IJESRT
INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

REDUCTION OF RESIDUAL STRESSES IN WELDED JOINTS USING MECHANICAL VIBRATIONS

Mr. P.Lakshminarayana
Associate professor, Mechanical Dept, Dadi Institute of Engineering and Technology, Anakapalle

DOI: 10.5281/zenodo.46505

ABSTRACT

In this study, Metal Arc Welding of rolled steel (JIS G3101 SS400) specimens were studied under computational conditions. The finite element methodology of residual stresses in butt welding of bi-linear plates is conducted using ANSYS software. The study includes a finite element structure for heat (thermal) and structural (mechanical) welding simulation. Additionally, it also comprises of a poignant heat foundation, material dumps, heat dependant material chattels, metal agility and flexibility, transient heat shift and mechanical investigation. The welding replication was measured as a sequentially coupled thermo-mechanical scrutiny and the element birth and death technique was engaged for the investigation of filler metal authentication. The residual stress dispersion and enormity in the axial course was obtained. A good agreement between the theoretical and computed results was attained.

KEYWORDS: Fabrication, Rolled Steel Joints, JIS G3101 SS400, Finite Element Method..

INTRODUCTION

WELDING is widely applied in the automotive industries to amass various metals. It is widely accepted that the welding procedure relies on a powerfully channelized heat input, which is predisposed to produce undesired residual strain and distortions in welded structures, notably in the case of lean plate structures. Hence, assimilation of the magnitude of welding distortions and illustrating the consequences of the welding conditions are made necessary. With latest computing

facilities, the finite element methodology (FEM) has become an applicable procedure for cognition and the forecasting of welding residual stress and deformation (Murthy, 1996). Moreover, the welding distortions are multiple with production dissimilarities such as measurements, welding materials and fabrication process considerations. Hence, swift and precise fabrication induced distortion for practical engineering application is more cumbersome.

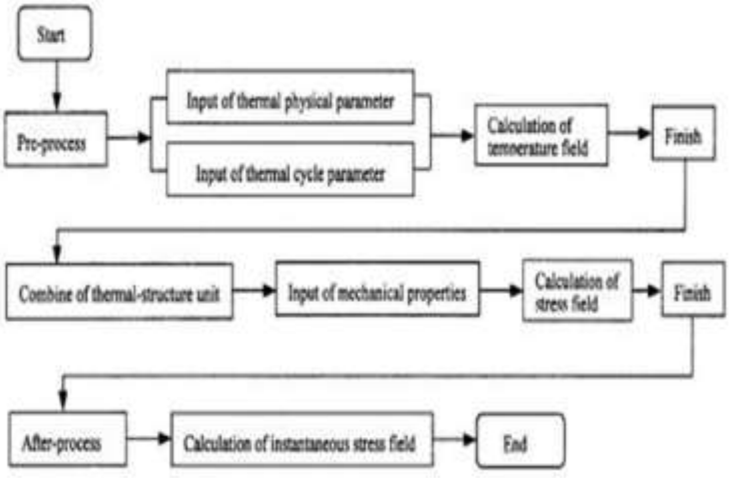
In multiple high temperature applications, it is mandatory to fuse modules of similar or varied chemical, physical and mechanical attributes. Unmistakably, the fusing of disparate metals is more painstaking than that of similar metals in lieu of variations in the properties of the fused base metals. Initially, fused structures are required to meet the potency pre-requisites and the probability of defect structure needs to be forecasted and scrutinized. Above this, lag stress could persist in any fused structure. Fabricating residual stress could be very complicated and their allotment is very difficult to forecast.

Numerous methodologies have been applied for the enumeration of residual stress in metals including stress easing methods, diffraction methodology, cracking procedures and methods by use of stress susceptible properties. These methodologies might not obtain complete stress allocation and an increasing number of these are high priced and time sensitive, while some of them are caustic to handle (Tritchkov, 2000). Off late, numerical methodology was established to address complicated engineering challenges and among them a prime place is occupied by the procedural evaluation of weld-induced residual stress. The finite element procedure is a preferred but conservative method of enumerating residual stress.

In this research, finite element analysis is adopted to execute welding replications and to forecast fabrication-induced residual stresses in butt welding of two similar rolled steel specimens. It also comprises of a poignant heat foundation, material dumps, heat dependant material chattels, metal agility and flexibility, transient heat shift and mechanical investigation.

FRAMEWORK

Illustration 1: Flow of research depicted



THEORETICAL DELIBERATION

Theoretically the welding procedure can be considered either by applying a thermal or mechanical analysis. Thermal stresses generated during a welding activity are obtained from the temperature distributions established by the thermal model. The residual stress output from each temperature increment is added to the nodal point location to establish the updated behavior of the model before the next temperature incremental.

There are two divergent methods: SEQUENTIAL and DIRECT, in a coupled-field study, and this procedure for a coupled field study will be applied, based on which fields are being coupled. The sequential method involves two or more sequential analyses that refer to a different filed.

On the contrary, the direct method normally involves just one analysis that is applicable to a coupled-field element type encompassing almost all required degrees of freedom. In this study, the process of welding is computed by the FE method. The welding procedure computation can be ideally split into two solution steps: thermal and mechanical analyses.

First, the temperature and phase evolution are pre-determined as a complete function of time in the thermal analysis model. Subsequently, mechanical analysis adopts the previous results to obtain displacements at nodes and stresses at multiple integration nodes. Since the thermal field has a strong influence on the stress field with little reverse influence, a sequentially coupled methodology works very well. Moreover, a 3 dimensional FE analysis is a highly accepted rational method of ascertaining the thermal cycle of welding. Therefore, in this study, the welding process is simulated using a methodically coupled 3-D thermo-mechanical FE formulation based on the ANSYS code. For both the thermal and mechanical analyses, temperature interlocked thermo-physical and mechanical properties of the used materials are included.

To simplify the welding simulation, it is computationally effective to conduct thermal and mechanical analyses disparately. It is an axiom that, alterations in the mechanical state do not cause a shift in the thermal state. But a change in the thermal state induces a change in the mechanical state. Firstly, the computation of the temperature history during fabrication and subsequent cooling is completed and this temperature field is adapted to the mechanical model as a body force to result in the viability to analyze residual stress.

This work includes FE models for the thermal and mechanical welding replication. In order to develop suitable welding numerical models it is mandatory to consider the process parameters (welding speed, number and sequence of passes, filling material supplying, etc. (Bonifaz, 2000)), the geometrical constraints, the material nonlinearities and all physical features and characteristics involved in welding.

Hence it is a great challenge to consider all relevant factors at the same time; on a more informal note, generally many models include some mathematical approximation: in the works (Fricke, 2002) it is identified that many serious attempts were made to reduce modeling efforts and system time. This work deals with the following main axioms and features about the thermal model:

1. The disproportionate movement of the parts, during the fabrication process, does not affect the thermal distribution of the specimens themselves;
2. All material properties are tabulated and validated till to the attainment of liquid state of metal;
3. Convection and radiation effects are additionally considered;
4. The element birth and death procedure is used, as explained previously.

Thermal analysis is the initial step and during this phase the distribution of temperature is tabulated and saved for each load step. Here it is taken that the thermal calculation at a given time is independent from the structural results obtained at a previous time according to the point (Wikander, 1994). So the thermal and the mechanical analysis can be uncoupled not only from a theoretical framework, but also computationally.

NUMERICAL EXAMPLE

The welding process of a butt-fabricated joint of two G3101 SS400 plates with the dimensions shown in Fig.1 was simulated. IN lieu of high temperature and stress gradients near the weld, the finite element model has a relatively fine mesh on both sides of the weld center line. The eight-node brick elements with linear shape functions are used to mesh the model. To simulate the dynamic heat source it is mandatory to model the heat source during each time increment. In this analysis the dynamic heat source is simplified by assuming the welding arc stayed at an element with a random constant specific volume heat generation, and then moved to the next element at the end of the load step as the welding was finished, see Fig.2. The element type SOLID70, which has a single degree of freedom, was used for thermal analysis. For structural analysis the element type SOLID45, with three translational degrees of freedom at each node, was used. Fig.3 shows the meshed model applied in the analysis.

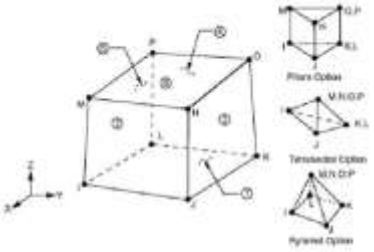


Figure 1: Geometrical Structural of SOLID 70

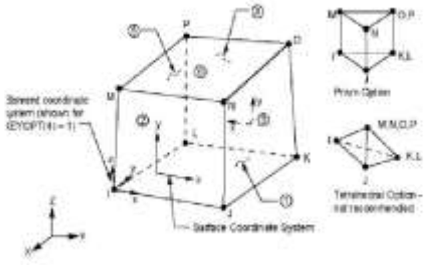


Figure 2: Geometrical Structure of SOLID 45

Figure 3: Geometry of the model used in the analysis

Figure 3(a): Meshed Model used in the anal ysis

The temperature interlocked thermal material properties for the specimens, heat affected area (HAA) and the filler weld material is assumed to be the same, see Table 1. For the mechanical material properties, same material models were applied for the weld beads and the base materials as per the yield strength. The plasticity material model used was von Misses rate-independent isotropic bilinear hardening.

MATERIAL PROPERTIES

MECHANICAL PROPERTIES	THERMAL PROPERTIES
YOUNGS MODULAS 200 MPA	thermal conductivity 36 w/mm0c
POSSION RATIO 0.26	Specific heat 711 j/kg0c
THERMAL EXPANSION 11.8E-8/C	
DENSITY 7.86E-6KG/mm3	

The heat input during welding is modeled in the ANSYS by the equivalent heat input which includes body heat flux. The amount of heat input was calculated as follows:

$$Q = \eta_V^{UI} \qquad \text{……….Eq. (A)}$$

where: η is the arc efficiency, V is the travel speed, U and I are the arc voltage and the current, respectively. In this analysis, the current was 181A, the voltage was 24.21V and the welding speed was 5.18 mm $(\text{sec})^{-1}$, while the arc effectiveness of the process considered to be 84.92 per cent. To simulate the dynamic heat source it is mandatory to model the heat source during each time incremental. In this analysis the dynamic heat source is simplified by assuming that the welding arc stayed at an element with constant specific volume heat flux, and then shifted to the next element at the end of the load step as the welding was completed. Figures 4 and 5 illustrate the temperature fields of the butt welding plates at t=6.66 sec after the start of welding, respectively.

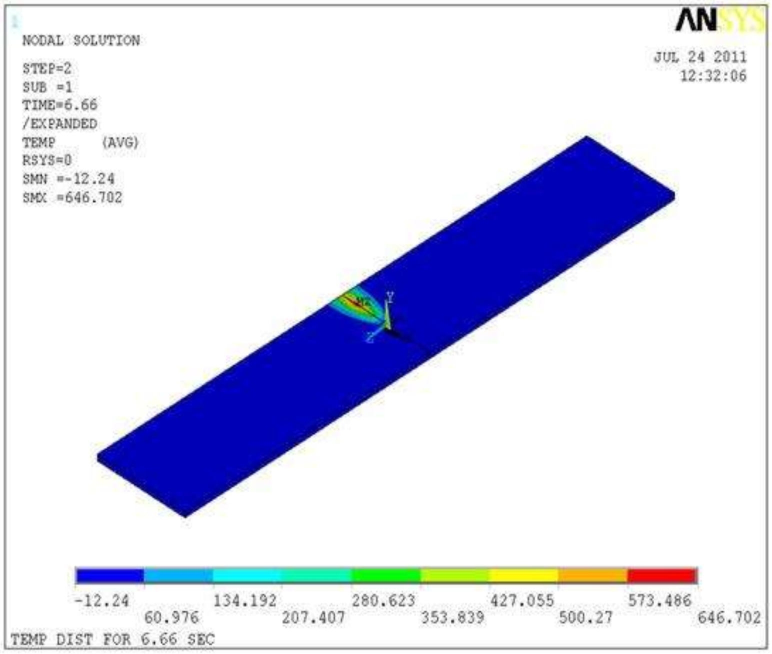


Figure 4: Temperature distribution at t = 6.66 sec

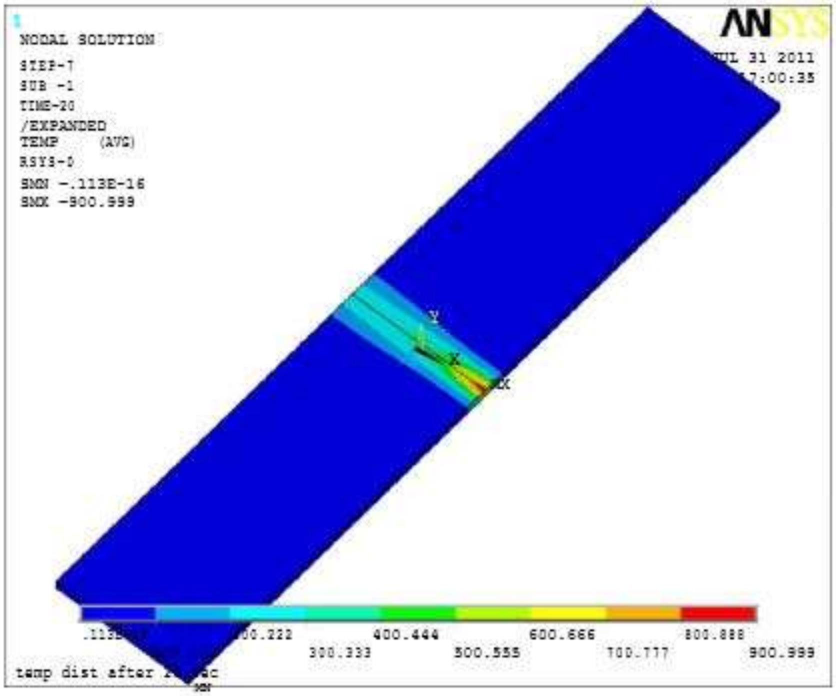


Figure 5: Temperature distribution at t = 20 sec

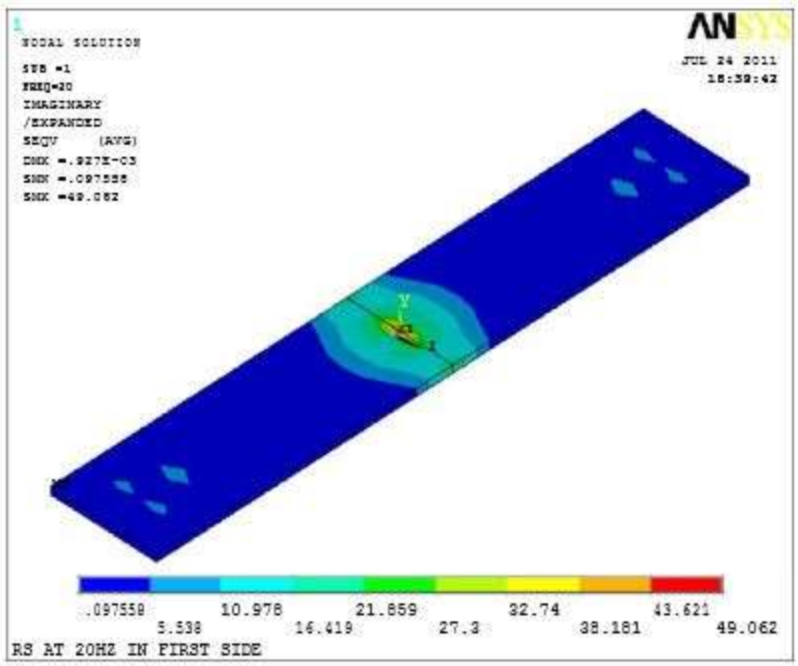


Figure 6: Residual stresses in the axial directions

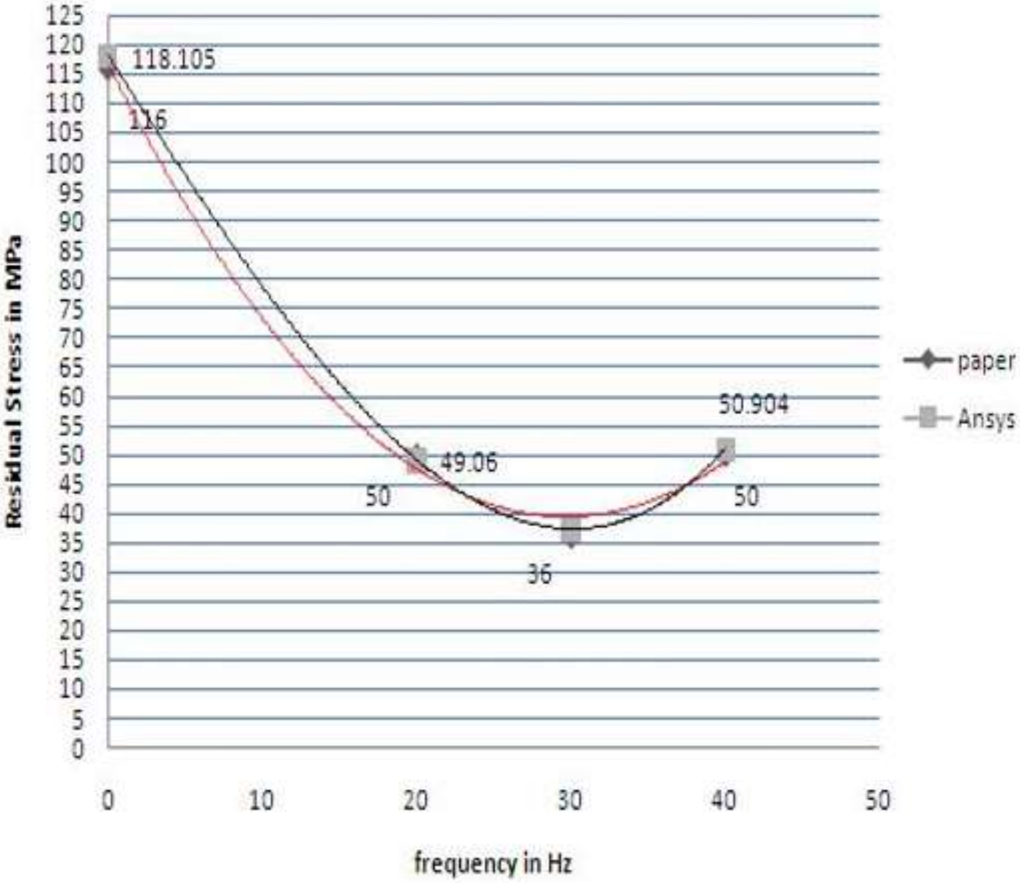
The values of axial residual stresses calculated by the finite element method are compared with the experiments, Fig.7, Fig.8; these results show that the computation finite element results are very close to the experimental results.

CONCLUSIONS

This work represents the FE model for numerical replication of welding residual stresses in high strength rolled steel butt welds (fabrication). The finite element analysis is an efficient technique in analyzing residual stresses in welding procedure. A three-dimensional finite element welding simulation was executed on a one-pass Manual Metal Arc Welding plain joint structure. The welding simulation was completed as a sequential coupled thermo-mechanical analysis and the element birth and death technique was adopted for the simulation of filler metal deposition. The finite element analysis results of the residual stress distributions of two butt welded plates in the axial directions are presented in Fig.6. Thus the values of the axial residual stresses tabulated and enumerated by the finite element method and experiment are shown in Fig.7.

These results reflect that the present results obtained by the finite element method are in close proximity to the experimental results. This method adopted by the welding simulation can be applied in other analyses. It could consider different process parameters, for instance welding speed, number and sequence of passes, filler material supplying, etc. However, multiple geometrical constraints and material nonlinearities could be included in the analysis. In order to achieve results using the finite element method very close to the results obtained in experiments, various mesh sizes could be taken into consideration.

Illustration 2: Comparison of FEM with ANSYS results on central line of the Specimen



REFERENCES

[1] MEDVEDEV,S.V.: Computer modeling of residual welding strains in technological design of welded structures, Welding International. - 2002, Vol.16(1), pp.59-65.
[2] BONIFAZ,E.A.: Finite element analysis of heat flow in single-pass arc welds, Welding Res. Supplement, 2000, pp:121-125.
[3] DENG,D., MURAKAWA,H.: Numerical simulation of temperature field and residual stresses in multi-pass welds in stainless steel pipe and comparison with experimental measurements, Comput. Mater. Sci., 2006, 37:269-277.
[4] HANSEN,J.L.: Numerical modeling of welding induced stresses, PhD thesis, Technical University of Denmark, ISBN 87-90855-52-3, 2003.
[5] JONSSON,M., KARLSSON,L., LINDGREN,L.E.: Deformations and stresses in butt-welding of large plates with special reference to the mechanical material properties, J. of Eng. Mat. and tech., 1985, Vol.107, pp.256-270.
[6] WIKANDER,L., KARLSSON,L., NASTROM,M., WEBSTER,P.: Finite element simulation and measurement of welding residual stresses, Modelling Simul.mater.Sci.Eng., 1994, Vol.2, pp.845-864.
[7] GOLDAK,J., BIBBY,M., MOORE,J., HOUSE,R., PATEL,B.: Computer modeling of heat flow in welds, Metallurgical transaction B, 1986, Vol.17B, pp.587-600.
[8] FRICKE,S.,E. keim and SCHMIDT,J.: Numerical weld modeling a method for calculating weld-induced residual stresses, Nucl.Eng.Des., 2001, 206:139-150.
[9] HSIANG,P.C., LIANG,T.T.: Numerical and experimental investigations on the residual stresses of the butt-welded joints, Int. J. Comput. Mater.Sci., 2004, 29:511-522.

[10] LINDGREN,L.E., KARLSSON,L.: Deformations and stresses in welding of shell structures, Int. J. Numerical Methods Eng., 1998, 25:635-655.
[11] SARKANI,S., TRITCHKOV,V., MICHAELOV,G.: An efficient approach for computing residual stresses in welded joints, Finite Element Anal. Des., 2000, 35:247-268.
[12] MURTHY,Y.V.L.N., RAO,G.V., IYER,P.K.: Numerical simulation of welding and quenching processes using transient thermal and thermo-elasto-plastic formulations, Comput. Struct., 1996, 60:131-154.
[13] Gunwant D.Shelake, Harshal K. Chavan, Prof. R. R. Deshmukh and Dr. S. D. Deshmukh, "Modeling and Analysis of Surface Grinding for Residual Stresses in Workpiece", International Journal of Mechanical Engineering & Technology (IJMET), Volume 3, Issue 3, 2012, pp. 344 - 349, ISSN Print: 0976 – 6340, ISSN Online: 0976 – 6359.
[14] Mr. Harshal K. Chavan, Mr. Gunwant D. Shelake, Dr. M. S. Kadam, "Finite Element Model to Predict Residual Stresses in MIG Welding", International Journal of Mechanical Engineering & Technology (IJMET), Volume 3, Issue 3, 2012, pp. 350 - 361, ISSN Print: 0976 – 6340, ISSN Online: 0976 – 6359.
[15] P. Govinda Rao, Dr. C L V R S V Prasad, Dr. D.Sreeramulu, Dr. V. Chitti Babu and M.Vykunta Rao, "Determination of Residual Stresses of Welded Joints Prepared under the Influence of Mechanical Vibrations by Hole Drilling Method and Compared by Finite Element Analysis", International Journal of Mechanical Engineering & Technology (IJMET), Volume 4, Issue 2, 2013, pp. 542 - 553, ISSN Print: 0976 – 6340, ISSN Online: 0976 – 6359.
[16] P. Govindarao, Dr. P. Srinivasarao, Dr. A. Gopalakrishna and C V Sriram, "Improvement of Tensile Strength of Butt Welded Joints Prepared by Vibratory Welding Process", International Journal of Mechanical Engineering & Technology (IJMET), Volume 4, Issue 4, 2013, pp. 53 - 61, ISSN Print: 0976 – 6340, ISSN Online: 0976 – 6359.