International Journal of Mechanical and Industrial Engineering

Volume 1 | Issue 1 Article 14

July 2011

Thermo Mechanical Analysis of Hard Faced Circular Grid Plate

Shashi Kumar
Engineering Design Division, Department of Mechanical Engineering, Anna University, Chennai, shashi.ceg@gmail.com

Balaguru S
Engineering Design Division, Department of Mechanical Engineering, Anna University, Chennai, balaguru_iit@yahoo.co.in

Velamurali .
Engineering Design Division, Department of Mechanical Engineering, Anna University, Chennai, Velamurali@gmail.com

Chellapandi P
Nuclear & Safety Engineering Group, IGCAR, Kalpakkam, Chellapandi@gmail.com

Recommended Citation
Kumar, Shashi; S, Balaguru; ., Velamurali; and P, Chellapandi (2011) "Thermo Mechanical Analysis of Hard Faced Circular Grid Plate," *International Journal of Mechanical and Industrial Engineering*: Vol. 1 : Iss. 1 , Article 14.
DOI: 10.47893/IJMIE.2011.1013
Available at: https://www.interscience.in/ijmie/vol1/iss1/14

Thermo Mechanical Analysis of Hard Faced Circular Grid Plate

Shashi Kumar[1] **Balaguru S**[2] **Velamurali**[3] **Chellapandi P**[4]
[1, 2, 3] *Engineering Design Division, Department of Mechanical Engineering, Anna University, Chennai-25.*
[4] *Director, Nuclear & Safety Engineering Group, IGCAR, Kalpakkam .*
E-Mail: shashi.ceg@gmail.com,balaguru_iit@yahoo.co.in

ABSTRACT

In this paper, plasma transfer arc welding of hard faced circular grid plate was studied. Hard face deposition made by Plasma Transferred Arc Welding (PTAW) on grid plate at relatively high temperature, generates residual stresses due to differential shrinkage of the molten deposit, process-induced thermal gradients and difference in coefficients of thermal expansion between the deposit and base material. However, the magnitude and distribution of the residual stresses vary depending on the preheat temperature, heat input, deposition process, and the geometry of the component. Finite element analysis of residual stress is performed with commercial FEA package ANSYS 12.0 which includes moving heat source, material deposit, temperature dependent material properties, metal plasticity and elasticity. Coupled thermo-mechanical analysis is done for welding simulation and the element birth and death technique is employed for simulation of filler metal deposition.

Key Words: ***Residual Stress, Plasma Transferred Arc Welding, Material Property, SS304, Colmonoy.***

1. INTRODUCTION

Development of residual stresses and distortion in a welded component are strongly affected by many parameters and by their interactions. In particular, there are structural, material and welding parameters. The structural parameters include geometry of the plates, thickness, width and the type of joint. Among the material parameters, mechanical and physical properties at various temperatures and the type of filler-metal are important parameters. Welding process parameters include type of process employed, welding procedure: current, voltage, arc travel speed, feed rate and nozzle to plate distance and also arc efficiency [1]. Hard facing is one of the popular surfacing techniques employed to enhance the surface property of metal for specific applications. It is a process of depositing a filler material on the surface of carbon and low alloy steel base metal [2]. In Nuclear Industrial applications, the grid plate made of structural steel (SS304) is used extensively to withstand high temperature and heavy load conditions. During opening and closing of the nuclear reactor, metal to metal contact takes place on both the sides, leading to wear and tear and oil leakage. This wear and tear reduces the service life of the grid plate.

Among many welding processes, PTA hard facing has been increasingly employed due to the advantages such as a higher deposition rate, minimum oxidation, relatively lower dilution, improved metallurgical bond between the substrate and finally the wide applicability of the material [3].

During hard facing deposition by PTA process, the components are often subjected to high temperature and heavy thermal load conditions. As a result of the non-uniform heating and cooling, a part of the material close to the weld is subject to different rates of expansion and contraction causing development of three-dimensional complex residual stresses [4, 5]. To understand the formation of residual stress, node temperature history during surfacing technique and temperature dependent physical and mechanical material properties must be known. In particular, during the process of depositing Colmonoy over structural steel, mechanical material properties change drastically, especially when the material approaches its melting point. Therefore, due to the temperature dependent material properties and large deformation, material and geometrical non-linearity have to be taken into account [9]. It has been reported that the initial expansion of material due to temperature increase is constrained by material placed away from the heat source, therefore generating compressive stress at the centre of the weld pool [10]. Plate stiffness affects strongly the magnitude and distribution of residual stress, in particular, by the boundary conditions applied during the hard facing process. In some cases, the residual stress may equal or exceed the yield stress of the base material. The plastic strains resulting from the heating causes stress, which in turn produces internal forces that may cause buckling, bending and rotation. These deformations are called distortion [4].

In recent years, advanced numerical analysis has been applied to resolve complex problems. The finite element method is the conventional means of calculating welding residual stresses. In 1971, the application of finite element methods to analyze the welding residual stress problem has been pioneered [6,7]. Bonifaz theoretically analyzed the butt weld on the basis of the finite element method, while considering the effects of changes with temperature in the modulus of elasticity, yield stress and the coefficient of linear thermal expansion [8].

The residual stress combined with distortion results in premature failure during service. It is found from literature survey that very few attempts have been made to analyse residual stresses in grid plate. The present work aims to predict the residual stresses in a hard faced grid plate with the help of finite element analysis.

2. Material properties

SS304 is austenitic *Cr-Ni* stainless steel. Stainless steels are two-phase alloys based on the iron-chromium-nickel system. These materials typically comprise approximately equal proportions of the body-cantered cubic (bcc) ferrite and face cantered cubic (fcc) austenite phases in their microstructure. The advantages of SS are strength is twice that of austenitic stainless steels, chloride stress-corrosion cracking resistance, and pitting corrosion resistance.

Stainless steel weld ability is generally good. But the performance of SS304 can be significantly affected by welding. Due to the importance of maintaining a balanced microstructure and avoiding the formation of undesirable metallurgical phases, the welding parameters and filler metals employed must be accurately specified and closely monitored. The balanced microstructure of the base material that is, equal proportions of austenite and ferrite will be affected by the welding thermal cycle. If the balance is significantly altered and the two phases are no longer in similar proportions, the loss of material properties can be acute.

Colmonoy is a complex *Ni-Cr-B-Si-C* alloy. The average content of *Cr-B-C* precipitates is 10-15 wt. %, dispersed in a solid solution of *Cr-Fe-Ni-B* matrix. The distribution of precipitates in Colmonoy is approximately as follows: chromium carbide Cr_7C_3, 60 wt.%; chromium boride mixed, 20 wt.%; chromium carbide $Cr_{23}C_6$, 10 wt.%; chromium carboboride $Cr_{23}BC_4$,10 wt.%. Cr_7C_3 (hexagonal close-packed structure) and $Cr_{23}C_6$ (face-centred cubic structure) are not pure phases but include Ni-Fe-B [11]. The alloy hardness increases with the boron content, present mainly as chromium boride, which is very hard and very brittle (4600 HV).

SS304 is used as Base metal and colmonoy are used as hard faced material on base metal. Chemical composition of SS304 and hard faced alloy colmonoy is shown in table 1 .

Table.1 Chemical composition of SS304 and hard facing alloy

Elements	% by weight	
	SS304	Colmonoy
C	0.1	0.6
Mn	1.6	0.1
Si	0.69	3.8
P	0.04	-
S	0.022	-
Cr	18.1	11.5
Ni	8.4	77.5
V	0.48	-
Fe	69.93	4.4
B	-	2.6
Cu	0.33	

3. Finite Element Analysis

Finite element modeling and analysis are performed by using ANSYS 12.0 to predict residual stresses and distortion within the hard faced grid plate.

3.1 Model Description

A Circular grid plate with inner and outer diameter of 780 mm and 1000 mm respectively and 30 mm thick were considered to build the 3D solid model as shown in Fig. 1. The hard faced material was modeled above and below the base metal with 4 mm thickness. The hard facing process simulated is a single pass plasma transferred arc hard facing. The finite element model in this study employs the technique of element birth and death. All elements have to be created, including those that are born in the later stages of the analyses

3.2 Mesh creation

Using 8-noded brick element the solid model is meshed. Totally 17863 nodes and 14650 elements are generated in the present model. The accuracy of the finite element method depends upon the density of the mesh used for the analysis. The temperature around the welding arc is generally higher than the melting point of the material, and it drops sharply in regions away from the weld pool. Therefore in order to obtain the correct temperature field in the region of high temperature gradients, it is necessary to have a more refined mesh close to the weld line while in regions located away from weld-line a coarse mesh is generated. The mesh used in the mechanical analysis is identical to that in the thermal analysis. Fig. 2 shows the final meshed model.

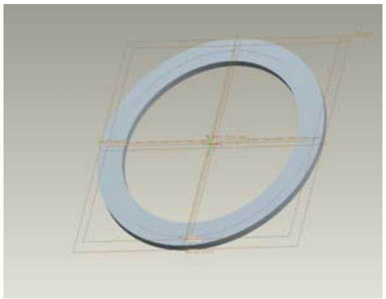

Fig.1. Solid model of hard faced circular grid plate

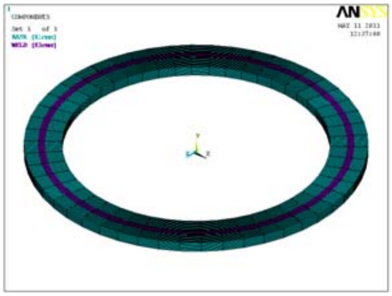

Fig.2. Meshed model used in analysis

3.3 Boundary Condition

The finite element method cuts the structure into small number of elements and interconnecting the elements through nodes. By assembling all the element matrices, it will give the total displacement of the structure. In transient thermal analysis, temperature field (T) of welded plate is a function of time (t). Thermal conduction will take place on the metal. Therefore three dimensional transient heat transfer equation is

$$k\left(\frac{\partial^2 T}{\partial X^2}+\frac{\partial^2 T}{\partial Y^2}+\frac{\partial^2 T}{\partial Z^2}\right)+Q_{int}=\rho c_p \frac{\partial T}{\partial t} \qquad (1)$$

Here Q_{int} is the internal heat source, ρ, c_p, k are density, specific heat and thermal conductivity of material respectively. During the welding, the heat will be lost due to radiation and convection. The convection and radiation heat loss are determined by the following equations:

$$q_{rad} = \mathcal{E}\sigma\,(T^4 - T_0^4) \qquad (2)$$

$$q_c = h\,(T-T_0) \qquad (3)$$

Here, T_0 is atmospheric temperature is considered as initial temperature, T is surface temperature of plate, h is convective heat transfer coefficient, σ is the Stefan-Boltzmann constant, $\mathcal{E}$ emissivity. Convection boundary condition applied to all surfaces of metal plate except bottom surface of the plate.

3.4 Thermal analysis

SS304 is used as a base metal and the nickel based hard faced alloy colmonoy used as filler metal. Temperature dependant material properties of SS304 is shown in table.2 and temperature dependent material properties of colmonoy is shown in table 3. The element type solid 70, which has a single degree of freedom having temperature at each node, is used for thermal analysis. Here the single pass welding simulation consists of 25 load steps. For filler metal deposition, element birth technique is employed. At every load step the corresponding elements will be activated and heat flux is applied. Heat flux is applied on the element after activating the element in single pass welding in each load step. The heat input for welding simulation is calculated as follows:

$$Q = \eta \frac{UI}{V} \qquad (4)$$

Where, □ is the efficiency, *V* is the travel speed; *U* and *I* are the arc voltage and the current. The PTA hard facing parameter value is given in table 4.

3.4.1 Element birth and death technique

In this model, the element birth and death technique is implemented to simulate the weld filler metal variation with time in single pass welded joints. All elements must be created, including those weld filler to born in later stages of the analysis. The method proposed here does not remove elements to achieve the ‘element death’ effect. Instead, the method deactivates them by multiplying their stiffness by a severe reduction factor. Although zeroed out of load vector, element loads associated with deactivated elements still appear in element load lists. Similarly, mass, damping, specific heat, and other effects is set to zero for deactivated elements. The mass and energy of deactivated elements are excluded from the summations of the model.

Table.2 Temperature dependent material properties of SS304

Temp. (K)	Specific heat (J/kgK)	Conductivity (W/mK)	Density (kg/ m^3)	Yield stress (MPa)	Thermal expansion coefficient (10^{-5}/K)	Young’s modulus (GPa)	Poisson ratio
300	492	15.0	7900	230	1.9	200	0.278
473	515	17.5	7830	184	1.9	185	0.288
673	563	20.0	7750	132	1.9	170	0.298
873	581	22.5	7660	105	1.9	153	0.313
1073	609	25.5	7560	77	1.9	135	0.327

1273	631	28.3	7370	50	1.9	96	0.342
1473	654	31.1	7320	10	1.9	50	0.350
1613	669	33.1	7300	10	1.9	10	0.351
1663	675	66.2	7270	10	1.9	10	0.353

Table.3 Temperature dependent material properties of Colmonoy

Temp. (K)	Specific heat (J/kgK)	Conductivity (W/mK)	Density (kg/m^3)	Yield stress (MPa)	Thermal expansion coefficient (10^{-5} /K)	Young's modulus (GPa)	Poisson ratio
300	475	10.5	7880	449	1.33	194	0.3
473	525	13.5	7830	390	1.33	182	0.3
673	553	16.6	7760	356	1.33	172	0.3
873	597	20.4	7690	339	1.33	169	0.3
1073	626	24.0	7640	305	1.33	136	0.3
1273	659	28.7	7610	274	1.33	87	0.3
1313	665	29.2	7570	265	1.33	70	0.3

An element strain is also set to zero as soon as that element killed. Similarly, when the elements are 'born' they are not actually added to the model, but are simply reactivated. When the element is reactivated, its stiffness, mass, element loads, etc. return to their full original values. Thermal strains are computed for newly activated element according to the current load step temperature and time.

3.4.2 Moving heat source

Table 4. PTA hard facing parameter value

Parameters	**Values**
Initial condition	30°C
voltage	22V
current	150A
Welding efficiency	85%
Welding speed	1.5mm/sec
Heat flow	1870 J/mm

A heat source in PTA welding is continuously travelling along the specified path on top surface of the work piece where fusion process should take place. The weld seam contains 600 elements. For the single-pass welding, the elements are grouped. The element size is 15mm in longitudinal direction. To simulate the moving heat Source, it is necessary to model the heat source during each time increment. In this analysis, totally 25 load steps are used to complete weld simulation. Time step size is increased from 0.5 to 2 seconds. The modified Newton-Raphson method is used in each time step for the heat balance iteration. The elements are activated in each load step and time given at the end of the each load step is the travelling speed of heat source. In this analysis, the moving heat source is simplified by assuming that welding arc stayed at an element with constant specific volume heat flux, and then moved to the next load element at the end of the load step. The fig.3 represents temperature distribution at the end of the welding. It is found that at end of the welding, the heat source is at the point when the welding gun completes one pass. At the point along the weld groove the temperature is getting reduced from its welding temperature of 1365K. Fig. 4 shows temperature at node 17162 which has affected by welding and raised its temperature to 1792K when the welding got reduced to this node. And the welding gun moves away from this node shows that temperature getting reduced with time.

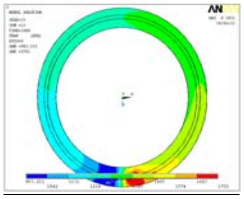

Fig. 3Temperature distribution after completion of welding

3.5 Mechanical analysis

The mechanical analysis is carried out to predict the stress and distortion in a hard faced grid plate.

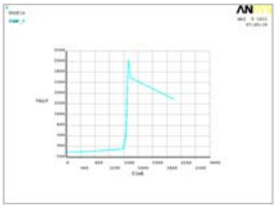

Fig.4 Temperature versus time graph after 1800 sec

The model used for thermal analysis is same as that of structural analysis. For the structural analysis, Solid 45 elements having 8 nodes and 3 degrees of freedom per node were used for static structural model analysis. With the available global temperature history from the thermal analysis, and with proper constraints, structural analysis has been carried out to predict the residual stresses induced and thereby distortion in the hard faced circular grid plate.

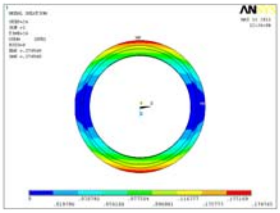

Fig.5 Distortion of welded plate

The temperature distribution obtained from the thermal analysis is used as an input for structural analysis. The thermal stresses and strains are calculated at each time increment. Distortion of the weld plate is shown in fig.5. The residual stresses from each temperature increment are added to the nodal point location to determine the updated behavior of model before the next time increment .After the FE analysis tensile residual stress at the magnitude of 179 MPa occurred inside the weld groove. And compressive stress at the magnitude of 96 MPa occurred away from weld groove. Residual stress generated in welding simulation is shown in fig. 6.

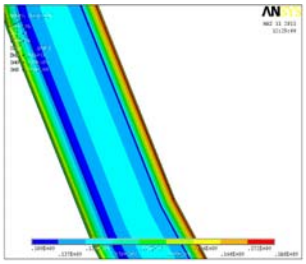

Fig.6. Max. Residual stress at weld groove

\

4. Conclusions

The finite element method is used to evaluate the residual stresses and distortion of single pass hard faced circular grid plate. Large tensile stress at the magnitude of 179 MPa occurs inside the weld groove and compressive stress at the magnitude of 96 MPa generated away from the weld. Main assumption and salient features of the work are material properties used up to melting point, convection and radiation effects considered, element birth and death technique, temperature dependant material property used and moving heat source. The welding simulation is considered as sequential coupled thermo-mechanical analysis.

We can obtain the numerous results by changing the welding parameters such as number of passes, different heat input, welding speed, changing the filler metals, changing the base metals, different mesh sizes, different weld geometry without any cost. We can also predict the heat affected zone and phase transformation of the material.

References

[1] M M Mahapatra, G L Datta and B Pradhan: Three-dimensional finite element analysis to predict the effects of shielded metal arc welding process parameters on temperature distributions and weldment zones in butt and one-sided fillet welds. IMechE 2006, 837-845.

[2] Q.X.Yang, M.Yao, J.K.Park: Numerical simulations and measurements of temperature and stress field in medium-high carbon steel specimen after hard-face-welding. Computational Materials Science 29 (2004) 37-42.

[3] Garg AK.: Plasma Arc Welding and Surfacing Process. Welding Research Institute, Department of Science and Technology, SERC School on Advances in Welding Technology. 2002: 101 - 114.

[4] Michaleris P., DeBiccari. :Prediction of Welding Distortion. Welding Journal, April 1997.

[5] Canas J, R Picon, F Paris, J Marin: Experimental and numerical analysis of residual stress in welded AL-5083-O Aluminum plates. Welding International 1994.

[6] Siva Prasad, Sankara Narayanan,.: Finite Element Analysis of Temperature Distribution during Arc Welding using Adaptive grid Technique. Welding Journal, April 1996.

[7] Murugan.S, Kumar.P.V, Gill.T.P.S, Raj.B, Bose. M.S.C.: Numerical Modeling and Experimental Determination of Temperature Distribution During Manual Metal Arc Welding. Journal of Science and Technology of Welding and Joining, Vol.4, No.6, March/1999, pp. 357-364.

[8] E.A.Bonifaz :Finite Element Analysis of Heat Flow in Single-Pass Arc Welds. Welding Research Supplement, May 2000, 121-125.

[9] Peng-Hsiang Chang, Tso-Liang Teng: Numerical and experimental investigations on the residual stresses of the butt-welded joints, Computational Materials Science 29 (2004) 511–522.

[10] Maeder G, Leburn JL and Sprauel JM: Present Possibilities for the X-ray Diffraction Method of Stress Measurements. NDT International 1981; 14 (5): 235 -248.

[11] Punitharani, K., Murugan, N., Sivagami, S.M : finite element analysis of Residual Stresses and distortion in Hard faced Component. Journal of scientific and Industrial research. VOL. 69, FEB 2010 , 129-34

[12] Metals Handbook, Vol. 3, American Society for Metals, Metals Park, OH, 1973.