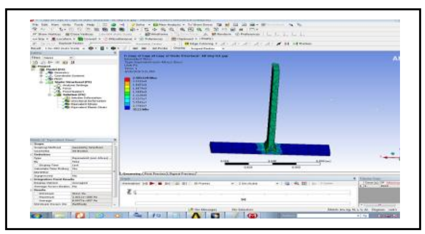
Fig. 8

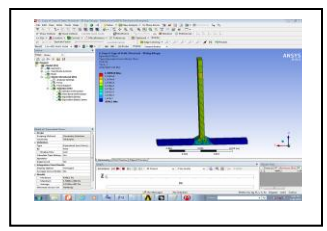
Fig. 9

The above figures represents the equivalent von misses stress value 195.54 for 45º, 150.09 Mpa for 30º and 250.11 Mpa for 60º with 0.6 mm gap between the parent material.

4.5 ANALYSIS RESULT

Vertical plate consisting of chamfer is taken onto account in performing FE analysis at 30º, 45º & 60º. The below table interprets the numerical values of Maximum breaking stress considering a varying space of 0.6 and 1.0 mm in relation with the chamfer angle on the plate resulting in Maximum von-misses stresses there in T-joint weldment.

Gap between parent plates(mm)	Breaking stress for 25KN (Mpa)		
	3	45º	60º
0.6	1	195.54	250.11
1	1 6 0	211.99	268.91

Table.5.1 Numerical values exhibited for Von misses stress with varying plate gap and chamfer angle.

4.6 EVALUATION

From the numerical calculations and finite element results we found the stress values matches around 85% approximately. When the result obtained by FEA analysis is compared with the numerical value through FE analysis of the T-welded joint of equivalent geometry, delivered approximately the same result of maximum Von-misses stress of 268.91 Mpa and that through FEA is 224.60 Mpa.

5. CONCLUSIONS

To determine the deformation breaking stress of weld T- joint, finite element analysis is employed. When the weldment with a gap between the plates was subjected to a tensile load during Static stress analysis resulted in maximum Von-misses, leading to a fruitful completion of design and analysis of welded T-joint. The theoretical results are compared with analysis (ANSYS work bench 19.0) result and both are acceptable.

Maximum stress values were observed when the weldment is exposed to axial tensile load with a gap in between the parent plates.

The FE analysis of T-weldedjoint for the same geometry revealed the maximum Von-misses stress of 268.91Mpa for 60 º degrees with a gap of 1mm in plates, which is almost equal to the numerical maximum breaking strength of 224.60Mpa.

REFERENCES

1. Dean Deng, Wei Liang, Hidekazu Murakawa(2007) "Determination of Welding deformation in fillet-welded joint by means of numerical simulation and Comparison with experimental measurements". Journal of Materials Processing Technolog.183:219-225.
2. Tso-Liang Teng, Chin-Ping Fung(2001) "Analysis of residual stresses and distortions in T-joint fillet welds" International journal of Pressure Vessels and Piping,78:523-538
3. Mohd Shahar Sulaiman, Chan Yin Chau(2011) "Simulation and experimental study on distortion of butt and T-joints using WELD PLANNER, Journal of Mechanical Science and Technology ,25:2641-2646.
4. M.N.Buradkar, Dr. D.V. Bhope, S. D. Khamankar(2013), "Experimental & photo elastic analysis of arc welded lap-joint" Kotiya, et al, International Journal of Advanced Engineering Research and Studies,2247:112-115.
5. Kim JR Rasmussen(2002), "Strength of Butt Welded Connections between Equal-width Rectangular Hollow Sections" Research Report No R817.
6. Shahram Sarkani, George Michaelov(2000) "An efficient approach for computing residual stresses in welded joints" Finite Elements in Analysis and Design. School of Engineering and Applied Science, The George Washington University, Washington, DC 20052, USA, 35:247-268.