Simulation of Arc Welding: Methodology Development and Its Application by Using Finite Element Analysis

Deepak Pacha, Suyog Shinde

Product & Technology Development Centre, Heavy Engineering IC, Larsen & Toubro, Mumbai

Abstract: Welding is an integral part of manufacturing process. Control of distortion and residual stresses caused due to welding had been a challenge for welding community in order to fabricate complex heavy engineering components with desired dimensional accuracy. In-house development of the Welding Simulation technology has been carried out to predict weld distortion and residual stresses to reduce time, cost and rework during fabrication.

This development is based on Computational Weld Mechanics to convert actual arc welding process in to numerical domain. The methodology simulates, heat input during multiple weld passes, by using moving heat source models such as double ellipsoidal heat source model with suitable heat distribution. The user defined functions are developed in Python Script to generate and solve the numerical model of the system, including base metal component, weld material and other welding aspects by using ABAQUS. Weld material deposition is simulated by using element activation and deactivation. Sequential thermal and structural simulations are carried out to obtain thermal profile, distortion and residual stresses during welding.

The methodology has been validated using welding experimentation. It is further simplified and successfully used for control of weld distortion in number of actual manufactured components such as tube-sheet, shell and support joints in pressure vessel equipment and distortion control of tube-sheets due to Electro Slag Strip Cladding process. The details of the simulation methodology, its validation and applications are described in this paper.

Keywords: ABAQUS, Distortion prediction, Residual stresses, Weld distortion, Welding simulation

1. Introduction

The numerical simulation of the process of arc welding has been the subject of research and publications for a few decades. Enhancement of plant capacities with progression in engineering technology in past decades has resulted in to larger overall size and geometric complexity of components in Heavy Engineering Industry. The construction of heavy engineering components requires joining of multiple parts together by fusion welding, involving intense thermal cycles. During this the material gets subjected to rapid heating and cooling, shrinkage and plastic deformation. This results in to the undesirable distortion of welded components and induces residual stresses in the weld material and the heat affected zone.

The large magnitude of distortion and residual stresses may cause misalignment during joining of adjacent components. It can also be detrimental to the mechanical performance of the equipment during service. This problem is usually overcome by post-weld heat treatment (PWHT) as it reduces the weld distortion and residual stresses up to certain extent. However it is costly and can be technically challenging/unfeasible to perform in many cases. Residual stresses developed from the process of welding may significantly affect the strength of the components with respect to stress corrosion cracking, hydrogen-induced cracking and to some extent, fatigue failure.

Considering these aspects, it is important to predict the weld distortion and residual stresses in critical components and understand their behavior under welding conditions. The predicted data can be used further to optimise the welding parameters and plan the PWHT accordingly to minimize the distortion and residual stresses in final component. [1, 2]

2. Welding simulation methodology

Welding is a highly dynamic process which involves various physical phenomena such as heat input, behavior of molten filler metal, weld pool dynamics, microstructural evolution, change of material properties and the overall thermo-structural response of the component under very dynamic thermal environment. Different domains representing these phenomena and their coupling between each other during welding are as shown in Figure 2.1.

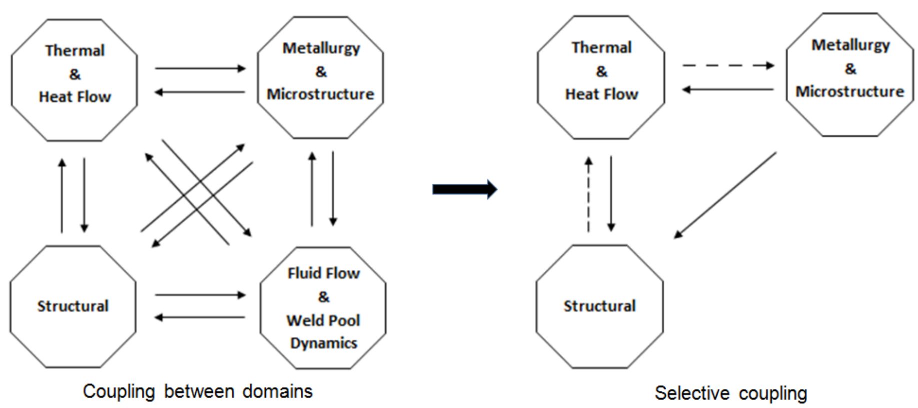


Figure 2.1 Domains in weld behavior

It is indeed very cumbersome to account for all the coupling effects. In computational welding mechanics very high degree of simplification is used to convert the actual complex welding process in to numerical form. Many of the couplings are neglected depending upon their influence on final weld distortion and residual stress and based on the application.

In the developed methodology, the effect of weld pool dynamics (Fluid flow) and it's coupling are neglected based on literature study. In this way, the overall coupling domain simplifies to the one with inclusion of heat flow (thermal), structural and metallurgical domain. This results in to a Welding simulation model in the form of thermal-structural coupling as shown in Figure 2.1. Since the effect of mechanical deformation on heat flow has been ignored, the fully coupled (bidirectional) thermo-mechanical phenomenon of welding can be safely broken down in to unidirectional coupled analysis. According to this simplified approach a thermal analysis is performed first to determine the temperature profile of the system over the complete duration of welding operation. This thermal analysis is followed by a structural analysis. The transient temperature profile determined by thermal analysis is used as thermal loading for structural analysis. In this way the effect of transient thermal conditions on structural behaviour of welded component is determined in the form of distortion and residual stress. During the thermal and structural analysis, the effects of thermal variations on metallurgical changes are included by using temperature dependent material properties in the range of room temperature to melting point of the material. However the effect of welding on metallurgical parameters such as microstrure and hardness are not considered in this methodology.

The user defined subroutines are generated using Python script to transform the overall methodology in to FEA based numerical domain. The model has been generated and successfully solved by using ABAQUS.

Some of the important aspects in the simulation methodology are as below,

2.1 Heat source modelling

Accurate consideration of heat input and its distribution is important to simulate the actual effect of welding. Many heat source models are developed in the past which represent the molten metal pool and its heat input distribution as welding progresses. The heat source is sequentially moved along the path of welding to simulate the heat input effect at high temperature molten metal. The most widely accepted model for heat source is double ellipsoidal heat source model, presented by Goldak et al [1,5]. The same has been primarily used in this methodology.

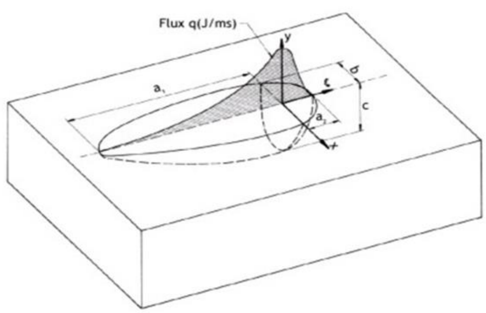


Figure 2.2 Goldak's double ellipsoidal model

As shown in Figure 2.2, this model is based on the consideration that, the power density distribution of welding heat occurs in the form of double ellipsoidal shape.
The heat flux throughout the heat source (representing molten pool) may not be the same across its volume. However it should be more at the front half and gradually decrease towards rare end.
Accordingly, the power density distribution in the heat source model is function of location (viz. x, y, z) in the heat source model and time (viz. t) during welding. This distribution is represented in terms of two different equations (viz. Equation 1 and Equation 2) for front and rare halves of the heat source model as below [5]

Front side:

$$q(x,y,z,t) = \frac{6\sqrt{3}\, f_f Q}{abc\pi\sqrt{\pi}} \quad e^{-3x^2/a^2}\, e^{-3y^2/b^2}\, e^{-3[z+v(\tau-t)^2/{C_1}^2} \qquad \text{------- (1)}$$

Rare side:

$$q(x,y,z,t) = \frac{6\sqrt{3}\, f_r Q}{abc\pi\sqrt{\pi}} \quad e^{-3x^2/a^2}\, e^{-3y^2/b^2}\, e^{-3[z+v(\tau-t)^2/{C_2}^2} \qquad \text{------- (2)}$$

In the present simulation methodology, the user defined subroutines are developed to calculate the centroid distance of nodes from the moving arc center corresponding to the arc position at any instant during welding. The power density distribution had been calculated and applied at relevant locations by using developed user defined subroutines using Python script [3].

2.2 Material modelling

The accurate implementation of computational weld mechanics depends mainly on the accurate modeling of temperature dependent material properties. But this is a difficult task due to insufficient material data available at elevated temperature i.e. in the range from room temperature up to melting temperature of material. In the past, several research efforts have been dedicated to the investigation of material properties and their effect on the structural response under transient thermal loading during welding.
In the computational weld mechanics, microstructure evolution is addressed either by a direct or an indirect method. In the direct method, calculations are made from thermal history, various phase fractions, properties of each constituent and deformation history. On the other hand in an indirect method calculations are made by considering the temperature dependent properties of the material at elevated temperatures, as those properties are effect of microstructural changes.
The indirect approach has been adopted in the developed methodology and desired material properties are obtained from various literatures. Suitable extrapolations of the properties are also carried out based on the study of material behavior. The user defined subroutines are generated in the form of material property data base. [1,2]

2.3 Numerical aspects and material deposition modelling

Welding simulation is highly nonlinear coupled thermo-mechanical phenomenon with elasto-plastic behavior of the material. Varies techniques have been studied and implemented in the methodology to solve the numerical model by using FEA tool ABAQUS.
Modeling of material deposition during welding process is also critical part of the simulation. This effect has been implemented by using element activation and de-activation techniques in ABAQUS [3].

3. Welding simulation of multi pass butt joint by using developed methodology

One of the welding simulation cases which was carried out along with the experimental validation is described in following sections, this includes welding of two 30 mm thick plates made up of SA 516, Gr 70 material. The simulation was carried out based on the developed methodology to weld these plates with butt joint with multiple number of passes as shown in Figure 3.1.

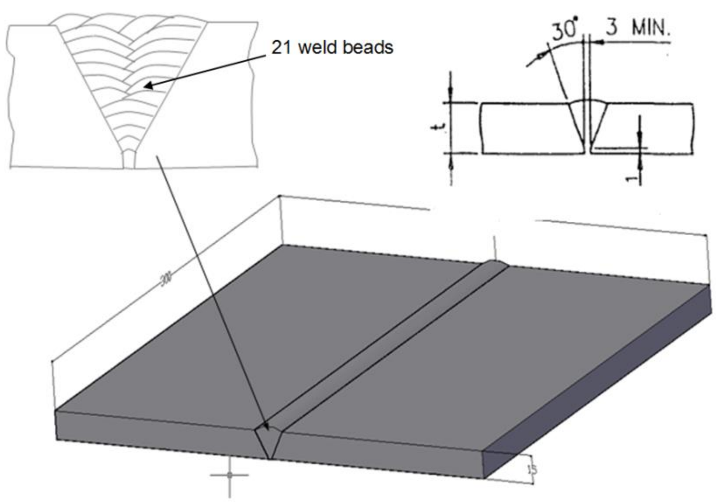


Figure 3.1 Welding setup and WEP

3.1 Simulation model and analysis

The sequential thermal and structural analysis has been carried out in ABAQUS platform using developed user defined functions to incorporate the methodology of welding simulation. Simulation model as shown in Figure 3.2 and 3.3 show two plates along with the weld beads. The plates are freely supported on the ground, however for simulation purpose, one of the plates is considered to be fixed at the end. This constraint simulates the actual support behavior as overall model is free to distort on the unconstrained end.

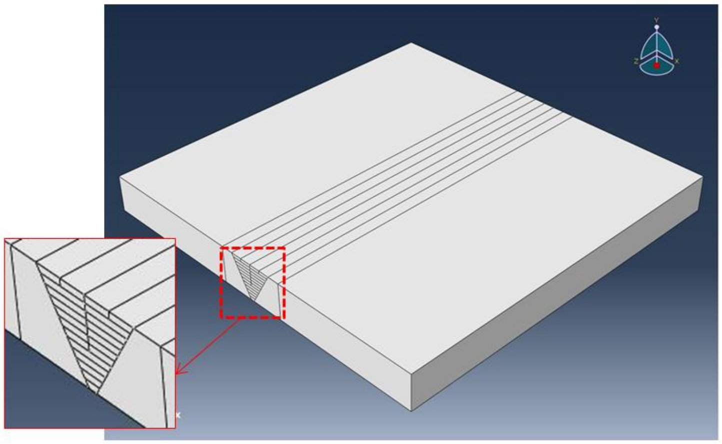


Figure 3.2 Simulation Model

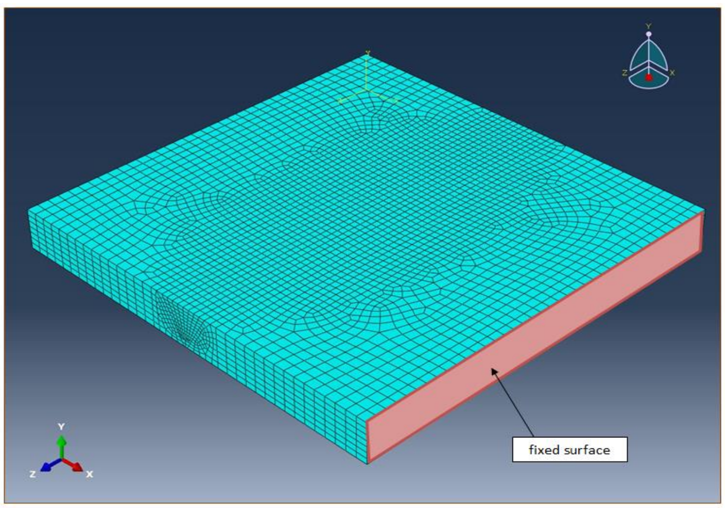


Figure 3.3 Finite Element Model with constraint

The thermal analysis includes considerations such as heat input through double ellipsoidal heat source model, application of preheat and interpass temperatures and heat loss from plate to surrounding. Around 1600 steps are generated corresponding to respective positions of heat source during welding to carryout complete transient thermal analysis.

The temperature profiles obtained from thermal analysis are used as input loading in subsequent transient structural analysis to achieve weld distortion and residual stresses. The element activation and deactivation has been defined to simulate the effect of material deposition in both thermal and structural runs.

3.2 Results and discussions

Thermal profile of molten pool and adjacent regions and its effect on the temperature across base metal can be seen in the temperature plots as shown in figure 3.4 and figure 3.5.

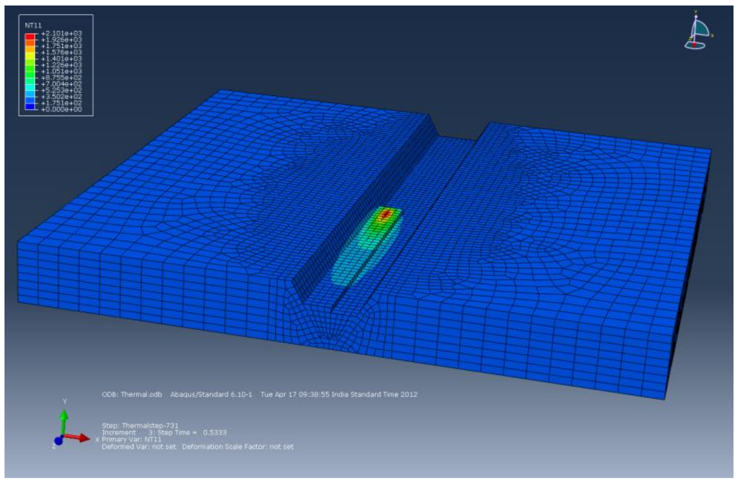


Figure 3.4 Temperature profile at intermediate stage

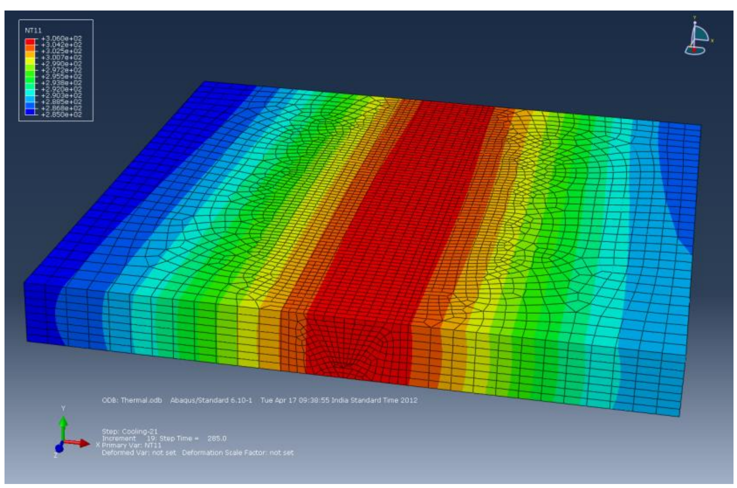


Figure 3.5 Temperature profile at the end of welding

Figure 3.6 and 3.7 shows the typical distortion and equivalent stress plots obtained at the end of the welding. Similar results can also be studied during the progression of the welding. It was found that the weld plates were distorted in vertical direction after each bead. The final vertical distortion at the free end of the plate was observed to be 9.2 mm.

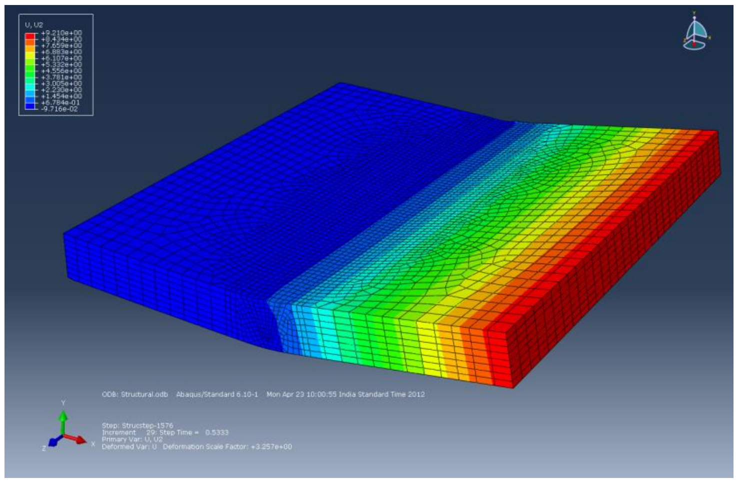


Figure 3.6 Vertical distortion (in mm)

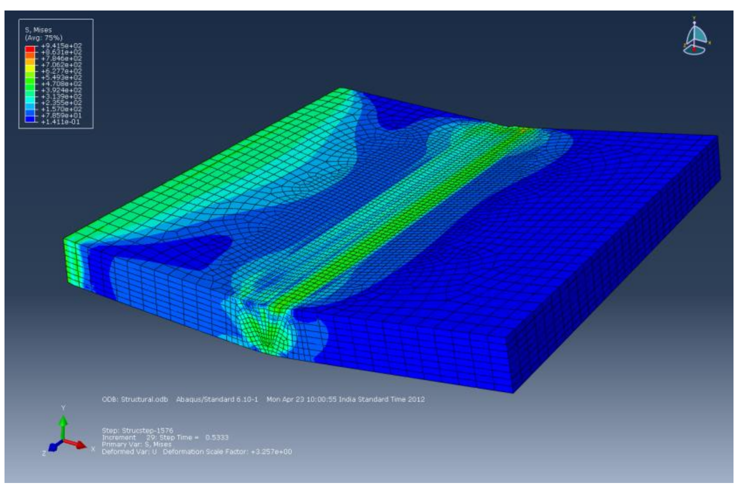


Figure 3.7 Equivalent stresses (in MPa)

3.3 Methodology validation

An experimentation was carried out by welding two actual plates of SA 516,Gr 70 material keeping all parameters such as geometry and weld specifications similar to that of simulated case.

The temperature profile, distortion and residual stress data were recorded during experimentation and at the end of welding as shown in figure 3.8. [8]

Figure 3.8 Temperature, distortion and residual stress measurement

A comparative study was carried out between the recorded data and simulated data. It was observed that, the temperature profile and distortion obtained by simulation was closely matching with the recorded data. The residual stress pattern was found similar to that of measurements and closely matching with desired pattern.
Figures 3.9 to 3.11 shows some of the results from evaluation study of simulated data vs. recorded data.

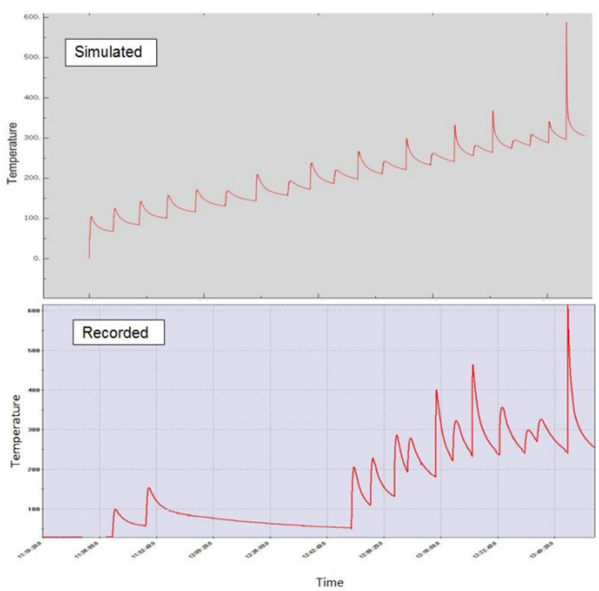


Figure 3.9 Simulated vs. recorded temperatures at typical location on plate

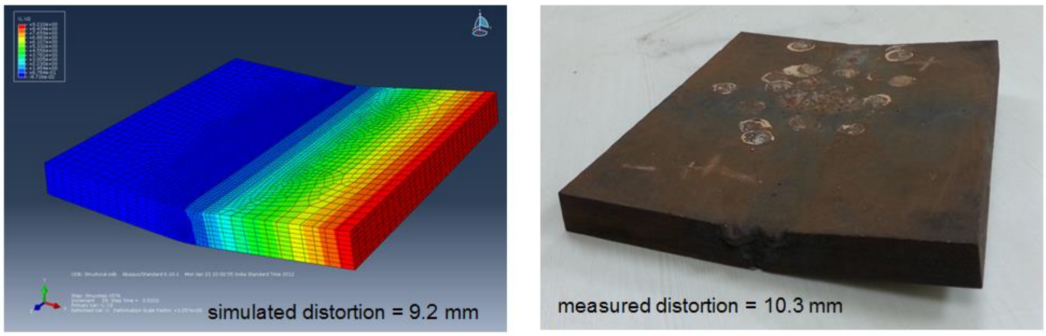
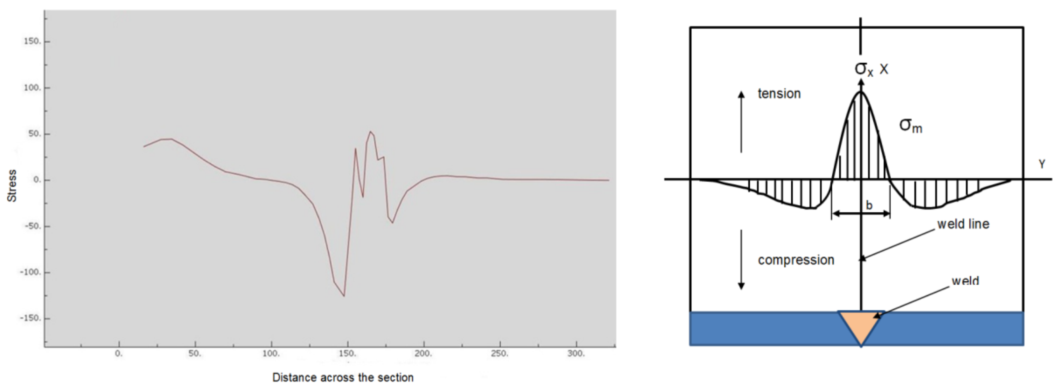


Figure 3.10 Simulated vs. recorded distortion at the end of welding



simulated residual stress pattern

standard residual stress pattern

Figure 3.11 Simulated stress pattern across transverse section against the standard stress pattern [2]

4. Technology application to actual manufactured components

The methodology described in earlier sections delivers satisfactory results. However it's implementation to the welding of actual components with complicated geometries and bigger size demands enormous amount of computational time and capability. Simple approaches are further developed based on the basic methodology to predict the weld distortion and to understand the behavior of welded components for optimisation of welding process. The methodology of 3-D simulation has been simplified in to 2-D approach based on applications. The heat source model with uniform heat distribution across bead volume has been used in some applications for quicker prediction of distortion. These modified methods are validated on simpler components and subsequently successfully implemented to predict the distortion behaviour of many actual welding cases.

In one of the cases, simulation of cladding on ‘Y’ forging of a Process Equipment had been successfully carried out. The simplified 2-D Axisymmetric approach had been utilized for this case. Many numbers of simulations were carried out to understand the behaviour of ‘Y’ forging (shell to support skirt junction) under different cladding sequence and configurations.
The simplified simulation approach had also been applied to the Electro Slag Strip Cladding of 115 mm thick tube-sheet as shown in figure 4.1. The simulations were carried out on ten different configurations with variation in cladding sequence.

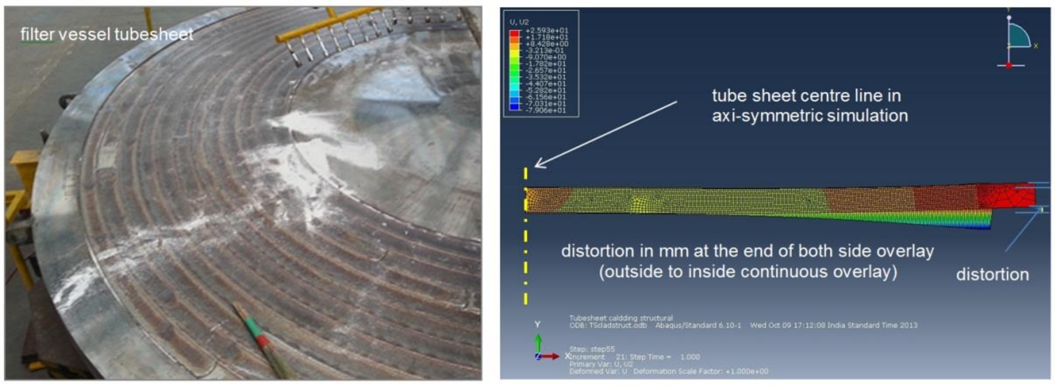


Figure 4.1 Tube-sheet cladding simulation

The results helped in determining the best possible configuration with least distortion of tube-sheet after cladding. With the help of this study the post cladding flatness of tube-sheet was achieved within the allowable limit of 3 mm and the total cycle time of cladding operation was reduced to one third of the original required time.
The methodology has also been applied for simulation of multiple number of identical type of welds on large equipment vessels. Figure 4.2 shows the base section of the vessel with multiple numbers of welds. It was challenging to carryout simulation of large number of welds. Hence one of the welds was simulated by using detailed simulation methodology. The results were obtained

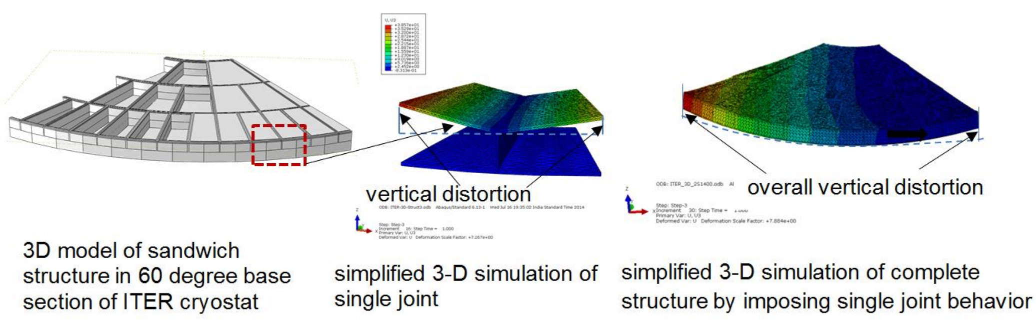


Figure 4.2 Simulation of sandwich structure of ITER cryostat base section

in terms of temperature distribution, distortion and residual stress caused due to single weld. This effect was imposed on the overall complex structure at respective welds to predict the distortion behavior of overall base section. This predicted behavior helped in optimization of actual welding procedure resulting in reduction of time, cost and rework during fabrication.

5. Conclusions

The methodology for simulation of Arc Welding is developed by combination of Computational Weld Mechanics and advanced numerical techniques. The prediction of distortion and residual stresses has been carried out for multiple pass welding of simple components and the developed methodology of weld simulation was successfully validated. In order to reduce the computational time, simplified procedures have been developed based on the case studies analyzed by using detailed simulation methodology. The results obtained by these simple analysis help in understanding the distortion behaviour of weld components by quick prediction of distortion patterns. Results achieved by such analysis along with engineering experience have been successfully applied to many practical problems to reduce the weld distortion and residual stresses. The next stage of this development is to make the simplified methodologies more accurate and implement those for various practical applications.

6. References

1. John A. Goldak , Mehdi Akhlaghi, Computational Welding Mechanics, Springer, USA, 2005
2. Masubhushi K, Analysis of Welded Structures, Pergamon Press, USA, 1980
3. ABAQUS User's Manual, Version 6.13
4. Goldak, J., Bibby, M., Moore, J., House, R., and Patel, B. Computer modeling of heat flow in welds. Metallurgical Transactions B, 17 (3) (1986) 587-600
5. Goldak, J., Chakravarti, A., and Bibby, M. A new finite element model for welding heat sources.Metallurgical Transactions B, 15 (2) (1984) 299-305
6. C.T. Karlsson, B.L. Josefson, Three-dimensional finite element analysis of temperatures and stresses in a single-pass butt-welded pipe, ASME J. Pressure Vessel Technol. 112 (1990) 76–84.
7. Hong J.K., Dong Y., Tsai C.L. and Dong P., Finite Element Modelling of Residual stress in Austenitic Stainless Steel Pipe Girth Welds, Welding Journal, 17 (10) (1997) 442-449
8. ASTM E837-01, Standard Test Method for Determining Residual Stresses by the Hole-Drilling Strain Gage Method