MODELLING AND SIMULATION OF VIBRATION ASSISTED ARC WELDING USING ANSYS

First Author

Ranjith K

Assistant professor, Dept. of mechanical engineering (APJ Abdul Kalam Technological University) college of engineering thalassery (APJ Abdul Kalam Technological University) Kannur, India

Amalraj S P

Student, Dept. of mechanical engineering (APJ Abdul Kalam Technological University) college of engineering thalassery (APJ Abdul Kalam Technological University) Kannur, India

Abhinand C

Student, Dept of mechanical engineering (APJ Abdul Kalam Technological University) college of engineering thalassery (APJ Abdul Kalam Technological University) Kannur, India

Sayooj Ramesh M

Student, Dept. of mechanical engineering (APJ Abdul Kalam Technological University) college of engineering thalassery (APJ Abdul Kalam Technological University) Kannur, India

Diljith P

Student, Dept of mechanical engineering (APJ Abdul Kalam Technological University) college of engineering thalassery (APJ Abdul Kalam Technological University) Kannur, India

***Abstract*— Nowadays welding is most commonly used in industrial and structural application. In this thesis are welding is used. Residual stress is the most important problem faced. Because of this residual stress quality of the welding is reduced. The main cause of stress is due to expansion and contraction while heating. There are many ways to reduce residual stresses. Residual stress in weld elements can be reduced by applying heat treatment and peening techniques. However these methods need special equipment and are time consuming. In this present project work, a comparatively new method for reducing the residual stresses using vibration during welding is proposed. For this, mechanical vibrations will be used as vibration load. The plates to be welded will be held fixed and excited by an external source such as motor at lower frequencies. In order to reduce residual stress experimental method were used before but simulation method was not used.**

In the present condition, finite element process has been used to control residual stress in the vibration condition. Demo version ANSYS workbench16.0 is used for simulation analysis. A Finite Element Model has been generated and thermal analysis, structural analysis, modal analysis, and harmonic analysis are carried out.

In this analysis MIG welding process is used and this welding commonly used in high deposition rate welding process. In this project, the finite element analysis is used to perform welding simulation and to predict weld-induced residual stress in arc welding of mild steel plate.

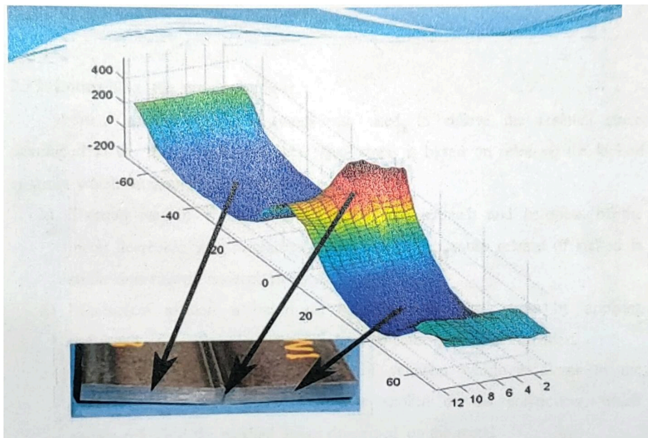
I. Introduction

Welding is the most important element in manufacturing process. It is a conventional technique for joining of two metals which requires less economical support but provides effective output. Welding has been applied to various industries in particular, automotive, aerospace and microelectronics. It is the process of using metal as melt to join two metal parts by applying heat. Thermal cycle produced near weld line generates residual stress and inhomogeneous plastic deformation in weldments. Residual stresses are formed in a structure due to differential expansion and contractions. Residual stresses have a significant effect on corrosion, fracture resistance and creep. Many methods are used for reduction of residual stress in manufacturing process such as heat treatment process, peening process etc. So these methods need special equipment and cost. In this project, a new method used for reduction of residual stress using vibration during welding is proposed. The plate will be held fixed and excited by an external source such as motor at lower frequencies which are closer to natural frequency of the work piece. In this work, finite element method (FEM) is used. Demo version ANSYS Workbench 16.0 has been used for simulation. A Finite Element Model has been generated and thermal analysis, structural analysis, modal analysis, and harmonic analysis are carried out. Excited frequency is closer to natural frequency.

The modal analysis is used to calculate the vibration characteristics such as natural frequency and deformed shape. The natural frequency generated in structure depends on Size and applied force. The output of the modal analysis is the input of harmonic response. Sinusoidal vibration is imparted. The main aim of the modal analysis is to find out the natural frequency of the component which is closer to vibration induced in the component. Middle frequency has been selected, and then the other frequencies are selected in equal distance based on middle frequency. The residual stress distributions were validated with the experimental result available in literature. The residual stresses are minimized compared to the specimen plates which are welded without any vibrations. In this project, the finite element analyses are used to perform welding.

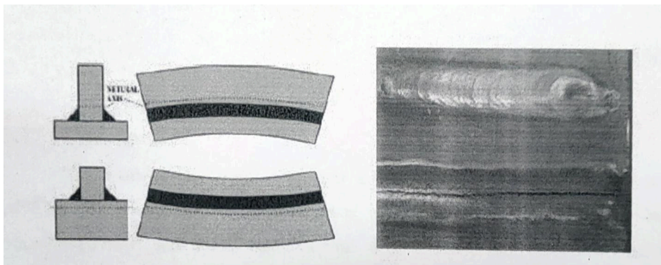
II. WELDING INDUCED RESIDUAL STRESS

Residual stress in welding joint is developed due to differential heating and cooling experienced by the weld metal and HAZ Magnitude of residual stresses varies under the differential heating and cooling conditions. Heat flowing from the weld area causes the joint area to expand. Thermal expansion and contraction due to welding will leave behind permanent stress. A compressive stress is developed in the region due to thermal expansion and also contraction of the low temperature surrounding metal. Tensile or compressibility influences the dimensional and mechanical performance of weld joints.



Maximum residual stress which acting weld center line

Since magnitude of residual stress is increasing gradually to maximum value as weld joint is cooled down to room temperature. So the magnitudes of residual stresses are observed near the last stage of welding. The presence of residual stress in the weld joint incites failure due to external loading condition and also crack developed on weld joint under the atmospheric conditions. Residual stresses are developed due to the corrosive condition which leads to the failure of weld joints; this phenomenon is called corrosion cracking contrary affect their load carrying capacity.
Therefore, residual stress must be removed from the weld joint before under taking any machining processes.



Solidification cracking

III. METHODOLOGY

The modeling of welding process and subsequent simulation was done in ANSYS Workbench 16.0. The work piece was of rectangular shape with a size of 300-75x10 mm and it was made of mild steel Different rate of convectional heat dissipation from the work piece to still air were assumed for the metal A fine mesh was provided on both side of weld center line. Meshing is provided near the high temperature and stress gradients of the weld, the finite element model was performed using the concept of adaptive meshing. The default material properties of mild steel as available in demo version ANSYS Workbench 16.0 were applied to the work piece. The process of welding was modeled such that the heat source started from one of the edges of the work piece and moved along the Centre line towards the other end. The heat source is moved under vibratory condition. The output of the modal analysis is the natural frequency of the body. The frequency of oscillation of the work pieces were 75 to 95 Hz

A. PROCESS DESCRIPTION

Description: Material used: Structural Steel as available in the material library of ANSYS.

Size of work piece: 300 x 75 x 10 mm

1. Develop a transient thermal model for Moving Heat Source.
2. Obtain a sequential coupling between this transient thermal model and a static structural model to determine the weld induced residual stress.
3. Do Modal Analysis to determine the natural frequencies for the vibrating specimen.
4. Develop a Harmonic response model for the work piece by which the specimen may vibrate as the result of external forced harmonic excitations at its natural frequencies.
5. Validate the model against the results in literature. If the proposed model reduces residual stress in comparison with un-vibrated specimen in the reported fashion, the model stands validated. The validated model can be used for further study.

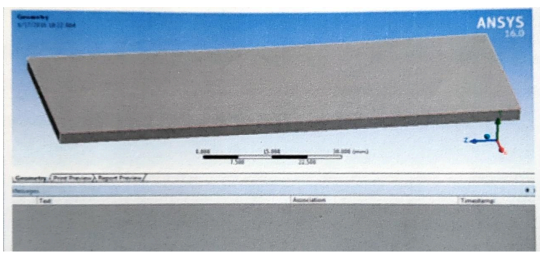
Work description:

Parameters: Frequency of vibration (F), are power of welding (P) and wekling speed(S). P = n x V x 1; where, n= Arc efficiency (assumed to be 80%),

V = wekling voltage = 24 V, I-welding current 150A

B. **MODELING AND ANALYSIS USING FEM**

The present work of modeling and simulation of arc welding process was carried out by using the commercial FE software package demo version ANSYS Workbench 16.0. Rectangular work piece made of structural steel was selected for the study, since steel is the most common material being used for structural purposes and arc welding is mainly used for making joints in structures. The size of rectangular work piece was 300 x 75 x 10 mm' and the welding process was without any consumable filler material..

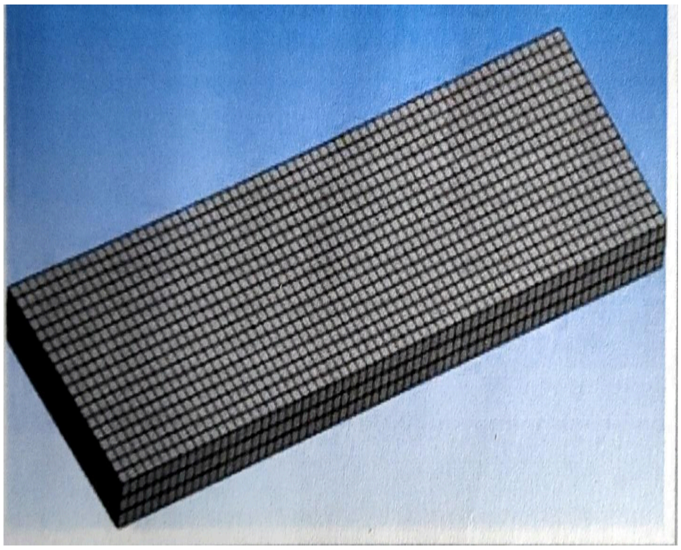


Volumetric source was modeled as moving from one end of work piece to the other end as shown in Fig.3.1. Sequential coupling between transient thermal model static structural model modal analysis and harmonic response are required to predict the residual stresses developed after welding. The resulting temperature distribution from thermal model was fed to the static structural model and harmonic response after assigning necessary boundary conditions to it.

frequencies	75hz,80hz,85hz,90hz,95
current	150A
voltage	25V

C. **MESHING**

A mesh is the Discretization of a component in to a number of small elements of defined size. Finite element analysis is dividing the geometry in to various small numbers of elements. These elements are connected to each other at points called nodes. Each node may have two or more than two elements connected to it. The collection of this element is called mesh.



IV. **MOVING HEAT SOURCE**

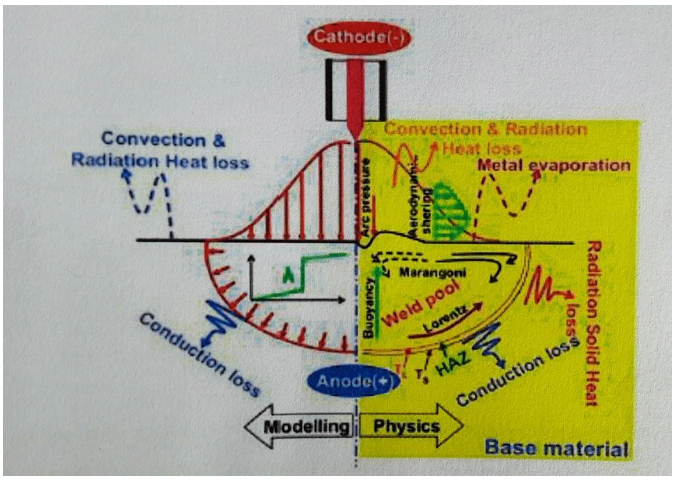
Modeling of a welding process starts with developing a suitable moving heat source model and there are several types of such models described in literature. Point heat source, line heat source, disc heat source and elliptical heat source are popular types of 2D heat sources. The evolution from point heat source to elliptical heat source is the journey of researchers to get a model better matched with the real life welding situations. The 2D heat sources have some inherent disadvantages as being described only along a plane and they are less effective to represent the real welding process since the real weld pools are 3 dimensional entities. But, the assumption of heat source as a planar one does not make much difference in arc welding processes like TIG welding where the depth of weld pool is not as much as that of laser welding. The planar heat source models may be uniform or distributed depending on the intensity of heat load applied on the planar area of heat source. The simplest is the uniform heat load, but it differs much from the reality where the heat intensity varies right from the geometric center of the area to its ends. In general, the intensity of heat flux is the maximum at the center of the area and fades towards the ends. The Gaussian Heat Source Model is a planar model in which the heat flux is distributed as Gaussian and its value is the maximum at the center of the disc and nearly 5% at the diametrical ends. Again, the Gaussian heat source may be approximated to a uniform circular one without any substantial error if the effective diameter of the heat source is very low.

A. **GAUSSIAN SURFACE HEAT SOURCE MODEL**

Since heat over a surface, surface heat source is closer to the real condition than point heat source. Instead of uniformly distributed, many researchers have used distributed surface disc heat source modeling according to Gauss distribution [4]. Surface heat source is 2-D one with either distributed.

B. **GAUSSIAN VOLUMETRIC HEAT SOURCE MODEL**

A volumetric heat source is 3-D source which is found to be closer to the real heat source. Pressure, surface tension and buoyancy forces have been known in practical welding. These phenomenon's are included in volume of material Golden introduced volumetric heat source.



C. **HEAT SOURCE SIMULATION**

The temperature-distance profile shows that the heat source travels along the weld in the direction A-A' at a constant speed.

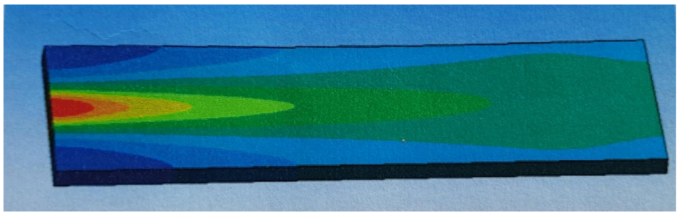


Fig 4.2 Moving Heat Source Model

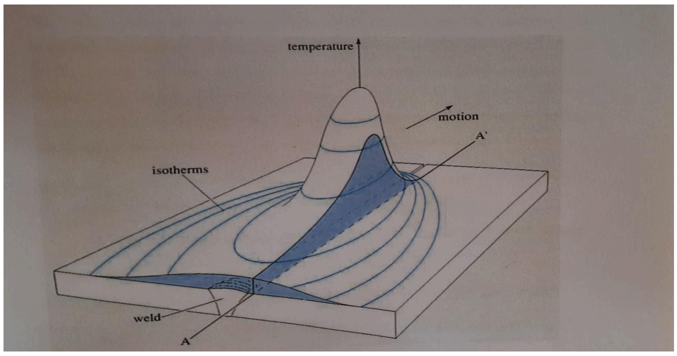


Fig 4.3 Temperature distribution round a typical weld [2]

As the heat source moves on the cooling rates around the weid are very high A more intense heat source will give a steeper profile and the HAZ, which will be confined to a narrower region Further, differential heating and cooling experienced of different zones of weld joint cause not only metallurgical heterogeneity but also non-uniform volumetric change which in turn produces the residual stresses. These residual stresses adversely affect the mechanical performance of the wet joint besides distortion in the wekled components if proper care is not taken

C.a) **MODELING MOVING HEAT SOURCE**

In this study first the transient thermal model for the circular heat source was developed and simulated using demo version of ANSYS workbench 16.0. Moving Heat Source ACT Extension 16.0 and the resulting temperature field was fed to a static structural model for obtaining the sequential coupling between the thermal and structural models. While simulating moving heat source we have inputted various data such as, velocity of weld which is 6 mm/sec, radius of beam as 5 mm, power, The work piece was given boundary conditions such that its movement was completely arrested in all the directions Currently transient thermal analysis module in Workbench can only be used to define time varying or spatial varying loading condition. To define a source which varies not only in space also with time requires MAPDL commands. Thus, an ACT extension has been created to facilitate the definition of a moving heat thux source in Mechanical with an Ul. In version 4, additional moving heat energy source is also created. This will facilitate the creation of moving energy source in the volume of the material. Additional moving heat energy source can now be created. This will facilitate the creation of moving energy source in the volume of the material.
New feature of removing the material based on temperature is introduced. New result just added to view the removed

material. New result object is added to view the mean temperature distribution.

D. EFFECT OF RESIDUAL STRESS UNDER THE VIBRATORY CONDITION

Vibration method, which is considered as alternative heat treatment has also influence on reduction on residual stresses.. In this thesis, the effect of vibrations with different frequencies on reduction of residual stress is examined. Work pieces are welded using harmonic vibration and without harmonic vibration for comparison. When thin plates are welded using vibrations with different frequencies, tensile residual stresses are reduced and reduction rate is largest compared with other conditions. Obtained results are examined by analytical method. A finite element model is developed for the simulation of vibration stress relief after welding. For non-resonant vibration, the reduction in stress strongly depends on the amplitude of vibration. For the resonant vibration, the vibration frequency is the key for residual stress relief. The vibration frequency should be close to the structure natural frequency for the desired vibration mode. Only small vibration amplitude is required, which will be amplified during vibration. Vibration time does not have major impact on vibration stress relief and the velocity of welding is 6mm/s. In order to examine the effect of excitation frequency on reduction of residual stress, excitation frequencies are chosen as 75, 80, 85, 90 and 95 Hz. Harmonic analyses are used to determine the steady-state response of a linear structure to loads that vary sinusoidal (harmonically) with time, thus enabling to verify designs will successfully overcome resonance, fatigue, and other harmful effects of forced vibrations. This analysis technique calculates only the steady-state, forced vibrations of a structure. In this analysis all loads as well as the structure's response vary sinusoidal at the same frequency. Residual stress is measured as the linearized normal stress across the Well Centre Line (WCL). The maximum residual stress was obtained in area near to WCL which is the heat affected zone. For the frequencies between 75 to 95 Hz coupled thermo- mechanical analysis was performed and it is found that residual stresses are greatly reduced.

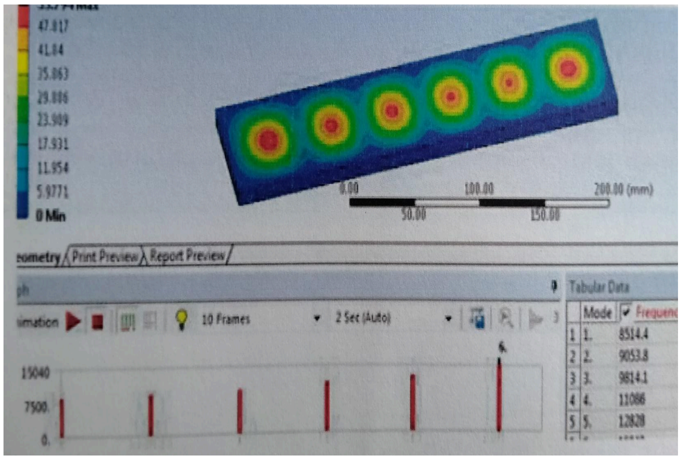
E. MODAL ANALYSIS

Natural frequencies of system are those frequencies at which the resonant response occurs under the right excitation conditions. Knowledge of these critical dynamic frequencies is an essential step in the design or evaluation of the system subjected to dynamic loading.

The modal analysis is used to calculate the vibration characteristics such as natural frequency and mode shape (deformed shapes) of a structure or a machine component. The output of the modal analysis can be further used as input for the harmonic and transient analysis. The shape of the component corresponding to frequency is known as mode shape. The mode shape is graphical representations of the deformation attained due to vibration. The main aim the modal analysis is to find whether the natural frequency of the component is closer to the vibration induced in the component. If the natural frequency of a system is closer to excitation frequency, the component can get resonance. Also, in actual practice, the displacement produced at resonance may not be infinite due to the presence of damping.

F. ANALYSIS STEPS

We can plot the mode shape (deformed shape) at each mode. However, before plotting the deformed shape, we need to specify the mode in the graph.



After conducting static analysis the results were fed into modal analysis to conduct prestressed modal analysis and captured the natural frequencies and its resulting mode shapes. The results from simulated the modal analysis was taken for consideration of frequency selection. Then conducted a harmonic response analysis with an applied force which is excited between selected ranges of frequencies obtained from modal analysis. This same model was considered for different frequency of vibration excitation.

In modal analysis output frequency which is equal to natural frequency of the specimen. The natural frequency of the component is closer to the vibration induced in the component. Equal distance frequencies are 75Hz, 80Hz, 90Hz and 95Hz selected with respect to middle frequency 85Hz. The output result of the modal analysis is the input of harmonic response. These 5 frequencies are applied as input values for harmonic analysis. Maximum resonance value obtained at 85 Hz. At resonance frequencies small periodic driving force have the ability to produce large amplitude oscillations. Then we have analyzed for the transverse residual stress and longitudinal residual stress values at specified paths on either side of the weld centers line. The residual stress distributions were validated with the experimental results available in literature.

G. RESULTS AND DISCUSSIONS

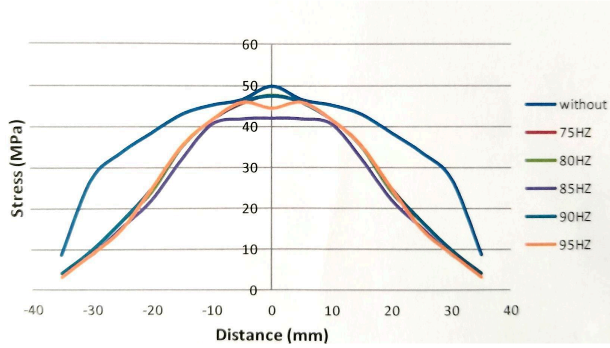
The welding process of steel plate with the dimension shown in fig.3.1 was simulated. Due to high temperature and stress gradients near the weld, the finite element model has a relatively fine mesh in both sides of weld center line. The eight node brick elements with linear shape functions are used in meshing the model. To simulate the moving heat source is necessary to model the heat source during each time increment. In this analysis the moving heat source is simplified by assuming the welding are stayed at an element with a constant specific volume heat generation, and then moved to the next element at the end of load step as the welding was finished

G.A) **EFFECT OF FREQUENCY OF VIBRATION**

Fig.7.1 shows that sample was welded without any vibration. The other four samples were vibrated while they were being welded, applying different frequency of vibration measured. The longitudinal and transverse residual stress of the samples was In the transverse residual stress, static condition value of stress is measured at 49.78 MPa, at weld center line. Vibration frequency of 75 Hz shows minimum residual stress value 47.58 MPa. Further increases in vibration frequency 80 and 85 caused the residual stresses to decrease by 47.58 MPa and 42 MPa respectively. A further frequency 90 Hz and 95 Hz shows 47.38 MPa and 44.38MPa.

The FEM analysis was done and following are the observations of residual stress values as shown in table 7.1.

Distance moved in traverse direction	Without vibration	Applying Vibrations				
		75HZ	80Hz	85Hz	90Hz	95Hz
(mm)	Stress value (Mpa)	Stress value (Mpa)	Stress value (Mpa)	Stress value (Mpa)	Stress value (Mpa)	Stress value (Mpa)
-35	8.73	4.13	4.133	4.1	4.1335	3.1335
-30	26.98	9.77	9.775	8.7	9.7759	8.776
-25	33.58	16.78	16.78	15.2	16.784	14.785
-20	38.38	24.81	24	22	24.814	24.815
-15	42.917	35.22	35	32	35.22	35.222
-10	45.17	41.58	41.58	40.5	41.58	41.585
-5	46.5	45.58	45.8	41.8	45.807	45.808
0	49.78	47.58	47.58	42	47.38	44.382
5	46.5	45.8	45.8	41.8	45.8	45.808
10	45.17	41.58	41.58	40.5	41.58	41.585
15	42.917	35.221	35	32	35.22	35.222
20	38.38	24.81	24	22	24.814	24.815
25	33.58	16.78	16.78	15.2	16.784	14.785
30	26.98	9.77	9.77	8.7	9.7759	8.776
35	8.73	4.133	4.133	4.1	4.1335	3.1335



COMBINED GRAPH

H. **EFFECT OF FREQUENCY**

In this investigation a high frequency vibration was applied to the specimens to observe its effect. Due to the high frequency, amplitude is very small. Vibration was started before welding. In case of 75 Hz and 80 Hz in transverse direction shows same stress values. Residual stress was observed in the specimen as a result of small amplitude of vibration. With increase in the frequency of vibration the residual stress were found to decrease further. Transverse residual stress showed different characteristics with increase in small frequency of vibration.

I. **DISCUSSION**

The above results strongly indicate that there may be a relationship between vibration frequency and residual stress. The modal analysis output is the natural frequency of the body. Some natural frequency values are getting by analyzing modal analysis. In harmonic analysis we have to give the different types of input. In this process we know the reaction of the structure. We can find stress, maximum and minimum displacement at each at each degree of freedom of the structure by this harmonic analysis. For the frequencies between 75 Hz to 95Hz coupled thermo mechanical analysis was performed and it is found that the residual stress are greatly reduced at a frequency of 85 Hz. The minimum stress value was obtained at 85 Hz because the resonance occurred at this frequency. For the longitudinal case, residual stress value is maximum at static condition. High frequency of vibration also did not result any particular trend of change in residual stress. Longitudinal and transverse residual stress will be increased. At statically condition, both longitudinal and transverse residual stress will be increased.

J. **CONCLUSION**

Finite element modeling and simulation of vibration assisted arc welding has been performed. Thermal model for the moving heat source was developed and it was sequentially coupled to structural model to find residual stresses. Modal and Harmonic analyses revealed the effects of vibration frequency on weld induced residual stresses.

Modal analysis was conducted for getting fundamental and higher natural frequencies of vibration and the fundamental frequency of 85 Hz were selected for the present study. Two equidistant frequency values on either side of 85 Hz were selected for constituting the frequency range of 75, 80, 85, 90 and 95 Hz. It was observed that there was considerable reduction in residual stress after establishing vibration during the process of welding. It was found that, within the range of frequency selected, the reduction in residual stress was the maximum at the middle point, 85 Hz, and it decreased on either side. Hence it can be concluded that the residual stress can be greatly reduced by maintaining frequency of forced vibration at or near to natural frequency of the specimen

K. **FUTURE WORK**

The present work did not consider the fluidic aspects of weld pool. This work may be extended to a wonderful research programmed by including the fluidic aspects of vibration of molten weld pool.

REFERENCES

[1] Murlidhar C. Ingale, Prof.S.L.Kolhe, Prediction of Residual Stress in Weld Joint under Mechanical Vibration, "International Journal of Engineering and Techniques, vol 2, Jan-Feb 2016.

[2] Ihab F. Z. Fanous, Maher Y. A. Younan, and Abdalla S. Wifi, 3-D Finite Element Modeling of the Welding Process Using Element Birth and Element Movement Techniques," Journal of Pressure Vessel Technology, vol. 125, May 2003.

[3] Nur Syahroni and Mas Irfan Purbawanto Hidayat, 3D Finite Element Simulation of T-Joint Fillet Weld: Effect of Various Welding Sequences on the Residual Stresses and Distortions," INTECH, 2012.

[4] Muhammad ABID and Muhammad Jawad QARNI, "3D thermal finite element analysis of single pass girth welded low carbon steel pipe-flange joints," Turkish J. Eng, vol. 33, 2009.

[5] C.S. Wu, Q.X. Hu, and J.Q. Gao, "An adaptive heat source model for finite element analysis of keyhole plasma arc welding" Computational Materials Science, vol. 46, 2009.

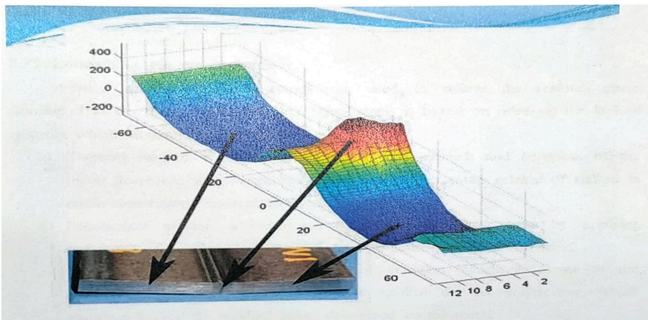
[6] C. S. PATHAK, L. G. NAVALE, A. D. SAHASRABUDHE, and M. J. RATHOD, "Analysis of Thermal Cycle during Multipass Arc Welding." WELDING JOURNAL, vol. 91, MAY2012.

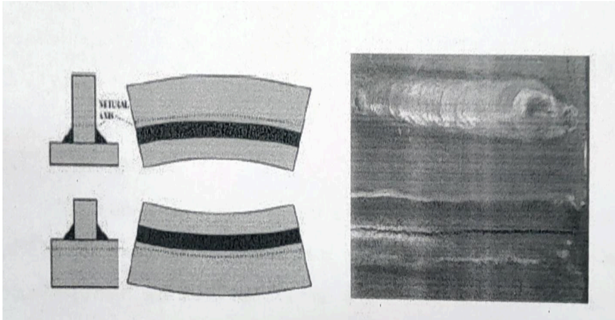
[7] JYOTI PRAKASH, DR. S.P. TEWARI, and BIPIN KUMAR SRIVASTAVA, "A REVIEW ON SOLIDIFICATION AND CHANGE IN MECHANICAL PROPERTIES UNDER VIBRATORY WELDING CONDITION," International Journal of Engineering Science and Technology, vol. 2, 2010.

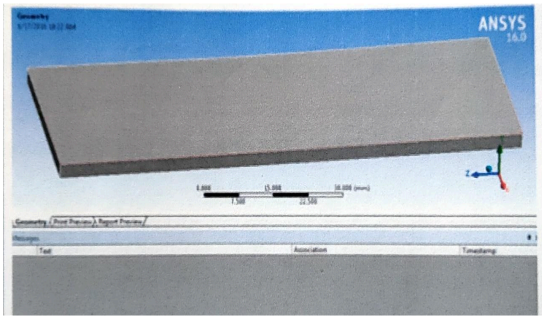
[8] M. SALCUDEAN, M. CHOI, and R. GREIF, "A study of heat transfer during Arc welding." Int. J. Heal Mars Transfer, vol 29, Jan. 1985.

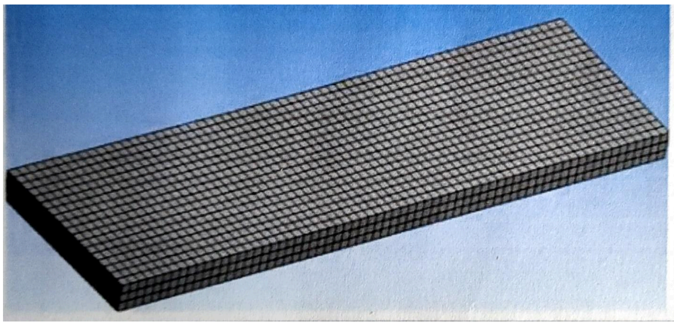
[9] S. Venkannah and J. Mazumder, "Changes in Laser Weld Bead Geometry with the Application of Ultrasonic Vibrations," Proceedings of the World Congresson Engineering, vol. 2, 2009.

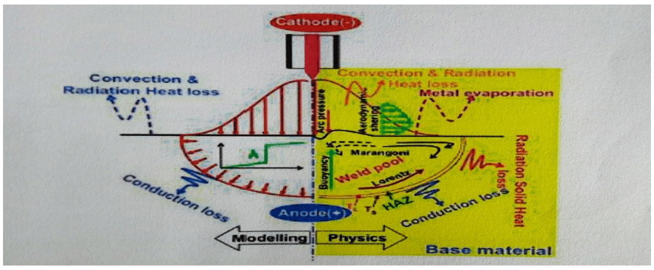
[10] B. Pucko and V. Gliha, "Charpy toughness and microstructure of vibrated weld metal" Institute of Materials, Minerals and Mining, vol 11, 2006.

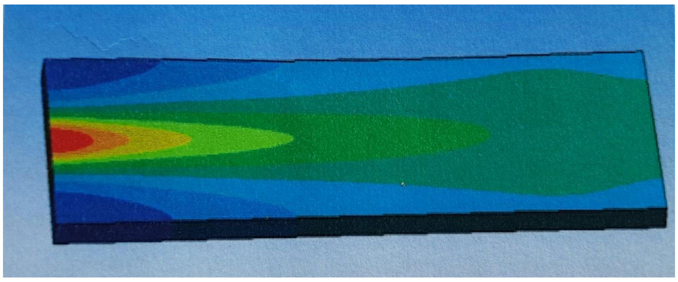

400
200
0
-200
-60
-40
-20
0
20
60
12 10 8 6 4 2

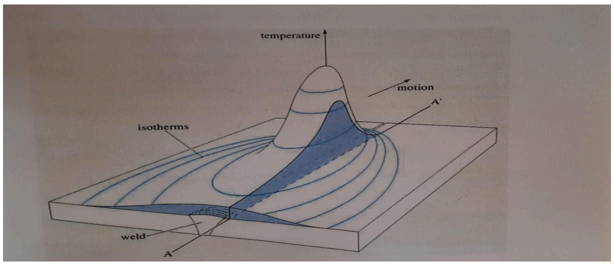


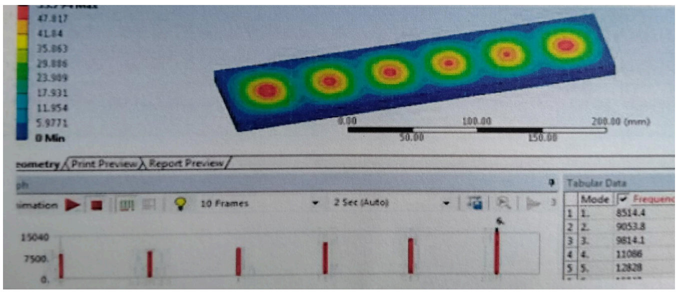

ANSYS
16.0
Geometry
Print Preview
Report Preview
Messages
Text
Association
Timestamp




Cathode(-)
Convection & Radiation Heat loss
Convection & Radiation Heat loss
Metal evaporation
Arc pressure
Marangoni
Weld pool
Buoyancy
Lorentz
HAZ
Radiation Solid Heat loss
Conduction loss
Conduction loss
Anode(+)
Modelling
Physics
Base material




temperature
motion
isotherms
weld
A
A'


47.817
41.84
35.863
29.886
23.909
17.931
11.954
5.9771
0 Min
0.00
100.00
200.00 (mm)
50.00
150.00
Print Preview
Report Preview
10 Frames
2 Sec (Auto)
Tabular Data
Mode
15040
7500.

Distance moved in traverse direction	Without vibration	Applying Vibrations				
		75HZ	80Hz	85Hz	90Hz	95Hz
(mm)	Stress value (Mpa)	Stress value (Mpa)	Stress value (Mpa)	Stress value (Mpa)	Stress value (Mpa)	Stress value (Mpa)
-35	8.73	4.13	4.133	4.1	4.1335	3.1335
-30	26.98	9.77	9.775	8.7	9.7759	8.776
-25	33.58	16.78	16.78	15.2	16.784	14.785
-20	38.38	24.81	24	22	24.814	24.815
-15	42.917	35.22	35	32	35.22	35.222
-10	45.17	41.58	41.58	40.5	41.58	41.585
-5	46.5	45.58	45.8	41.8	45.807	45.808
0	49.78	47.58	47.58	42	47.38	44.382
5	46.5	45.8	45.8	41.8	45.8	45.808
10	45.17	41.58	41.58	40.5	41.58	41.585
15	42.917	35.221	35	32	35.22	35.222
20	38.38	24.81	24	22	24.814	24.815
25	33.58	16.78	16.78	15.2	16.784	14.785
30	26.98	9.77	9.77	8.7	9.7759	8.776
35	8.73	4.133	4.133	4.1	4.1335	3.1335