Finite Element Analysis for Residual Stress, Strain and Temperature Characteristics of Butt Welded Steel Plate – a Review

Deepak M.Badgujar	P.G. student S.S.B.T's C.O.E.T.,Bambhori City, Jalgaon-425001,State-Maharashtra,India
S.P.Shekhawat	Academic Director , S.S.B.T's C.O.E.T.,Bambhori City, Jalgaon-425001,State- Maharashtra, India

ABSTRACT   In this paper, the finite element analysis of residual stresses in butt welding of two similar plates is reviewed with the ANSYS software. For this paper work thermal structural direct coupled field analysis is used to simulate welding process. The residual stress distribution and magnitude in the axial direction was obtained. Finite element analysis is the effective method to carry out the stress analysis and to identify the stress gradient in the component. In this project temperature variation and stress, strain characteristics of the butt welded steel plate were studied and results were derived by Finite element analysis and coupled field analysis. This analysis is very useful for the engineers worrying in the field of welding technology and processes. Comparison between experimental residual stresses and FEA residual stresses is done. These results are found to be very much similar

I.   Introduction

Welding is widely used in automotive industries to assemble various products. It is well known that the welding process relies on an intensely localized heat input generate undesired residual stresses and deformations in welded structures, especially in the case of thin plates. Therefore, estimating the magnitude of welding deformations and characterizing the effects of the welding conditions are being necessary. Due to modern computing facilities, the finite element (FE) technique has becaome an effective method for prediction and assessment of welding residual stress and distortions . However, the welding deformations are various with production variations such as dimension, welding materials and welding process parameters. Therefore, rapidly and accurately predicting welding induced distortion for real engineering applications is more challenging.

The joining of dissimilar metals is more challenging than that of similar metals due to differences in the properties of the base metals welded. Firstly, welded structures must meet the strength requirements and the probability of defect formation must be estimated and monitored. Moreover, residual stresses may exist in any welded structures. Residual stresses due to welding may be very complex and their distribution is very difficult to predict. Many techniques have been used for measuring residual stresses in metals including stress diffraction techniques, relaxation techniques, cracking techniques and techniques

by use of stress sensitive properties. These techniques cannot obtain complete stress distribution and most of them are costly and time consuming and some of them are destructive. In recent years, numerical analyses are established to solve the complex engineering problems and among them evaluation weld-induced residual stresses. The finite element method is the conventional means of calculating residual stresses. In this paper, the finite element analysis is used to perform welding simulation and to predict weld-induced residual stresses in butt welding of two similar carbon steel plates. This analysis includes heat source which moves, temperature dependant material properties, material deposit, metal plasticity and elasticity, heat transfer which is transient and mechanical analysis.

II.   Simulation of welding process

Theoretically the welding process can be considered either using a thermal or a mechanical analysis. Thermal stresses induced during a welding process are obtained from the temperature distributions determined by the thermal model. From each temperature increment, the residual stresses caused are added to the nodal point location to determine the updated behavior of the model before the next temperature increment. [4]

There are two distinct methods: sequential and direct, in a coupled-field analysis. Which procedure for a coupled- field

analysis will be used depends on which fields are being coupled in this work; the process of welding is simulated with the FE method. The welding process computation can be split into two solution steps: thermal and mechanical analyses. First, the temperature and phase evolution are determined as a function of time in the thermal analysis. Then, the mechanical analysis employs the previous results to get displacements at nodes and stresses at integration points. Since the thermal field has a strong influence on the stress field with little inverse influence, a sequentially coupled analysis works very well. Moreover, a 3-D FE analysis is the optimum method of ascertaining the thermal cycle of welding. Therefore, in this paper, the welding process is simulated by using a sequentially coupled 3-D thermo-mechanical FE formulation based on the ANSYS code. For both the thermal and mechanical analyses, mechanical and temperature-dependent thermophysical properties of the materials are incorporated. To understand the welding simulation, it is very much efficient to perform thermal and mechanical analyses separately. It is assumed that changes in the mechanical state do not cause a change in the thermal state. On the other hand, a change in the thermal state results in a change in the mechanical state. Firstly, the computation of the temperature history during welding and subsequent cooling is completed and this temperature field is applied to the mechanical model as a body force to perform the residual stress analysis. This work includes FE models for the thermal and mechanical welding simulation. It is necessary to consider the process parameter (number and sequence of passes , filling material supplying, welding speed, etc.), the geometrical constraints, physical phenomena involved in welding, the material nonlinearities and all to develop suitable welding numerical models [3-5]

A.   Material Properties

The temperature dependant thermal material properties for the plates, heat affected zone (HAZ) and the filler weld material were assumed to be the same, see Table 1. [2] For the mechanical material properties, same material models were used for the weld beads and the base materials according to the yield strength. The plasticity of the material model used was von Misses rate-independent isotropic bilinear hardening.

III. Calculation of heat generation rate

The amount of heat input, QR has been calculated by using empirical relation shown in Eq. 1. [1] Arc efficiency is denoted by η, arc voltage by V, arc current by I . Typical welding parameters taken in this study are , arc voltage 30 volts, arc current 200 amp and arc efficiency 70%.

Heat input = efficiency x voltage x current

$$Q_R = \eta \times V \times I \quad (1)$$

B.   Why ANSYS?

The commercial analysis software ANSYS is used to simulate the

welding process using finite element analysis because ANSYS supports pre-processing solutions and post-processing. The complete simulation can be carried out using ANSYS software. Geometry is created as per dimensions .Then the solid model is meshed using solid 5 element by creating the FEA model having no. of nodes 12393 and no. of elements 8250 as shown in fig.1 and is used for further processing then in FEA solution the FEM model created is analyzed for coupled field analysis (also called as coupled field analysis).After completion of simulation the results are read and tabulated using post-processing stage in ANSYS.

TABLE 1
Material Properties [2]

| Temperature (°C)	Specific heat (J/kg°C)	Conductivity (W/m°C)	Density (kgm⁻³)	Yield stress (MPa)	Thermal expansion coefficient (10⁻5/°C)	Young's modulus (GPa)	Poisson's ratio
0	480	60	7880	380	1.15	210	0.3
100	500	50	7880	340	1.2	200	0.3
200	520	45	7800	315	1.3	200	0.3
400	650	38	7760	230	1.42	170	0.3
600	750	30	7600	110	1.45	80	0.3
800	1000	25	7520	30	1.45	35	0.3
1000	1200	26	7390	25	1.45	20	0.3
1200	1400	28	7300	20	1.45	15	0.3
1400	1600	37	7250	18	1.45	10	0.3
1550	1700	37	7180	15	1.45	10	0.3
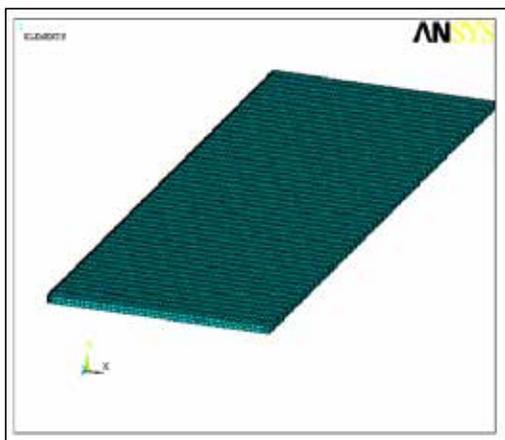
Figure 1: Meshed model

IV. REVIEW OF RESULTS
The different results obtained by using ANSYS are as below

C. Temperature at different distances across the centre line of weld with FEA
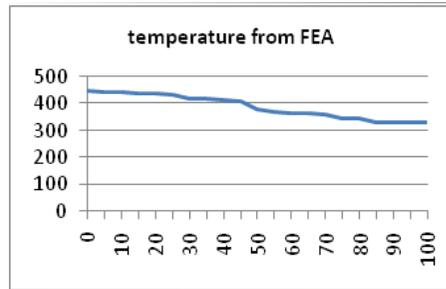A graph showing the relation between temperature and distance from weld at 100

seconds after the completion of welding process is shown in fig. 2.The relation between temperature and distance from weld can be predicted by studying the graph. We can see that temperature of welded plate goes on decreasing drastically from weld center line from 0 to 85 mm and after 85 mm it becomes almost constant up to 100 mm.

D. Strain calculation with ANSYS
The value of strain for different distances from weld center line are shown in fig.3 with the help of graph .It can be seen from the graph that the value of strain goes on decreasing as the distance from the center of weld gets increased. As stress is directly proportional to strain the value of stress gets varied according to the value of stress.

E. Comparison of the FEM with experimental results for axial residual stresses
The values of axial residual stresses calculated by the finite element method are compared with the experimental results.[1] A graph showing Comparison of the FEM with experimental results for axial residual stresses can be seen in fig. 4.These results

Distance From Weld (mm)

Figure 2: Graph between temperature and distance from center line of the weld at 100 sec. after welding .
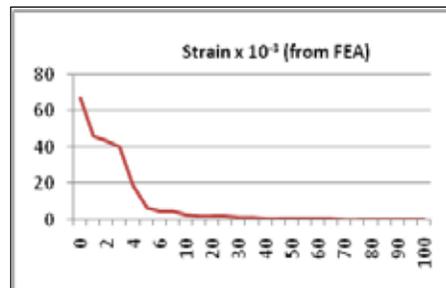
Distance from Weld (mm)

Figure 3: Graph between strain x10-3 at mid section perpendicular to weld of the ASTM36 steel plate along its length direction
are tabulated as shown in table no.2. These results show that the finite element results are very close to the experimental results. There is a little variation between the two results because in actual practice there are different types of heat losses and error in manual welding process which causes

Table no.2 stress (Mpa),(results from FEA) v/s Distance From Weld(mm)

Sr. no.	Distance From Weld(m)	Experimental Stress(Mpa)	FEA Stress(Mpa)
1	0	350	270
2	5	390	320
3	10	300	290
4	15	100	150
5	20	-120	-110
6	25	-200	-180
7	30	-125	-220
8	35	-100	-80
9	40	-50	-55
10	45	-25	-30

11	50	-5	-10
12	55	0	-5
13	60	5	10
14	65	10	20
15	70	15	20
16	75	18	30
17	80	20	30
18	85	22	35
19	90	25	40
20	95	28	40
21	100	30	40

Distance from Weld (mm)

Figure 4: Comparison of the FEM with experimental results for axial residual stresses against distance from weld ununiform distribution of heat through welding path.

V. Conclusion

The time and cost required to find residual stress in the parts due to welding process using FEA software is very less than the experimental method. Calculation of Residual stress for any complicated welding process as well any complicated structure is possible without altering the physics of the problem. Experimental stress analysis require the prototype of the structure to be analyzed while analysis FEA software eliminates that so the cost & time of product design gets reduced.With the FEA software the behavior of the structure can be seen for any number of welds. Using different mesh density the results are obtained with great accuracy. Thus FEA is only the tool for analysis which gives results very fast.

REFERENCE

]M.Jeyakumar,T.Christoper,R.Narayana-n & B. Nageswararao Residual Stress Evaluation In Butt-Welded Steel Plates, Indian Journal Of Engineering & Material Sciences,Vol.18,Pp.425-434December 2011. | [2]Dragi Stamenković & Ivana Vasović Finite Element Analysis Of Residual Stress In Butt Welding Two Similar Plates, Scientific Technical Review,Vol.LIX,No.1,Pp.57-60,2009 . | [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., Pp.269-277; 2006. | [4]Gurinder Singh Brar, Finite Element Simulation of Residual Stresses in Butt Welding of Two AISI 304 Stainless Steel Plates, International Journal on Theoretical and Applied Research in Mechanical Engineering (IJTARME) , ISSN : 2319 – 3182, Volume-2, Issue-1, Pp.75-79;2013. | [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., Vol.107, Pp.265-270; 1985.