Validation and Modeling of FEA Welding Simulation and Parametric Study of TIG Welding Parameter

Akash B. Chovatiya¹, P. M. Ravanan², H. S. Mahadik³
¹M.tech Student, ²Associate Professor, Veermata Jijabai Technological Institute, Mumbai, Maharashtra, India
³Manager, Larsen & Toubro Limited, Powai, Mumbai, Maharashtra, India

Abstract—One of the major problems in welded structures is the welding residual stress and welding distortion due to local heating. This work has validated maximum stress on weld bead due to welding process in ANSYS v14.5 simulation software and Experimental Procedure. Residual stress after welding is measured by X-Ray Diffraction non-destructive method. This validation helps for predicting the temperature distributions and maximum stress induced in Titanium plates due to the welding process. A Gaussian mathematical model of transient thermal process in line heating, suitable boundary conditions and non-linear material properties have been used to simulate the transient thermal analysis with moving heat source model by using finite element method. Finite element models have been developed. The effect of different parameters on the line heating response (temperature distribution) including the torch speeds, heat inputs, thicknesses of plates has been studied.

Keywords—FEA, Residual Stress, Validation, Welding simulation, X-Ray Diffraction method

I. INTRODUCTION

Welding is the fundamental process in manufacturing marine structures, ship building, process equipment etc. Welding deformation is inevitable due to non-uniform distribution of temperature and plastic yielding during welding. It depends on various factors such as plate thickness, welding speed, heat input, material properties etc. The deformation due to welding is one of the principal obstacles in enhancing the productivity in manufacturing procedure of process equipment like heat exchanger, nuclear equipment, aero-space equipment. It is more important to know how much deformation and residual stress produced due to welding with different welding parameters. The problem of welding distortions during large titanium fabrications lead to dimensional inaccuracies and misalignment of structural members. This increases the cost of production and leads to delays. Therefore, the problems of distortion and residual stresses are always of great concern in welding industry.

In order to deal with this problem, it is necessary to predict the amount of distortion resulting from the welding operations.

One way to predict the distortion and shrinkage of titanium welding is through numerical analysis such as finite element analysis (FEA).

The research activity in welding simulation started decades ago. Rosenthal (1946) was among the first researchers to develop an analytical solution of heat flow during welding based on conduction heat transfer for predicting the shape of weld pool for two and three-dimensional welds.


Pankaj Biswas et al. [6] analyzed transient temperature profile of laser welding Ti-6Al-4V Alloy. He did a 3-D finite element analysis considering effect of latent heat of fusion and convective and radiative boundary conditions.

Nirbhay L Ranpariya et al. in the year of 2015 [7] analyzed the welding process using Gaussian heat distribution model. He did temperature field analysis and used 5 mm thick AA 5083 plates for experiment model and derived good agreement between them.

Different factors affect the distribution of residual stresses in welded joints of structures. In this work, parametric studies based on numerical simulations are conducted to establish the effects of critical welding process parameter on weld induced residual stresses, temperature-time history. To validate the accuracy of the developed finite element simulation strategy, maximum residual stress produced in shop-floor TIG welded titanium plates at longitudinal direction of weld bead which is measured using X-Ray Diffraction method is compared with FEA result. In industrial applications, the fundamental process of joining of two metallic components is welding. Welding produces highly tensile residual stresses near the weld itself due to molten weld metal cooling and contracting. This pulls the surrounding material away from its equilibrium position and generates a residual stress field.
II. EXPERIMENTAL STUDIES & FEA

A. Experiment and measurement:

1. Specification of TIG welding machine: Specification of welding machine are shown in Table I which has been used to weld two plates of same dimensions of 250*75*6 mm.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Specification</th>
</tr>
</thead>
<tbody>
<tr>
<td>Supply voltage</td>
<td>8-20 volt</td>
</tr>
<tr>
<td>Supply Current</td>
<td>80-220 amp</td>
</tr>
<tr>
<td>Shielding Gas</td>
<td>Argon</td>
</tr>
<tr>
<td>Tungsten Electrode</td>
<td>2% thoriated Tungsten</td>
</tr>
<tr>
<td>Polarity</td>
<td>DC-DCEN</td>
</tr>
<tr>
<td>Work piece</td>
<td>250<em>150</em>6 mm</td>
</tr>
</tbody>
</table>

2. Experimental parameter: Parameters that have been used during TIG welding are shown in TABLE II. Also Figure I shows the welded Titanium plate.

<table>
<thead>
<tr>
<th>Sr. No</th>
<th>Parameter</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Welding Current</td>
<td>amp</td>
<td>80</td>
</tr>
<tr>
<td>2</td>
<td>Welding Voltage</td>
<td>volt</td>
<td>15</td>
</tr>
<tr>
<td>3</td>
<td>Thickness of the plate</td>
<td>mm</td>
<td>6</td>
</tr>
<tr>
<td>4</td>
<td>Welding speed</td>
<td>mm/s</td>
<td>3</td>
</tr>
</tbody>
</table>

3. Heat input: Heat Input is typically calculated as the ratio of the power (i.e. voltage*current) to the velocity of the heat source (i.e. the Arc) as follows:

\[ H = \frac{nVI}{1000 \times v} \]

\[ = \frac{15 \times 80 \times 0.6}{1000 \times 3} \]

\[ = 0.24 \text{ kJ/mm} \]

where,

- \( H \) = heat input (kJ/mm)
- \( V \) = arc voltage (volts)
- \( I \) = current (amps)
- \( v \) = travel speed (mm/s)

4. X-Ray Diffraction Method: Diffraction methods are based on determining the elastic deformation which will use changes in the inter-planar spacing \( d \) from their stress free value \( d_0 \). Then, the strain could be calculated by using Bragg’s law.

Electro-polishing had been done before putting piece of plate in X-Ray diffractometer machine. Weld bead is marked in figure 1 and that 25*25*5 mm piece was cut as shown in figure 2.

B. FEA Procedure:

There are three phases to perform FEA analysis:

i) Pre-processor for input of simulations: Creating CAD model, Application of Mesh, The loading of boundary conditions and the applied force
ii) **Solver**: executing numerical solution i.e. interpolation functions, internal virtual work of element, creating element matrices, calculations of nodal displacements, assembly of system matrices

iii) **Post-processor for evaluation of results**: Deformation and stress graphics, contour plot, natural frequency results, analysis of heat, temperature. It contains animation service.

1. **3D-Coupled Thermo-mechanical Analysis**: Figure 3 shows 3D-coupled thermo-mechanical analysis to perform welding simulation in ANSYS v14.5.

   ![Figure 3: Coupled analysis](image)

2. **CAD Model**: The overall dimensions of plate for analysis taken are 27*15*6 mm due to ease of analysis, saving computer processing time. The welding procedure is modeled as a single pass in this analysis. No overfill of the weldment were considered.

   Length of the weld bead taken is 27 mm and it is divided into 9 segments. So we can able to apply torch speed 3 mm/s giving heat flux to the area.

   ![Figure 4: CAD model](image)

3. **Meshing**: A free mesh has no restrictions in terms of element shapes and has no specified pattern applied to it. Mapped mesh is restricted in terms of the element shape and the pattern of the mesh. A mapped area mesh contains either only quadrilateral or only triangular elements while a mapped volume mesh contains only hexahedron elements.

   ![Figure 5: Mapped mesh](image)

4. **Material used and its properties**: Ti6Al4V material is used. It is recognized for its high strength-to-weight ratio. The relatively high melting point makes it useful as a refractory metal.

   Temperature dependent material properties are needed in the analysis. The modulus of elasticity is a measure of the stiffness of a material. A higher modulus material is more likely to resist distortion. The amount of expansion or contraction of a metal will undergo during a heating or a cooling cycle depends on the coefficient of thermal expansion. Thermal conductivity gives a measure of the ease of heat flow through a material.

   The melting and boiling point temperature of Ti is 1668˚C and 3287˚C respectively. Density of Ti is 4506 kg/m³. Other temperature dependent thermo-physical material properties are shown in the following tables:

   ![Table III: Temp.-dependent material properties](image)

<table>
<thead>
<tr>
<th>Tempe. (°C)</th>
<th>Specific Heat (J/kg K)</th>
<th>Tempe. (°C)</th>
<th>Thermal expansion Coefficient (E-06)/°C</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>565</td>
<td>0</td>
<td>9.0</td>
</tr>
<tr>
<td>20</td>
<td>580</td>
<td>250</td>
<td>9.5</td>
</tr>
<tr>
<td>205</td>
<td>610</td>
<td>500</td>
<td>9.9</td>
</tr>
<tr>
<td>425</td>
<td>670</td>
<td>750</td>
<td>10.5</td>
</tr>
<tr>
<td>650</td>
<td>760</td>
<td>1000</td>
<td>10.8</td>
</tr>
<tr>
<td>870</td>
<td>930</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
5. Heat flux calculation: Here, top surface area of weld bead shown in figure 4 is 11.892 mm².

\[
\text{Heat flux} = \frac{V \times T}{A} \text{ W/mm}^2
\]

\[
= \frac{15 \times 80 \times 0.6}{1000 \times 11.892}
\]

\[
= 60.54 \text{ W/mm}^2
\]

6. Transient Thermal Analysis: The room temperature was taken 22°C to all surfaces of the plate. The value of the convective film coefficient is 100 W/m²°C. Element birth and death technique has been used for applying heat flux (W/mm²) and creating moving heat source.

Boundary conditions applied are 1) frictionless support to allow translational displacement in all directions except into and out of the supported plane and 2) Compression only support to restrict movement in bottom direction and allow free movement in upper direction.

III. Results & Discussion

A. Temperature-time history graph

Here, we got temperature time history graph. Welding is a local heating process. Heat is concentrated on one point and as torch passes from one point to other point temperature drop at previous point is high.

Here, from graph we can observe that global maximum temperature at 9 sec i.e. last second for heat input is 4330.5 °C and that temperature drops to 1028.9 °C. These information can help us to decide cooling method to control metallurgical characteristic of weldment.
B. Static structural result

Static structural analysis gave the maximum residual stress in longitudinal direction of weld bead and that value is 146.62 Mpa. Figure 10 shows the condition of welded plate when it comes to the room temperature.

C. Parametric study on Temperature variation

1. Voltage: By keeping welding current, thickness, efficiency factor, welding speed constant variation of voltage vs. maximum temperature after welding gives graph shown in figure 11.

2. Welding Speed: Graph of variation of welding speed vs. maximum temperature after welding is shown in figure 12.

3. Thickness of the plate: Graph of variation of thickness of plate vs. maximum temperature after welding is shown in figure 13.

4. Efficiency: Graph of variation of efficiency factor vs. maximum temperature after welding is shown in figure 14.
Developments In Fem Prediction Of Welding Residual Stress And...  

IV. CONCLUSION  

The following conclusions are drawn from the present work:  

1. Development of 3-dimensional coupled thermo-mechanical finite element model of a TIG welding process is described in this work.  
2. Maximum residual stress in longitudinal direction of weld bead obtained from FEA procedure is 146.62 MPa which is quite near to the value 169.9 MPa i.e. residual stress measured by experiment procedure. This validate the FEA procedure.  
3. Parametric study i.e. Variation of temperature by varying welding voltage, efficiency of joint, welding speed and thickness of the plate is observed.  
4. The end temperature is gradually increasing from 1302 to 1414°C as the voltage is varied from 15 to 17 V.  
5. The end temperature is gradually decreasing from 1670 to 1080 ° C as the welding speed increases from 2 mm/s to 4 mm/s.  
6. The end temperature is gradually decreasing from 1302 to 1095 ° C as the thickness of plate is varied from 6 to 15 mm.  
7. The end temperature is gradually increases as efficiency of joint is increases.  

REFERENCES  


[15] Welding Engineering And Technology By Dr. R.S.Parmar, Chapter 2 & 4  

[16] Computer Simulation Of Welding Processes  