

## CHARACTERISTICS OF RESIDUAL STRESSES ON WELDED TUBULAR T-JOINTS

CH. SIVA RAMAKRISHNA<sup>1</sup> & B. N. DHANANJAYA RAO<sup>2</sup>

<sup>1</sup>Associate Professor, Vignan's Institute of Information Technology, Visakhapatnam, Andhra Pradesh, India

<sup>2</sup>P.G. Student, Vignan's Institute of Information Technology, Visakhapatnam, Andhra Pradesh, India

### ABSTRACT

Welding is a wide range multistate permanent fastening method of fabrication. Globally welding technology has numerous applications. The present paper deals with the analysis of distribution of the residual stresses that act on the welded tubular T-joints. Residual stresses vary according to the varying temperatures on the joint. The equivalent stresses and other stresses are calculated from the theoretically calculated inputs. The theoretical inputs are given to the analysis software ANSYS Workbench i.e. heat flux, temperature and young's modulus of the material selected for the joint. The model i.e. T-joint including with the weld region is designed in designing software CATIA and the model is imported to ANSYS for analysis.

**KEYWORDS:** Welding, ANSYS Workbench, CATIA, Residual stresses, Tubular Joint

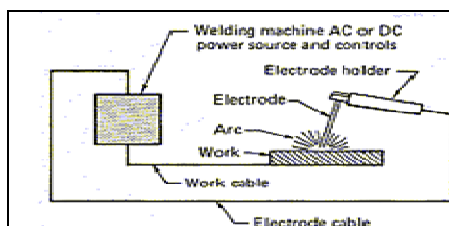
### 1. INTRODUCTION

#### 1.1. Introduction to Welding Technology

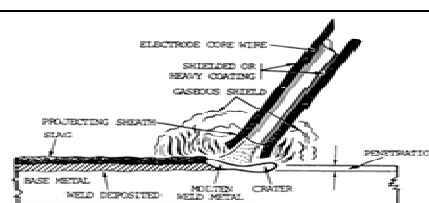
Welding is a science of joining the metals by the application of heat. Welding has been an important fabrication process in modern industry outdating the other means of joining the metals like bolting, riveting etc. The welding is widely used as a fabrication and repairing process in industries. Some of the typical applications of welding include the fabrication of ships, pressure vessels, automobile bodies, off-shore platform, bridges, welded pipes, sealing of nuclear fuel and explosives etc. Welding has been an important fabrication process in modern industry outdating the other means of joining the metals like bolting, riveting etc.

#### 1.2 Manual Metal Arc Welding (MMAW)

It is a manual arc welding process that uses a consumable electrode coated in flux to lay the weld. **Operation of Manual Metal Arc Welding:** To strike the electric arc, the electrode is brought into contact with the work piece in a short sweeping motion and then pulled away slightly. This initiates the arc and thus the melting of the work piece and the consumable electrode, and causes droplets of the electrode to be passed from the electrode to the weld pool.



**Figure 1: Equipment of Manual Metal Arc Welding**



**Figure 2: Manual Metal Arc Welding Process**

As the electrode melts, the flux covering disintegrates, giving off vapours that protect the weld area from oxygen and other atmospheric gases. In addition, the flux provides molten slag, which covers the filler metal as it travels from the electrode to the weld pool. Once part of the weld pool, the slag floats to the surface and protects the weld from contamination as it solidifies. Once hardened, it must be chipped away to reveal the finished weld. While polarity affects the penetration and burn-off rate, the electrode coating also has a strong influence on arc characteristics.

### 1.3 Residual Stresses at Welded Joints

The residual stresses can be sufficient to cause a metal part to suddenly split into two or more pieces after it has been resting on a floor or table without any external load being applied. This is not a common occurrence but experienced people in the metal working industry have witnessed this phenomenon. Residual stresses are the stresses that are inside or locked into a component or assembly of parts. The internal state of stress is caused by mechanical or thermal processing of the parts or both. Common examples of these are bending, rolling or forging a part. Also the thermal stresses are induced during welding. Residual stresses can play a significant role in explaining or preventing failure of a component at times. Thermal residual stresses are primarily due to differential expansion when a metal is heated or cooled. The two factors that control this are thermal treatment (heating or cooling) and thermal restraint. Both the thermal treatment and restraint of the component must be present to generate residual stresses. **Sources of Residual Stresses:** Due to the non-uniform temperature distribution during the thermal cycle, incompatible strains lead to thermal stresses. These incompatible strains due to dimensional changes associated with solidification of the weld metal, metallurgical transformations and plastic deformation are the sources of residual stresses and distortion. Welding induced residual stresses and distortion can play a very important role in the reliable design of welded joints and welded structures. **Effects of Various Factors on Residual Stresses:** Residual stresses mainly depend on the following factors (I) Effect of heat input value. (II) Effect of temperature dependency on material properties. (III) Effect of radiation and phase changes. **Development of Welding Residual Stresses:** Residual stresses are formed by the plastic deformations during the thermal cycle of welding. During welding, the high temperature at the weld region causes compressive stress due to thermal expansion. This compressive stress exceeds the yield stress at the corresponding temperature. Away from the weld, the stress is tensile equilibrating the compressive stress over the analysis plane. The stress reverses sign from compressive to tensile at locations that have plasticized during welding, and a band of residual stresses at the material yield level (at room temperature) is developed at the weld region. Away from the weld the residual stress is compressive.

## 2. LITERATURE REVIEW

**SK. Yeoh [1]** presented his research on the 'behaviour of Tubular T-joints that are subjected to combined loading' in the year 1995 and concluded that a round-to-round tubular t-joint was subjected to axial load, in-plane bending, out-of plane bending and combination of three loads. The peak hot spot position on the T-joint under combined loads shifts in location from that of the basic load cases.

**L. Sundar [2]** presented a paper on 'butt joints through manual metal arc welding' in 2006 and proved that MMAW process is being successfully used for welding of cast irons, low alloy steels, high alloy steels, stainless steels, manganese steels, copper and its alloys, aluminium and its alloys, nickel, etc. the best parameter of MMAW is selection of welding parameters i.e., current and voltage. **S. Shengzhi [3]** found the results of 'static strength of lap joints' in 2011 and concluded that as stress concentration exists near the weld toe, failure generally occurs near the weld. Thus, by increasing the thickness of chord near the weld, the static strength of tubular T-joint is improved. These are few papers that include

results on various welded joints and their behaviour when stresses are induced in them. But, we have evaluated the 'characteristics of residual stresses on welded tubular T-joints' with the help of analysis software, ANSYS workbench and the suitable working conditions as inputs.

### 3. METHODOLOGY

#### 3.1 Introduction to Analysis

The objective of this paper is to investigate the characteristics of the residual stresses induced in two tubes in arc welding in order to minimize residual stresses in the work piece. More specifically, the choice of voltage and current applied has been sought with a mechanical model designed in CATIA and imported to ANSYS Workbench. The thermal model is based on a heat source description which in essence is governed by the heat input i.e. heat flow (depends on voltage and current) and weld area. This model in turn delivers heat flux which is helpful to find out thermal stresses and finally residual stresses in the work piece. In this model the heat generation is expressed as a surface heat flux from the electrode into work piece, however it is the function of the electrode diameter and voltage and current applied as follows.

#### 3.2 Calculation of Heat Flux (q)

$$\text{Total heat flow, } Q = V * I * \eta$$

$$\text{Heat flux, } q = Q/A$$

$$\text{Area, } A = \text{perimeter of the joint} \times \text{throat thickness of the weld}$$

$$A = \pi \times D \times t$$

Where, V=Voltage supplied, I=Current supplied,  $\eta$ =Weld efficiency (usually take 0.92), A=Weld area. D=Diameter of Joint, t=Throat thickness.

#### 3.2.1 Material Used -Low Carbon Steel

Low carbon steel approximately contains 0.05% - 0.29% carbon content (e.g. AISI 1018 steel). Mild steel has a relatively low tensile strength, but it is cheap and malleable; surface hardness can be increased through carburizing. It is a hypo eutectoid steel containing 0.2 percent carbon. In the austenitic range this alloy consists of a uniform interstitial solid solution. Each grain contains 0.2 percent carbon dissolved in spaces of the F.C.C iron lattice structure. It contains 75% proeutectoid ferrite and 25% pearlite.

#### 3.1.2 Material Properties

Different material have been used in structures where welding is involved, with low carbon steel being the most common. The material used for weldment is also low carbon steel. Two sets of temperature dependent material properties (Table 1) were needed in the analysis. The modulus of elasticity is a measure of the stiffness of a material. A higher modulus material is more likely to resist distortion. The amount of expansion or contraction of a metal will undergo during a heating or a cooling cycle depends on the coefficient of thermal expansion. Thermal conductivity gives a measure of the ease of heat flow through a material.

**Table 1: Temperature Dependent Material Properties for both Pipe and Weld**

T (Celsius)	0	100	300	450	550	600	720	800	1450	1510	1580
E(MPa)	315	349	440	460	410	330	58.8	58.8	1.29	1.0	0.01
$\alpha_0(10^{-6}/^{\circ}\text{C})$	10	11	12	13	14	14	14	14	15	15	15
K(W/m <sup>2</sup> K)	51.9	51.1	46.1	41.05	37.5	35.6	30.64	26	29.45	29.7	29.7
C(J/Kg <sup>2</sup> K)	450	499.2	565.5	630.5	705.5	773.3	1080.4	931	437.93	400	735.25

## 4. CALCULATIONS

### 4.1. Thermal Model

The heat flow is constant for the joint of varied temperature and thermal flux is calculated by software automatically by using heat flow input which is calculated theoretically.

In this case, voltage (V = 30 volts) and current (I = 300 Amps) are taken from the above table for calculating the heat flow.

**Table 2: Heat Flow Parameters**

Voltage, V (volts)	80	80	100	100	66	30
Current, (Amperes)	131	102	125	156	125	300
Heat, Q=VIη (Watts)	9641.6	7507.2	11500	14352	7590	8280

Calculation of Heat Flow (Q),  $Q = V * I * \eta = 30 \times 300 \times 0.92 = 8280\text{W}$

The residual stress of different welded joints is obtained by conducting Steady State Thermal Analysis in which the calculated heat flow value is taken as input.

Area of the joint at welded region =  $\pi (D) t$

Where,  $t = \text{Throat thickness} = 0.707 \times s$

$s = \text{Thickness of the weld} = 7.5 \text{ mm}$

Now,  $\text{Area, } A = \pi (D) t = \pi \times (250) \times (0.707 \times 7.5) \text{ mm}^2 = 4164.57 \times 10^{-6} \text{ mm}^2$

and  $\text{Heat flux, } q = Q/A = 8280/(4164.57 \times 10^{-6}) = 2.00001 \times 10^6 \text{ W/mm}^2$

## 5. DESIGN & ANALYSIS

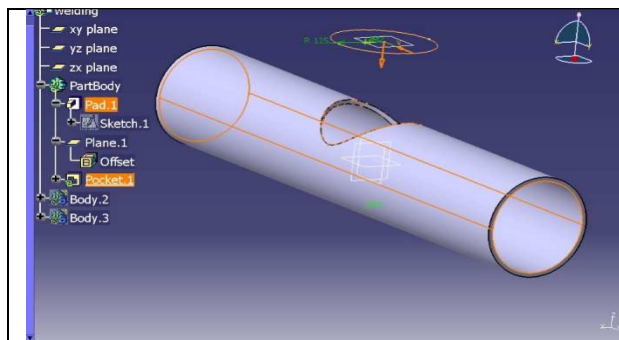
### 5.1. Introduction to CATIA

CATIA (Computer Aided Three-dimensional Interactive Application) is multi-platform software developed by the French company Dassault Systems. CATIA supports multiple stages of product development (CAX), from conceptualization, design (CAD), manufacturing (CAM), and engineering (CAE). CATIA facilitates collaborative engineering across disciplines, including surfacing & shape design, mechanical engineering, equipment and systems engineering. CATIA enables the creation of 3D parts, from 3D sketches, sheet metal, composites, and moulded, forged or tooling parts up to the definition of mechanical assemblies. CATIA can be applied to a wide variety of industries, from aerospace and defences, automotive, and industrial equipment, to high tech, shipbuilding, consumer goods, plant design, consumer packaged goods, life sciences, architecture and construction, process power and petroleum, and services.

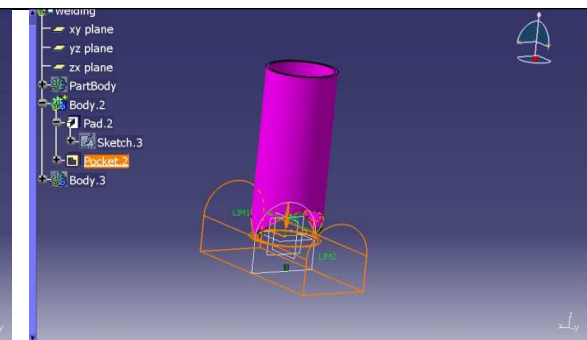
### 5.1.1. Design Steps

**Table 3: The Following Are the Steps to Be Followed to Model in CATIA V5**

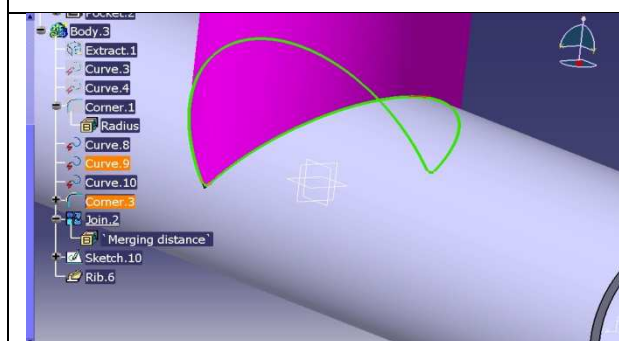
1	Go to 'START' window and select 'RUN' and type 'CNEXT' in the 'RUN' window.
2	After getting opened, Choose Start > Mechanical Design > Part Design to make sure that you are in the part design work bench. A new file in the Part Design workbench is displayed on the screen, as shown in the Figure. In which there is specification tree (i.e. axis tree) in which there are 3 different planes of which one is to be selected as reference plane.
3	Select any one of the plane (xy or yz or zx) on the on the specification.
4	Using 'profile' tool bar and selecting circle shown below and by selecting circle draw the required no. of circles (two concentric circles).
5	The required dimensions (diameter) are given by selecting the object and by using the constraint tool bar.
6	By clicking on the 'Exit work bench' command, it facilitates to get the 3D object. The 3D object is extruded by selecting the pad option in the sketch-based features.
7	Give the line at which angle the other pipe is to be projected and click on Exit sketcher to give reference plane. Here, draw the two concentric circles of required dimension and pad up to the surface of the pipe. Now, again select the reference plane which is previously drawn and draw the circle with inner diameter. This is used to pocket, which provides the opening at the joint.
8	For the weldment, create a bead and extract it using the extract tool. Now, create two curves and use corner tool to avoid sharp edges and finally join them using the join tool. Use rib for the weld joint.
9	The completed drawing is saved by using the save management and firstly by creating .CAT part file. While using this object in ANSYS workbench, the file is again saved by creating .stp file.



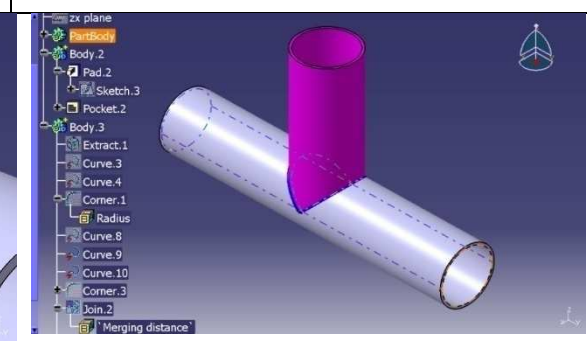
**Figure 1: Pocket Operation on Horizontal Pipe**



**Figure 2: Pocket Operation on Vertical Pipe**



**Figure 3: Joining Process**



**Figure 4: Assembled T-Joint with Weldment**

The model is imported to ANSYS and evaluated for stresses at the various temperatures values.

### Introduction to Finite Element Method (FEM)

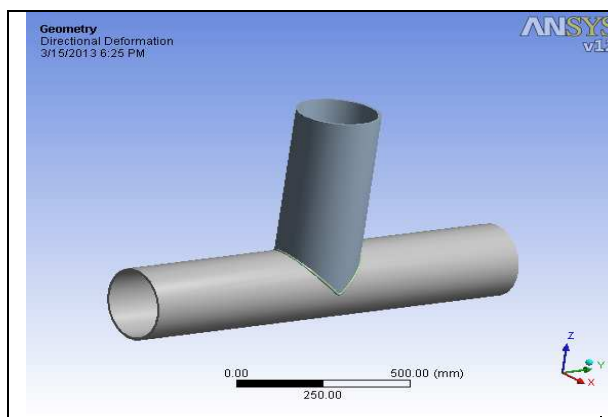
The basic idea in the Finite Element Method is to find the solution of complicated problems with relatively easy

way. The Finite Element Method has been a powerful tool for the numerical solution of a wide range of engineering problems. In the Finite Element Method, the solution region is considered as many small, interconnected sub regions called Finite elements. For simple members like uniform beams, plates etc., classical solutions can be sought by forming differential and/or integral equations through structures like machine tool frames, pressure vessels, automobile bodies, ships, aircraft structures, domes etc., need some approximate treatment to arrive at their behaviour. The classical differential equation solution approach leads to intractability. **Introduction to ANSYS Workbench:** The ANSYS Workbench represents more than a general purpose engineering tool. It provides a highly integrated engineering simulation platform. It supports multi-physics engineering solutions. Provides bi-directional parametric with most available CAD and CATIA systems. It provides solutions for structural, thermal, modal, linear buckling, and shape optimization studies.

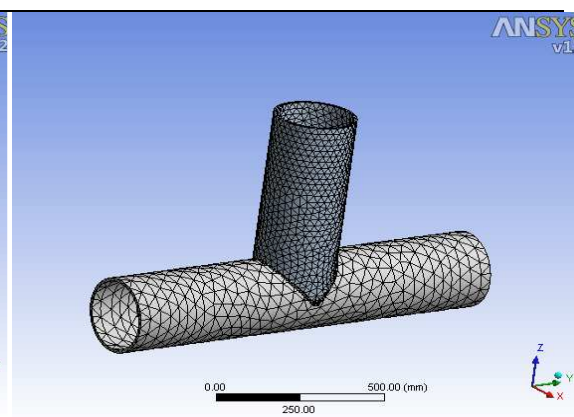
**Table 4: Steps to Solve Problem in ANSYS Work Bench**

1.	Launch ANSYS Workbench
2.	Select GEOMETRY>FILE>IMPORT EXTERNAL GEOMETRY After importing the STP file
3.	Select NEW SIMULATION>MESH>GENERATE MESH
4.	Select NEW ANALYSIS>STEADY-STATE THERMAL ANALYSIS
5.	STEADY-STATE THERMAL ANALYSIS > ANALYSIS > INSERT > TEMPERATURE
6.	STEADY-STATE THERMAL ANALYSIS > ANALYSIS > INSERT > TEMPERATURE
7.	STEADY-STATE THERMAL ANALYSIS > ANALYSIS > INSERT > HEAT FLOW/HEAT FLUX.
8.	Select SOLUTION>TOTAL HEAT FLUX >DIRECTIONAL HEAT FLUX >TOTAL DEFORMATION >EQUIVALENT STRESS >NORMAL STRESS >SHEAR STRESS >DIRECTIONAL DEFORMATION
9.	Right click on the SOLUTION > GENERATE > STEADY-STATE THERMAL
10.	Then go to setup and right click for transfer data from new and then mix up two fields i.e., STATIC STRUCTURAL ANSYS and STEADY STATE THERMAL ANSYS
11.	Click ANALYSIS SOLUTION in STATIC STRUCTURAL ANSYS to solve the problem
12.	SAVE the FILE in the WORD DOCUMENT.

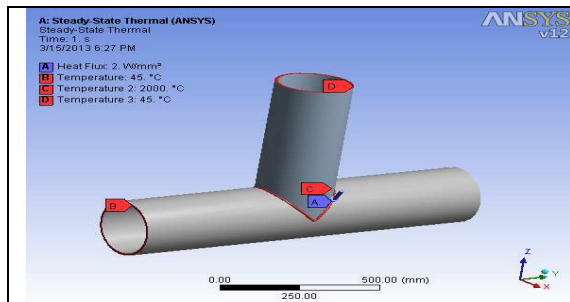
After designing all four models in CATIA software and saved as. STP files the models on which analysis is to be done using ANSYS Workbench are as shown below.



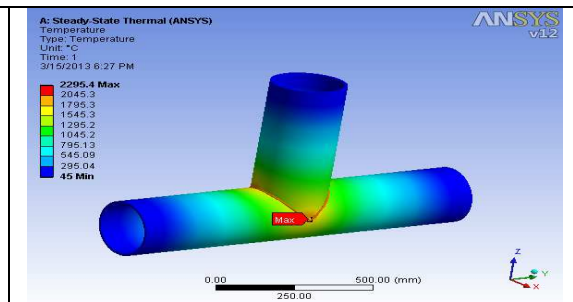
**Figure 5: Imported Model From CATIA**



**Figure 6: Model Meshed in ANSYS**



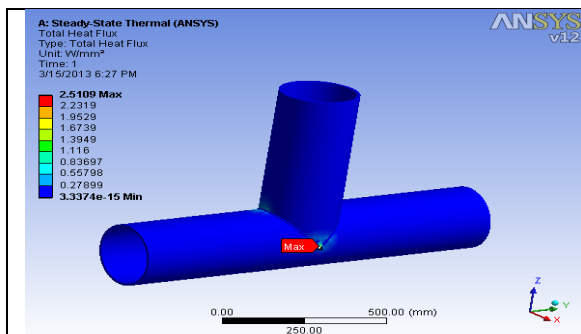
**Figure 7: Importing Temperature Values to the Model in ANSYS**



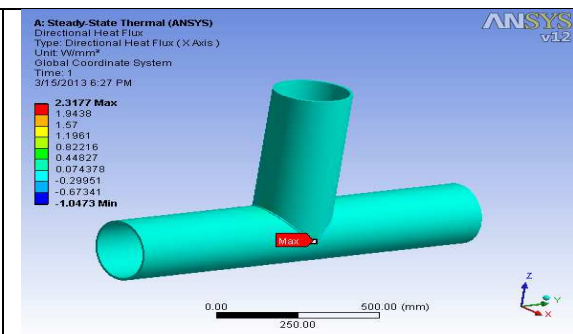
**Figure 8: Temperature Values Distributed to the Model**

## 6. RESULTS

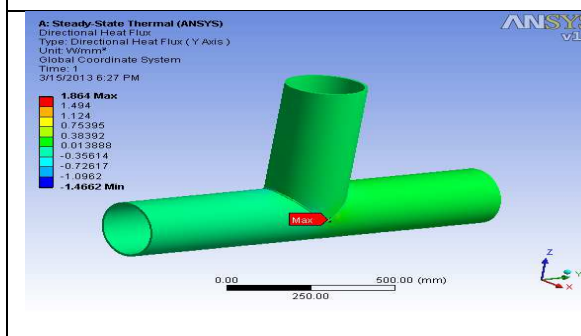
Based on the results from ANSYS Work bench, the simplified modelling process for simulating welding-induced residual stresses using a general-purpose FE package described here is reliable and instructive. The main feature of the simulation is by adopting steady state thermal analysis, the temperature history results are read from the thermal analysis as loading to the stress analysis. The thermal analysis should be steady-state to maintain constant temperature with time while the thermal analysis can be adopted for the stress analysis. However, the heat flow is calculated which is taken as input for the steady state thermal analysis and thereby we get different stresses and deformations for different temperature values. The Figures in this paper include: Total heat flux distributed, Directional heat flux, Directional deformation, Stress distribution on pipes. For the given various temperature and flux values to the model: Variation of colours represents the variation of stresses in pipes. The different values of stress (in MPa) are shown in below Figures.



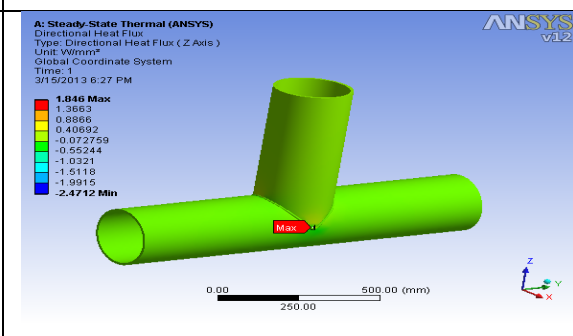
**Figure 9: Total Heat Flux Variations in the Pipe Joint**



**Figure 10: Directional Heat Flux Through X-Axis in the Pipe Joint**



**Figure 11: Directional Heat Flux - Y-Axis in the Pipe Joint**



**Figure 12: Directional Heat Flux-Z-Axis in the Pipe Joint**



Now the steady state thermal analysis is mixed with another field i.e., static structural analysis to evaluate the various stresses induced in the pipe joints. Now the temperature values are imported to the static structural analysis from the steady state thermal analysis as shown Imported.

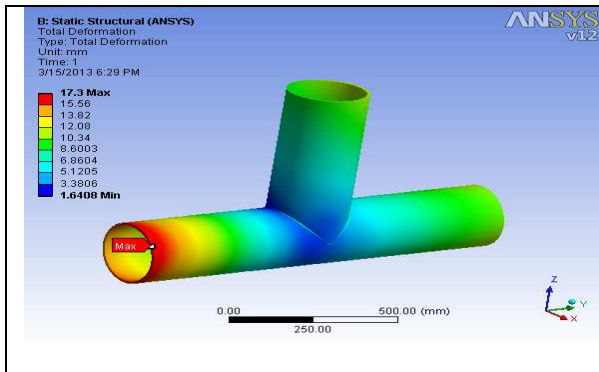


Figure 13: Total Deformations in the Pipe Joint

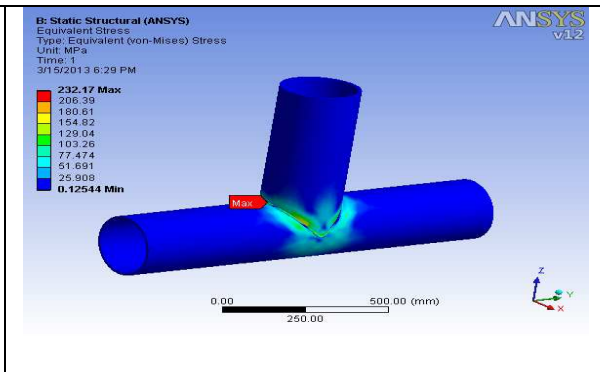


Figure 14: Equivalent Stress in the Pipe Joint

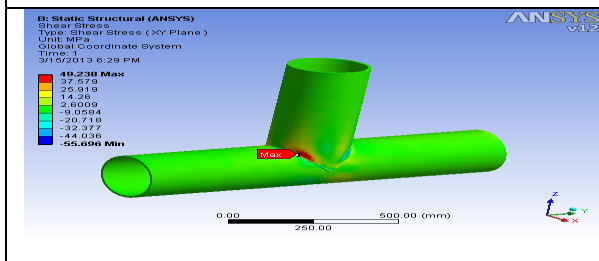


Figure 15: Shear Stress Along XY Plane in the Pipe Joint

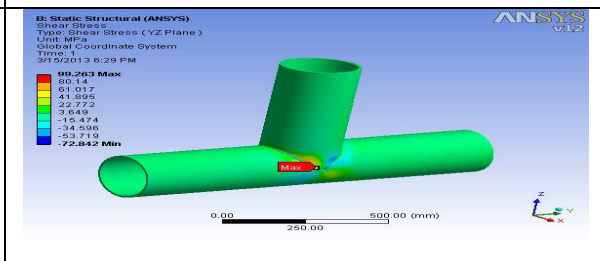


Figure 16: Shear Stress Along YZ Plane in the Pipe Joint

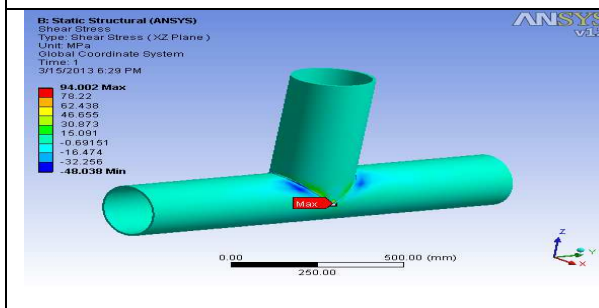


Figure 17: Shear Stress Along XZ Plane in the Pipe Joint

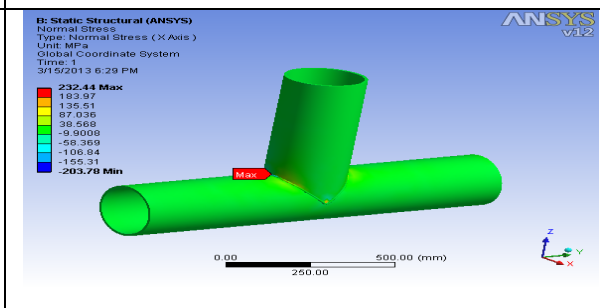


Figure 18: Normal Stress Along X-Axis in the Pipe Joint

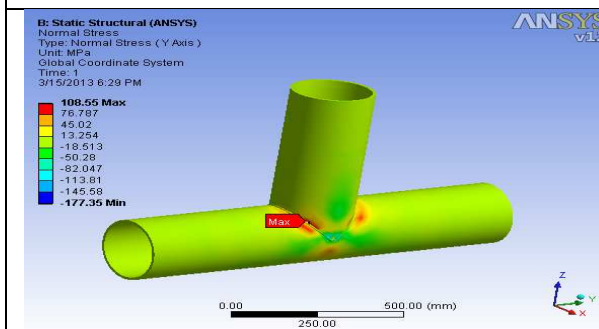


Figure 19: Normal Stress Along Y-Axis in the Pipe Joint

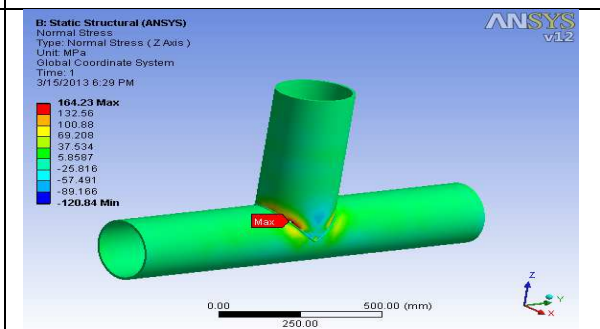
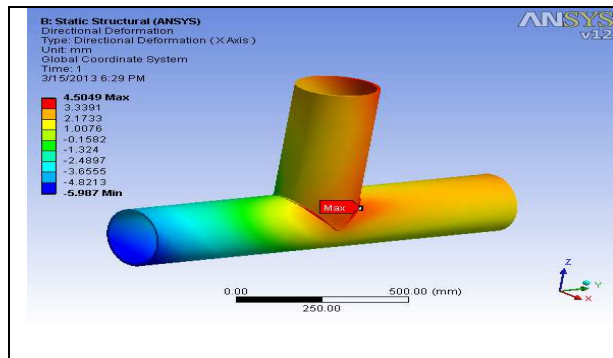
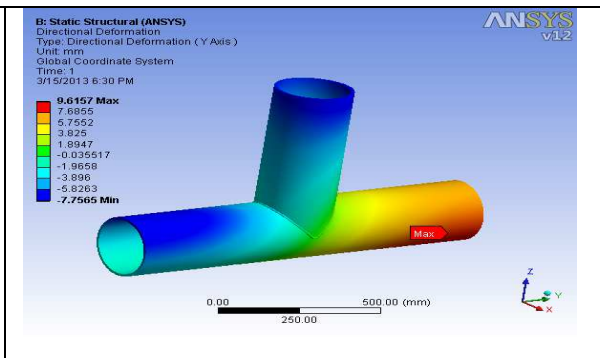


Figure 20: Normal Stress Along Z-Axis in the Pipe Joint

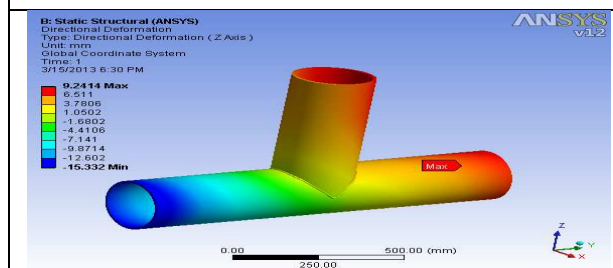




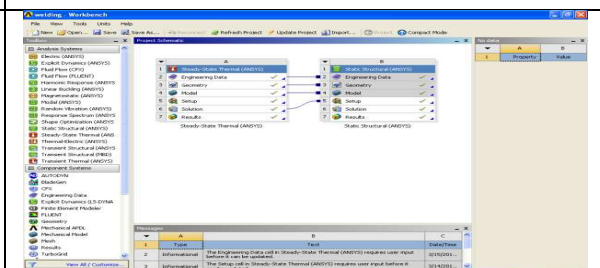
**Figure 21: Directional Deformation Along X-Axis in the Pipe Joint**



**Figure 22: Directional Deformation Along Y-Axis in the Pipe Joint**



**Figure 23: Directional Deformation Along Z-Axis in the Pipe Joint**



**Figure 24: Coupled Field Analysis for Welding**

The below table gives the comparison between the results of the pipe joints at different temperature

Temperature & orientation: 45°C & 2000°C (Horizontal pipe & Vertical Pipe)

**Table 5**

Max/Min Values	Total Heat Flux (W/mm <sup>2</sup> )	Equivalent Stress (MPa)	Normal Stress (MPa)	Shear Stress (MPa)	Directional Heat Flux (W/mm <sup>2</sup> )	Directional Deformation (mm)
Min.Value	3.3374 e-15	0.12544	-120.84	-48.038	-1.0473	4.5049
Max.Value	2.5109	237.17	232.44	99.263	2.3177	9.2414

## 7. CONCLUSIONS

By conducting analysis with weld parameters of the heat flow, which are calculated and are used as input for the analysis in arc welding of pipe joints. By the analysis using above cases it is observed that Total heat flux, Directional heat flux, equivalent stress, Normal stress, Shear stress are increased at the welded region in the joint. The Stress value is minimum of is 0.12544Mpa increased to 232.44MPa at 2000°C which indicates that the increase in temperature increases the stress with decrease in the joint efficiency. Steady state thermal analysis is carried out to estimate the residual stresses. The shear stress impact is less compared to normal stress. The Directional deformation is more in z direction compared to X and Y directions. The directional heat flux is more impact on X axis. The high temperature leads to increase in stresses in the joint. The temperature reduction methods are to be followed to reduce the effect.

**Future Scope of Work:** In future, the same model can be generated using another type of Welding processes like Friction stir welding, laser beam welding etc and analysed using ANSYS workbench.

## 8. REFERENCES

1. SK. Yeoh presented his research on the 'behaviour of Tubular T-joints that are subjected to combined loading' published by McGraw-Hill in the year 1995.
2. L.Sundar presented a paper on 'butt joints through manual metal arc welding' in 2006.
3. S. Shengzhi found the results of 'static strength of lap joints' in 2011
4. Vallance, Alex; Doughtie, Venton Levy, 'Design of machine members', 4<sup>th</sup> edition; published by McGraw-Hill in the year 1964. Page-(543-562).
5. Smith, William F.; Hashemi, Javad, 'Foundations of material science and engineering', 4<sup>th</sup> edition, published by McGraw-Hill, ISBN 0-07-2953586.
6. Cary, Howard B; Scott C.Helzer, 'Modern Welding Technology', published by Pearson Education in 2008.
7. Hicks, John, 'Welded joint design', published in Industrial press in 1999.
8. LincolnElectric, 'procedure Handbook of arc welding' in 1994.
9. WemanKlas, 'Welding processes Handbook', published by CRC Press LLC in 2003.
10. [www.millerwelds.com](http://www.millerwelds.com) – Images of arc welding and welding technology.
11. Wikipedia – Tubular T-joints design.
12. CATIA V5 – Design of Tubular T-joint.
13. ANSYS 10.0 Workbench Tutorial ANSYS, Inc.275 Technology Drive Canonsburg, PA 15317.