iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 7 **Issue:** V **Month of publication:** May 2019

DOI: https://doi.org/10.22214/ijraset.2019.5413

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue V, May 2019- Available at www.ijraset.com

Influence of Fillet Gap and Weld Angle on T-Joint

Anand S[1], Blessen G Mammen[2], Braida Sara Biju[3], Rahul Raj R[4], S. Sarath[5]
[1, 2, 3, 4, 5] *Student, B-Tech Mechanical Engineering,St Thomas College of Engg. And Technology, Chengannur*

Abstract: Welding plays a major role in the fabrication and erection of structures and components of machines. Arc welding is one of the most important manufacturing processes such as joining of structural elements for a wide range of applications including guide way for trains, ships, bridges, building structures, automobiles etc. It is necessary to ensure that the quality of the welded joints, because failure in welded joints lead to various direct and indirect losses such as increase in maintenance cost, work cost, increase in the possibility of accidents, increase in production cost, decrease in production and it also affects the company image etc. So it is necessary to understand the failure of the joints and it could be minimized at design stage, through which we can reduce the possibility of failure. Stress concentration in the welded joints to be analyzed during the formation of weld in arc welding and influence of metal gap between weld is looked upon. Design of welded joint can be done through numerical analysis with different design parameters. The same analysis is done by Finite Element Method [FEA] through SOLIDWORKS and results are compared. An optimum design parameter can also be predicted.

Keywords: fillet weld, von misses stress, FEM, joint gap, SOLIDWORKS

I. INTRODUCTION

Material joining is the important process used for assemble metallic and non-metallic parts. There are number of alternative welding technologies that enable the more use of lightweight and high performance material. The high temperature differences in the welding process results more temperature strains which causes the contact pressure distribution between the structural items. Welding plays a major role in fabrication of structures and machine components.

Welded joint design can be done through numerical analysis. This numerical analysis can be done through Finite Element Analysis (FEM) and the results are compared. In this analysis the optimum design also can be predicted.

In engineering application T-Joint welding is most commonly used. It is used in Transport vehicles, Marine ships, mobile plant devices. Welded structure analyses are challenge for the designer to get the desired output. It is important to ensure the quality of welded joints, because the failure causes direct and losses like increase in maintenance cost, work cost, increase the possibility of accidents, increase the production cost, decrease the production and it also affect the companies face. So it is needed to understand failure at the design stage and it could be minimized at these stages by which we can reduce the possibility of failure. The aim of the study is to carry out FEA and Numerical evaluation method to determine the welded joints strength and its influence on various design conditions. Finite element analysis (FEA) is a computer based method for analysing the behaviour of engineering structures and components under varying conditions. The analysis of welding is carried out with FEA software to validate simulation method and compare it with numerical evaluation so as to determine the dynamic property of structures under tensile load on non-complete joint fillet welding. Dean Deng et.al [1] performed experiments to investigate the characteristics of welding deformation in the fillet-welded joint. In order to precisely predict welding deformation by numerical method, a 3-D thermal elastic plastic finite element computational procedure is developed. The simulated results are in a good agreement with the experimental measurements. The influence on welding deformation of the flange thickness is investigated by experiment and numerical simulation. In addition, the generation mechanism of angular distortion is clarified through numerical simulation. Chottapathy A et.al [2] proposed a method that enables the determination of the stress concentration and the stress distribution in the weld toe region using a special shell finite element modelling technique. The procedure consists of a set of rules concerning the development of the finite element mesh necessary to capture the bending and membrane structural stresses. The structural stress data obtained from the shell finite element analysis and relevant stress concentration factors are subsequently used to determine the peak stress and the non-linear through-thickness stress distributions. The peak stress at the weld toe is subsequently used for the determination of fatigue crack initiation life. The stress distribution and the weight function method are used for the determination of stress intensity factors and for the analysis of subsequent fatigue crack growth. Tso-Liang Teng et.al [3] had carried out study that describes the thermal elasto-plastic analysis using finite element techniques to analyse the thermo mechanical behaviour and evaluate the residual stresses and angular distortions of the T-joint in fillet welds. Furthermore, this work employs the technique of element birth and death to simulate the weld filler variation with time in T-joint fillet welds. Also discussed are the effects of flange thickness, welding penetration depth, and restraint condition of welding on the residual stresses and distortions

Welding plays a major role in the fabrication and erection of structures and components of machines. It is necessary to ensure that the quality of the welded joints, because failure in welded joints lead to various direct and indirect losses So it is necessary to understand the failure of the joints and it could be minimized at design stage, through which we can reduce the possibility of failure. Main objective of this project is to identify and optimize best welding technique based upon metal gap and fillet angle Comparison of theoretical and finite element method is established to define better results.

II. NUMERICAL ANALYSIS

Numerical analysis is carried out to find out the plane of maximum shear stress which is acting on the load at right angles to the weld and followed by the analysis of weldment under tensile load is done on the same T-weld joint.

A. *Plane Of Maximum Shear Stress - Load At Right Angles To The Weld*

Let,

Leg size = r

Length of weld = h

Let α = angle of plane of maximum shear stress

d = throat thickness

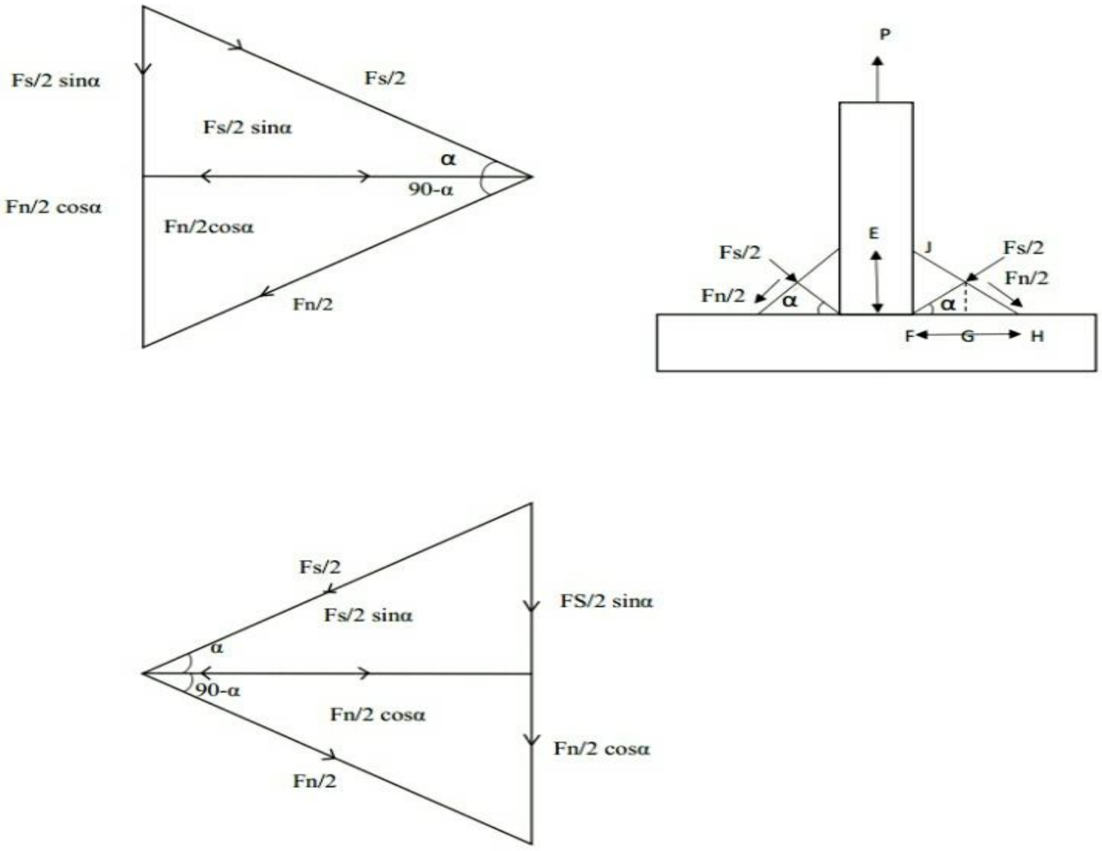


Fig. 1 Resolved figure showing horizontal and vertical components of force acting on the T- joint weld

$F = \frac{FS}{2}\sin\propto + \frac{FS}{2}\cos\propto + \frac{Fn}{2}\sin\propto + \frac{Fn}{2}\cos\propto$

$= Fs\sin\propto + Fn\cos\propto$…….(4.1)

Assuming that the resultant of $\frac{Fs}{2}$ and $\frac{Fn}{2}$ is vertical, then the horizontal components are equal and opposite.

Horizontal component of $\frac{Fs}{2} = \frac{Fs}{2}\cos\propto$

$$\frac{Fn}{2} = \frac{Fn}{2} sin \propto$$

$$\frac{Fs}{2} cos \propto = \frac{Fn}{2} sin \propto$$

Substituting the value of Fn in equation 1 we have

$$F = Fs\ sin\propto + \frac{Fscos\propto \times cos\propto}{sin\propto}$$

Multiplying throughout by sin$\propto$, we have

$$Fsin\propto = sin^2 \propto + Fs\ cos^2 \propto$$

$$= Fs\ (sin^2 \propto + cos^2 \propto) = Fs……..(4.2)$$

From the geometry,

FH = FG+GH

$$R = d\ cos\propto + dsin \propto = d(cos \propto + sin \propto)$$

There fore throat thickness, $d = \frac{r}{cos\propto + sin\propto}$

$A = 2d \times h$

$$= 2 \times \frac{r}{cos\propto + sin\propto} \times h = \frac{2r \times h}{cos\propto + sin\propto}$$

Max shear stress, $\tau = \frac{Fs}{A} = \frac{Fsin\propto(cos\propto + sin\propto)}{2 \times r \times h}$………………(4.3)

For max shear stress, differentiate the above expressions with respect to $\propto$ and equate to zero

$$\frac{d\tau}{d\alpha} = \frac{F}{2 \times r \times h}[sin \propto (-sin \propto + cos \propto) + (cos \propto + sin \propto)cos \propto] = 0$$

$$cos^2 \propto - sin^2 \propto + 2sin \propto cos \propto = 0$$

$2sin\propto cos \propto = sin2 \propto$

Therefore $cos2\propto + sin2 \propto = 0$

$Sin2\propto = -cos2 \propto$

$$\frac{sin2 \propto}{cos2 \propto} = -1$$

$$2 \propto = 135^{o}$$

Or $tan2\propto = -1$

$\propto = 67.5^{o}$

Eventhough plane of maximum stress angle is identified, the significant area of stress concentration is the priority to understand the problem.

B. Weldment Under Tensile Load

The below Fig.2 represents the T-joint fillet weld sections with dimensions (100mmx50mmx5mm) with fillet angle 30º and gap of 0.3mm. Here 25kN tensile acting upward on the top of the section.

F – Tensile load on vertical plate (N) =25 KN
W – Leg length of weld = 4mm
l – Length of weld or size

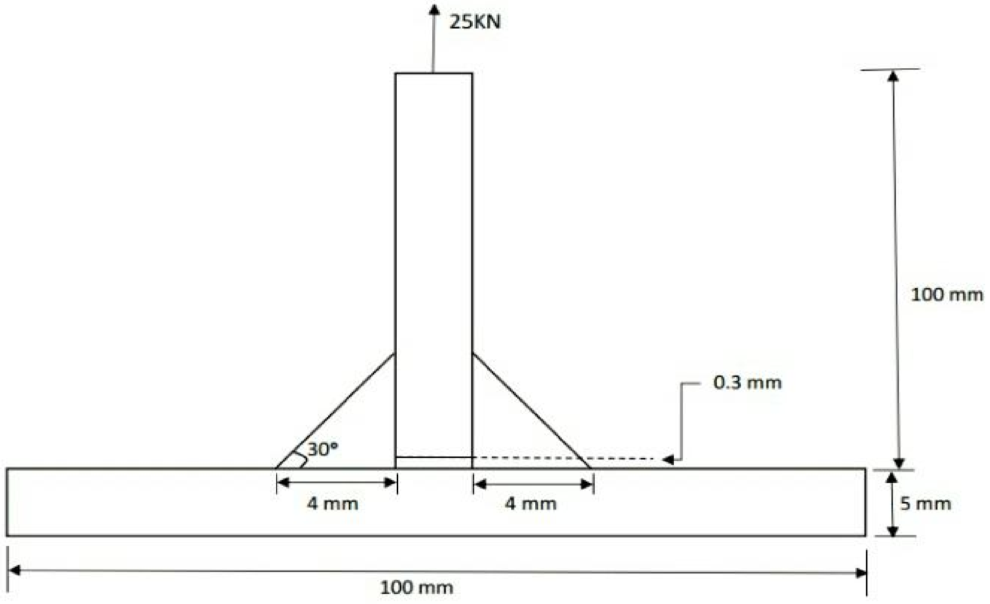


Fig 2 T-Joint with 30^0 fillet angle and 0.3mm gap

f weld = 50mm
lr - Length of top load section = 5 mm
B_r - Breadth of top load section = 50 mm
l_t - Throat length = 4mm
t - Throat thickness = 0.3,0.6,0.9
A - Area of weld section
The material properties are specifed
Total area where stress is directly acted = $A_f + A_l + A_t$
A_f - Area of fillet section
A_l - Area of load section
A_t - Throat area
<u>1st case</u> (0.3 thickness)
Area = $A_f + A_l - A_t$
<u>30^0</u>
A = 2 [4 Cos 30×50] + [5×50] – [0.3×5]
= $594.91mm^2$
Breaking stress = P/A ×K =[25000÷594.91] × 4
= 168.092MPa
<u>45^0</u>
A = 2 [4 Cos 45×50] + [5+50]-[0.3×5]
=$531.34mm^2$
Breaking stress = [25000÷531.34] ×4
=188.202MPa
<u>60^0</u>
A = 2 [4 Cos 60×50] + [5×50] – [0.3×5]
= $448.5mm^2$
Breaking stress = [25000÷448.5] × 4
= 222.96MPa
<u>2nd case</u> (0.6 thickness)

A = 2 [4 Cos30×50] + [5×50]-[0.6×5]
=593.41mm^2
Breaking stress = [25000÷594.41] × 4
= 168.51MPa
A = 2[4 Cos 45×50] + [5×50] – [0.6×5]
= 529.84mm^2
Breaking stress = [25000÷529.84] ×4
= 188.71MPa
A = 2 [4 Cos 60×50] +[5×50] – [0.6×5]
= 447mm^2
Breaking stress = [25000÷447] ×4
= 223.71MPa

3rd Case (0.9 thickness)
A = 2 [Cos 30×50] + [5×50] – [0.9×5]
=591.91mm^2
Breaking stress = [2000÷591.91] × 4
= 1698.94MPa
A = 2 [4 Cos 45×50] + [5×50]-[0.9×5]
= 528.34mm^2
Breaking stress = [2000÷528.34] × 4
= 189.271MPa
A = 2[4 Cos 60×50] + [5×50]-[0.9×5]
= 445.5mm^2
Breaking stress = [25000÷445.5] × 4
= 224.466MPa

1st case (0.3 mm gap)
Breaking stress(30^0) = 168.092MPa
Breaking stress(45^0) = 188.202MPa
Breaking stress(60^0) = 222.96MPa

2nd case (0.6 mm gap)
Breaking stress(30^0) = 168.51MPa
Breaking stress(45^0) = 188.71MPa
Breaking stress(60^0) = 223.71MPa

3rd case (0.9 mm gap)
Breaking stress(30^0) = 168.944MPa
Breaking stress(45^0) = 189.271MPa
Breaking stress(60^0) = 224.466MPa

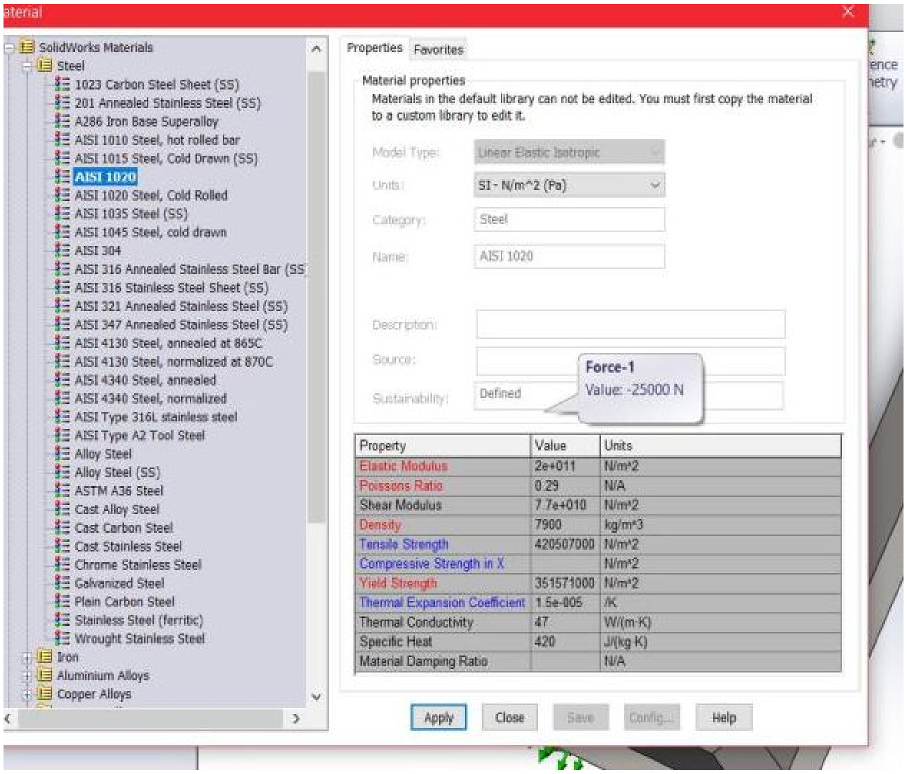
Table 1 Numerical results for breaking stress with different angle and gap betweenparent materials

Gap between parent plats (mm)	Breaking stress for 25 kN (MPa)		
	30^0	45^0	60^0
0.3	168.09	188.20	222.96
0.6	168.51	188.71	223.71
0.9	168.94	189.27	224.46

Table.1 shows the breaking stress for gap between parent plates. The maximum breaking stress 224.46MPa for 60° angle with 0.9mm gap welded section.

III.FINITE ELEMENT ANALYSIS (FEA)

The solid works 13 simulation (SWS13) is used to analyze distortions and residual stresses that can occur during the welding process. The software is based on Finite Element Analysis and the software is used to increase the overall efficiency of the system by possibly predicting the useful life of the welded structures along with the possible improvement modifications. It is a computer-based numerical technique which makes the structure under analysis in to finite elements and calculates its strength individually. So the results will be more accurate. It can be used to calculate the effects of structures by the application of stresses which cause for the phenomena like deflection, stress, vibration, buckling behavior etc. The joining points of finite elements are known as nodes in finite element method. The variable like temperature, displacement, and velocity or pressure inside the nodal region is unknown, so approximation function is necessary, such functions are also called as interpolation models and are known in terms of field variable at the nodes.



Material properties
Materials in the default library can not be edited. You must first copy the material to a custom library to edit it.

Model Type: Linear Elastic Isotropic
Units: SI - N/m^2 (Pa)
Category: Steel
Name: AISI 1020
Description:
Source:
Sustainability: Defined

Force-1
Value: -25000 N

Property	Value	Units
Elastic Modulus	2e+011	N/m^2
Poissons Ratio	0.29	N/A
Shear Modulus	7.7e+010	N/m^2
Density	7900	kg/m^3
Tensile Strength	420507000	N/m^2
Compressive Strength in X		N/m^2
Yield Strength	351571000	N/m^2
Thermal Expansion Coefficient	1.5e-005	/K
Thermal Conductivity	47	W/(m·K)
Specific Heat	420	J/(kg·K)
Material Damping Ratio		N/A

Fig 3 Material properties

Fig 3 showing the material properties of AISI 1020, the elastic modulus, poissons ratio, shear modulus, density, tensile strength, compressive strength, yield strength, thermal expansion coefficient, thermal conductivity, specific heat, material damping ratio.

Table 2 Analysis results for Von misses stress with different angle and gap

Gap between parent plats (mm)	Breaking stress for 25 kN (MPa)		
	30^0	45^0	60^0
0.3	163.38	190.51	248.92
0.6	166.97	212	276
0.9	170	235	285

Table.2 shows the breaking stress for gap between parent plates. The maximum breaking stress 285MPa for 60° angle with 0.9mm gap welded section.

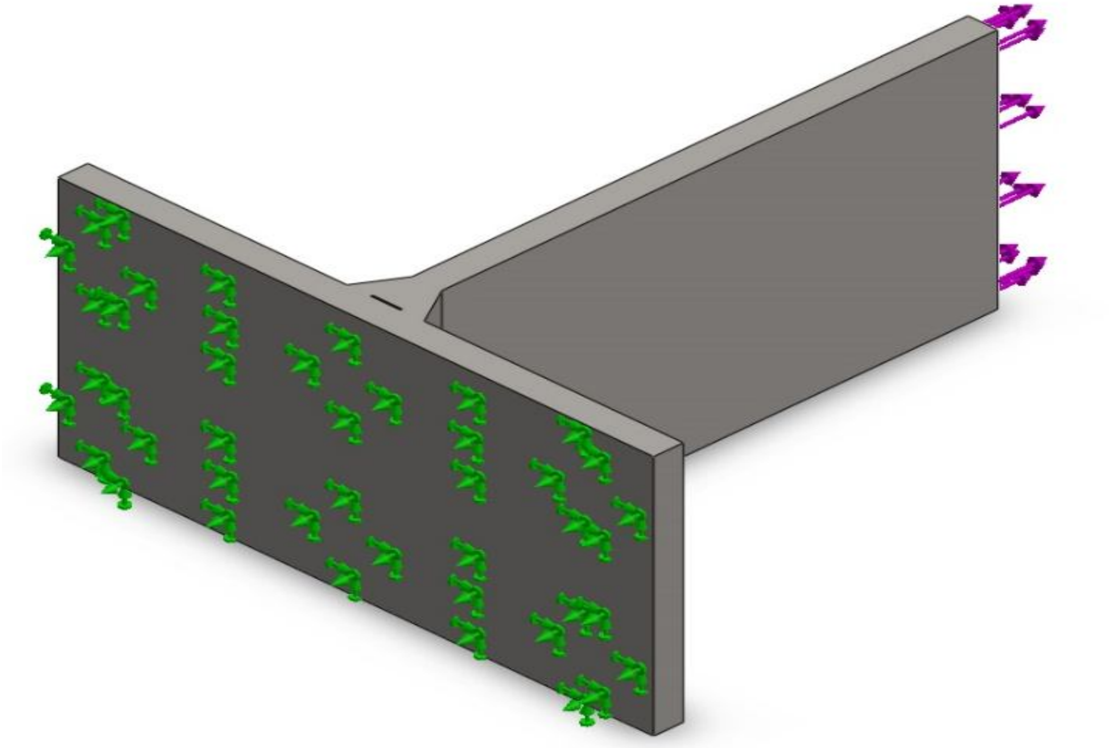


Fig.4 load and fixtures (boundary conditions)

As shown in figure 4 the same scenario is undertaken as numerical simulation. Load of 25kN is applied as shown with lower plate fixed.

Table 3 Mesh Information.

Mesh type	Solid Mesh
Mesher Used:	Standard mesh
Jacobian points	4 Points
Element Size	3.71898 mm
Tolerance	0.185949 mm
Mesh Quality	High

Table 3 showing the mesh informations of the model. In this work a solid mesh of standard mesher. There is 4 jacobian points are present in this mesh and having element size of 3.71898mm with tolerance of 0.185949mm hence the mesh quality is high.

Table 4 Mesh Information Details.

Total Nodes	15009
Total Elements	8703
Maximum Aspect Ratio	14.71
% of elements with Aspect Ratio < 3	98.3
% of elements with Aspect Ratio > 10	0.839

Table 4 showing the mesh information details of the model from that the total no of nodes are 15009 and the total no of elements are 8703 in this mesh. Having Maximum Aspect Ratio of 14.71 ,% of elements with Aspect Ratio < 3 is 98.3 and % of elements with Aspect Ratio > 10 is 0.839 with 0 distorted elements(Jacobian).

With uniform load of 25000N at different parameters shows following results.

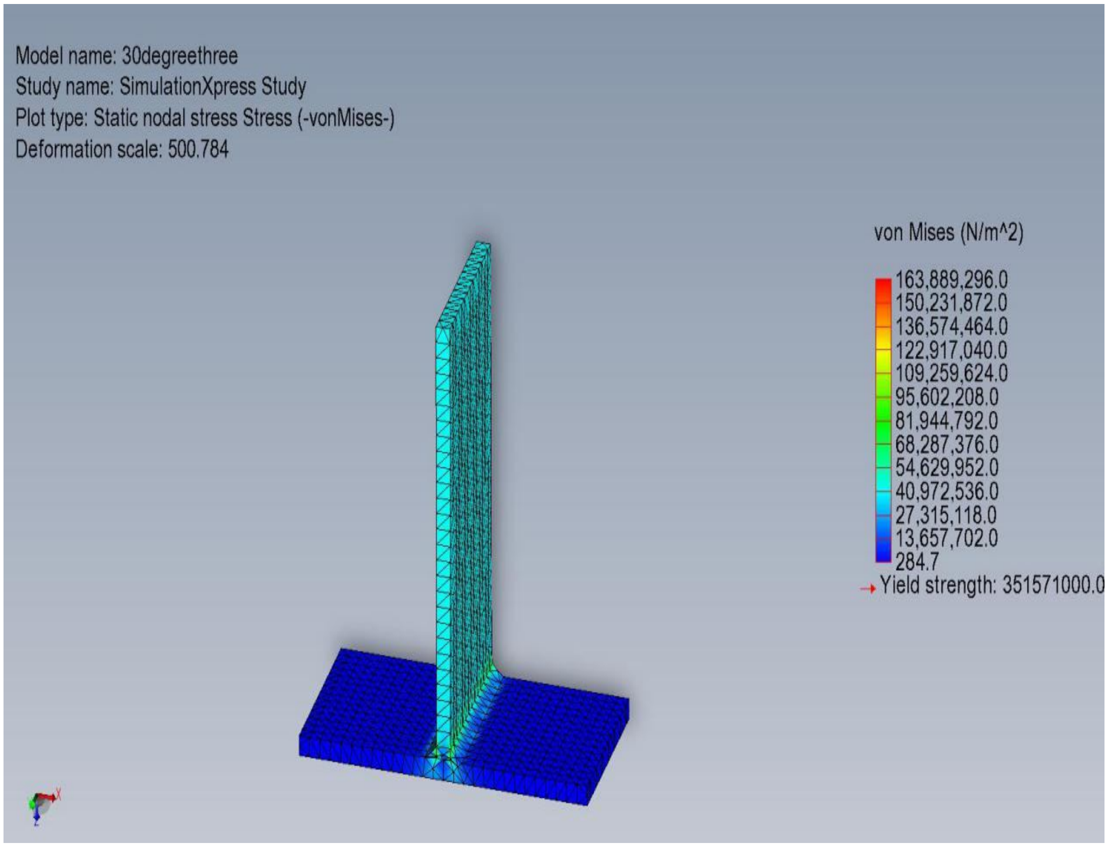


Fig.6.Equivalent von-misses stress for 30º degree and 0.3mm gap

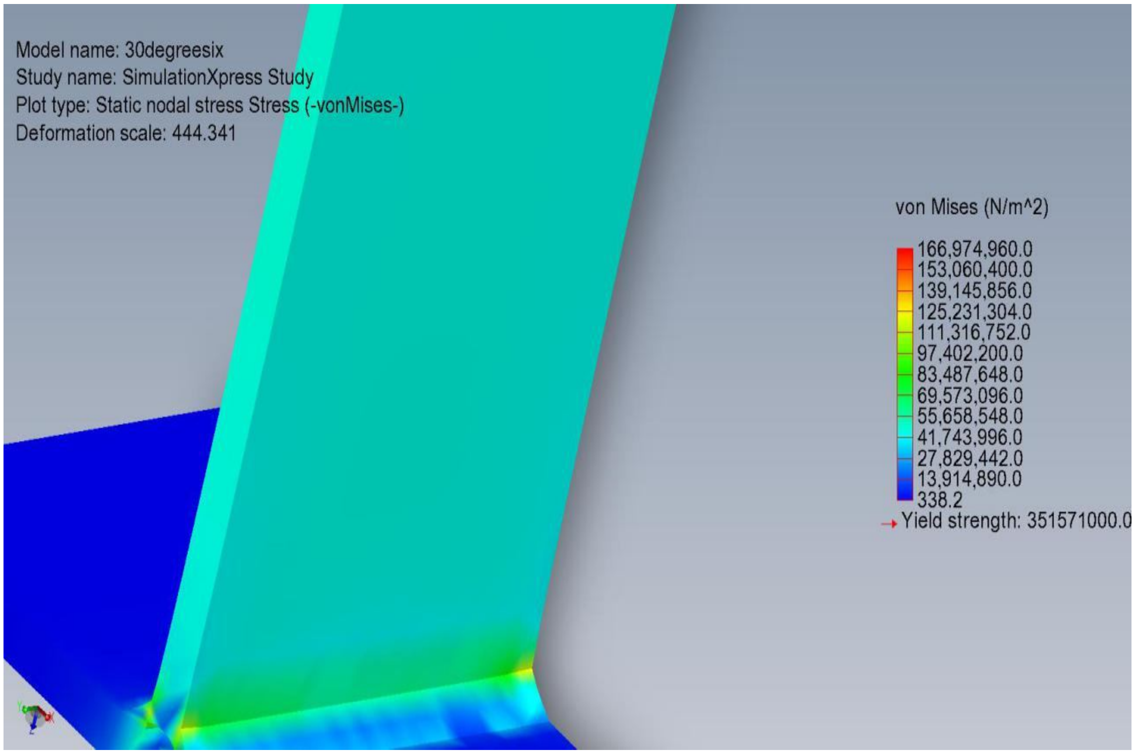


Fig.7.Equivalent von-misses stress for 30º degree and 0.6mm gap

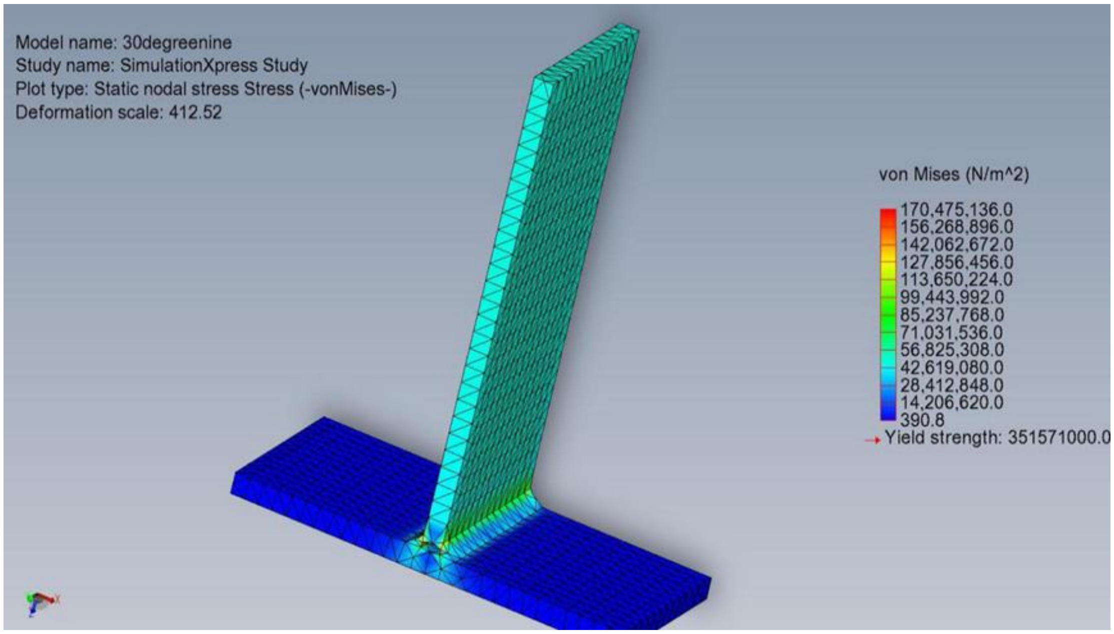


Fig.8.Equivalent von-misses stress for 30º degree and 0.9mm gap

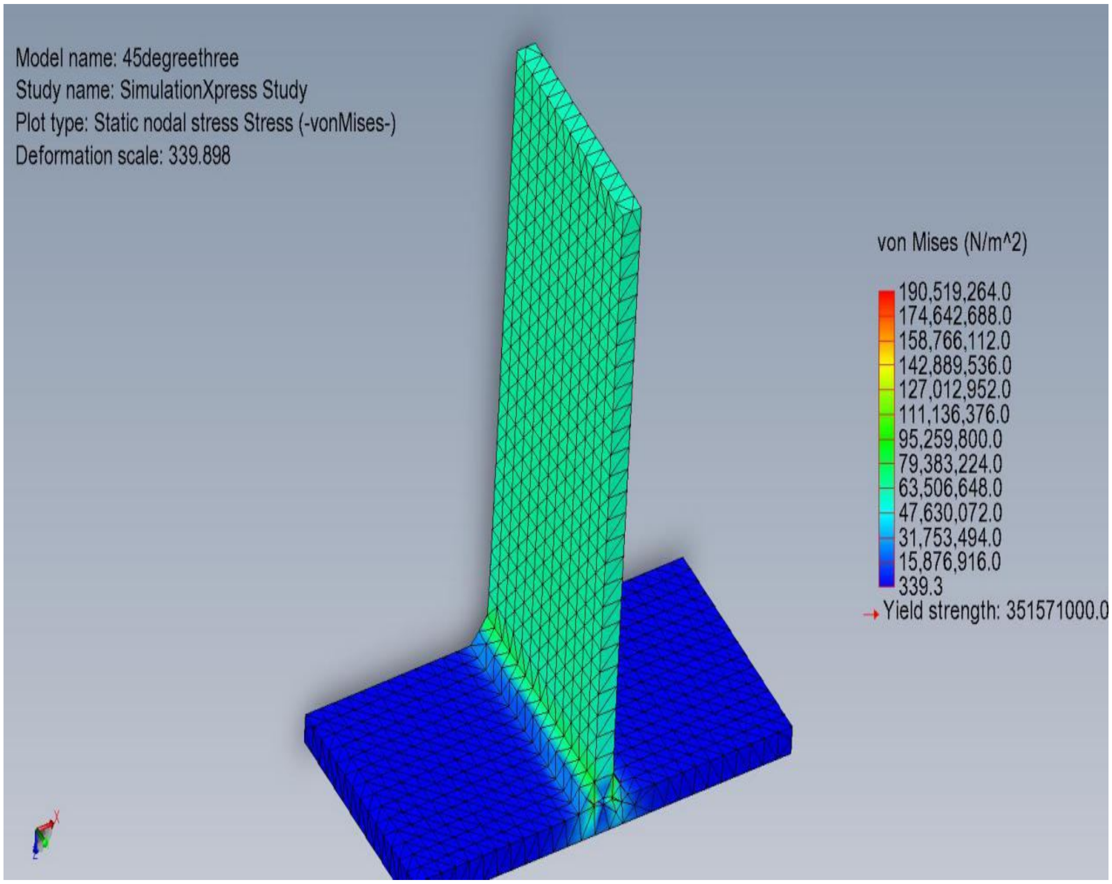


Fig.9.Equivalent von-misses stress for 45º degree and 0.3mm gap

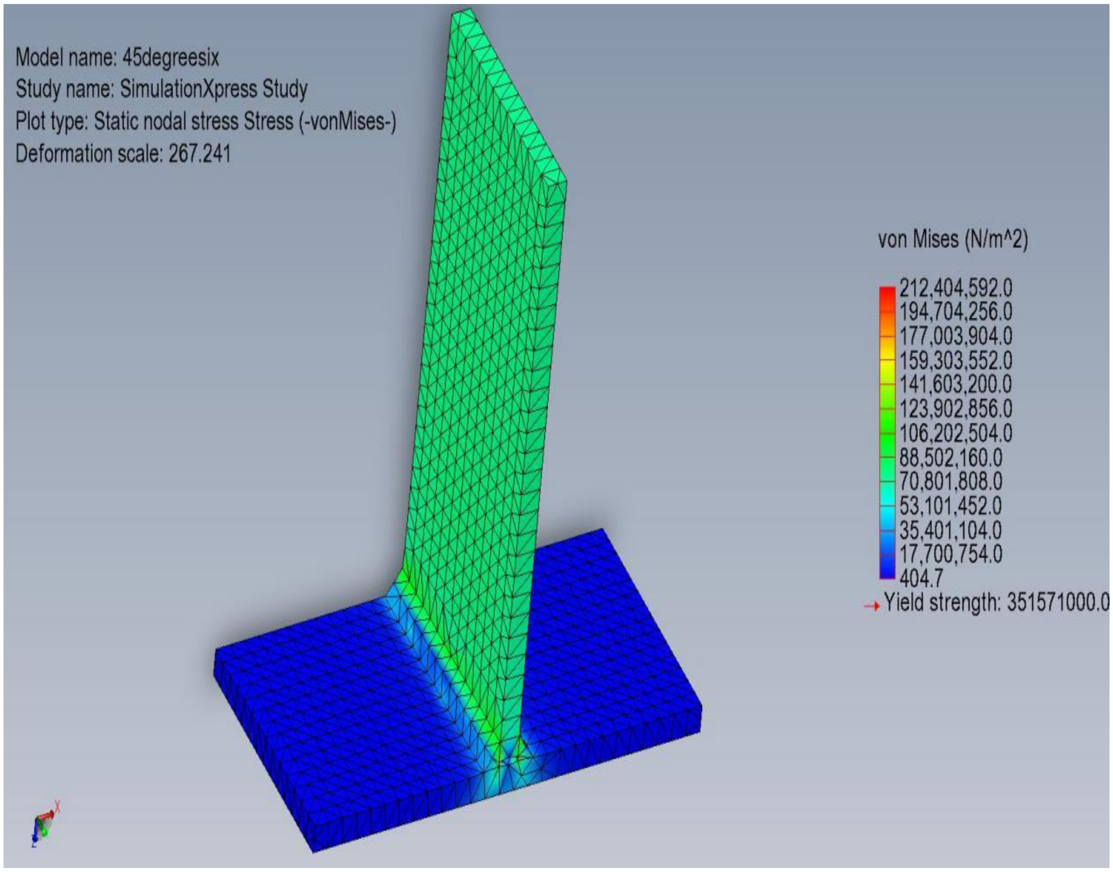


Fig.10.Equivalent von-misses stress for 45° degree and 0.6

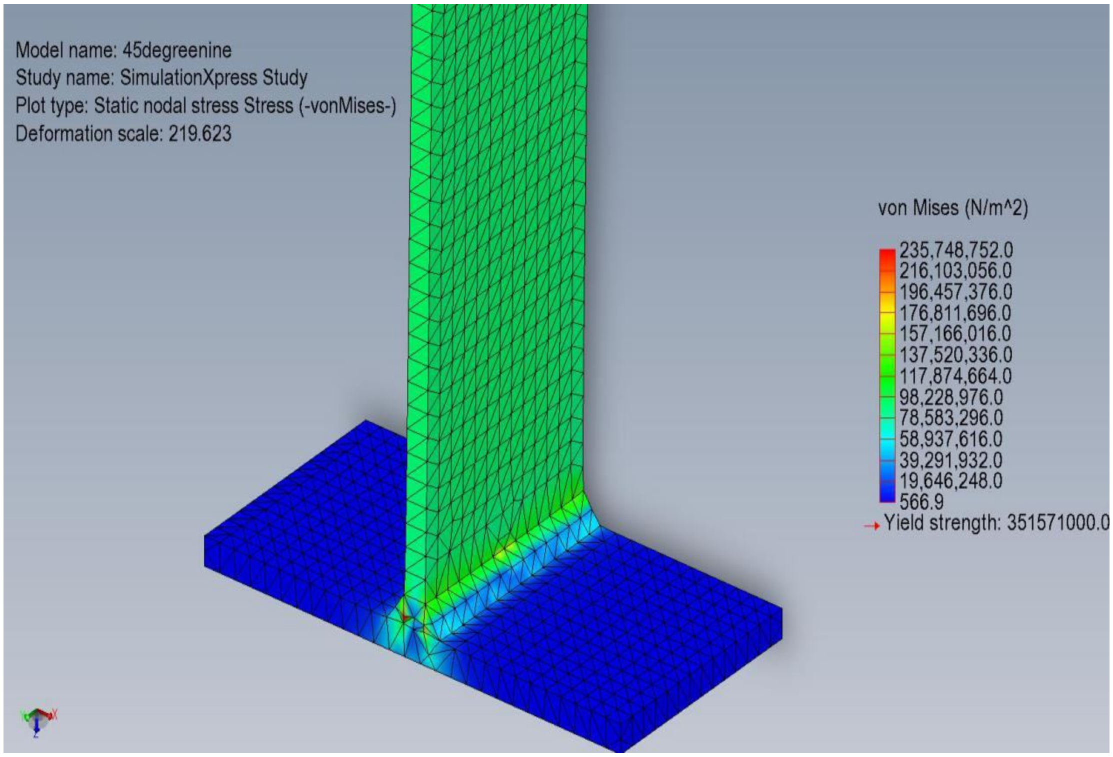


Fig.11.Equivalent von-misses stress for 45° degree and 0.9

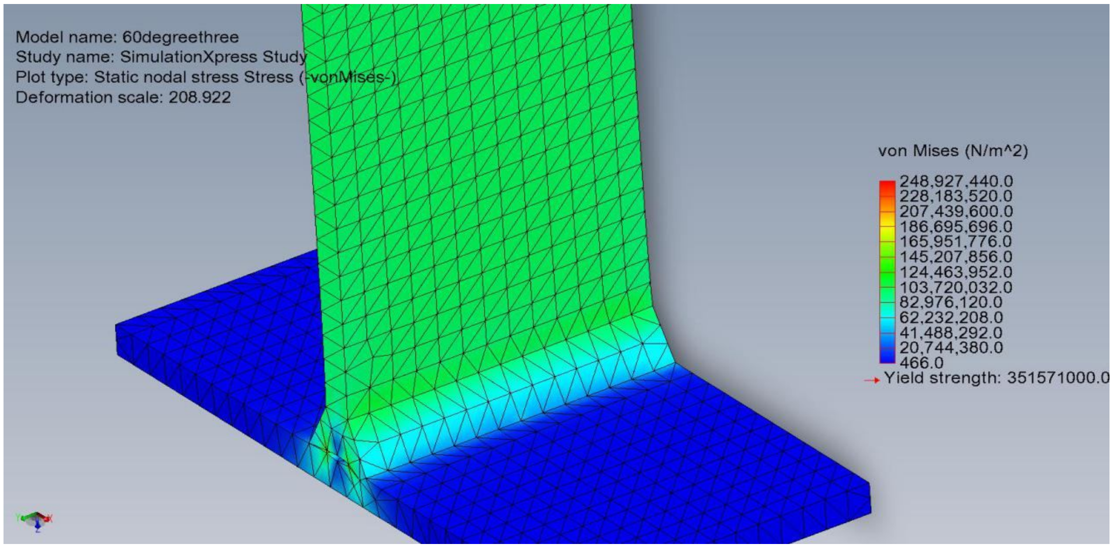


Fig.12.Equivalent von-misses stress for 60° degree and 0.3

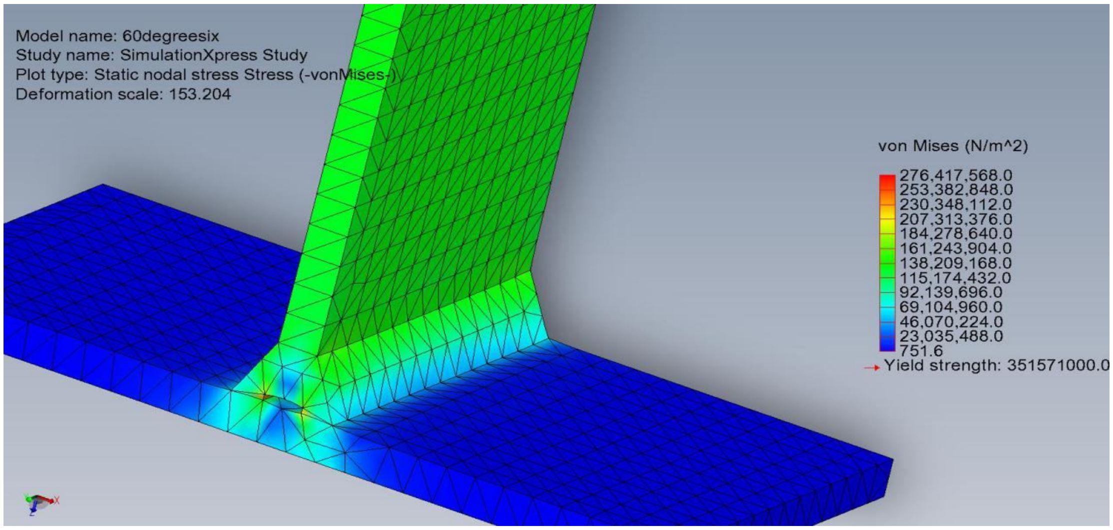


Fig.13.Equivalent von-misses stress for 60° degree and 0.6

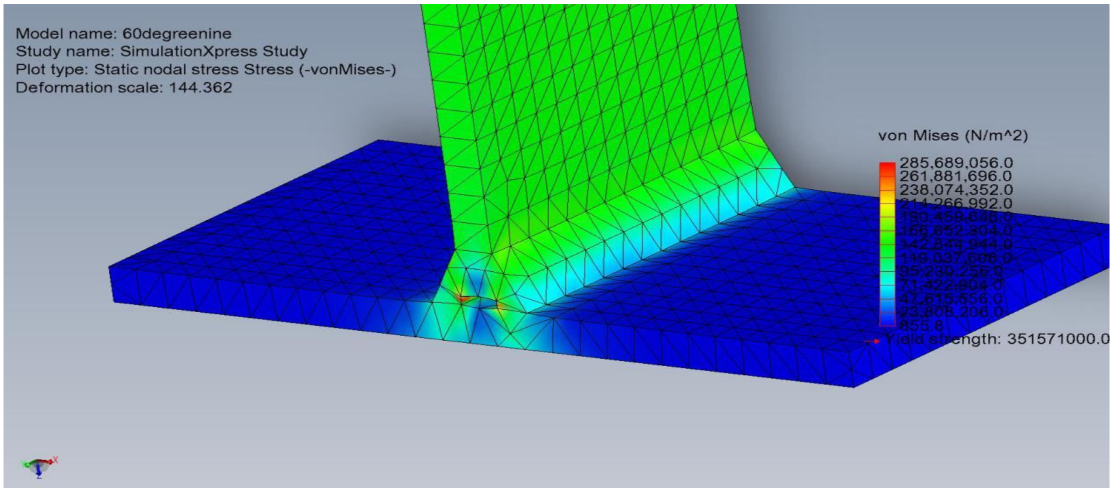


Fig.14.Equivalent von-misses stress for 60° degree and 0.9

IV. RESULTS AND DISCUSSION

The finite element and numerical analysis results of equivalent von misses stress are carried out and it's approximately equal to the numerical breaking stress. The FEA analysis of T-welded joint for the same geometry revealed the maximum Von-misses stress of 285MPa it's approximately equal to numerical maximum breaking strength of 224.46MPa.

Table.5 Von misses Stress Analysis results

Gap	Angle	Numerical analysis (MPa)	FEA (MPa)
0.3	30^0	168.09	163.88
0.3	45^0	188.22	190.51
0.3	60^0	222.96	248.92
0.6	30^0	168.51	166.97
0.6	45^0	188.71	212
0.6	60^0	223.71	276
0.9	30^0	168.94	170
0.9	45^0	189.27	235
0.9	60^0	224.46	285

Table 5 showing the stress analysis of the T-weld joint with different fillet angle and gap. From the numerical analysis maximum breaking stress is 224.46MPa (0.9mm gap and 60º) And from the finite element analysis maximum breaking stress is 285MPa (0.9mm gap and 60º).

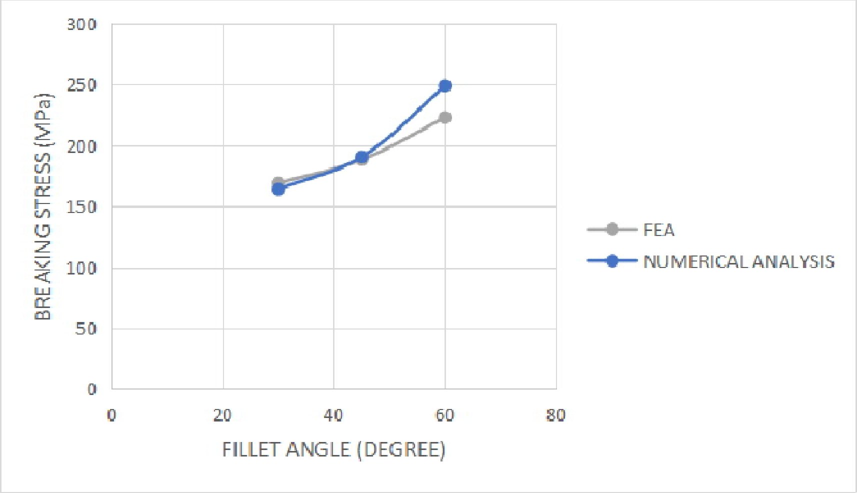


Fig.15 Relation between fillet angle v/s breaking stress With 0.3 mm gap.

V. CONCLUSION

The finite element analysis is used in this work to evaluate the deformation breaking stress of weld T-joint to restrict the weldment failure. Static stress analysis performed on the weldment under tensile load revealed the maximum Von-misses stress with respect to the gap between parent plates using solid works 13.The design and analysis of welded T-joint has been done successfully.

The numerical analysis are compared with FE analysis and both are approximately acceptable. The conclusion obtained by this project work are summarized below,

A. The 60º angle parameter gives better breaking stress results in numerical analysis and FEA compared then 30º and 45º.
B. The results shows that 0.9 gap between parent materials with 60º angle gives a maximum breaking stress.
C. If the gap is increased means breaking stress will gradually increases.
D. While comparing Numerical and Finite Element Analysis results a 90% accuracy is reported.
E. In future an experimental study also can be under taken.

REFERENCES

[1] Dean Deng, Wei Liang Hidekazu Murakawa, Determination of welding deformation in fillet welded joint by means of numerical simulation and comparison with experimental measurements, Journal of Material Processing Technology,183, 2007, 219-225.

[2] Chattopathay A ,Glinka G, El-Zein M, Qian J and Formas R:Stress Analysis and Fatigue of welded structures journal of welding in the weld,17,2011,234-237.

[3] Tso-Liang Teng Chin- Ping Fung, Analysis of residual stresses and distortions in T- joint fillet welds" International journal of Pressure and Piping,78,2001,523-538.

[4] Shahram Sarkani, George Michaelov, 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, 2000, 247-268.

[5] Mohd Shahar Sulaiman, Chan Yin Chau, Simulation and experimental study on distortion of butt and T-joints using Weld Planner, Journal of Mechanical Science and Technology, 25, 2011, 2641-2646.

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