# CHAPTER 1
# INTRODUCTION

## 1.1 Background of Study

Residual stress is a type of stress that remains after the original cause of the load (external force/thermal gradient) has been removed. The stress remain along the cross sectional area of the structure, even without the external cause. Residual stresses occur due to many reasons. It can possibly occur due to inelastic deformation and heat treatment. Heat source, which comes from the welding arc that varies from the temperature of 800 $^0$C to 1000 $^0$C, may cause localized expansion at the region of weld, which is taken up by either the molten metal or the placement of parts being welded. After the welding process end, some area that was affected by the heat, called heat affected zone (HAZ) experience uneven cooling compared to other region, creating a different phase transition that leaves residual stresses there.

Residual stresses are more difficult to predict than the in-service stresses on which they may come together with. These include the axial stress, hoop stress and radial stress that may come up along with residual stress component. Simulation using ANSYS to predict the behavior of residual stresses will assist the personnel involved to observe the trend of the stress concentration spot.

Heat affected zone (HAZ) is a region of the base material, which had its microstructure and properties altered due to the thermal load which include welding, heat treatment or intense cutting environment. The change in base material may include the change of microstructure and phase transition after being applied with severe heat. The extent and magnitude of the effect is mainly affected by the thermal diffusivity that is dependent on type of materials. If the diffusivity is high, the cooling rate is high and creates smaller HAZ region, while low diffusivity lead to slower cooling, result in a bigger HAZ region.

## 1.2 Problem Statement and Motivation of Study

Welding process induce a lot of heat input to the material at the affected region. Since heat is applied, concerns on the formation of residual stresses due to thermal load arise. Residual stress due to thermal load might show up upon the HAZ of the base material. Stainless steel pipelines that are used widely in oil and gas industry have been selected to become the subject of experiment. Since the pipelines are materials that are always connected by welding, there is concern on how the residual stresses behave in the different environment of oil and gas industry. The pipelines may be put on the sea beds that are exposed to the corrosive environment of saltwater. So, there is a need to simulate the residual stress activity at the welding region especially at HAZ which is caused by the thermal load. ANSYS will be used to simulate the weld region in 3D. Then, the HAZ of the stainless steel can be observed and analyzed.

## 1.3 Objective

1- To investigate the behavior and trend of residual stress due to thermal load in stainless steel pipeline
2- To observe the stress concentration spot on HAZ and the area of weldment
3- To simulate the welding region using ANSYS in 3D by using different heat input as the variable and to observe the effect of it on the residual stresses formation on the stainless steel pipelines

## 1.4 Scope of study

The analysis of residual stresses due to thermal load will be on the stainless steel material. The heat variable from the welding process will be set at the melting point of the material and will be moved up for the next test. The type of welding selected is the SMAW welding process which is widely used to weld pipelines. The simulation will be conducted using ANSYS software.

# CHAPTER 2
# LITERATURE REVIEW

Fusion welding is a joining process commonly used in many industries. Among them are construction, naval, steel bridges and oil and gas structure. Welding is preferred as it is a very efficient joining process. Among the advantages of using welding as the joining method is high efficiency, simple and easy, flexible and low in cost. However, when structures are manufactured by the mean of welding, there is a non-uniform temperature distribution created during the process. Initially, this temperature distribution caused a rapid thermal expansion and followed by the thermal contraction in the weld region and other area mainly HAZ. This generates plastic deformation and residual stresses in the welded area when it is cooled.

In this case of welding, there is a concern of welding on the super high strength steel. Steel that has ultimate tensile strength exceeding 1200 MPa is regarded as super high strength steel. This type of steel also has high impact toughness but the weld crack easily formed in the HAZ region and caused the possibility of brittle fracture in the welded zone when it is in the service. HAZ region is always given close attention by many investigators. The welding joint of this steel also experience different weld thermal phase, which cause change of microstructure in the welded affected region, mainly at HAZ. At present, the research on microstructure of steel ranging to the super high strength steel is not really common. There is nearly no welding research on this type of steel also. So, this is why alternative methods that can investigate this problem should be initiated especially for the welding process.

The control on the weld heat input has been discussed in many researches. According to D.Roylance [1], the weld heat input must be at the range where the molten metal is capable to constantly dissipate the heat for cooling process without severe uneven cooling. By applying the heat input under the range of 20 kJ/cm, the metal microstructure will change in a ductile phase. If it is higher, the tendency of plastic distortion happening is high and the existence of brittle fracture could happen. It has been proved that the larger the weld heat input, the longer the cooling time and thus, it is easier for the deterioration of the metal resulting in much severe residual stresses. A method to predict other super high strength steel welding behavior must be created.

Several experimental methods have been conducted to test and analyze the residual stresses and microstructure behavior of the material. There are several non destructive technique and destructive technique for directly measuring the residual stresses which have been developed inside a specimen. These techniques include X-ray diffraction method, neutron diffraction method, layer- removal method, sectioning method, ultrasonic and magnetic method and also holes drilling method. However, the applications of these various experimental techniques are limited by either their cost or accuracy. Method like x-ray diffraction is very accurate but also high in cost. There is a demand for an alternative method to verify these issues that can provide balance between cost and at the same time the accuracy of the results obtained.

Also, birth and death technique is utilized in this simulation. The reason being for using this approach is to simulate the welding process exactly to the real welding. Birth and death technique simulate how the filler metal deposited and joined together at the instantaneous welding spot or region. By this way, the heat input would be assigned independently to each of the selected elements or nodes so that the weld heat input would act as a single load at any given time. In this case, the value of the weld heat input would be varied to study the effect of different value of the weld heat input. By applying this method, the condition of exact welding process would be handy, and thus help create more reliable results. The birth and death technique has been used frequently in a thermo-mechanical analysis for welding related case study in ANSYS or any other finite element software. This includes software such as ABAQUS and other.

According to Shu Guo, K. Him Lo, Benjamin T. A. Chang [2], by using scanning electron microscope, it was found that when the weld heat input (E) ranges from 9.2 kJ/cm to 18.6 kJ/cm, there were a lot of dimples on the HAZ impact fracture of HQ130 steel. It also exhibits an obvious ductile fracture. When the heat input is elevated to 26 kJ/cm, the fracture morphology in the HAZ was a quasi-cleavage fracture that has a river pattern features. Judging by this pattern, it is identified that the crack propagation would be severe at any spot along the line. It showed obvious brittleness fracture feature as a much lower impact energy was observed compared to the others that have been applied with much lower weld heat input.

A proper welding method using SMAW technique has been discussed by Weman, Klas [3]. There are many important parameters that need to be looked after to obtain good welding result. Top priority in doing welding process would be the safety issue. Safety equipments must be complete to do welding job. For welding stainless steel metal, the electrode used would be 308L type. The penetration in welding stainless steel pipeline would be best to keep it at low rate. This is to prevent premature brittle crack. The current type used is alternating current (AC). In this process, the electrode connectivity must be considered as the positive electrode (reverse polarity). The suitable range of voltage that could be utilized is around 50 Amps to 150 Amps. The electrodes diameters that are suitable according to the type of weld passes and weld angles are in 3/32, 1/8 and 5/32 inches range. To produce good weld beads, several things must be taken care of. First, the angle of electrode must be within 10° - 30° from vertical axis. Arc length must be suitable so that the weld beads do not splatter.

Referring to the description by American Petroleum Institute (API) guidelines, burn through will occur when the metal beneath the weld pool can no longer withstand the pressure within the pipeline. In oil and gas industry, the precaution is to follow the guidelines to prevent the burn through accident. The guidelines stated that " burn through will not occur as long as the temperature level on the inside of inside surface never exceeds 980°C " (API 1995). This guidelines stress the important of weld heat input in welding so that the process can take place safely without any failure or incident. Influence of the existing thermal stresses and mechanical stress due to previous work on the pipeline should also be examined carefully. The effect caused by the inner pressure of in service pipeline is also need to be judged by the way it should be for a safe welding condition.

Back in 1930, a numerical method using Fourier's heat transfer equation has been developed to study the temperature distribution profile for a butt welding infinitely long plate (Parmar, 2002). In the model itself, the governing heat transfer equation has been solved in quasi stationary condition. The results were not accurate due to many assumptions made for the calculation. The result between the numerical method and experiment observation method yield several errors. Hibbit and Marcall (1973), have used finite element method to overcome this numerical method done before. They also studied the effect of temperature gradient on stress distribution on butt welds.

Godlak (2005) has used Double Ellipsoidal Power Density Distribution method to model the heat input during welding process. Brickstad and Josefson (1998) has used ABAQUS software and succeed in obtaining the residual stress in multi pass butt weld. Vahiki Tahami *et al* (2009) also used element birth and death technique to estimate the due weld residual stress in a 3D finite element base model.

# CHAPTER 3
# METHODOLOGY

In order to complete this simulation project, several tasks and method have been utilized and practiced. In general, the methods can be classed into three major steps. The first one would be the understanding and the interpretation of the literature review regarding the topic. The first step is crucial in order to plan the project. Here, the details of the projects would be identified and judged. The second step would be the execution step. In this step, the simulation is done. Two analysis would be performed which are thermal analysis and structural analysis. This step is important to simulate the welding process and also to determine the residual stresses. The last part would be the data interpretation. This step would be the process of the data comparison between several data sets and also with the results from the literature.

## 3.1 Planning

The simulation would be the welding process that takes place on a cylinder. This simulation is to simulate the welding process that happened on the pipeline in oil and gas industry or any other industry that experience this kind of situation. The welding method chosen is the SMAW (Shielded metal Arc Welding). This kind of technique is used frequently in welding pipeline. The material chosen is Stainless Steel 316. These are the basic parameters for this simulation.

### 3.1.1 Welding

SMAW process is a very common type of welding method that have been practiced in connecting metals together. The capability and the simplicity of the process are good. Using this method, metals are welded together with good quality. In oil and gas industry, most of the pipelines are connected using this method. The concept of SMAW method is to utilize the heated arc that was created due to huge potential difference between the electrode tip and the metal surface. When this happened, the arc is capable to melt the metal to join them together. By far, the connection quality resulted from this welding process can withstand high load and force that the pipeline need to withstand during its useful life. Figure 3.1 below shows how the arc penetrates the metal to form a pool of molten metal. This is where the two metals are connected.

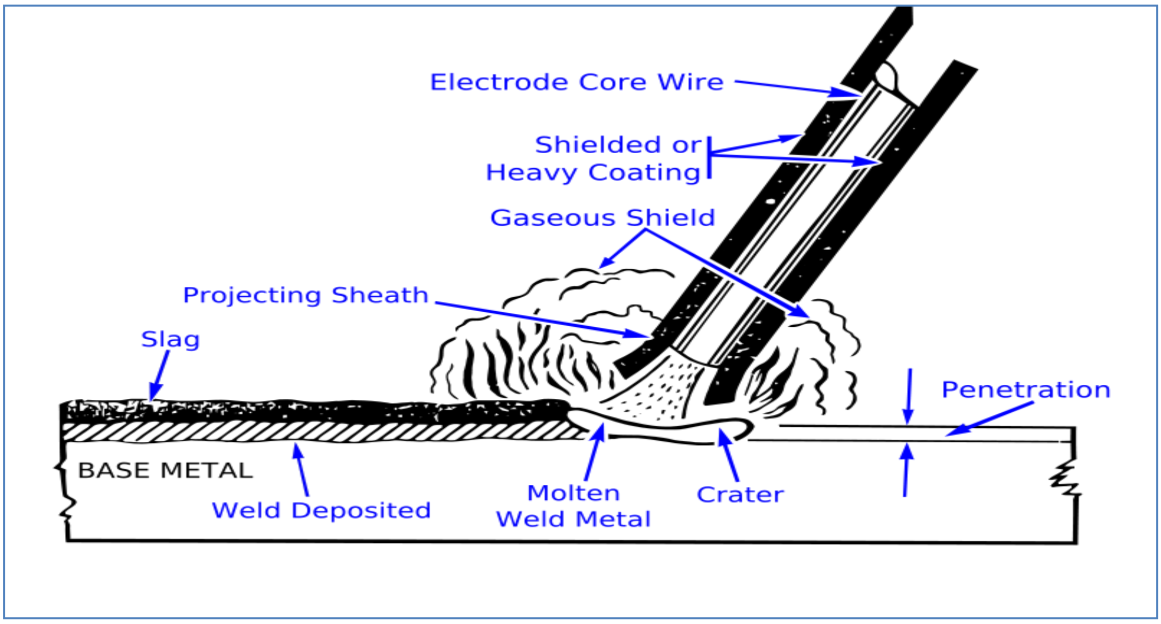


**Figure 3.1: Typical SMAW welding in progress**

### 3.1.2 Thermal and Structural Properties of 316 Stainless Steel

The parameter and details of the materials (Stainless Steel 316) can be referred to the tables below.

**Table 3.1: The thermal properties of 316 Stainless Steel**

| | | | | | | | | |
|---|---|---|---|---|---|---|---|---|
| Thermal Conductivity ( W/m.K) | 17.14 | 19.68 | 22.22 | 24.76 | 27.3 | 29.84 | 32.38 | 34.92 |
| Temperature (K) | 473 | 673 | 873 | 1073 | 1273 | 1473 | 1673 | 1873 |

| | | | | | | | | |
|---|---|---|---|---|---|---|---|---|
| Enthalpy ( J ) | 1.75E +09 | 2.59E +09 | 3.48E+ 09 | 4.41E+ 09 | 5.40E+ 09 | 6.49E+ 09 | 7.54E+ 09 | 8.03E+ 09 |
| Temp (K) | 473 | 673 | 873 | 1073 | 1273 | 1473 | 1673 | 1873 |

**Table 3.2: Structural properties for 316 Stainless Steel**

| | |
|---|---|
| Yield Strength (MPa) | 245 |
| Tensile Strength (MPa) | 515 |
| Poisson Ratio | 0.3 |
| Young Modulus (GPa) | 193 |
| Melting Temperature (K) | 1648 - 1672 |

**Table 3.3: Dimension of the pipeline model**

| | |
|---|---|
| OD (m) | 0.4 |
| ID (m) | 0.36 |
| Thickness (m) | 0.02 |
| Length (m) | 0.8 |

### 3.1.3 Pre-Analysis

The analysis will start with a thermal analysis using ANSYS. The aim is to obtain the thermal distribution of the welded line and the HAZ region. This method will be repeated with different welding heat input to observe the effects. After the required results are obtained, then the simulation will proceed to the structural analysis. Here, the residual stresses due to the thermal distribution would be performed. The result would be plotted and compared to see the effect of the different weld heat input value upon the residual stresses in the metals

## 3.2 Modeling

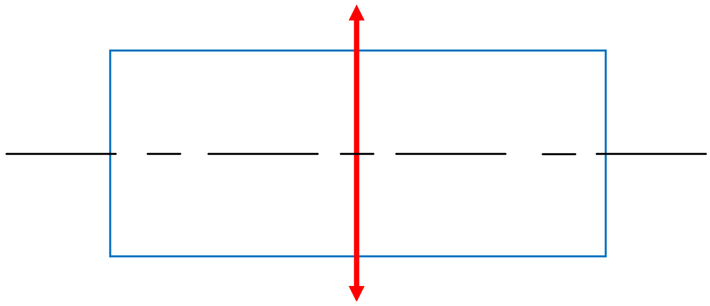

**Figure 3.2: Pipeline model shape configuration**

The line at the middle of the pipe length indicates where the welded line is situated in Figure 3.2. The welding process will be simulated by applying birth and death technique along the circumference of the middle line. The reason for choosing the middle line is for the ease of analysis. Also, HAZ region can be observed in symmetry by utilizing this configuration.

## 3.3 Assumptions

Several assumptions have been considered in order to perform the simulation. These assumptions are considered for the ease of the related analysis and calculation. Also, several assumptions are made so that the simulation will be smooth in ANSYS working plane. Assumptions made are as follows:

i. Convection boundary condition is applied on the outer surface of the model only
ii. Heat source is moving at constant speed, which is 1 cm/s
iii. Material properties of the filler and electrode are the same with material properties of the welded metal
iv. Displacement of metal does not affect the temperature distribution
v. Initial temperature will all be set to 300K
vi. Heat transfer coefficient, *h* is set to be 25W/m.K
vii. Radiation heat transfer is neglected
viii. Temperature distribution along the element is constant at all surfaces of the element

### 3.4 Flow Chart

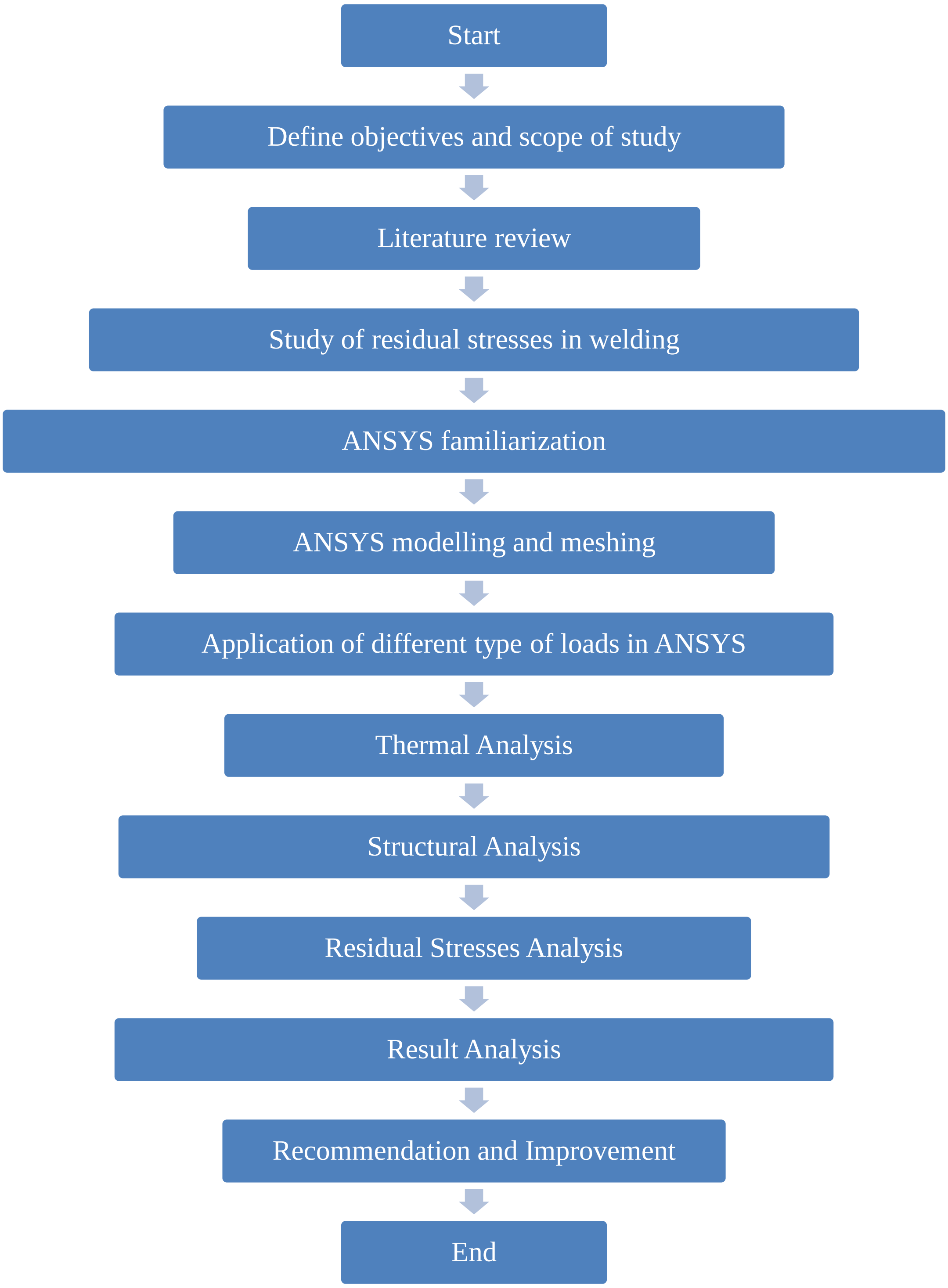


**Figure 3.3: Flow chart of overall steps taken in completing the project**

## 3.5 Thermal Analysis

The simulation will be start with the material defining process. The metal tested is Stainless Steel 316. The element selected would be SOLID70. The parameters and details about the Stainless Steel 316 would be inserted into the preprocessor. The data inserted include the properties like thermal conductivity, heat capacity, density, enthalpy, emissivity and thermal expansion coefficient. Basically, all the thermal dependent properties are included.

Then, the pipe model would be created. The hollow cylinder shape is selected. The parameters of this cylinder are according to the parameters that have been defined earlier in the preliminary data. Then the model would be meshed. Mesh density would be determined by the selection of the user and how the load would be applied later.

After that, the process continues to solution phase. The analysis would be transient analysis. The surface heat convection would be applied on the area of the cylinder. Then, heat flux will be applied on the weld pool. Birth and death technique would be utilized here to simulate how the real welding process happens. Birth and death technique help users to simulate how the metals connected together when the heat flux is applied on them.

The process of applying the heat flux would be repeated along the welded line. After all the loads have been applied, then solve command would be initiated. The result required would be the thermal/temperature distribution along the selected path along the welded line and HAZ region. These results of temperature distribution would be saved in .rth files extension to be used in the later structural analysis.

## Define Element

The element that is used for thermal analysis is SOLID70 type. It has a 3-D thermal conduction capability. The element has eight nodes with a single degree of freedom, temperature, at each node. The element is applicable to a 3-D, steady-state or transient thermal analysis. SOLID70 is a simple element to be utilized yet can produce reliable elements results. The element selected is shown in Figure 3.4.

Graphical Method **: Main Menu>Preprocessor>Element Type>Add/Edit/Delete**

Command Script **: ET**,1,70,,,1 ! SOLID70 option selection

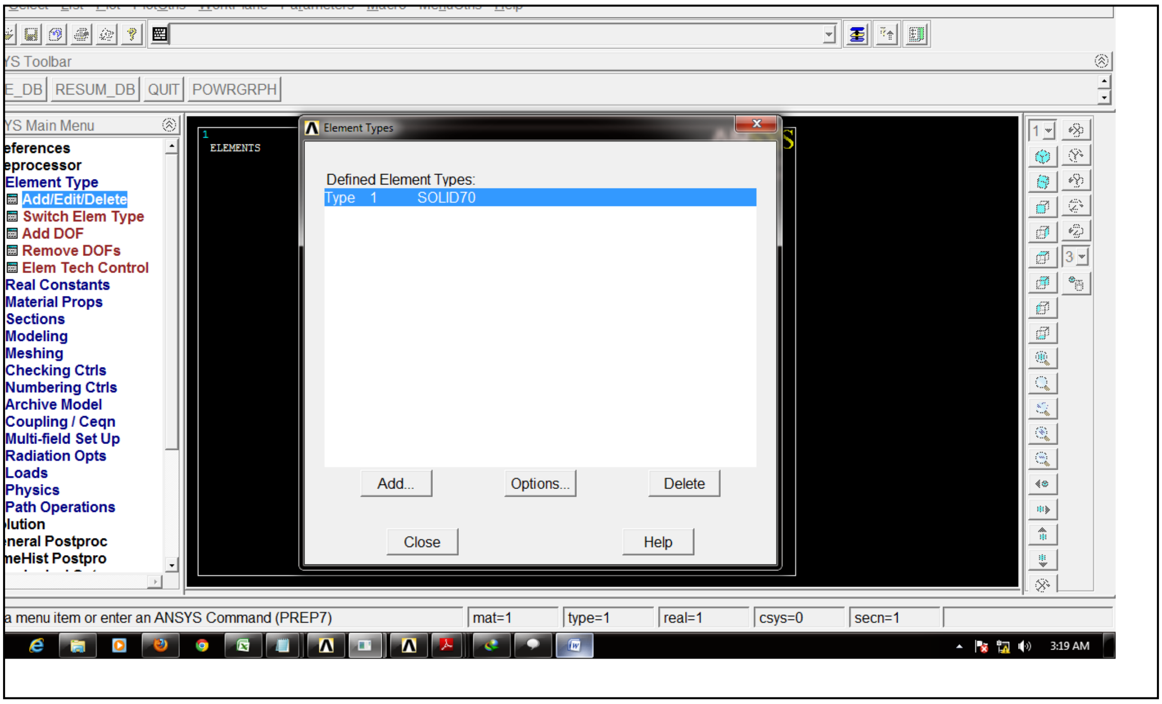


Figure 3.4: Define the elements

## Define Material Properties

Material properties information needs to be keyed in into ANSYS database are shown by Figure 3.5, to define the materials that are simulated. Data like thermal conductivity, density, heat capacity and enthalpy value are the defined values in this process.

Graphical Method **: Main Menu>Preprocessor>Material Props>Material Models**

Command Script **:** mp,kxx,1,24.2        ! Thermal conductivity (kxx) = 24.2 W/m.K

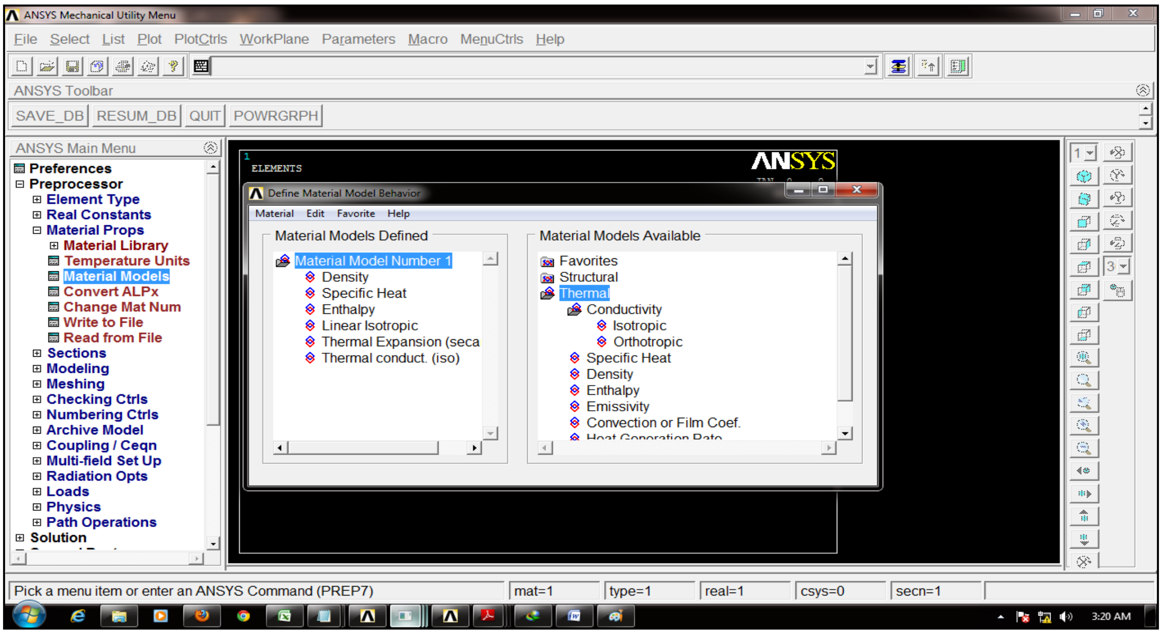


Figure 3.5: Define material properties

## Create Model

Model used is a cylinder shape 3D object. This is the shape of pipeline. Typical cylinder shape is defined in modeling command with full extrude option through 360$^{o}$ to create a full cylinder. Figure 3.6 show the exact model of the pipeline.

Graphical Method **:**
**MainMenu>Preprocessor>Modeling>Create>Volumes>Cylinder>By Dimensions**

Command Script **: CYLIND**, *0.2, 0.18, Z1, 0.8, 0, 360*        ! Cylinder modeling with outer radius = 0.2 m, inner radius = 0.18 m, length = 0.8 m with full extrusion of 360$^{o}$

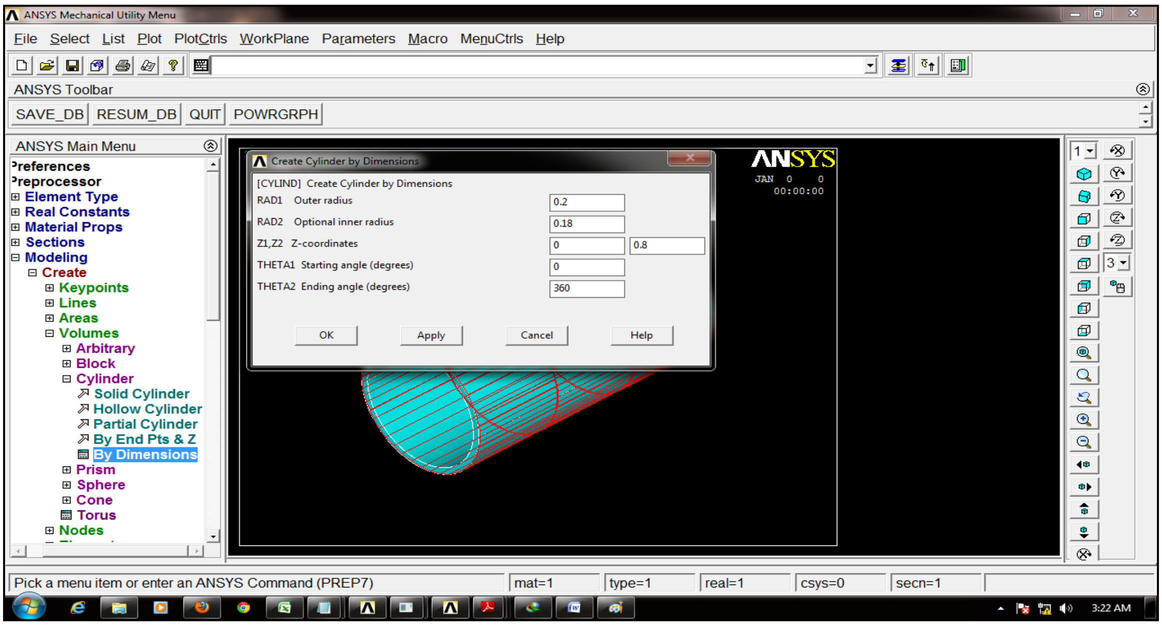


**Figure 3.6: Define the cylinder dimension**

## Meshing

The cylinder model is meshed with coarse mesh preference. This is due to the ease of operation and loading process in the next process. To obtain the mapped shape of mesh in cylinder, volume sweep mesh command has been used. By not using free mesh, the process is simpler and the orientations of meshes are also in standard. Figure 3.7 shows the meshes of the pipeline model using ANSYS volume sweep command.

Graphical Method : **Main Menu> Preprocessor> Meshing> Mesh> Volume Sweep> Sweep**

Command Script : **VSWEEP**, *VNUM, SRCA, TRGA, LSMO* *! Volume sweep command, VNUM = 1*

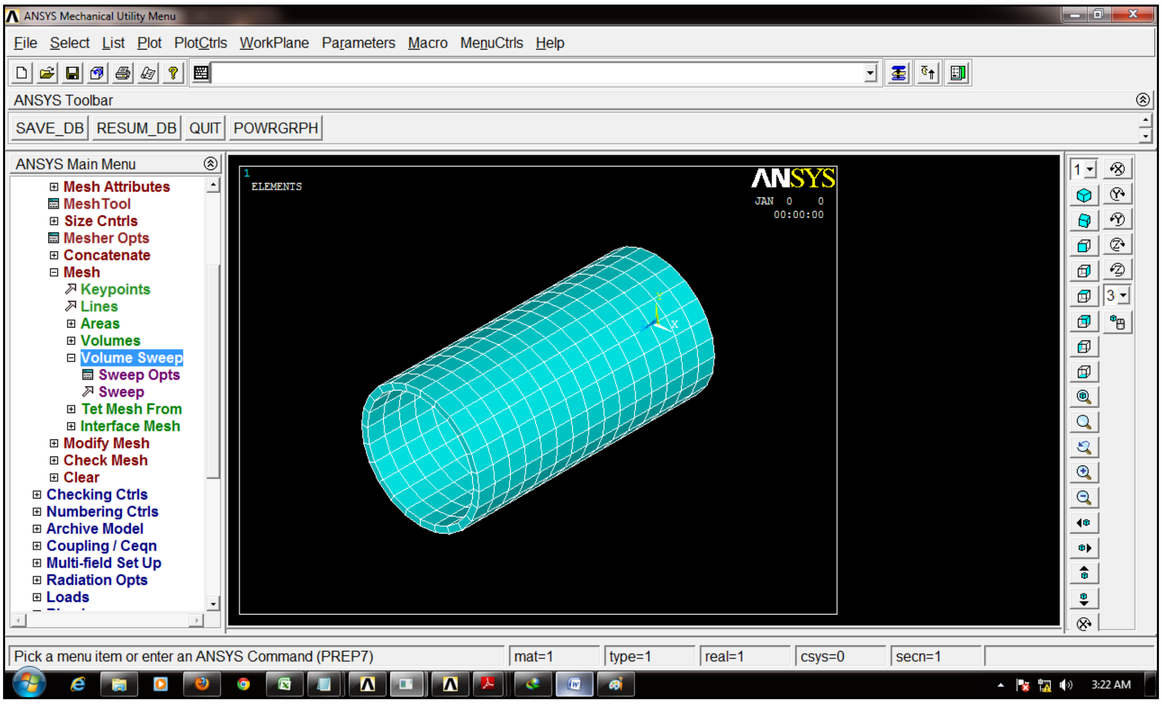


Figure 3.7: The meshes on cylinder using volume sweep command

## Analysis Type

The analysis done in the thermal analysis is basically transient analysis that is concerned with the temperature profile distribution over the time of welding. Full transient analysis mode is turned on by default. To obtain good result with birth and death application, the Newton-Rahpson mode with full mode is also turned on. Transient analysis option is chosen as shown in Figure 3.8.

Graphical Method : **Main Menu>Preprocessor>Loads>Analysis Type>New Analysis**

Command Script : **ANTYPE**, *Antype*, *Status*, *LDSTEP*, *SUBSTEP*, *Action ! Antype = 4/TRANS and Status = NEW*

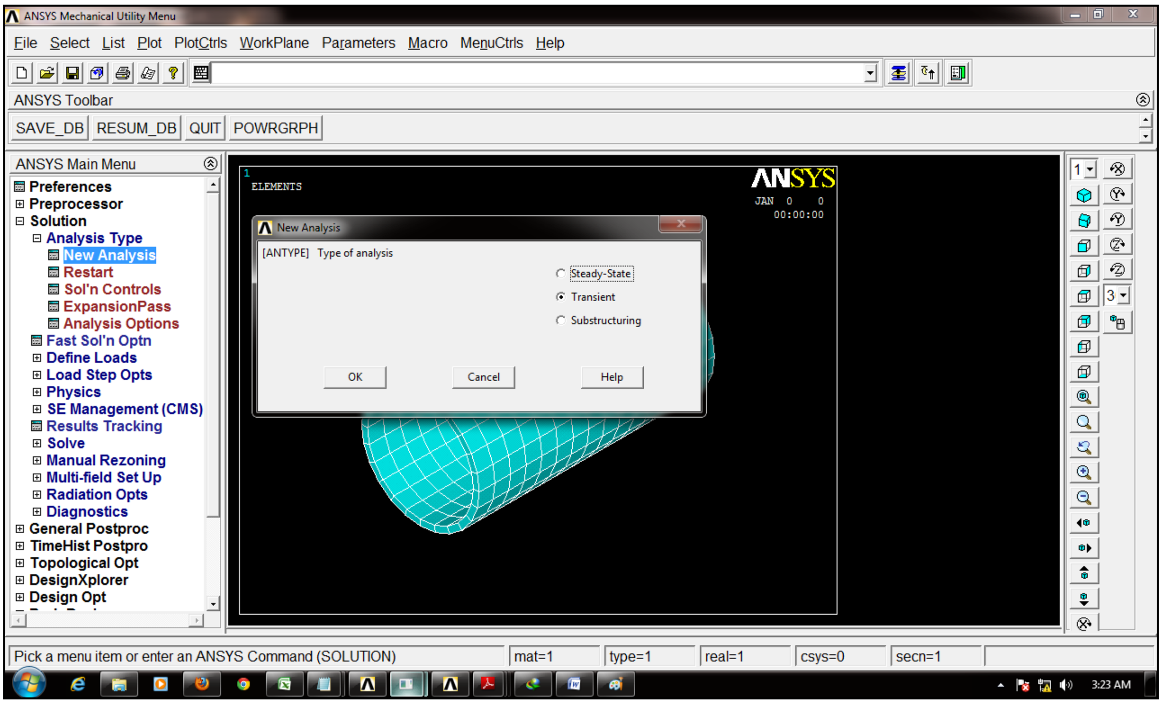


Figure 3.8: Choosing analysis type in thermal analysis

## Define Constraint/Initial Condition

There are two initial condition parameters that have been applied onto the cylinder model. The first one is ambient temperature. The value of this is set to be 300K with all elements is experiencing 300K temperature when the analysis is initiated in the SOLVE command later. The other initial condition is the convection surface load. This is applied on the outer are of the cylinder model shown in Figure 3.9. ANSYS read initial condition parameters for any analysis. Without initial condition defined, ANSYS will produce errors and random results. User must know well about the related constraint for the analysis conducted. As for this analysis, the radiation effect is not taken into consideration

Graphical Method :

**Main Menu>Solution>DefineLoads>Apply>Thermal>Temperature>Uniform Temp**

Command Script : **TUNIF**, *TEMP* *! Temp = 300*

Graphical Method :

**Main Menu>Solution>Define Loads>Apply>Thermal>Convection>On Areas**

Command Script : **SFA**, *AREA, LKEY, Lab, VALUE, VALUE2 ! Area = outer area only, VALUE = 25W/m.K and VALUE2 = 300K which is bulk temperature*

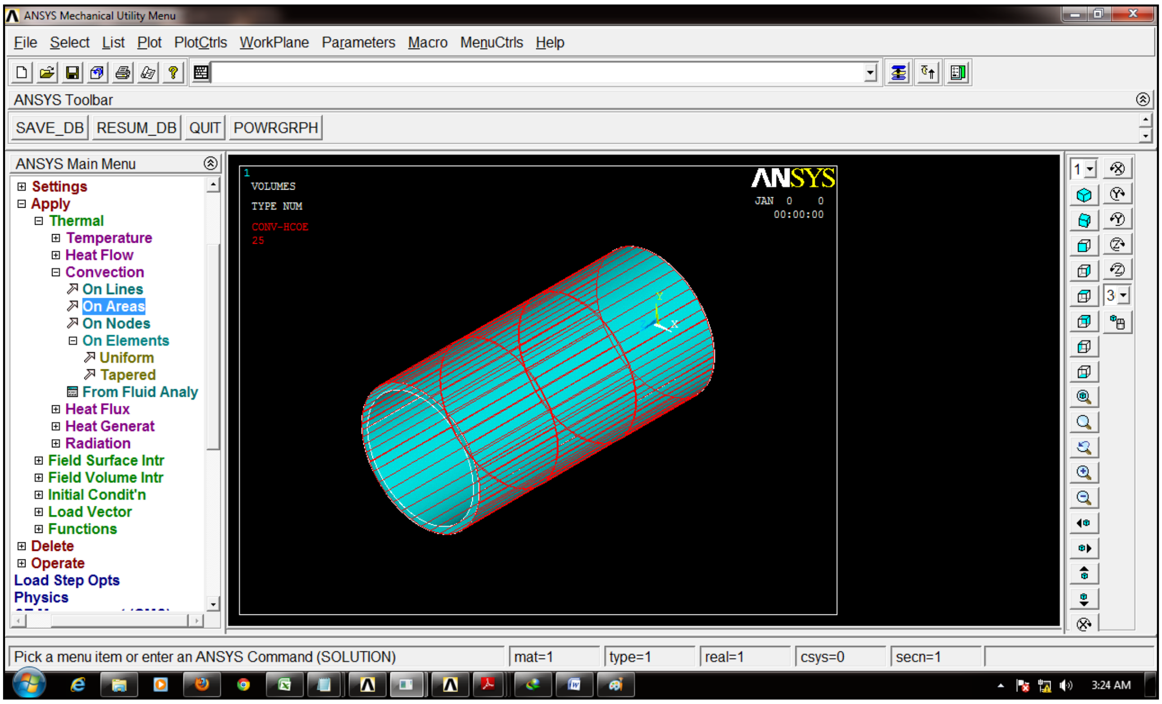


**Figure 3.9: Applying convection surface load boundary condition**

## <u>Define Load</u>

Weld heat input in this simulation is applied as the surface load on element. The type of load chosen is heat flux. Heat flux is applied on the selected element on the welded line shown in Figure 3.10. Heat flux need to be defined on the elements through birth and death approach. By using this approach, the elements which are not used will be issued EKILL command and the elements that need to be used will be issued EALIVE command. Three values of weld heat input used were 33 kW, 34 kW and 35 kW. These values were found from the equation of $= \eta \frac{VI}{v}$, where V = voltage, I = current, v = electrode travel speed and η = efficiency.

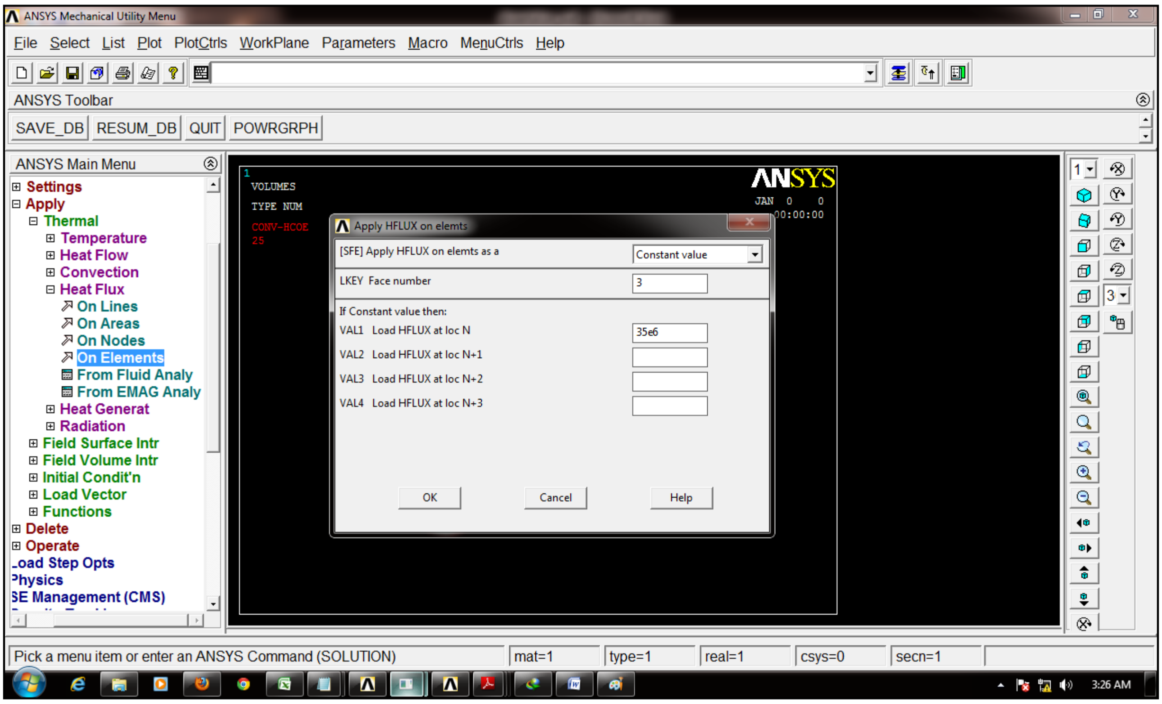


Figure 3.10: Define the loads and apply them onto the nodes and elements

## Solve

Solve command will be initiated after all the preprocessor commands needed for this analysis are completed. In this analysis, the SOLVE command is applied by two method which are by using current LS file and also increment LS file. Basically, any convergence in the result of the analysis is observed. Judging by the convergence data line, any proper modification can be made for quicker convergence. Method like refined meshes and increment of the load step for a particular load is usually utilized.

## Result Reading

The data that is anticipated here is the temperature distribution profile of the welded line and also the HAZ region. Temperature distribution in this case is shown by Figure 3.11. If any temperature distribution plotted seem to be irrelevant, the pre define loads parameters must be revised. Some cases include improper temperature distribution which shows the value of temperature below melting point of the metal at the weld pool. In case this happen, the value of heat flux must be revised, the meshes need to be improved in order to get more refined result here. Basically, the contour plot will be plotted to show the temperature distribution.

**Main Menu> General Postproc> Plot Results> Contour Plot> Nodal Solution**

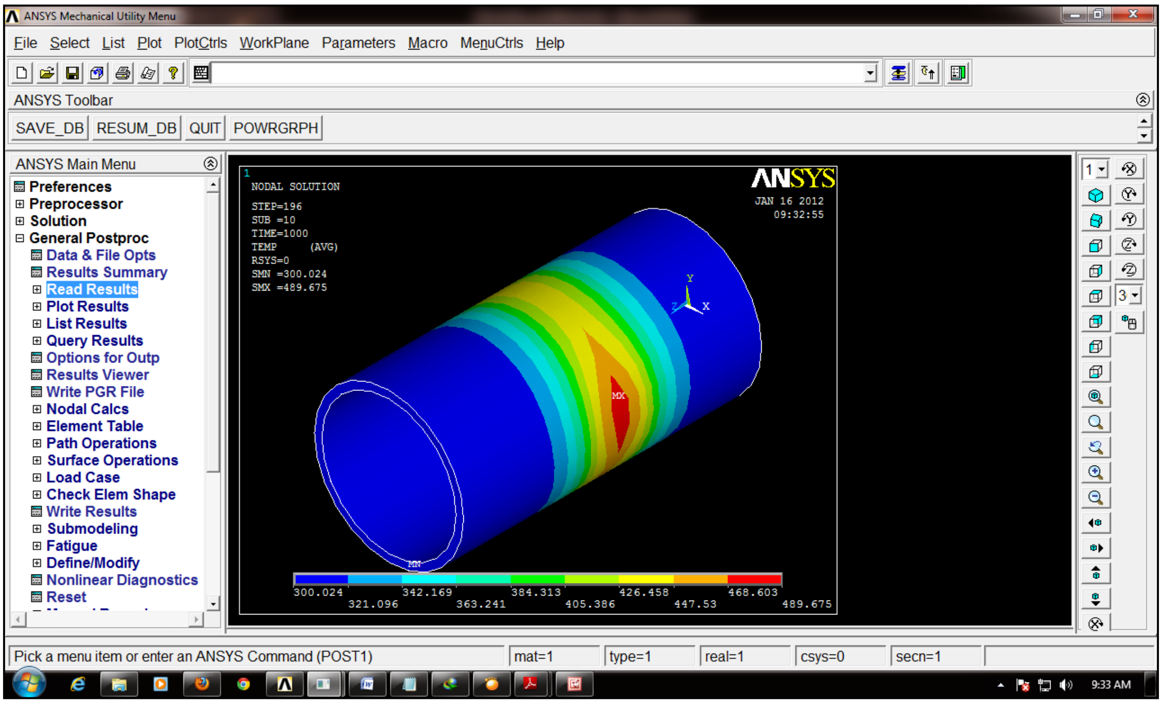

Figure 3.11: Temperature distribution of the cylindrical model

## Path Operation

Path is a method which can be utilized to collect specific data set. By using this command, user will be prompt to define path. A set of defined nodes or elements will be defined as a specific path just like the circular line shown in Figure 3.12. In this analysis, the data needed is the temperature distribution along the affected nodes which are nodes along welded line and HAZ region. Path operation will define and record the data between the starting point and the finish point. The path in this analysis is the path between two nodes, which collectively include all nodes in the welded region. By using path operation, the specific data set can be saved in external file and later on, will be issued to be used in structural analysis.

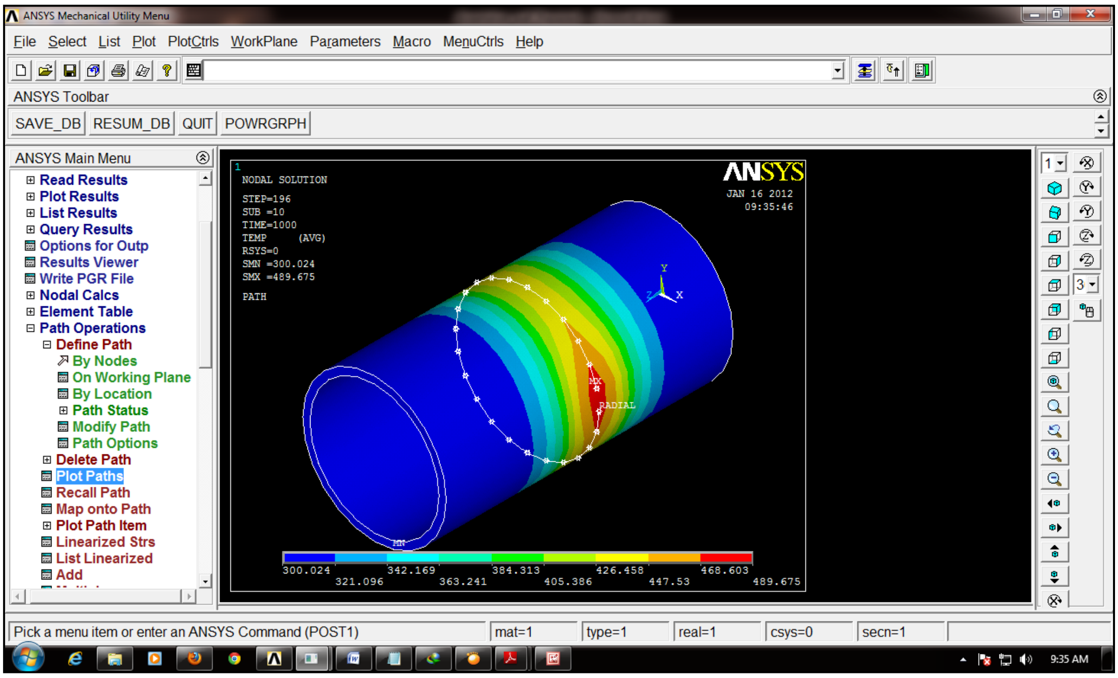

**Figure 3.12: The white line represent 24 nodes in a single nodes path**

The path defining process is by the command below

Graphical Method : **Main Menu>General Postproc>Path Operations>Define Path>By Nodes**

Command Script : **PATH**, *NAME*, *nPts*, *nSets*, *nDiv*

## Data Interpolation

After the selected data set is defined by a path, next, the data set will be interpolated in the graph according to the data type. In this analysis, the data set will be defined as TEMP for temperature. By this, the path is finally defined and will be recognized by the system as a thermal profile file. At this point, this path can be saved and used to plot graphs. This will be the one that is used in the next structural analysis simulation. Figure 3.13 shows the temperature distribution along the welded line that has been defined as a path shown by Figure 3.12.

Graphical Method : **Main Menu>General Postproc>Path Operations>Map onto Path**

Command Script : **PDEF**, *Lab*, *Item*, *Comp*, *Avglab ! Item = TEMP*

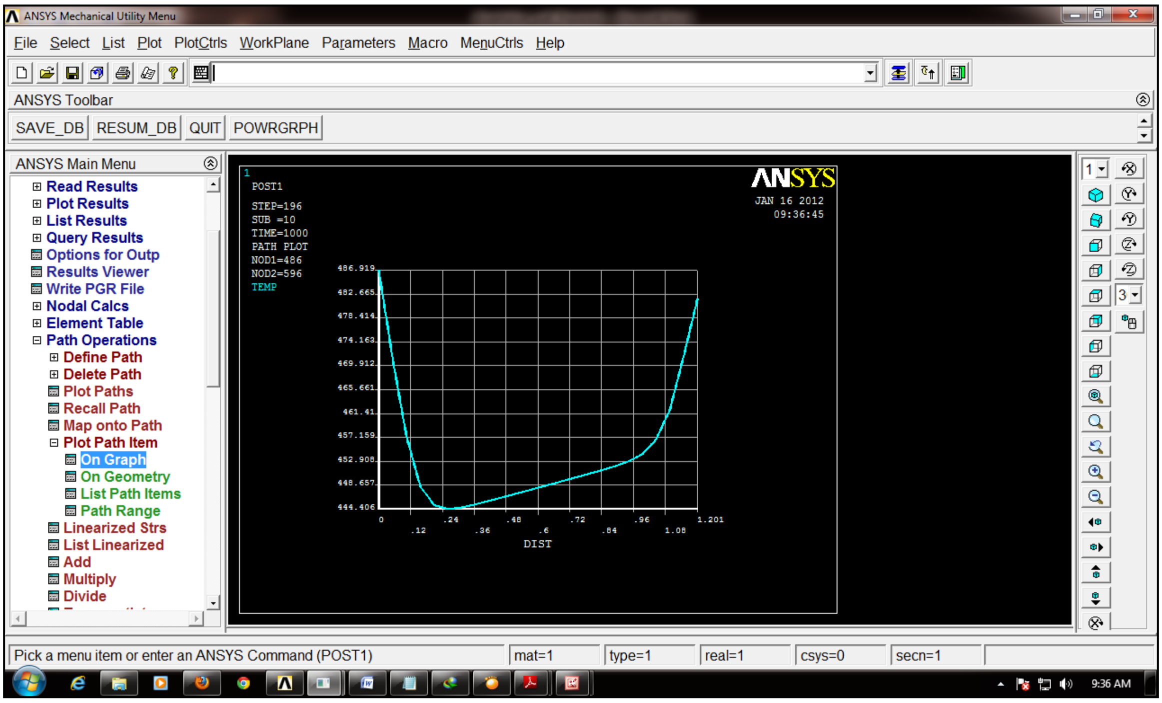

**Figure 3.13: Temperature distribution versus distance that reside along the path defined**

## Data Storage

To save the temperature profile, the path file will be saved to an external file. Select proper file and name to be saved.

Graphical Method **: Main Menu>General Postproc>Path Operations>Archive Path>Store>Paths in file**

Command Script : **PASAVE**, *Lab*, *Fname*, *Ext*, --

To plot the temperature profile for a recheck, simply choose the path and select the TEMP phrase in the command below. Up to 6 different parameters can be plotted in a single graph axis.

Graphical Method : **Main Menu>General Postproc>Path Operations>Plot Path Item>On Graph**

Command Script : **PLPATH**, *Lab1, Lab2, Lab3, Lab4, Lab5, Lab6*

Lab1 until Lab6 are the parameters that the users must define in order to plot the data in the graph. In this case, Lab1 is TEMP which stand for temperature.

**Finish**

Issue the FINISH command in the utility menu to complete the current analysis.

## 3.6 Structural Analysis

In this analysis, the additional properties of the material are added in the material model tables. The required properties are tensile strength, Poisson ration and Young Modulus. In this process, the element of the analysis would be changed to SOLID185. Then, using the same model used during the thermal analysis, the loads applied are the temperature distributions that have been obtained from the earlier thermal analysis. Using the data as the loads, the residual stresses are determined.

The residual stresses would be compared for different weld heat input. The effect of the weld heat input would be analyzed in graph and compared with the literature. If all the results obtained are significantly comparable with the literature, then this method of simulation can be accepted as reliable and practical. For this to happen, it should be noted that the simulation parameters must be as exact as the real process. By moving closer to real simulation, the results obtained would be much reliable and acceptable.

## Change Element

The SOLID70 elements that were used before need to be changed to proper structural elements in order to read and produce structural results. The elements chosen by default are SOLID185. The process is shown by Figure 3.14. SOLID185 is used for 3-D modeling of solid structures. It is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyperelasticity, stress stiffening, creep, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials. This element is superior and better than SOLID45.

Graphical Method : **Main Menu>Preprocessor>Element Type>Switch Elem Type**

Command Script : **/PREP7** et,1,185,,,1 ! element SOLID185 changes

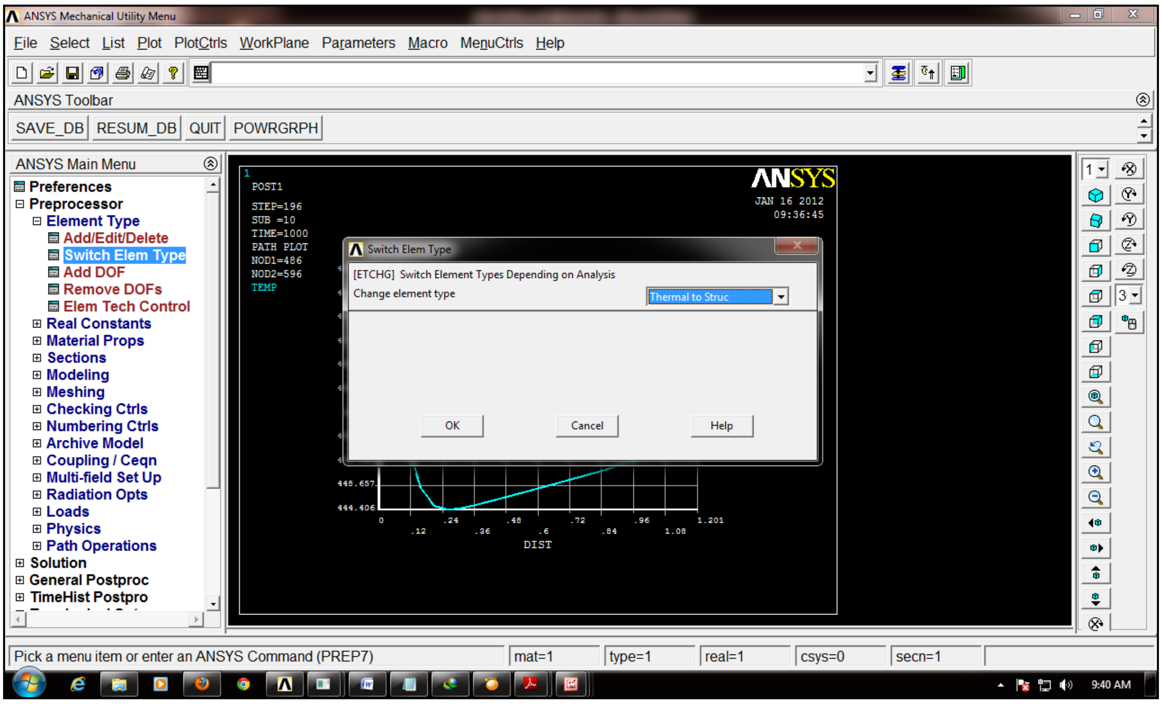


**Figure 3.14: Changing analysis environment from thermal to structural analysis**

## Define Properties

Additional structural properties need to be defined in the material properties tab. The required properties are modulus of elasticity, Poisson ratio and thermal expansion coefficient. By adding these properties, it is enough to conduct an analysis for stress computation. In Figure 3.15, the data such as Young Modulus and Poisson Ratio are inserted into the material properties tab.

Graphical Method : **Main Menu>Preprocessor>Material Props>Material Models**

Command Script :

**MP**,ex,2,10.6e6 ! Define modulus of elasticity

**MP**,alpx,2,1.35e-5 ! Define thermal expansion

**MP**,nuxy,2,.33 ! Define Poisson Ratio

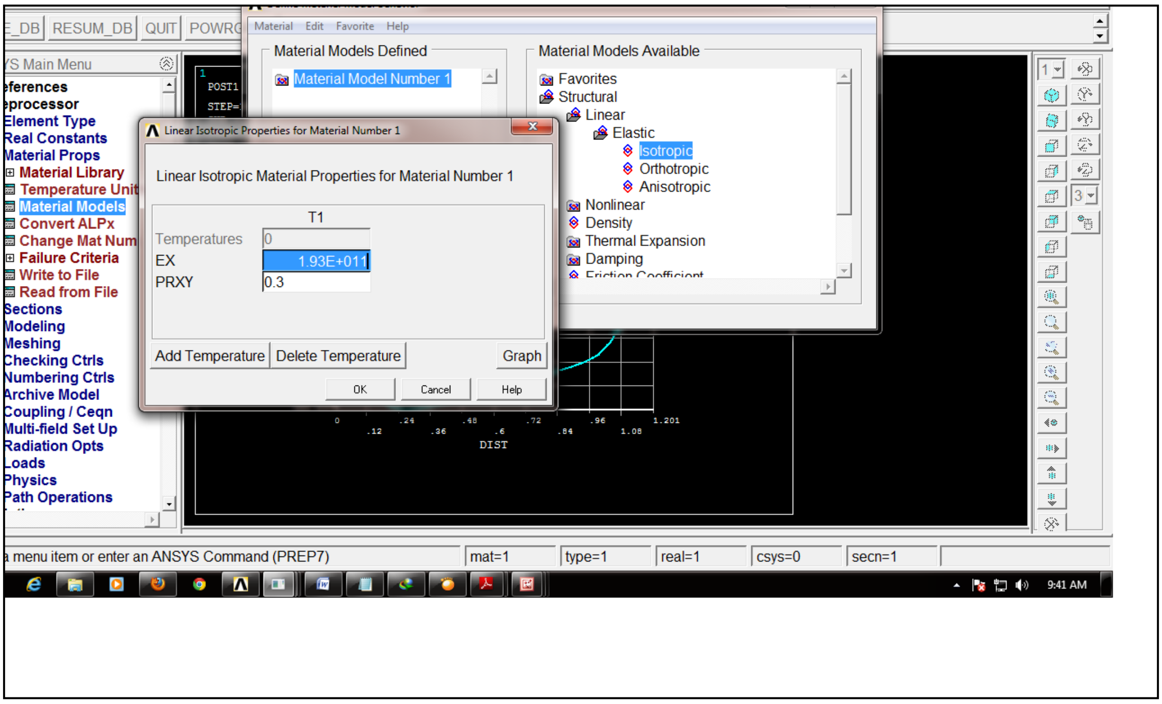


**Figure 3.15: Define the structural properties**

**<u>Specify Initial Condition</u>**

Like the thermal analysis, initial condition and constraint must be specified. For this analysis, the cylinder model will be assumed to be clamped at the two end of the cylinder. The circular surface area will be issued with no displacement command. Different way and place of imposing initial condition parameter might be considered to see the effect on the stress developed later.

Graphical Method :
**MainMenu>Preprocessor>Loads>DefineLoads>Apply>Structural >Displacement>On Areas**

**<u>Specifying Reference Temperature for Strain</u>**

This is the reference temperature for strain calculation. It should be the same as the bulk temperature of the initial condition. In Figure 3.16, the strain temperature parameter in the process is defined as 300K.

Graphical Method : **Main Menu>Solution>Define Loads>Settings>Reference Temp**

Command Script : **TREF**,300 ! Bulk temperature = 300K

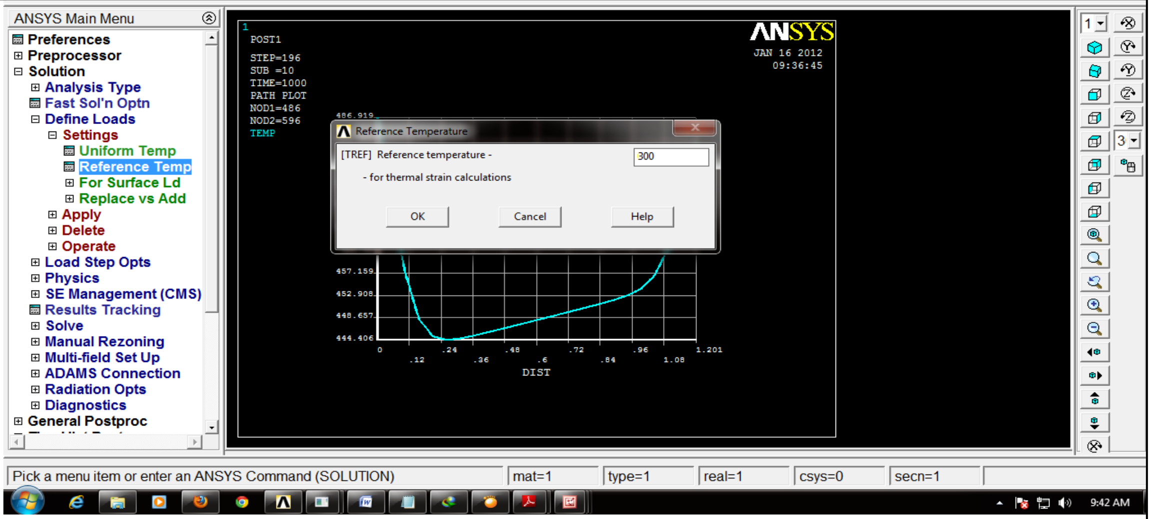


**Figure 3.16: Specify the temperature for strain calculation**

## <u>Solve</u>

This is the part where the temperature profile file that has been defined in thermal analysis operation will be used. To recall the file, the LDREAD command will be used. This command will read the temperature distribution file earlier and apply the temperature as body loads on the corresponding elements or nodes in the structural analysis. By doing this, the cylinder model is now loaded with temperature profile similar to the one captured in the thermal analysis. Note that the temperature parameters are selected under structural loading. It is a body temperature that have been affected by the heat input that cause structural changes.

Graphical Method :

**Main Menu>Solution>Define Loads>Apply>Structural>Temperature>From Therm Analy**

Command Script : **LDREAD**,temp,25,10,,,,rth ! Read in temperatures from thermal analysis run

Issue SOLVE command to initiate the structural analysis after the LDREAD command read the file without any error.

To recall the path from the file saved, the command below is used :

Graphical Method :

**Main Menu>General Postproc>Path Operations>Archive Path>Retrieve>Paths from file**

Command Script : **PARESU**, *Lab*, *Fname*, *Ext*, -- ! Lab = path name, Fname = file name

# CHAPTER 4
# RESULTS AND DISCUSSION

In structural analysis, the result reading will be mostly from the contour plot and vector plot. These two plots will show the stress behavior as well as stress vector computation among the elements in the cylinder model.

## 4.1 Stress Contour Plot

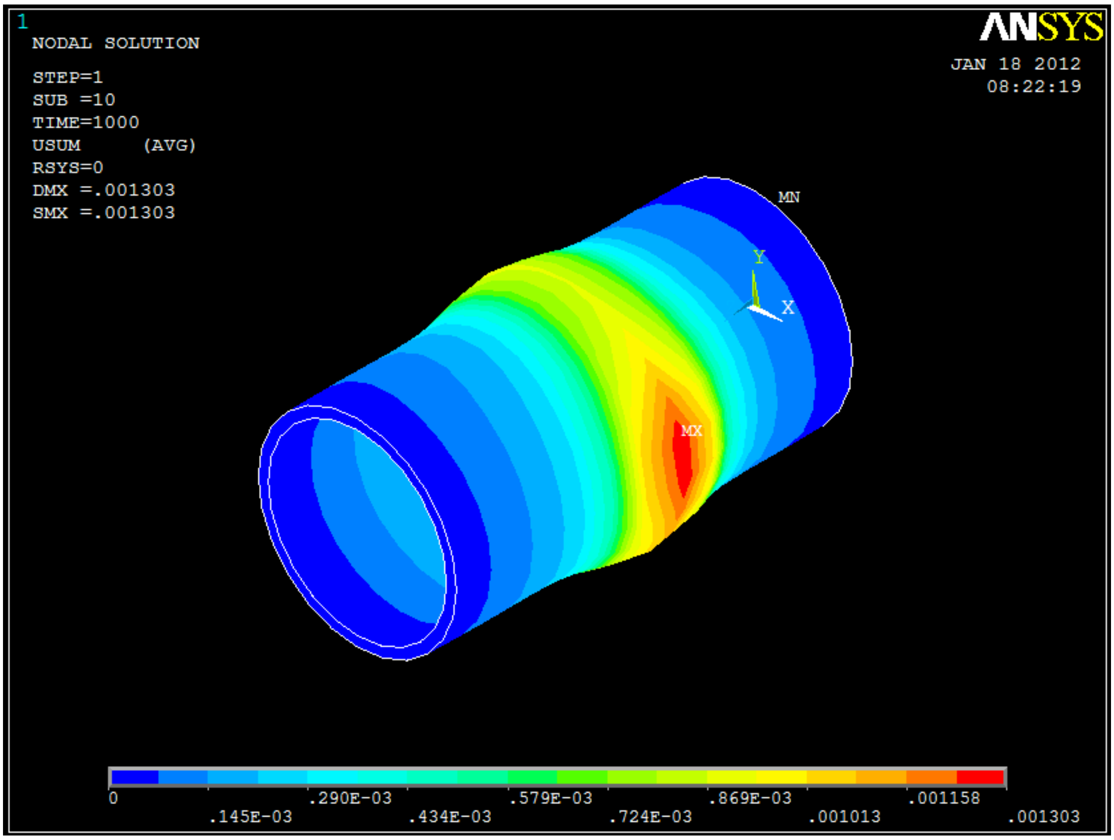


**Figure 4.1: Sum of Displacement/Distortion on the cylindrical pipeline**

From Figure 4.1, maximum displacement occurs at node 486. The maximum magnitude of the displacement is 1.303 mm. This shows clearly how maximum temperature affects the displacement of the metal. Since higher temperature creates more energy, the metal structure would gain much energy to distort weaker structure around. That is why the distortion is the most at the maximum temperature. Also, the distortion magnitude is affected by the Thermal Coefficient Expansion parameters of the corresponding materials.

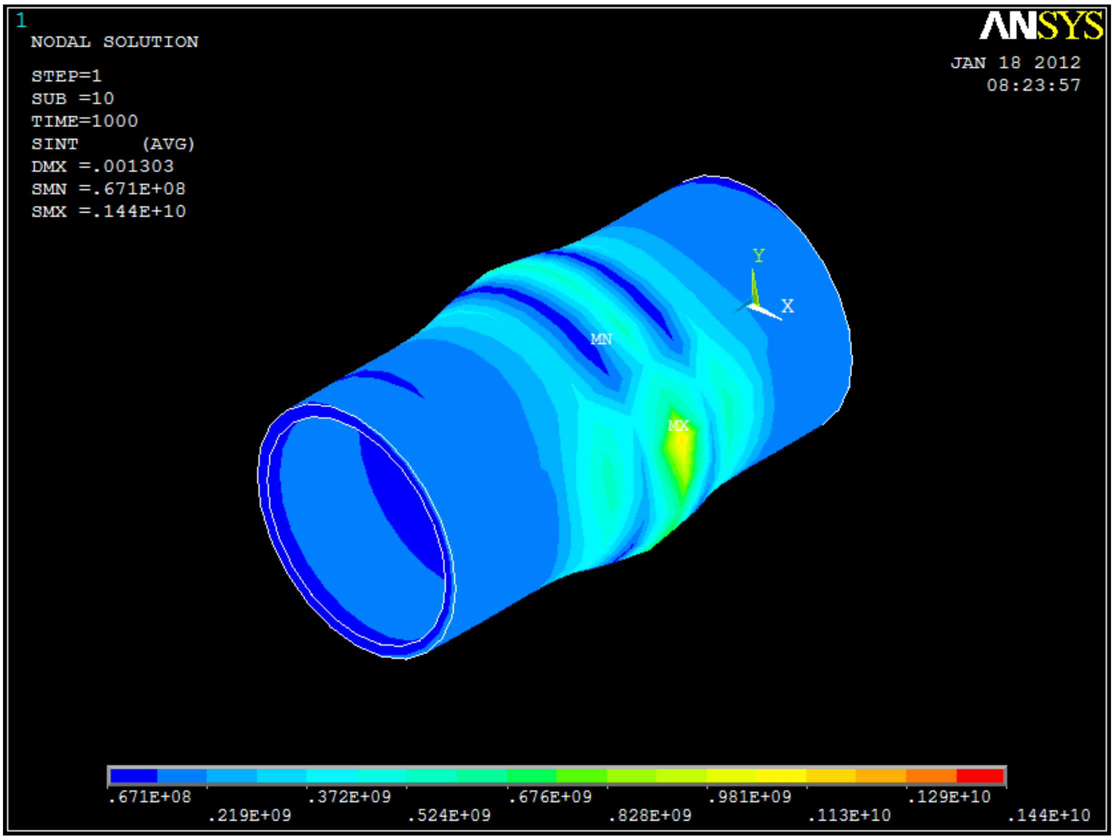


**Figure 4.2: Stress intensity plot for the cylindrical model**

Stress intensity plot shown in Figure 4.2 indicated that accumulation of stresses have been plotted at node 486. This is where all the stresses are concentrated. By this plot, stress concentration point is the point where the metal is the weakest. Any micro crack or distortion can lead to later crack propagation and also corrosion initiation. Since the spot is in the welding line, it is crucial to know that the weakest point of the pipeline is at the welding region. By knowing this, precautions such as heat relieving process and grains structure repair by the means of heat treatment could be done to reduce the stress concentration severity at this point.

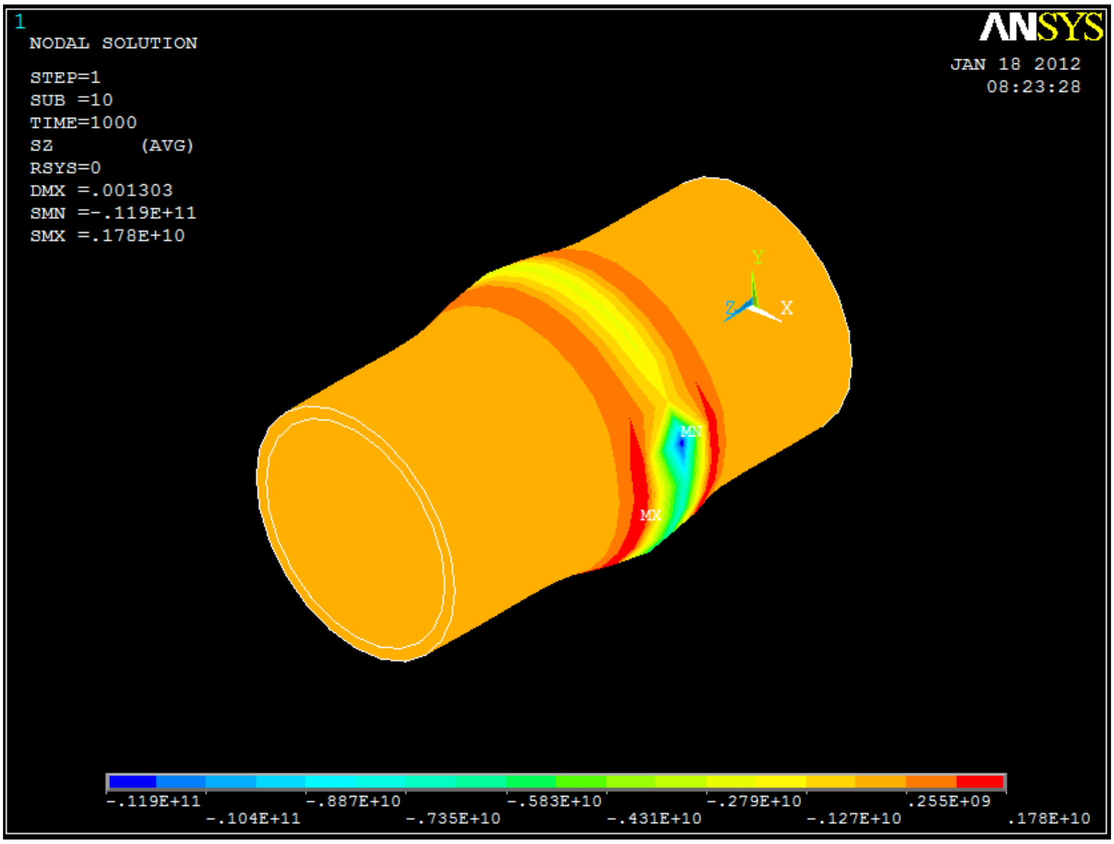


**Figure 4.3: Stress in Z-axis of the cylindrical model**

Figure 4.5 shows the stresses in Z axis which are the axial stresses acting on the pipeline. The axial stresses initiated from the tensile stress from the welding line, which acted normal with the welding line. The maximum axial stress is -1190 MPa. This is the compressive stress initiated when the cooling process takes place. The maximum stress is at node 486. Here, the accumulation of weld heat input occurred as the last weld heat input is applied on the first point of welding process. The maximum tensile stress plotted is 1780 MPa, situated at the region along the welded line. This indicates the gradual movement of tensile stress outwards when the last weld heat input acted upon the same node. In the simulation, the tensile stresses gradually move toward the two end of the pipeline and dissipated slowly later on. Where cooling initiated, then the compressive stress will arise.

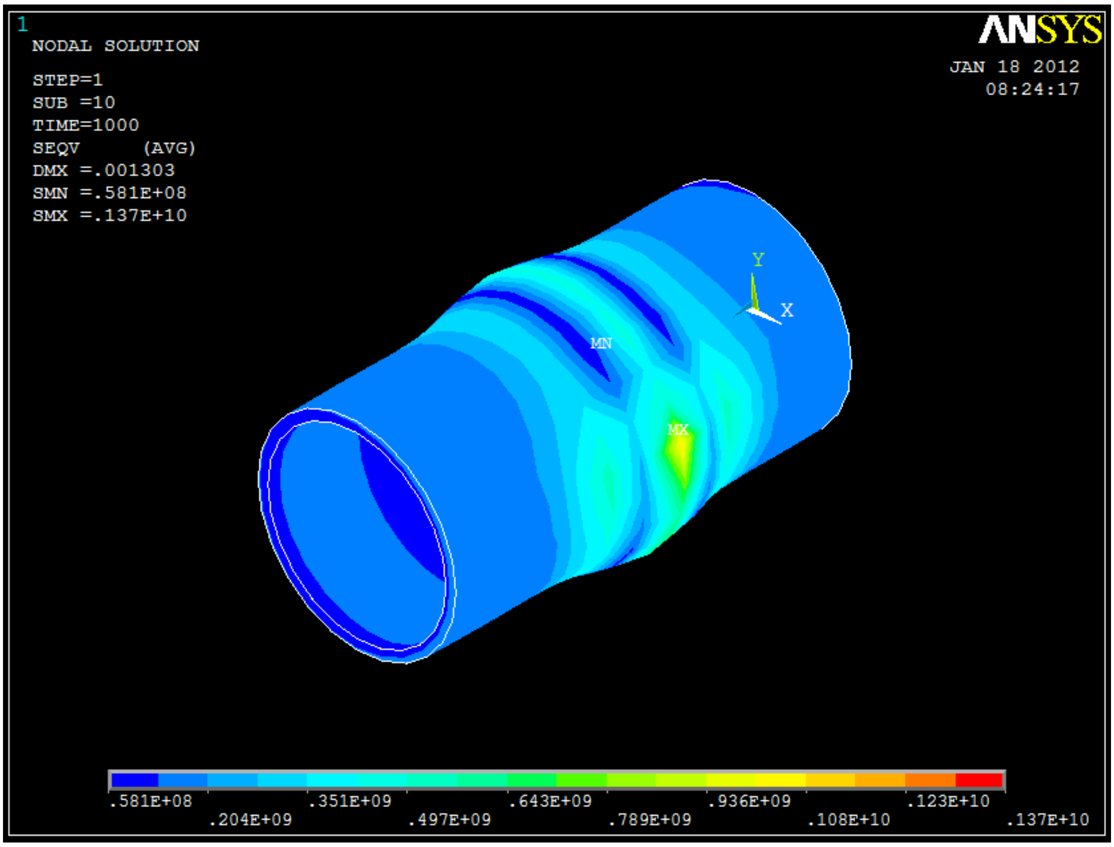


**Figure 4.4: Von-Mises Stress Distribution**

Figure 4.6 shows Von Mises Stress on the pipeline. Von Mises Stress is used to predict the failure of the material. Simple use of this theory is the material will undergo plastic deformation when the Von Mises stress exceeds the Yield Strength. Based on the figure, the maximum stress plotted is 1370 MPa. This is above the Yield Strength which is at 245 MPa. This indicates the welding heat input has yield plastic deformation on the pipeline welding line. The plastic deformation occurs mostly at the welded region and some of it is in the HAZ region.

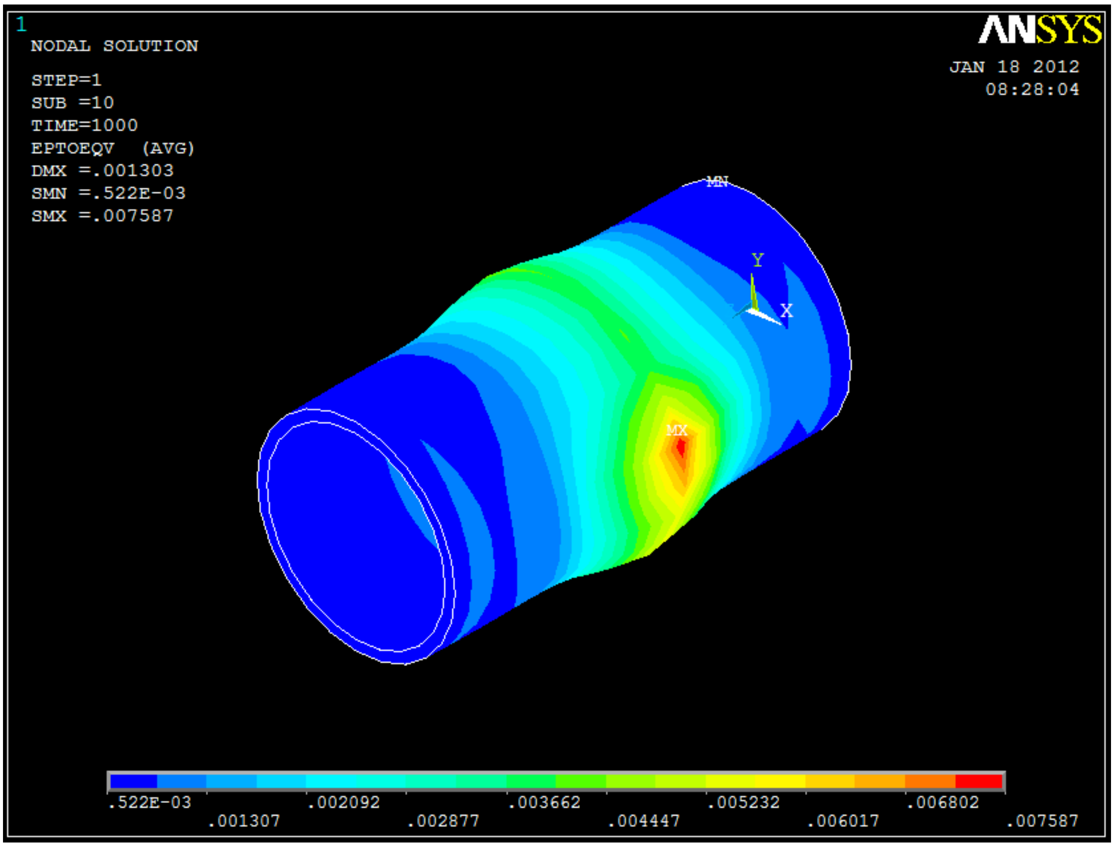


**Figure 4.5: Von Mises total mechanical strain on the model**

Figure 4.7 shows the Von Mises mechanical strain on the model. Note that a region of strain contour is created around the welded line. This is one of the ways to predict the formation of HAZ region across the welded region. The maximum strain stated in the plot is 0.007587 m/m which is at node 486 with temperature of 2277K. Strain indicates distortion along the region of node 486, and based on the stress value, mostly will be undergo plastic deformation.

## 4.2 Stress Vector Plot

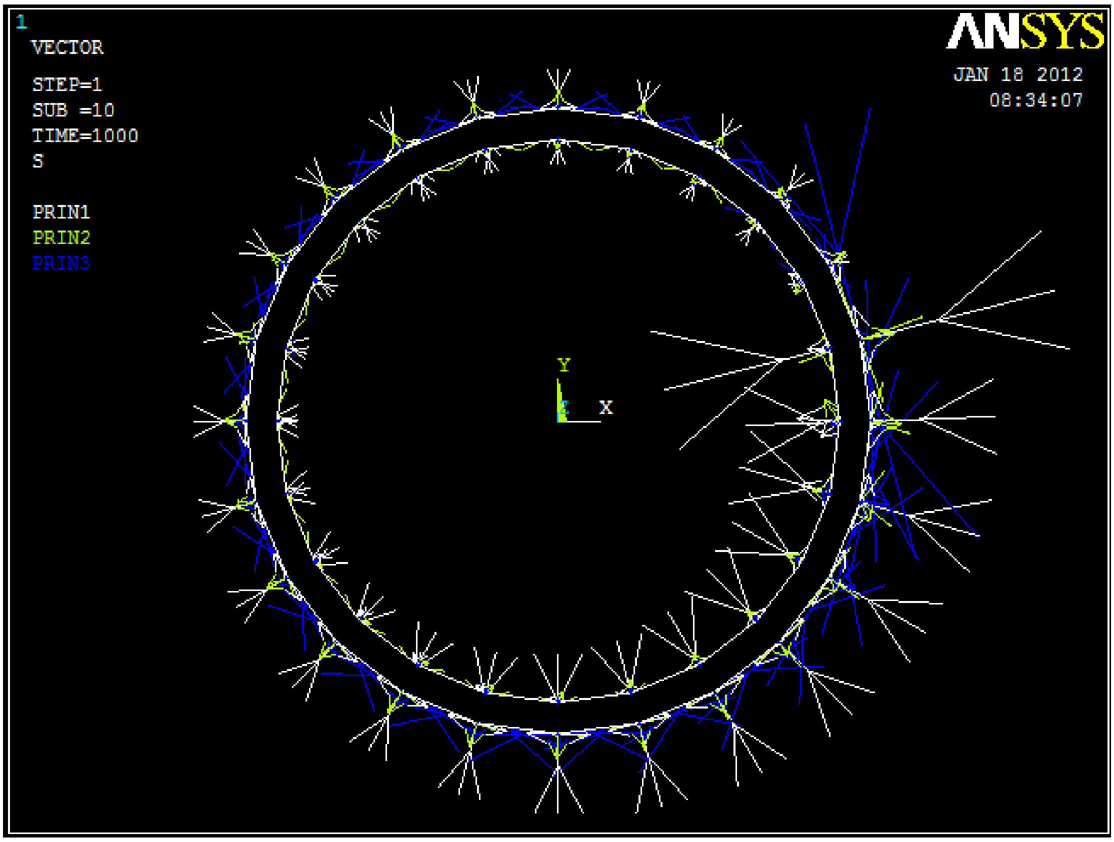


**Figure 4.6: Stress in vector plot**

Figure 4.8 shows a vector plot of the stress in the circumference of the pipeline. The directions of arrows in the plot indicate stresses from the inner surface of the pipeline are counter acted by the arrow from the outer wall of the pipeline. It is observed when the welding occurs, tensile stresses acted on the pipeline. When the cooling process starts to take place, gradual compressive stresses are initiated. The arrow pointed outward is the tensile stress due to the high temperature, and the arrow pointed inward is the compressive stress when the cooling process takes place.

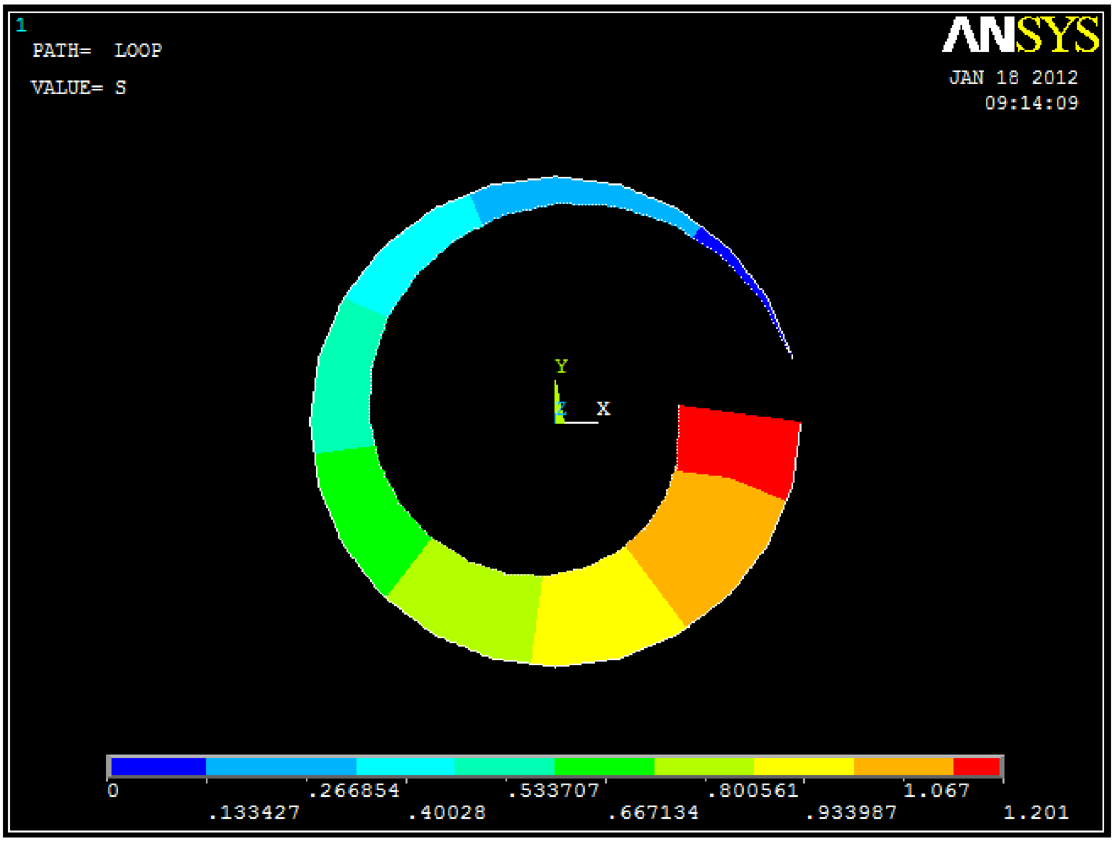


**Figure 4.7: Stress distribution along the welded line**

Figure 4.9 is a graphical plot showing the stress intensity distribution on the geometry of welded line. The value plotted is the ratio of the stresses magnitude with other stresses in the path line. The accumulation of stress is the greatest at node 486 which experience the temperature of 2277 K. This clearly indicate how higher temperature would affect the magnitude of stresses on the pipeline.

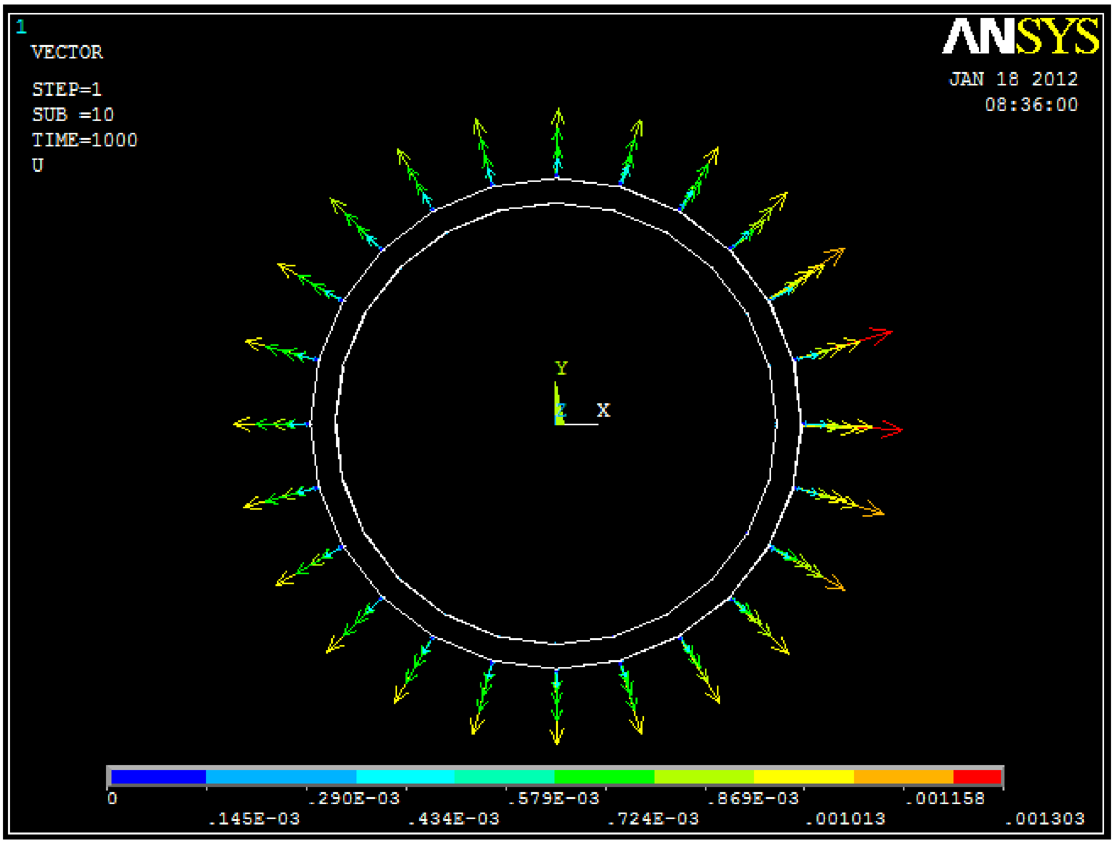


**Figure 4.8: Displacement/distortion in a vector plot**

Vector plot in Figure 4.10 shows the trend of metal distortion along the welded line. Note that the highest distortion trend is at node 486 which experienced the greatest temperature rise which up to 2277 K when the heat input is accumulated. Maximum distortion at node 486 is 1.303 mm. The distortions of metals are most likely to form the weld beads that are accumulated at the top of the welded line. Proper surface finishing and machining across the surface will remove the excess metal to discard any stress concentration spots.

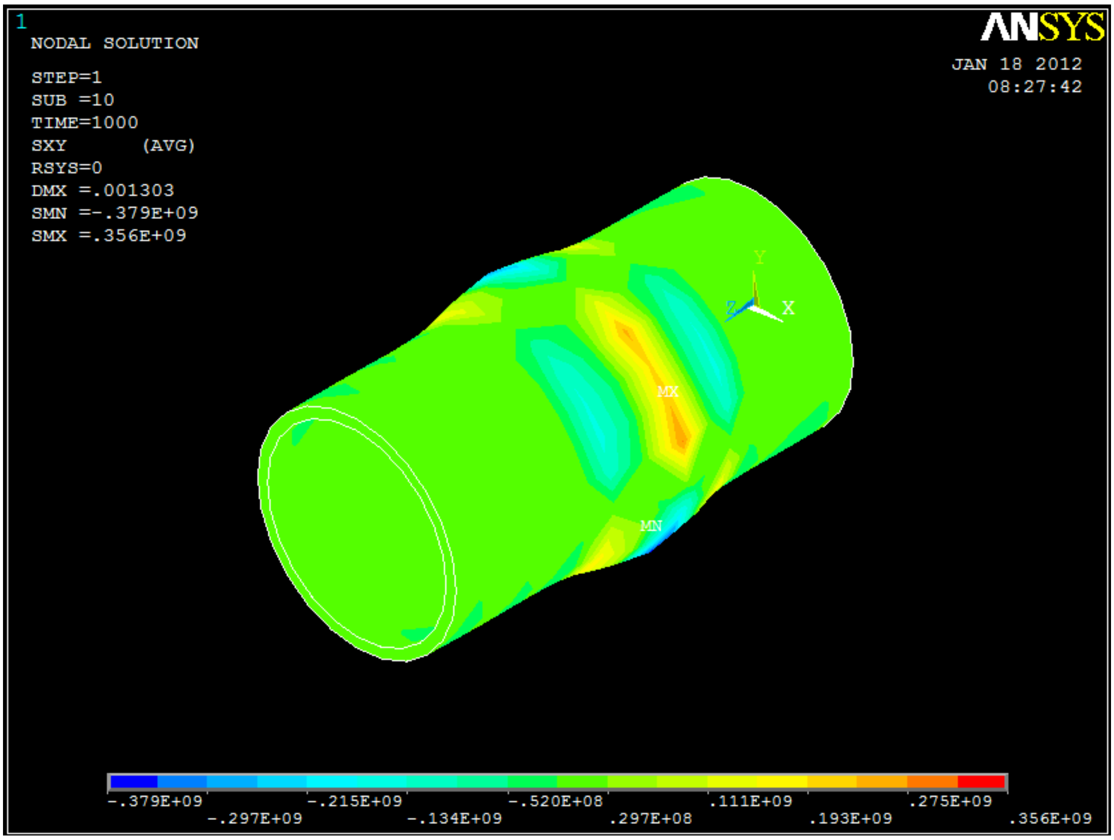


**Figure 4.9: Shear Stress in XY plane**

Among the three shear stresses plotted which are in XY, XZ and YZ plane, the highest value is – 379 MPa shown in Figure 4.11. Shear stress is related to the total distortion of the model as the load is parallel to the cross section of the pipeline. As the heat input move from node 486 towards the end of the welding line, non uniform stresses are created along the line.

## 4.3 Data Results Comparison

**Table 4.1: Summary of Plotted Results in ANSYS Simulation**

| Weld Heat Input (W) | 33000 | 34000 | 35000 |
|---|---|---|---|
| Max. Tensile Stress in X axis (MPa) | 1810 | 2340 | 2810 |
| Max. Tensile Stress in Y axis (MPa) | 1680 | 1900 | 2120 |
| Max. Tensile Stress in Z axis (MPa) | 1600 | 1780 | 1970 |
| Maximum Stress Intensity (MPa) | 1340 | 1440 | 1720 |
| Von Mises Stress (MPa) | 1150 | 1370 | 1890 |
| Von Mises Mechanical Strain (m/m) | 0.0070 | 0.0076 | 0.0081 |
| Maximum Stress Distribution Ratio | 1.17 | 1.20 | 1.32 |
| Shear Stress in XY Plane (MPa) | 247 | 356 | 414 |
| Maximum Temperature (K) | 2007 | 2145 | 2277 |
| Maximum Distortion (mm) | 1.227 | 1.303 | 1.401 |

The equation used to calculate the weld heat input is given by $= \eta \frac{VI}{v}$ , where V = voltage, I = current, v = electrode travel speed and $\eta$ = efficiency.

Voltage, V = 12V

Current, I = 140A, 145A and 150A

Efficiency, $\eta$ = 1.0

Electrode travel speed, $v$ = 0.05m/sec

By using 33 kW, 34 kW and 35 kW, the values are being standardized with constant increment in power output.

## 4.4 Discussion

**Stress**

Most of stresses plotted in the cylinder model structural model are concentrated at the point where the temperature is the highest. The highest temperature at the welded line is 2277 K which is at node 486. It is where the last welding welds heat input finish the closed circular welding process. This temperature is mainly due to the sum up of the new weld heat input with the earlier weld heat input that been applied there as the first weld heat input. The stress intensity show the most stress concentrated at the same point too.

**Axial Stress**

Axial stress in this pipe model is basically the stress in z component. This stress accumulation is due to the initial condition that is applied. The highest stress plotted is at the node which received the highest temperature as body load. The stress value is -1190 MPa which is a component of compressive stress. Due to the melted metal at the welded line, the axial load is moving toward the welded line and induced the greatest axial stress at the node 486 where the metal phase change occur. Since this stress in higher than then yield strength of the 316 stainless steel, it is expected that distortion could occur here. The distortion will be reduced over time as the metal cooled down to room temperature. Along the cooling process, the metal distort in less severe than when it is under high temperature load.

**Stress Intensity**

The stress concentration is the greatest at the node 486 which is the node that experience the biggest change in the temperature profile. First, the node is given a weld heat input value of 35E6 W/m.K that make the node temperature rise to 1800 K. After 120 seconds, the weld heat input once again is applied on the same node with the same weld heat input value. This gives the nodes a rise of temperature of up to 2277 K. This is why the stress is concentrated at the spot mostly. When an element experiences greater temperature rise, the residual stress which caused distortion and expansion as well as contraction will accumulated here. It is the weakest spot when the metal in molten form, so all displacement will move towards this point.

**Vector Plot Stress**

Stress by vector plot shows a common stress distribution due to temperature profile. The stress is the greatest at the highest temperature value available. Note that the arrow of the stress is pointing toward the welded line. This shows a good sign of compressive stress towards the welded line. Two compressive forces from opposite direction are moving towards the welded line, thus provide the necessary stress to make the welding joint intact. The stress arrows are also pointing in opposite direction with each other in the circular line across the pipeline. This indicated a balance tensile and compressive force between each other along the line of welding. Only the stress value is differing due to temperature difference. The good thing is that the joint of welding is assured to be intact with this vector stress movement. Even it is good for welding condition, it should be note that the distortion will be the highest at the welded lien due to all stress accumulated here.

**Vector Plot Displacement**

The distortion occur the most at the node 486 and most of the welded line along the circular path. The distortion indicates a stress accumulation that exhibit a constant movement towards outer side of the pipe. The distortions are most likely will form the weld beads that accumulated at the top of the welded line. The distortion is not too severe with a value of 0.001303 m which is around 1 mm only. Proper surface finishing and machining across the surface will remove the distorted metal at the upper layer of the welded line

## 4.5 Recommendations and Improvements

**ANSYS SOFTWARE**

ANSYS has shown what it is capable to simulate. Basically, the simulation of welding is working smoothly and the results are reliable. Nevertheless, there are several things that can be taken care of in the next simulation tasks.

In the simulation, refined mesh can help to achieve better results and more accurate data plot. In this analysis, coarse meshes are used to simplify how the loads are applied to the selected elements and nodes. This will help the beginner users to slowly adapt on how ANSYS works. By using refined meshes in the next analysis, ANSYS read the data iterations more frequent than when the coarse meshes are used. This is how the data result would be much more reliable. More iteration means more data processing, so the result would be more accurate. Although using refined mesh is good, it will also consume a lot of time. So, choosing proper mesh for the analysis conducted is essential. Also, other alternative is to refine mesh only at the point of interest. This way, it will not just saves the time needed for an analysis, but also help users to capture the results they need the most at the given point of interest.

One way that can help users to apply the loads to the model in easier way is by mastering and utilizing the command text language. ANSYS provide this platform so the advanced users can utilize it to conduct more complex analysis. It is typically irrelevant to specify loads at thousand of nodes by doing the graphical picks and choose. It is time consuming and also need huge efforts. By mastering the command text technique, users can specify loads in a more sequential ways. Errors can also be traced easier with the help of software that is used to write the command text.

ANSYS read most of the text easily and even Notepad software can be made as the text writing platform. It is recommended for users to master this command text so for the next analysis, it would be easier to perform. Although ANSYS provide thousands of command texts, it is still beneficial for users to master several basic commands.

**Command script for applying loads**

```
TIME,5                  ! Time taken for a single step, this represent 5 seconds
NSUBT,10,10,10          ! This represent 10 substeps option
F,486,HEAT,35000        ! The heat input with the value of 35kW at node 486
LSWRITE,1               ! Write the corresponding load file into database
FDELE,ALL,HEAT          ! Delete current heat input

TIME,10
NSUBT,10,10,10
F,487,HEAT,35000
LSWRITE,2
FDELE,ALL,HEAT

TIME,15
NSUBT,10,10,10
F,488,HEAT,35000
LSWRITE,3
FDELE,ALL,HEAT

TIME,20
NSUBT,10,10,10
F,489,HEAT,35000
LSWRITE,4
FDELE,ALL,HEAT
```

**WELDING**

It has been shown by the simulation of what the weld heat load can do towards the metal. Basically, weld heat input create distortion and also stress at the affected region.

In the simulation, the process is a single weld pass method. This approach creates a large distortion of metal towards the outside region. The effect of this distortion is the geometry profile of the cylinder is altered. This will affect the flow of fluid in the pipeline when it is in service later. Also, since distortion exists, there will be stress concentration point along the circumference of the pipeline. This could lead to a pipeline failure like sudden burst under high pressure if not given immediate attention. So, the welding process needs to be modified. To improve welding, the process of welding by several weld passes should be implemented. Weld the metal using several weld pass approach, starting with the root pass, second layer, third layer and so on until the top of weld bead. This method isolate critical region from unnecessary heat load input. This also prevents excessive metal distortion as previous weld pass reduce the total metal affected by the weld heat input.

# CONCLUSION

- Finite element analysis using ANSYS as the medium is proved to be reliable and accurate
- High tensile stress is indicated near the welded line and reduce gradually over time
- Compressive stress arise when cooling phase started
- Metal distortion is basically the weld beads formation around the welded line
- The higher the weld heat input, the higher the residual stress magnitude
- Most of the structural changes occur in welding process is in the plastic region of the metals
- Careful assumptions must be made so the results obtained are reliable

# REFERENCES


[1] David Roylance, “Finite Element Analysis”, Department of Materials Science and Engineering, Massachusetts Institute of Technology, Cambridge, MA 02139, 2001

[2] Shu Guo, K. Him Lo, Benjamin T. A. Chang, “Residual Stresses in 3LPP Pipeline Coatings”, NACE International, Corrosion Conference Expo, 2011

[3] Jeremy E. Scheel, N. Jayaraman, Douglass J. Hornbach, “Engineered Residual Stress to Mitigate Stress Corrosion Cracking of Stainless Steel Weldments” Lambda Technologies, USA, NACE International, Corrosion Conference Expo, 2011

[4] A. Scaramangas, R.F.D Porter Goff, “Residual Stress in Cylinder Girth Butt Welds”. Cambridge U, 1985

[5] P Delfin, I. Sattari Far, B. Brickstad, “Effect of Thermal and Weld Induced Residual Stresses on J-Integral and CTOD in Elastic-Plastic Fracture Analysis”, SAQ Kontoll AB, 1998

[6] P. J. Withers, H. K. D. H Bhadeshia, “Residual Stress, Part 1-Measurement techniques”, 2001

[7] Weman, Klas, “Welding Processes Handbook”, New York: CRC Press LLC. ISBN 0-8493-1773-8, 2003

[8] Argyris, J. H Szimmat, and William K. (1983), “Finite Element Analysis of the Arc Welding Process”, Seattles WA, USA

[9] Peng-Hsiang Chang, Tso-Liang Teng, “Numerical and Experimental Investigations on the Residual Stresses of the Butt-Welded Joints”, Da Yeh University, Taiwan (2003)

## APPENDIX 1: EXAMPLE OF COMMAND SCRIPT FOR FIELD-COUPLED THERMO-MECHANICAL ANALYSIS

```
/batch,list
/show
/title, thermal stress in concentric cylinders - indirect method
/prep7
et,1,plane77,,,1              ! PLANE77 axisymmetric option
mp,kxx,1,2.2                  ! Steel conductivity
mp,kxx,2,10.8                 ! Aluminum conductivity
rectng,.1875,.4,0,.05         ! Model
rectng,.4,.6,0,.05
aglue,all
numcmp,area
asel,s,area,,1                ! Assign attributes to solid Model
aatt,1,1,1
asel,s,area,,2
aatt,2,1,1
asel,all
esize,.05
amesh,all                     ! Mesh model
nsel,s,loc,x,.1875
d,all,temp,200                ! Apply thermal loads
nsel,s,loc,x,.6
d,all,temp,70
nsel,all
finish
/solu
solve
finish
/post1
path,radial,2                 ! Define path name and number of path points
ppath,1,,.1875                ! Define path by location
ppath,2,,.6
pdef,temp,temp                ! Interpret temperature to path
pasave,radial,filea           ! Save path to an external file
plpath,temp                   ! Plot temperature solution
finish
/prep7
et,1,82,,,1                   ! Switch to structural element, SOLID82
mp,ex,1,30e6                  ! Define structural steel properties
mp,alpx,1,.65e-5
mp,nuxy,1,.3
mp,ex,2,10.6e6                ! Define aluminum structural properties
mp,alpx,2,1.35e-5
mp,nuxy,2,.33
nsel,s,loc,y,.05              ! Apply structural boundary conditions
cp,1,uy,all
nsel,s,loc,x,.1875
cp,2,ux,all
```

```
nsel,s,loc,y,0
d,all,uy,0
nsel,all
finish
/solu
tref,70
ldread,temp,,,,,,rth         ! Read in temperatures from thermal run
solve
finish
/post1
paresu,radial,filea          ! Restore path
pmap,,mat                    ! Set path mapping to handle material discontinuity
pdef,sx,s,x                  ! Interpret radial stress
pdef,sz,s,z                  ! Interpret hoop stress
plpath,sx,sz                 ! Plot stresses
plpagm,sx,,node              ! Plot radial stress on path geometry
finish
```

## APPENDIX 2: COMMON CURRENT USAGES IN SMAW WELDING FOR DIFFERENT ELECTRODE