GE-International Journal of Engineering Research
Vol. 3, Issue 11, Nov 2015 IF- 4.007 ISSN: (2321-1717)
© Associated Asia Research Foundation (AARF) Publication
Website: www.aarf.asia Email : editor@aarf.asia , editoraarf@gmail.com

THEORITICAL AND FINITE ELEMENT ANALYSIS OF T-JOINT IN ARC WELDING PROCESS

Arunkumar.A[1], Vijayashankar.R[2],

[1]PG scholar, Department of Mechanical engineering,

KIT-Kalaignarkarunanidhi Institute Of Technology, Coimbatore.

[2]Assistant Professor, Department of Mechanical engineering,

KIT-Kalaignarkarunanidhi Institute Of Technology, Coimbatore.

ABSTRACT

Welding is a process used to join two or more pieces of similar or dissimilar metals by creating a strong metallurgical bond between them by heating or pressure or both. Arc welded structures are widely used in automobiles, constructions & power plants. As the main cause of weldment failure is design defect and overload, hence it is necessary to analyze the maximum stresses in the weldment. This work deals with optimizing the fillet angle and gap between the parent material of arc welded T joint with plain carbon steel and copper(Cu) material. Here finite element analysis (ANSYS14.5) which is carried out on welded T-joint under tensile load to determine the von-misses stress in the weldment. The stress distribution along weld size & throat thickness is evaluated and also compared with the results from the analysis. Further T-joint weld is analyzed under static tensile load condition by varying the gap between parent plates with chamfer at weldment edges.

KEYWORDS-Arc Welded Structures, Finite Element Analysis, T-joint weld, Weldment, metallurgical bond

1. INTRODUCTION

The American Welding Society [AWS] defines welding process as “Welding is a mechanical machining process used to join the two or more metal pieces, which may be similar or dissimilar creating a strong metallurgical bond between them by heating or pressure or both”. It is distinguished from other forms of mechanical connections, such as riveting or bolting, which are formed by friction or mechanical interlocking. It is one of the oldest and reliable methods of joining.

Welding offers air tight and water tight joints thereby it is ideal for oil storage tanks, ships etc. Welding could be employed for joining metals of different shapes and sizes. Welded structures are stronger compared to structures made of riveted joints and other permanent joints. A truly continuous structure is formed by the process of fusing the members together. Generally welded joints are as strong , thereby placing no restriction on the joints. Stress concentration effect is also considerably less in a welded connection

Some of the disadvantages of welding are that it requires skilled manpower for welding as well as inspection. Also, non-destructive evaluation may have to be carried out to detect defects in welds. Welding in the field may be difficult due to the location or environment. Welded joints are highly prone to cracking under fatigue loading large residual stresses and distortions are developed in welded connections.

2. PROBLEM IDENTIFICATION

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. As the main cause of weldment failure of T-joint is 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.

3. SPECIFICATION OF THE PROBLEM

For the FE analysis on the weldment of T joint two plates of uniform dimensions of (100 mm X 50mm X 5mm) thick are welded with weld size of 4 mm\with the gap between them to take into account the positional error which occurs during welding or manufacturing. The analysis is considering tensile load. Where the weld material is plain carbon steel and filler material taken as a copper(Cu). The gap between parent plates is varied from 0.2 mm to 1.0 mm in the step of 0.2 mm to take into account the 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 is carried out.

4. OBJECTIVE

Objective of this research work is to find 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 SOLID WORKS 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.

5. 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 widely in structural and automobile applications. These are the one class of carbon steels. Carbon steels are alloys of iron and carbon in which carbon does not exceed 1%, manganese does not exceed 1.65%, and copper and silicon each do not exceed 0.6%. Plain carbon steels with 0.15 to 0.20% carbon have excellent weldability.

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 face centered cubic crystal structure. They have good formability and malleability. They have welding factors of high electrical and thermal conductivities and certain high copper alloys have marked effect of weldability.

Copper have high thermal conductivities, so welding heat is quickly conducted into base. This may cause lack of fusion in weldment. Preheating of these alloys will reduce welding heat requirements for good fusion. For this reason plain carbon steel and copper is selected as base and filler material.

6. NUMERICAL CALCULATION

6.1 Weldment under Tensile Load

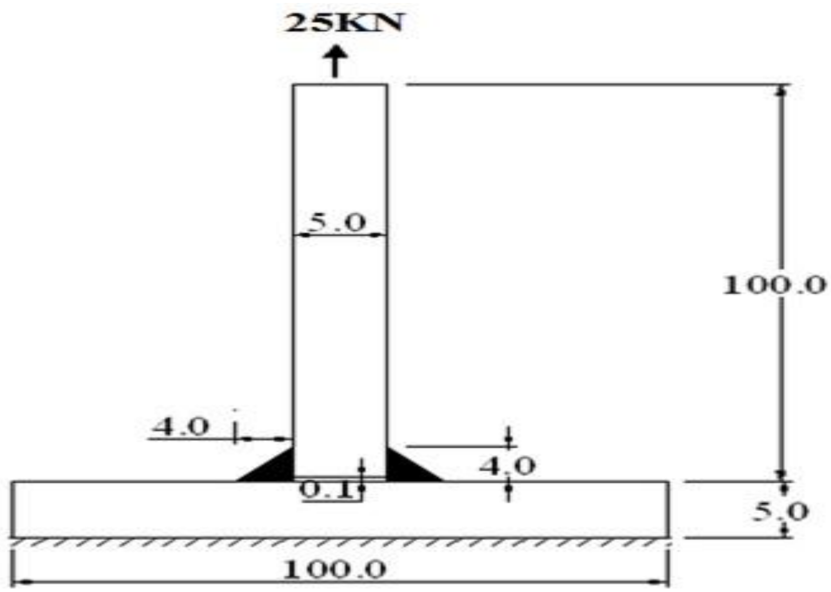

Fig. 6.1 Arc weld of T-joint under tensile load

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) = 25KN
- w -Leg length of weld (mm) = 4.0 mm
- h -Throat of fillet weld (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
- b_T-breath of top load section = 50mm
- l_t -throat length = 4mm
- t - throat thickness = 1mm
- A -Area of weld section

The material properties are specified are as follows,

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

Formula used,

Area of weld section(A) = $2\,A_f + A_l - A_t$

Where,

A_f – Area of fillet section = $2\times(\cos\theta\times w)\ x\ l$

A_l – Area of load section = $l_T \times b_T$

A_t – Throat area = $t \times l_t$

Therefore,

Breaking stress = (Breaking Load /Throat Area) X Stress Concentration Factor

$$= \frac{P}{A_t} \times k$$

Stress concentration factor (k) parent material given by Karl Heinz Frank is the range of 3.5 to 4.

7. NUMERICAL RESULTS

7.1 Tabulated Breaking Stress Values from Numerical Calculation

The Maximum breaking stresses present in T-joint weldment at the throat thickness with gap variation and the variation of Maximum breaking stress with respect to gap is also shown in Table. 7.1. Where 30º, 45º & 60º chamfer is provided on the vertical plate by varying the gap of 0.2 to 1.0 mm in the steps of 0.1 mm.

Gap between parent plates (mm)	Breaking stress for 25KN(Mpa)		
	30°	45°	60°
0.2	130.07	187.96	222.62
0.4	130.20	188.25	223.01
0.6	130.34	188.50	223.41
0.8	130.48	188.68	223.81
1.0	130.60	189.90	224.60

Table7.1 Numerical results for breaking stress with different angle and gap between parent materials

8. FINITE ELEMENT ANALYSIS

Finite Element Analysis (FEA) is a computer-based numerical technique for calculating the strength and behaviour of engineering structures. It can be used to calculate deflection, stress, vibration, buckling behaviour and many other phenomena. It also can be used to analyse either small or large-scale deflection under loading or applied displacement. It uses a numerical technique called the finite element method (FEM). In finite element method, the actual continuum is represented by the finite elements

In this project finite element analysis is carried out using the FEA software ANSYS. The primary unknowns in this structural analysis are displacements and other quantities, such as strains, stresses, and reaction forces, are then derived from the nodal displacements.

8.1 Design Model

1. Boundary Conditions

The finite element model of welded T-joint is shown in Fig.8.1 Bottom end is fixed and 25 KN load is applied at top end the plate.

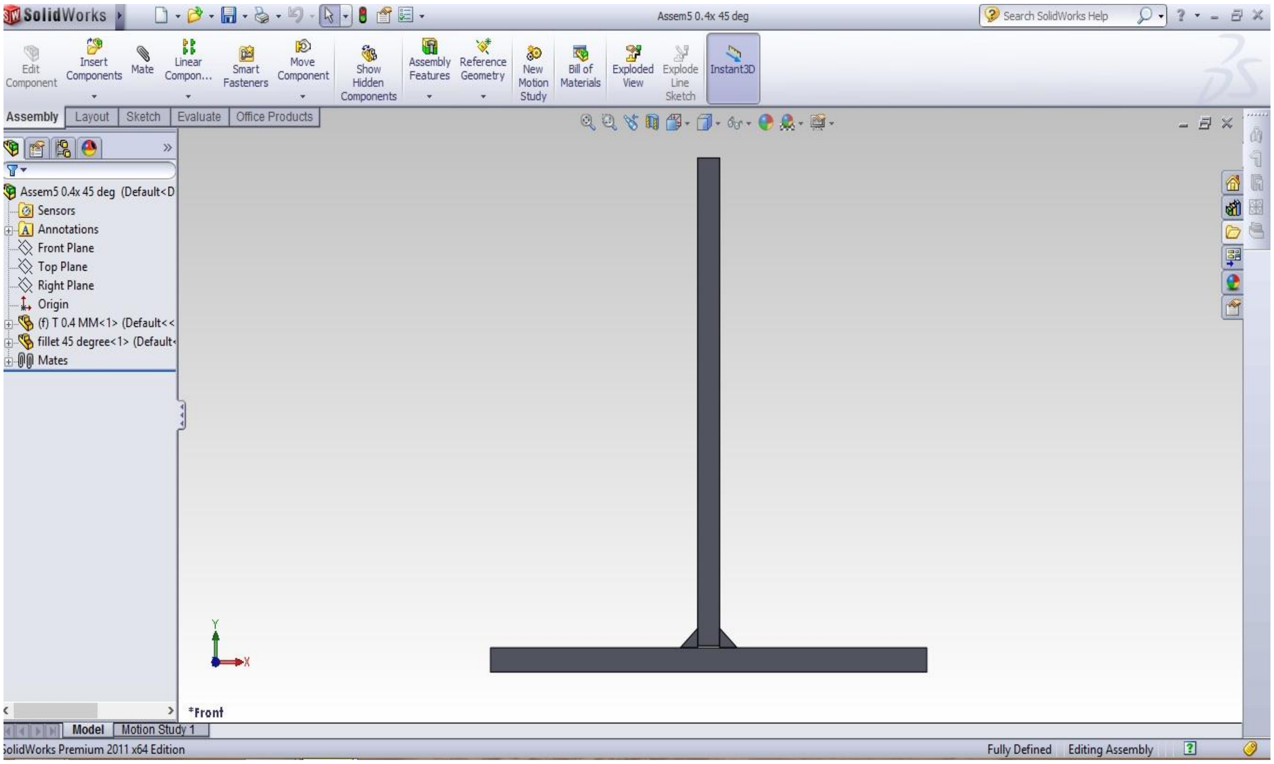

Fig.8.1 T-joint of design model

Above this Fig8.1 represents the design model of T-joint welded section with (100×50×5mm) and the fillet angle 45 degree. This design model element is created by using SOLID WORKS software

2 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 14.5 workbench software.

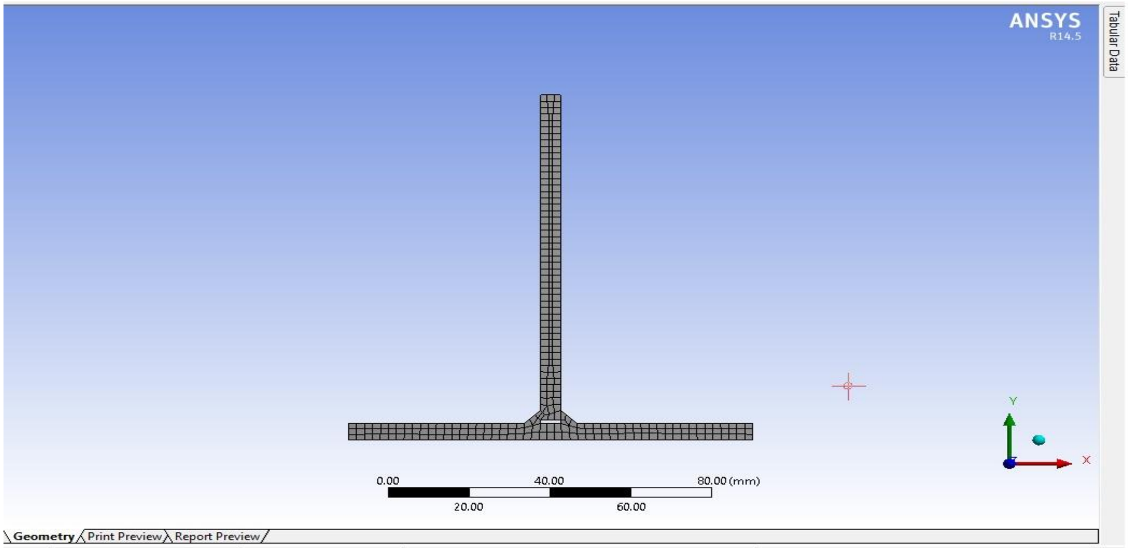

Fig.8.2 Meshed Element

3 Remote Force

This 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.

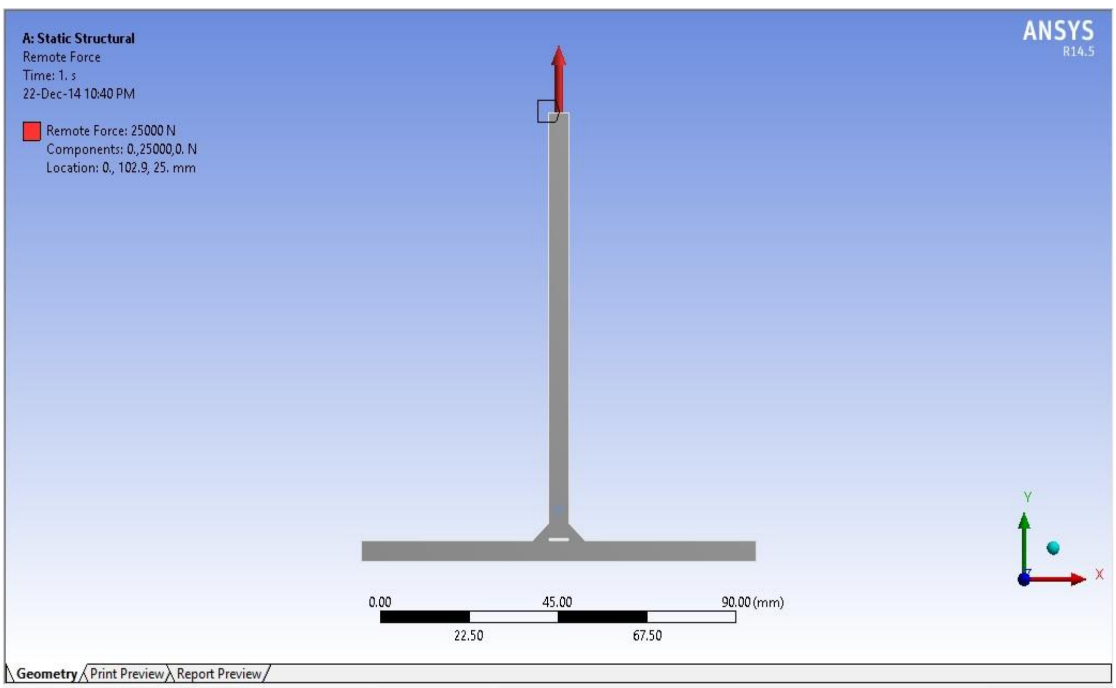

Fig.8.3 Applied tensile force on top of the plate

8.2 Equivalent (Von-Misses) Stress

Von misses stress is widely used by designers to check whether their design will withstand a given load condition.

Von misses stress is considered to be a safe haven for design engineers, if the maximum value of von misses stress induced in the material is more than strength of the material. Its work well for most cases, especially when the material is ductile in nature.

1. For 25KN load, θ = 45º and 1 mm gap

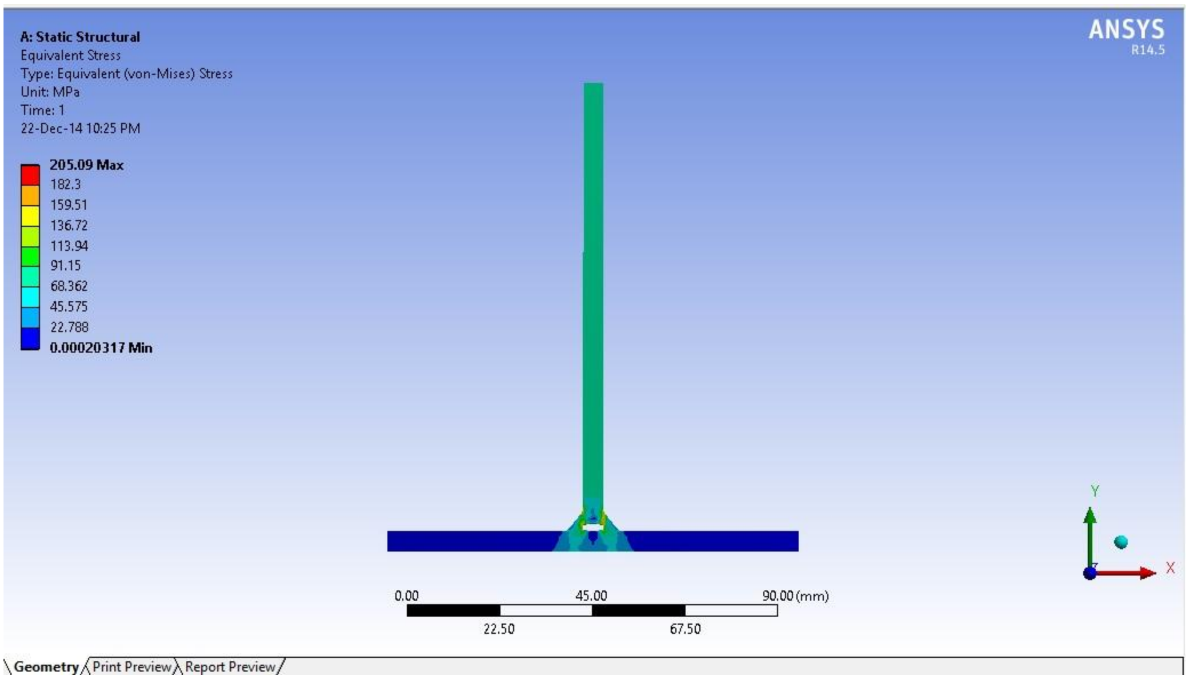

Fig8.4 Equivalent von-misses stress for 45º degree and 1mm gap

The above Fig.8.4 represents the equivalent von misses stress value is 205.09Mpa for 45º and 1mm gap between the parent materials.

2. 25KN load, θ = 60º and 1mm gap

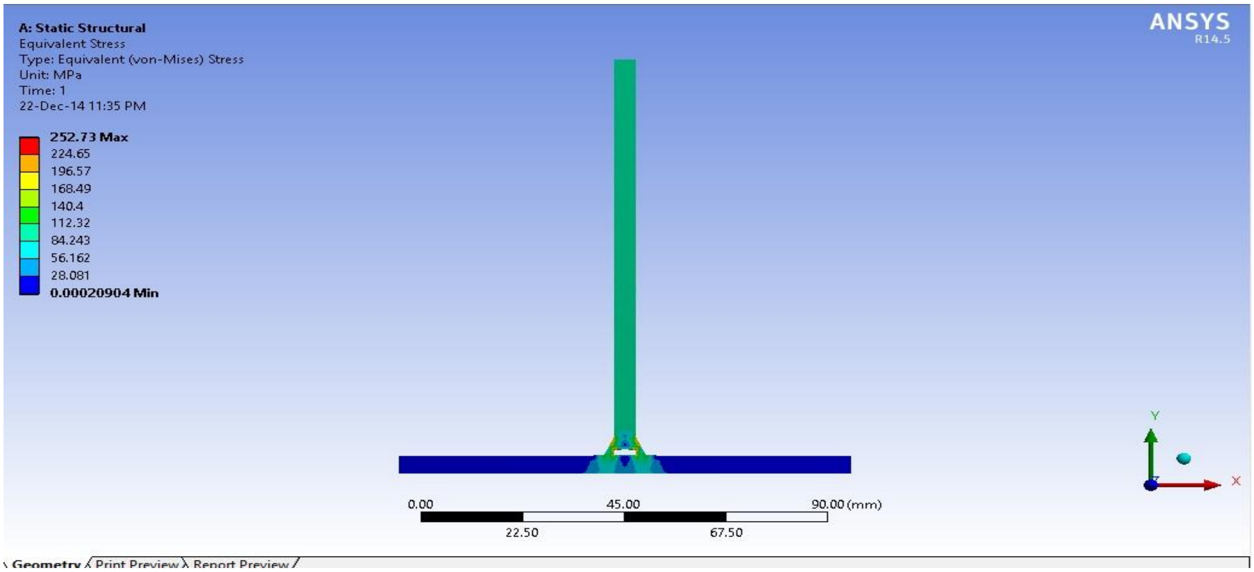

Fig8.5 Equivalent von-misses stress for 60º degree and 1mm gap

3. 25KN load, θ = 30º and 1mm gap

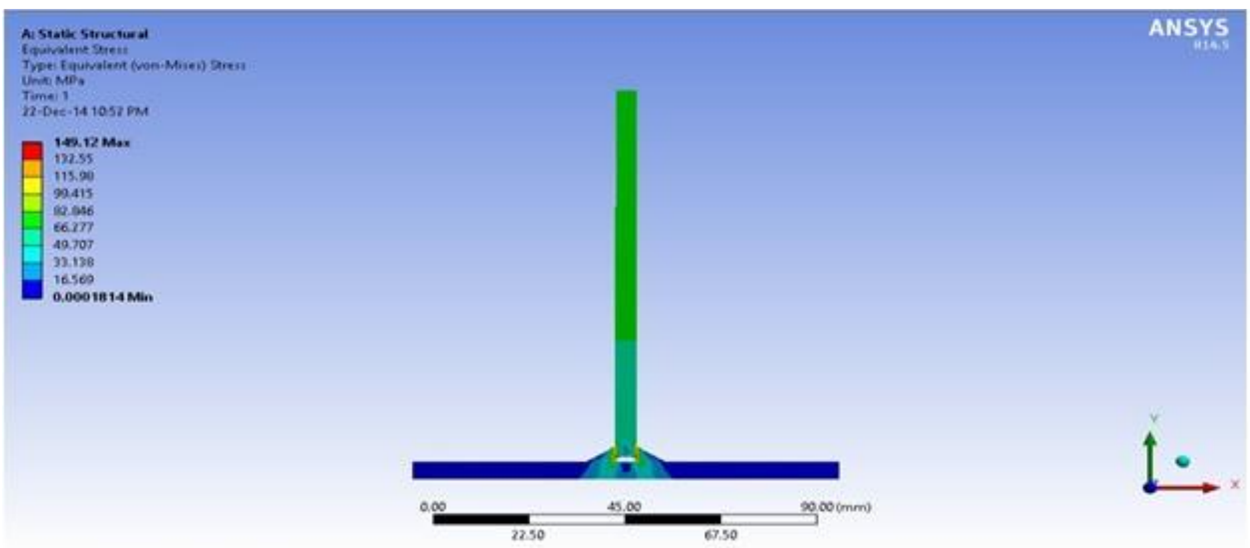

Fig8.6 Equivalent von-misses stress for 30º and 1mm gap

The above two figures 8.5 and 8.6 represents the equivalent von misses stress value is 252.7 Mpa for 60º and 149 Mpa for 30º with 1mm gap between the parent material.

9. ANALYSIS RESULT

FE analysis is also carried out by considering the chamfer on vertical plate. The Maximum von-misses stresses present in T-joint weldment at the throat thickness with gap variation and the variation of Maximum breaking stress with respect to gap is also shown in Table 9.1. Where 30º, 45º & 60º chamfer is provided on the vertical plate by varying the gap of 0.2 to 1.0 mm in the steps of 0.2 mm.

Gap between parent plats(mm)	**Breaking stress for 25KN (Mpa)**		
	30º	**45º**	**60º**
0.2	137.23	192.04	236.62
0.4	140.2	194.25	242.01
0.6	142.89	197.50	245.41
0.8	144.74	199.68	247.81
1	149.12	205.06	252.76

Table.9.1 Analysis results for Von misses stress with different angle and gap

10. EVALUATION

From the analytical calculations and finite element results the equivalent von misses stress 75% approximately equal. The FE analysis of T-welded joint for the same geometry revealed the maximum Von-misses stress of 252.76 Mpa is approximately equal to numerical maximum breaking strength of 224.2Mpa.

11. CONCLUSION

The finite element analysis is used in this work to evaluate the deformation breaking stress of weld T- joint. Static stress analysis performed on the weldment under tensile load revealed the maximum Von-misses stress with respect to the gap between parent plates. The design and analysis of welded T-joint has been done successfully. The theoretical results are compared with analysis (ANSYS work bench 14.5) result and both are acceptable.

Static stress analysis performed on the weldment under axial tensile load revealed the maximum Von-misses stress w. r. t. gap between parent plates. The FE analysis of T-welded joint for the same geometry revealed the maximum Von-misses stress of 252.76 Mpa for 60º degree and 1mm gap, it's approximately equal to numerical maximum breaking strength of 224.2 Mpa.

The gap between parent plates increased from 0.2 mm to 1.0 mm, the maximum Von-misses stresses decreases from 252.34MPa to 236.22MPa.

12. SCOPE AND FUTURE WORK

From the theoritical and finite element analysis results are approximately equal. The design and analysis of welded T-joint has been done successfully for T-joint fillet weld under tensile load with angle and gap between parent materials.

In future same analysis is carried out for experimental investigation will do for later with plain carbon steel and copper under tensile conditions.

13. 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.

[7] A.chottapathay, G.Glinka, M.EI-Zein, J.Qian and R.Forams(2011), "stress analysis and fatigue of welded structures" journal of welding in the world.17:234-237.

[8] N.B. Mostafa, M.N. Khajavi(2006), "Optimisation of welding parameters for weld penetration in FCAW" journal of Achievements in Materials and Manufacturing Engineering. 16:132-137.

[9] A. Unt, E. Lappalainen, A. Salminen(2013), "Autogeneous laser and hybrid laser arc welding of T-joint low alloy steel with fiber laser systems" Lasers in Manufacturing Conference, 41:141-143.

[10] M. Islam, A. Buijk(2014) "Simulation-based numerical optimization of arc welding process for reduced distortion in welded structures. journal homepage: www.elsevier.com/locate/finel,84:54-64.

[11] Jichao SHEN, Zhen CHEN(2014), "Welding Simulation of Fillet-welded Joint Using Shell Elements with Section Integration" Journal of Materials Processing Technology, 34:01-47.

[12] Hyeong soon moon, suck joo- na(1997), "Optimum design based on mathematical model and neural network to predict weld parameter for fillet joints "journal of manufacturing system,16:13-22.

[13] M. E. Scholar(2013), "A Review on Parametric optimization of MIG Welding for Medium Carbon Steel using FEA-DOE Hybrid Modeling" IJSRD - International Journal for Scientific Research & Developmen,30:1843-1846.

[14] Matt I. Dawson(2008) "Modeling and Analysis of Welding Processes in Abaqus using the Virtual Fabrication Technology (VFT) Analysis Software developed by Battelle and Caterpillar Inc" .Abaqus Users' Conference.20:1-14.

[15] Klaus-Dieter Schoenborn(2012)"Study of Fatigue Analysis of a Welded Assembly Using ANSYS Workbench Environment". ANSYS Service @ CADFEM GmbH, Germany, 21:31-44.