3rd Specialty Conference on Material Engineering & Applied Mechanics
3e Conférence spécialisée sur le génie des matériaux et mécanique appliquée

Montréal, Québec
May 29 to June 1, 2013 / *29 mai au 1 juin 2013*

ESTIMATION OF RESIDUAL STRESSES IN THICK STEEL PLATES DUE TO WELDING THROUGH FINITE ELEMENT SIMULATION

Omar A. Ibrahim, Dimitrios G. Lignos, Colin A. Rogers
Department of Civil Engineering and Applied Mechanics, McGill University

Abstract: The utilization of thick steel plates, to construct built-up structural components, has become a common practice in modern steel construction around the world. This is due to the need to build taller structures. Additionally, the design specifications require subjecting the structural components of these structures to extreme loading due to earthquakes, wind and/or blast. In thick plates, due to high restraints and long welding time, the welding process induces stresses in adjoined steel plates. The likelihood of occurrence of cracks and dislocations in these plates depends on the plate thickness. These cracks and dislocations may propagate under these induced stresses. In this paper, a detailed finite element model is developed to simulate the welding process of heavy built-up box sections. The simulation is divided into two phases: 1) heat flow and temperature distribution through the welding process; 2) temperature data from phase one are transferred to a stress analysis model that estimates strains and corresponding stresses at each stage of the welding process.

1. Introduction

Heavy built-up sections involving thick steel plates (> 50mm) (AISC 360-10), are commonly used in modern steel building construction around the world. Using standard steel shapes of higher strength is not always the best choice, because typically with increased material strength comes reduced plastic deformation capacity. In some cases no significant improvement in resistance is achieved when structural stability is the main concern. For these reasons built-up box columns constructed of thick plates that are welded together are often selected for use by engineers.

The heat applied to the base metal through welding causes it to expand according to the heat flow. Regions with higher temperature expand more than the ones with a lower temperature distribution. The high temperature regions are restrained by those with lower expansion which produces compression stresses. During the cooling phase the base metal contracts and is restrained by lower temperature regions producing tension stresses in the restrained regions (Figure 1). The magnitude of these tension stresses may reach the steel yield stress (Bate et al. 1997, Chen et al. 1993). In thick plates, due to high restraints and long welding time, the stresses induced by the welding process are significant and in the range of the yield stress of the steel material. The likelihood of occurrence of cracks and dislocations in these plates increases with the plate thickness. These cracks and dislocations may propagate under these induced stresses. In the past, cracks reported in the base metal were associated with these residual tensile stresses (Doty 1987, Fisher et al. 1987, Blodgett et al. 1993). This research addresses the residual stresses that are generated in heavy built-up box columns due to welding procedures.

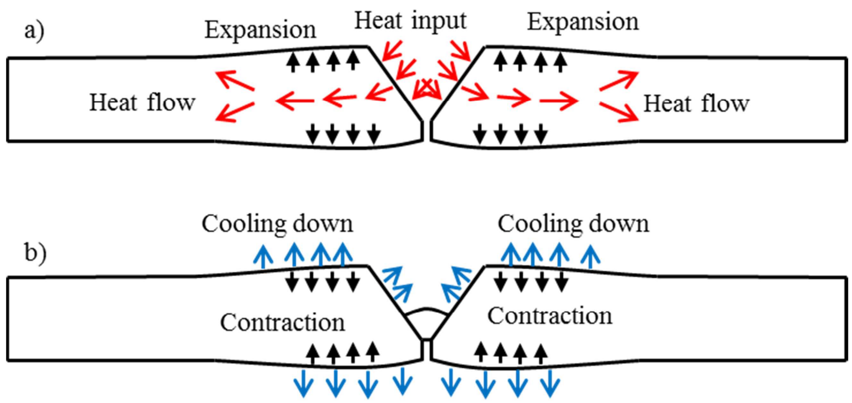


Figure 1: Illustration of the stresses induced through welding; a) Shows the material expanding, which results in compression stresses near the weld. b) Shows the material shrinking, which results in tension stresses near the weld

For heavy built-up box columns an automatic submerged arc welding procedure (SAW) is typically used in the industry. Each corner is divided into a number of passes according to the electrode size, with two corners welded simultaneously. In practice and according to AISC Guideline 21 (Miller 2006) it is recommended to decrease the number of passes as much as possible, but also to not increase by much the welding pass size to keep the heat input within acceptable limits such that the size of the heat affected zone is not increased. In order to keep the restraints as low as possible during the welding process, one pass is performed on one side. The built-up box column is then flipped and the other side is welded. The built-up box column is then flipped again to the first side for the next weld pass. This process is continued until the entire built-up shape has been fully welded, with the number of flips increasing with thicker plates.

In summary, the main parameters of the submerged arc welding procedure affecting the magnitude and distribution of the residual stresses in a built-up box column are as follows: (1) the steel plate thickness, (2) the steel grade (3) the number of passes, (4) the welding sequence, (5) the welding temperature, (6) the preheat temperature, (7) the welding speed, (8) the time between passes, (9) the presence of stiffeners and their spacing, (10) the cooling rate, and (11) the structural element setup in the fabrication plant. Miller (2010) has also identified the importance of the aforementioned parameters during the welding process of built-up members.

This paper contains a discussion of the development of a 2-D and 3-D finite element simulation model that replicates the welding procedure of built-up steel columns. This model is employed in order to investigate how the 11 parameters that were summarized above affect the induced residual stresses in heavy built-up box columns. The proposed finite element model and method are presented and verification is completed of their applicability to the study of the influence of the welding parameters on the stress output.

2. Physical Model

A number of heavy built-up box columns with plate thicknesses 75mm A572 steel grade 50 were scrapped by a well-known steel fabricator in North America due to the development of cracks after the welding procedure. Figure 2 shows a schematic representation of the cross section of such column. It is worth mentioning that the fabricator followed the recommendations given by AISC Guideline 21 (Miller 2006) for welding thick plates; however these recommendations are of a qualitative nature.

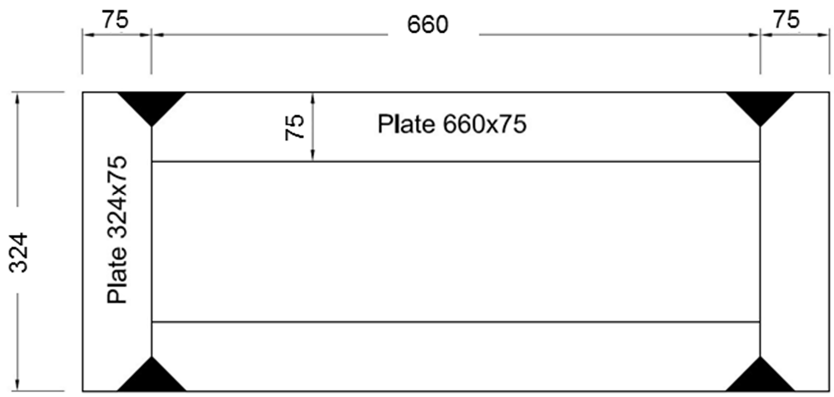


Figure 2: Cross-section dimensions of the built-up box column used in this research

The welding of each two corners at the same side of the built-up column shown in Figure 2 is completed simultaneously. Each corner consists of 13 weld passes (see Figure 3b). For this thickness, the welding sequence according to the industry and AWS D1.1 (2010) guidelines is summarized as follows:

- Weld one pass on one side (side-1) of the built-up box column.
- Flip the built-up box column and weld the other side (side-2) to pass number 4.
- Flip again and weld side-1 to pass number 7.
- Flip and weld the other side-2 to pass number 7.
- Flip and weld the other side-1 to pass number 13.
- Flip and weld the other side-2 to pass number 13.

The region to be welded is pre-heated to the welding temperature (1500 ˚C) allowing the heat to flow through the base metal. A weld pass is then added, after that the pass is left in contact with the room temperature for certain time until the welding electrodes are returned to the start position and the pass surface is cleaned. Then the region is heated again for the second pass and so on. During this heating and cooling the base metal experiences expansions and contractions (strains) resulting in stresses according to the material law at the corresponding temperature. An investigation of the effect of the 11 listed parameters on the stress output due welding is the general aim of the paper. It was not practical to carry out such an investigation of heavy built-up box columns experimentally; therefore, a finite element simulation was developed incorporating the effect of the different welding parameters on the stress output.

3. Finite Element Model

In order to investigate the effect of the welding sequence and to identify an optimal procedure to minimize the effect of residual stresses during the welding process of the built-up section discussed in Section 2, a finite element model was developed in ABAQUS v6.11 (ABAQUS 2011). The modeling procedure involved two numerical models. The first one was a transient heat transfer model to measure the temperature distribution at all times through the welding process. The second one was a static model to determine the stresses induced by the welding process. The two models were integrated together to simulate the whole process. Figure 3a shows the finite element model of the built-up box column. The sequence of passes at one corner joint is shown in Figure 3b. The geometry and the locations of the mesh elements have to be exactly the same in both modeling stages. The subsequent sections discuss specific details of both models.

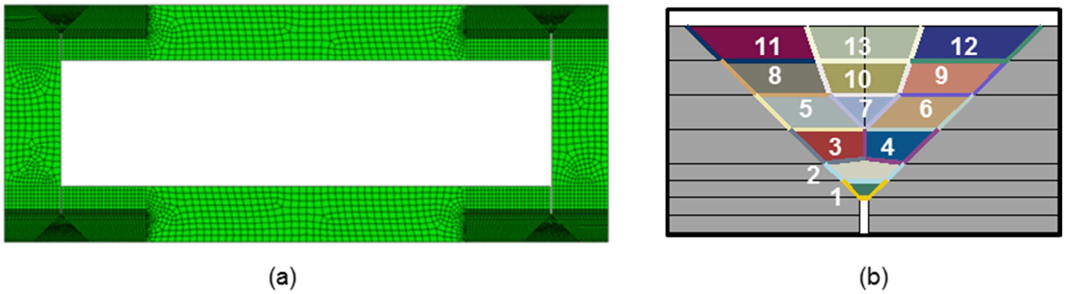


Figure 3: Numerical 2-D model of the built-up box column; (a) finite element model; (b) sequence of passes at a corner joint

3.1 Heat Transfer Model

The output of the heat transfer model is the temperature distribution due to the welding of each pass. For this model important material's parameters such as the thermal conductivity and the specific heat were defined as temperature dependant, also the heat transfer coefficient and the emissivity were defined. After the welding of each pass, heat convection and radiation boundary conditions were assigned to the surfaces in contact with the atmosphere. The first step of this model was to deactivate all the passes from the model and then reactivate them according to their order in the welding process; this is called the "death and birth" technique (Brickstad et al. 1998). The temperature distributions through the welding pass and cooling phase were generated by means of this numerical analysis process.

3.2 Stress Analysis Model

The stress analysis model imports the temperature distributions due to the welding and cooling of each pass from the heat transfer model. These temperature distributions were then translated to strains according to the defined thermal expansion coefficient at different temperatures. Subsequently, these strains were translated to the corresponding stresses according to the defined stress-strain relationship. The modulus of elasticity and yield strength of the steel material were defined as temperature dependent parameters. Figure 4 shows the reduction in the elasticity and strength of steel according to Eurocode-3 for fire design (CEN 2003).

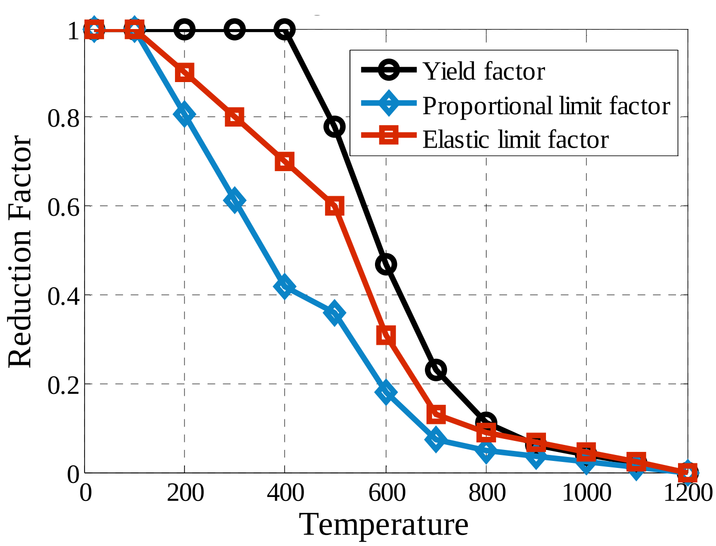


Figure 4: Steel properties reduction factors at elevated temperatures (Eurocode-3, CEN 2003)

Figure 5 displays a chart that summarizes the modeling method discussed earlier to simulate the welding procedure:

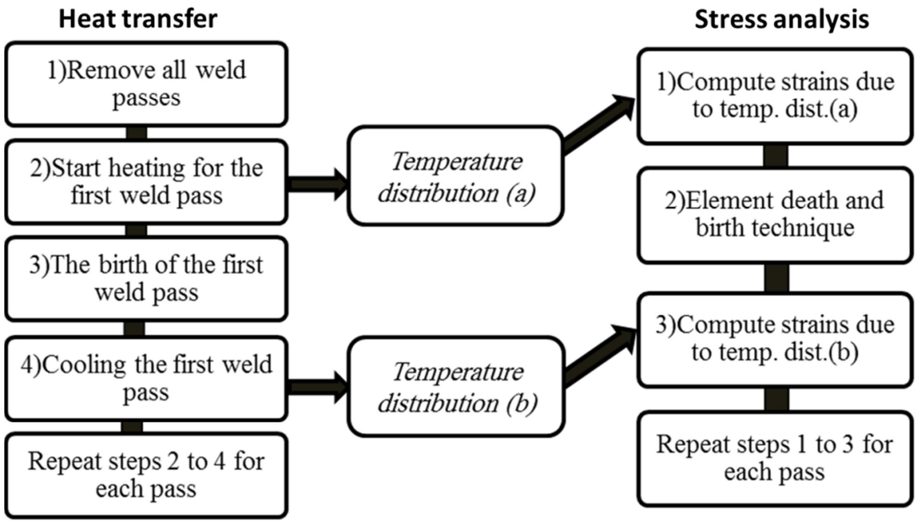


Figure 5: Summary of the technique used for simulating the welding procedure using finite element analysis

3.3 Justification of Modeling Approach

During the initiation of the welding process in the physical model, the welding passes do not yet exist. To simulate this as realistically as possible in the heat-transfer model at the first step, all the passes were deactivated in the numerical model. However, this is not possible in the stress analysis model; instead, the passes were all activated at the initial time before welding and they were assigned the temperature of welding (1500 °C). These passes were then activated according to their order in the welding procedure. This was done since the stress analysis model, unlike heat-transfer model, experiences deformations after each pass. If the stress model was treated in a similar fashion to a heat-transfer model it will experience numerical errors due to the deformation of the boundaries of the base metal in contact with the weld pass before a welding pass activated. This will result in the nodes of the welding pass not coinciding with the base metal. The attempts done to adjust the nodes of the passes with the adjacent nodes in the base metal will induce artificial stresses. This is shown schematically in Figure 6.

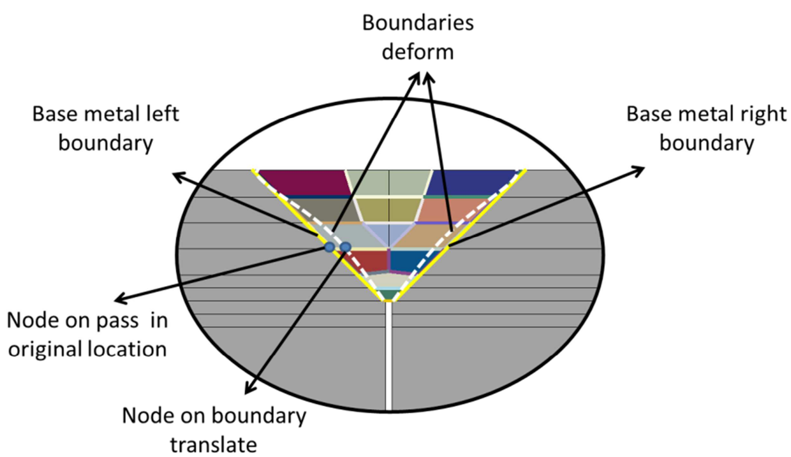


Figure 6: Illustration of the deformations between the base metal and the weld when removing all the weld passes at the initial time of the stress analysis model

To verify that the proposed modeling technique would not affect the stress values and distributions through the welding process another stress analysis model was developed by applying the “death and birth” technique as in the heat-transfer model. This new model was set to run and stopped after finishing the fourth pass on the bottom side. The results at this stage were compared with the original stress analysis model. In the following discussion, the original model is referred to as model-1 and the second model as model-2. A corner joint in both models after the completion of welding pass number four (immediately before model-2 was stopped) is shown in Figure 7; this figure also illustrates the location of the nodes that were studied to verify the simulation technique. Node A is located in model-1 at the last pass to be welded, and node B is located in both models on the base metal.

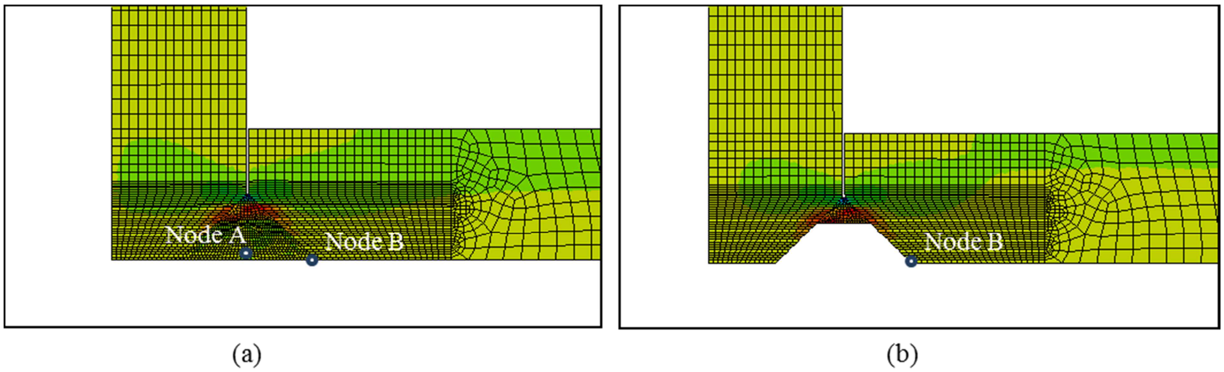


Figure 7: Finite element simulation of the welding passes; Corner joint at: a) model-1 b) model-2.

Figure 8 displays the variation of the stress values at a node in the last weld pass of model-1 (Node A) shown in Figure 7a through the welding process until this pass is activated. The stress values are normalized with respect to the yield stress of the steel material (f_y=360MPa). The value of the maximum stress at this node was about 0.3% of the material’s yield stress. From this figure it is confirmed that although all the passes were activated in the simulation of the welding process; the passes that haven’t been welded were not affected by the prior weld passes. Figure 9 shows the Von Mises stress after finishing the 4th pass in model-1; it is clear that the stress at the passes that haven’t been welded yet is almost zero.

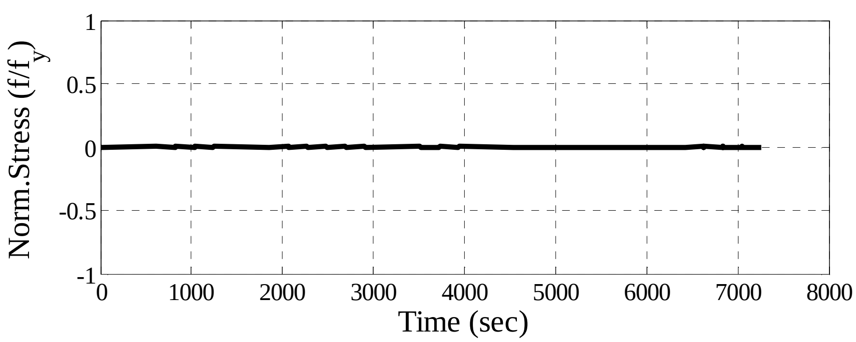


Figure 8: Normalized transverse stress to yield stress ratio at node (A) in model-1 along the welding process

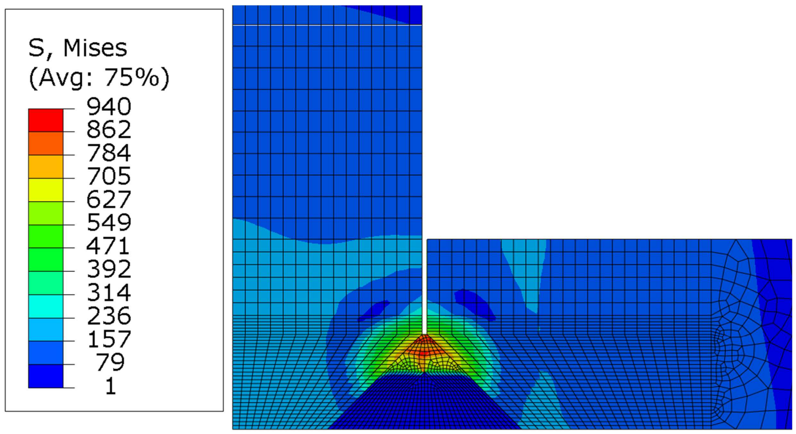


Figure 9: Von Mises Stress distribution (MPa) after finishing the 4th pass in model-1

Figure 10 displays a comparison between the transverse stresses at node B in both models up to the welding of the 4th pass (the point model-2 was stopped). The two models gave almost identical results. This demonstrates that the modeling technique used for the simulation had no effect on the stress output in the built-up box column, and that this technique is acceptable to be used for the simulation of the welding procedure.

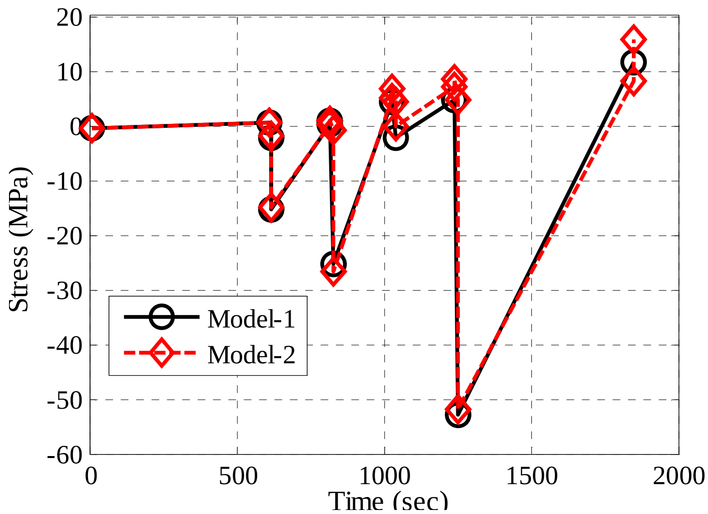


Figure 10: Transverse stress comparison between model-1 and model-2 at node (B)

4. Mesh Sensitivity Study

In order to validate the mesh size to be employed for the detailed finite element analysis of the welding sequence of built-up box columns a mesh sensitivity study was conducted and is summarized herein. This sensitivity study was conducted for both the 2-D and 3-D welding processes using a simpler model than that shown in Figure 2. A quarter of a box section composed of 28 mm thick plates connected with a complete joint penetration (CJP) weld consisting of 10 passes was utilized for this purpose. Figure 11 shows the mesh sizes that were implemented according to the mesh sensitivity study. Four models were established in 2-D and in 3-D; the mesh varied from a coarse mesh to a very dense mesh. The results of each two consecutive meshes were compared until the difference was no longer significant, then the coarser mesh was selected. For the 2-D model, the elements used were linear quadrilateral; the selected mesh was of minimum element size 1mm and the maximum 7mm. For the 3-D model the elements used were linear hexahedral, the selected mesh was minimum element size 2mm the maximum 9mm. Linear elements were selected for the analysis because in ABAQUS time integration in transient problems was done with the backward Euler method, which is unconditionally stable for linear problems. Shear locking was avoided because the deformations in this analysis were very small compared with the size of the model; also quadratic elements were tried and showed no significant difference in the results from using the linear elements. Hourglassing was avoided by not using reduced integration and also by comparing the result with that of models built with quadratic elements.

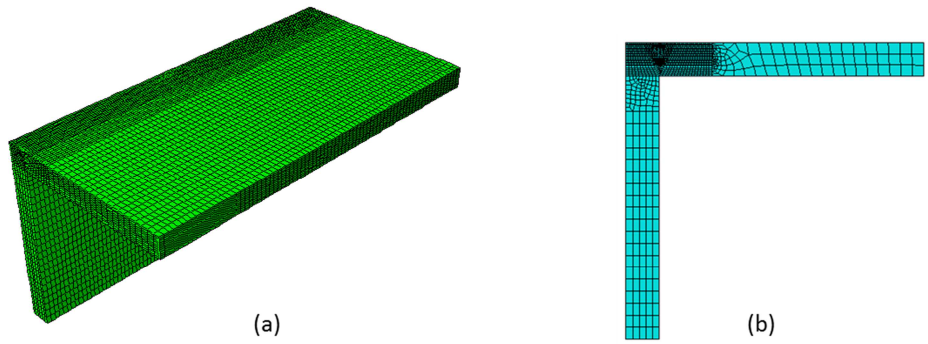


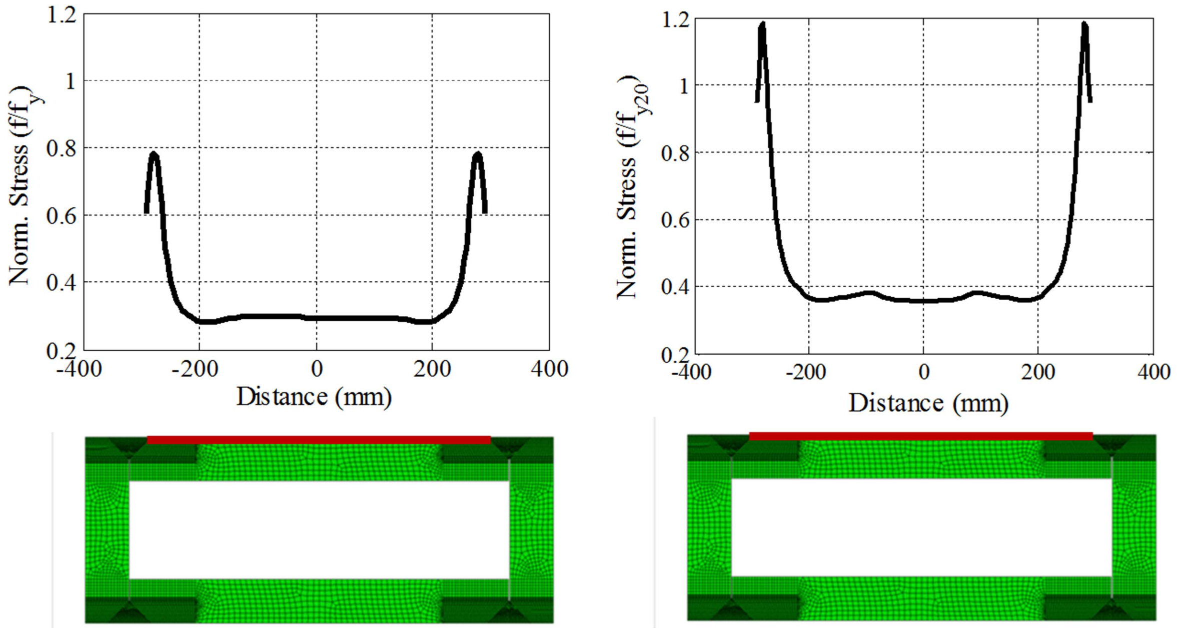
Figure 11: Suggested meshes to be used in simulation of the welding process; a) 3-D model, b) 2-D model

5. Results Discussion

A 2-D model of the built-up box column was developed (Figure 3a) based on the modeling principles discussed herein. The box dimensions were 810x320x75 mm with a 45˚ groove for the partial joint penetration (PJP) weld. The steel yield stress and ultimate strength at 20˚C are 360 and 520 MPa respectively. Based on industry feedback, the number of passes for this weld should be twenty-four; such that the first two passes are done with one electrode. Two grouped electrodes are used for the later passes, as such; the number of passes in the model was thirteen (see Figure 3b).

The time it takes to weld a single pass in the 2-D model was 10sec; the time between passes was assumed to be 200 seconds and the time it takes to flip the built-up box column upside down was 600sec. The boundary conditions were considered according to the setup used in the fabricator shop, such that the built-up box column was supported laterally from one side only. These parameters were assumed based on feedback from steel fabricators.

Figure 12 shows the stress distribution on the outer surface of the horizontal plate normalized to the yield stress of the steel at room temperature (20˚C). Figure 12a shows the stress distribution after finishing all the welding and cooling to room temperature. It can be seen that the stress levels are higher near the weld than away from the weld. The stress levels reach about 80% of the yield stress of the steel material. Due to the large thickness of the steel plate, its outer surface is all under tensile stresses. Figure 12b shows the stress distribution at a point through the process such that the welding is finished on one side of the built-up section. From this figure, it is clear that the stress levels are higher than the yield stress of the steel material. This indicates the formation of plastified zones near the weld. Figure 13 shows the size of the plastified zones formed as a result of the welding process.



a) Temp. everywhere is 20˚C   b) Temp. is higher than 20˚C

Figure 12: Stress distribution on the outer surface of the horizontal plate; a) after finishing all the welding at room temperature, b) after finishing only one side

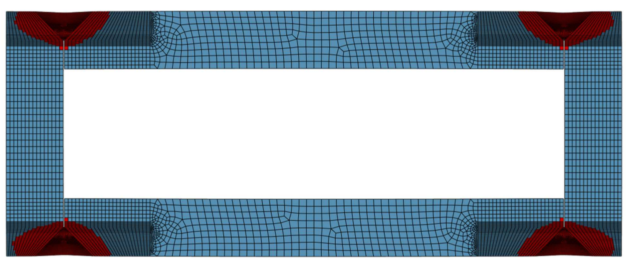


Figure 13: Plastified zones due to the welding process

6. Summary and Future Work

In summary, the method of simulating the welding procedure of heavy built-up box columns has been established and verified for detailed finite element analysis. The simulation was divided into two phases: 1) heat transfer analysis, 2) stress analysis. The temperature distributions from the heat transfer analysis were exported to the stress analysis model through the welding process. The stress analysis model transforms the differences in the temperature distributions to strains and then to corresponding stresses. In the stress analysis model all the passes were activated in the model at the onset of the analysis to avoid numerical errors, which was then followed by the “death and birth” technique analysis. This modeling technique was verified. Additionally, a mesh study was conducted for the welding simulation and mesh sizes were recommended for 2-D and 3-D simulations. Preliminary results have been established at different stages through the welding process.

Furthermore, tests of the plate material from the scrapped columns at different temperatures will take place to calibrate the material properties used in the detailed finite element model. The specimens will be extracted from different locations in a fully welded box-column, a partially welded box-column and a non-welded steel plate. The thickness of the plates forming the built-up box columns is 75mm of A572 grade 50 steel. The locations of the material test specimens will be set on the outer surface and in mid-thickness of the plates to capture the difference in material strength through the thickness; specimens will also be taken near and far from the weld to determine the effect of heating and cooling on the material strength near the weld. Through the subsequent welding of the existing partially welded box column, thermo-couples will be set on the plates being welded to determine the temperature distribution during the welding process.

After verifying the modeling procedure used for simulating the welding process, this model will be used to study the effect of each welding parameter on the stress output. Additionally, the welding procedure for heavy built-up box columns will be re-evaluated to decrease its effect on the residual stress developed in built-up box columns

Acknowledgments

The authors sincerely thank the ADF Group Inc. and DPHV Structural Consultants for providing information about the welding procedures and also providing the test specimens. The authors also acknowledge the financial support from the Natural Sciences and Engineering Research Council of Canada. Any opinions, findings, and conclusions or recommendations expressed in this paper are those of the authors and do not necessarily reflect the views of sponsors.

References

ABAQUS Ver. 6.11 2011. © Dassault Systèmes*, 2010 Dassault Systemes Simulia Corp., Providence*, RI, USA.

American Institute of Steel Construction. 2010. ANSI/AISC 360-10*. Specification for Structural Steel Buildings*, AISC, Chicago, IL, USA.

AWS D1.1. 2010. *Structural Welding Code—Steel. 22nd Edition*, American Welding Society, Miami, FL, USA.

Bate, S. K., Green, D. and Buttle, D. 1997. A Review of Residual Stress Distribution in Welded Joints for the Defect Assessment of Offshore Structures, *Health and Safety Executive – Offshore Technology Report.*

Blodgett, O.W. and Miller, D.K. 1993. The Challenge of Welding Jumbo Shapes, *Welding Innovation, James F. Lincoln Arc Welding Foundation*, Vol. X, No.1, pp. 3–16.

Brickstad, B., Josefson, B.L. 1998. A Parametric Study of Residual Stresses in Multi-Pass Butt-Welded Stainless Steel Pipes, *Journal of Pressure Vessel and Piping 75*, pp. 11-25.

CEN -European Committee for Standardization. 2003. Eurocode 3-Design of steel Structures- Part 1-2; *BS EN 1993-1-2, General Rules- Structural Fire Design.*

Chen, S. and Chang, S. C. 1993. Residual Stresses in Welded Jumbo Box Columns, *J. Construct. Steel Research 25*, pp. 201-209.

CWB/Gooderham Centre. 2005. Welding for Design Engineers, *Canadian Welding Bureau*, 7250 West Credit Ave., Mississauga, ON, Canada L5N 5N1.

Doty, W.D. 1987 Procedures for Thermal Cutting and Welding Heavy Structural Shapes, *Proceedings of the National Engineering Conference and Conference of Operating Personnel*, New Orleans, USA, pp. 16-1–16-6.

Fisher, J.W. and Pense, A.W. 1987. Procedures for Thermal Cutting and Welding Heavy Structural Shapes, *Proceedings of the National Engineering Conference and Conference of Operating Personnel*, New Orleans, USA, pp. 18-1–18-47.

Miller, D.K. 2006. AISC Steel Design Guide 21 Welded Connections – A Primer for Engineers, *American Institute of Steel Construction*, Chicago, IL, USA.

Miller, D.K. 2010. Welding heavy Structural Steel -- Successfully, *2010 NASCC: The Steel Conferenc*e, Orlando, FL, USA.