

Modelling the Welding Process of an Orthotropic Steel Deck

Evy Van Puymbroeck^{1,a,*}, Nouman Iqbal^{1,b} and Hans De Backer^{1,c}

¹Ghent University, Department of Civil Engineering EA15, Technologiepark 904, 9052 Ghent, Belgium

^aEvy.VanPuymbroeck@UGent.be, ^bNouman.Iqbal@UGent.be, ^cHans.DeBacker@UGent.be

Keywords: Finite Element Modelling, Welding Simulation, Residual Welding Stresses, Orthotropic Bridge Deck

Abstract. A three-dimensional finite element welding simulation procedure is developed with the software Siemens NX and solver type SAMCEF in order to determine the residual stresses of a welded component of an orthotropic bridge deck. The welding process of a deck plate which is welded to a closed trapezoidal stiffener is simulated. A decoupled thermal-mechanical analysis is performed. During the thermal analysis, the temperatures introduced by the passage of the welding torch are calculated for different time steps. This temperature field is used during the thermal analysis to determine the residual welding stresses for the same time steps. The decoupled thermal-mechanical analysis gives the distribution of the residual stresses in the longitudinal direction. Only this direction is discussed since this is the direction that follows the welding path. On the deck plate there are tensile yield residual stresses near the weld region. In between the welded webs of the stiffener, there are compressive residual stresses. For the longitudinal stiffener, there are again tensile yield residual stresses near the weld which decrease at a greater distance and turn into compressive residual yield stresses. The finite element model results in a residual stress distribution introduced by the welding operation. This distribution can be used in future research to determine the effect of residual stresses on the fatigue life of this welded component in the orthotropic bridge deck.

Introduction

During the welding operations of bridge components, residual stresses are introduced due to local plastic deformation. The presence of residual stresses affect the final in-service performance of the weldment, such as fatigue and brittle fracture behavior [1]. Welding residual stresses may be beneficial or detrimental with respect to the load-induced stresses, depending on the magnitude, sign and distribution of the residual stresses. Tensile residual stresses, especially in the magnitude of the material's yield stress are detrimental because they increase crack growth and prevent crack closure. On the other hand, compressive residual stresses increase the crack closure, thus decreasing the crack growth rate. To increase the accuracy of fatigue life prediction of welded bridge components, it is essential to incorporate the effects of the residual welding stresses into the structural integrity and fatigue assessment [2]. Therefore, reliable residual stress prediction has to be developed. The advantage of a reliable finite element model is that it can be used for a wide range of welding conditions and bridge component geometries [3].

A three-dimensional finite element welding simulation is developed with the software Siemens NX in order to determine the residual stresses of a welded component of an orthotropic bridge deck. The welding process of a deck plate which is welded to a closed trapezoidal stiffener is simulated. A decoupled thermal-mechanical analysis is performed. During the thermal analysis, the temperatures introduced by the passage of the welding torch are calculated for different time steps. This temperature field is used during the thermal analysis to determine the residual welding stresses for the same time steps.



Geometry

The welding process of the longitudinal stiffener-to-deck plate weld from an orthotropic bridge deck plate will be determined. The geometry of a large scale test specimen is used. It is a steel bridge deck with closed trapezoidal stiffeners constructed with constructional steel S235 with a yield strength of 235N/mm. The cross section of the longitudinal stiffener connected to the deck plate is the only interest for the welding simulation procedure. The closed trapezoidal longitudinal stiffeners have a height of 300mm and on top, they are 300mm wide while a width of 125mm is present at the lower soffit. The deck plate has a thickness of 15mm and the longitudinal stiffeners are 6mm thick. The dimensions of this connection is shown in Fig. 2. This cross section will be further considered for the welding simulation and a length of 250mm is assumed. This is the distance in between two tack welds.

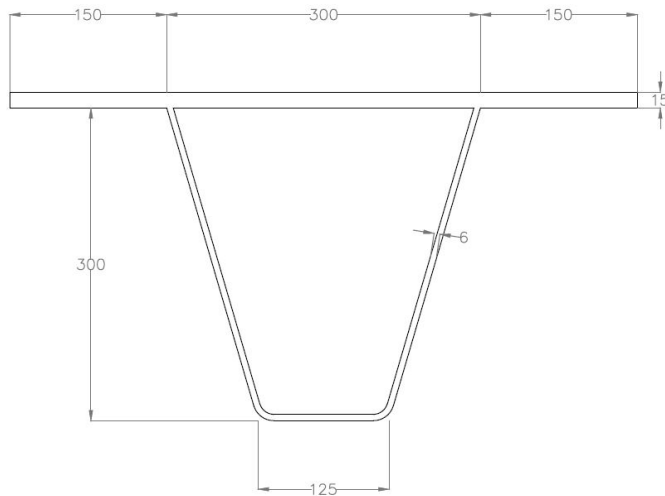


Fig. 2 Dimensions of longitudinal stiffener connected to deck plate (dimensions in mm)

Welding procedure

To construct the orthotropic bridge deck, the longitudinal stiffeners are welded to the deck plate with twin wire submerged arc welding. The orthotropic bridge deck is inverted to execute the welding operation. The stiffeners are kept on the right position and before they are entirely welded to the deck plate with the submerged arc welding, tack welds are provided to ensure their position on the deck plate and to provide distortion control of the stiffener. These tack welds are ground down before the welding is executed to minimize the tack weld profile. The length of the tack welds is equal to 25mm.

The welding can only be executed from the outside of the stiffener since the inside cannot be reached with a welding torch. Therefore, the welding process has to result in a weld melt-through towards the inside of the stiffener to obtain a good connection of the stiffener with the deck plate.

The parameters of the welding procedure for the simulated orthotropic bridge deck are given by the manufacturer. The diameter of the electrodes D is equal to 2mm while the extension of an electrode L is equal to 30mm. The welding is executed with a current I of 780A and a voltage of 29V. The advancing speed of the welding torches S is 950mm per minute or 15.83mm/s. DC+ submerged arc welding is used, this means that the current flows from the electrode to the base metal.

Finite element model

To determine the residual stress distribution caused by welding the longitudinal stiffener to the bridge deck plate, a decoupled thermal-mechanical analysis is performed with the software Siemens NX with solver SAMCEF. First a thermal analysis will be performed to calculate the temperature field introduced by the welding process. Subsequently a mechanical analysis is performed to determine the residual stresses. Solving the thermal and mechanical fields

simultaneously is not recommended since it requires a large un-symmetric system of equations to be solved which is computationally demanding [4]. Therefore, the welding simulation is described by decoupled quasi-static thermo-elasticity equations. Thus the deformations are dependent on temperatures but temperatures are independent of deformation. The thermal field as a function of time is determined during the thermal analysis. This temperature field is used as input for the mechanical analysis [3].

A finite element mesh with three-dimensional 8-node brick elements is used in the thermal and mechanical analysis. The mesh comprises 27624 elements and 38870 nodes. Linear brick elements are the basic recommendation in plasticity which will arise due to the large temperatures of the steel during welding [4]. A true stress-plastic strain behavior is specified, this is natural strain which is conjugated to Cauchy stress and an isotropic hardening behavior is assumed.

The welding process will be modelled by subjecting the elements of the deck plate and the stiffener to a heat flux. The weld material itself will not be modelled but the width of the weld is taken into account by applying heat flux to the adjacent elements on the deck plate and stiffener.

Thermal analysis

In the analysis of the mechanical effects of welding, the fluid flow is not taken into account. This high temperature region is approximated by a given heat input in the thermal analysis. Thus, the actual physics that take place in the weld pool are simplified considerably and replaced by a heat input model [4]. During the thermal analysis, the temperature field is obtained by specifying the heat input.

The parameters necessary to describe the heat input to the weldment from the arc are essential to accurately compute the transient temperature field. The heat input model of Goldak [5], the so-called double ellipsoid heat source is used. This is a three-dimensional double ellipsoidal heat flux model used to examine the three-dimensional temperature, stress and strain fields. The heat flux can be determined based on the weld geometry [6].

For the thermal material behavior, the microstructure change is ignored and it is assumed that the material properties depend only on temperature [7]. The thermal conductivity and specific heat are specified in function of temperature. A constant mass density of the material is specified since the change in density is computed simultaneously with the temperatures when deformations are calculated. The mass density is 7872 kg/m³.

Boundary conditions must be employed to account for surface heat losses. Natural convective heat transfer will be present and a convection coefficient of 15 W/m²°C is specified. The emissivity in function of temperature is also considered. An initial temperature of 18°C is assumed for the whole structure.

Initially, the meshes of the bridge deck and the longitudinal stiffener are not connected to each other. Only after the passage of the welding torch, when a temperature of 1440°C is reached, the meshes are connected. This is realized by the finite element model with the TWELD command which defines the source welding temperature threshold that initiates the gluing and connecting of the meshes.

The heat input distribution is used to model the welding torch with a certain advancing speed. The heat flux for elements subjected to the welding process is calculated with the heat input distribution taking into account the dimensions of the elements. The heat flux is calculated for different time steps to simulate the passage of the welding torch with the considered welding speed.

The thermal analysis simulates the heat introduced in the orthotropic bridge deck by welding the longitudinal stiffener to the deck plate and calculates the temperatures of the steel bridge deck during the welding process. The applied heat flux is the largest in the connection of the

deck plate with the stiffener where the welding torch is present and it gradually fades out when the distance to the connection increases. The welding torch also preheats a very small area in front of the torch where the heat source is going to pass. The melting temperature of the material is defined as 1440°C, at this temperature the melted material is located in the fusion zone. High temperatures are present at immediate vicinities of the fusion zone which defines the heat affected zone. The calculated temperatures for the front face of the welded connection for two time steps are displayed in Fig. 3.

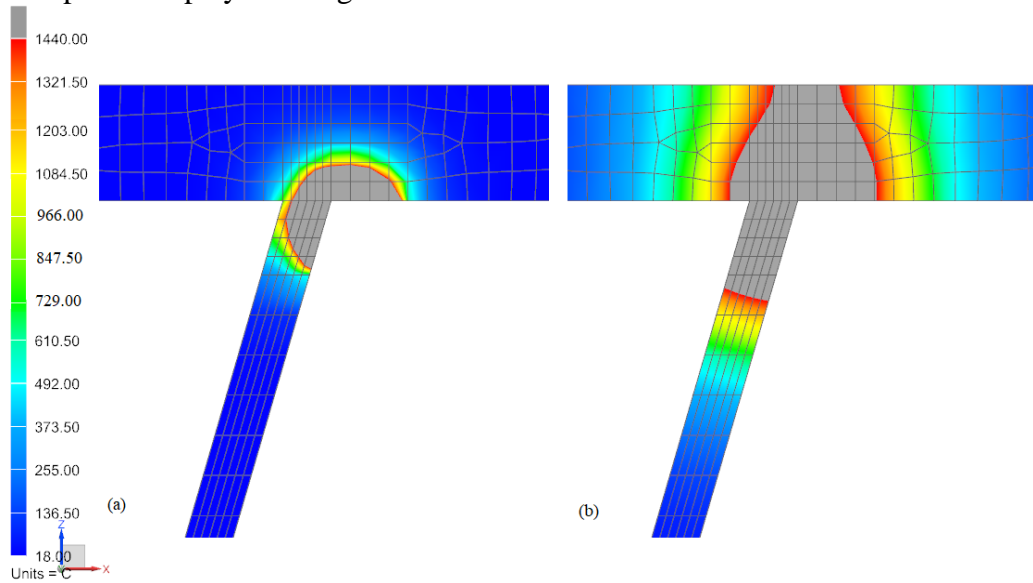


Fig. 3 Results from thermal analysis for (a) $t=2.289s$ and (b) $t=15.2055s$ for the front face

The fused zones and the weld penetration can clearly be recognized and is indicated by the grey zone which represents temperatures higher than 1440°C. This is the melting temperature which activates the command TWELD resulting in a connection of the meshes of the stiffener and the deck plate. It can be clearly seen that the temperature for the entire connection area is higher than this temperature resulting in a complete weld melt-trough. The zone with high temperatures increases when the time step is increased. This is due to the passage of the welding torch and the penetration of the weld into the material.

When the welding torch is passed by the entire length of the bridge component, the temperatures have to decrease due to the interaction of the model with the environment. The heat losses caused by natural convective heat transfer and radiation result in a decrease of the temperature for the bridge deck. This decrease in temperature can be recognized in the temperature results for later time steps.

Mechanical analysis

After the thermal analysis, the temperatures introduced during the welding of the orthotropic bridge deck are stored. The subsequent mechanical analysis introduces these temperatures as a time dependent load onto the model. A quasi-static mechanical analysis is sufficient since the inertia effects in the welding process is negligible due to the simplification of the weld pool region. Large deformation and rotation effects are the result of the large welding deformations. These large deformations are taken into account during the mechanical analysis which makes it more complex than the thermal analysis. Accounting for large deformations is realized by making use of the logarithmic natural strain definition [4].

It is important to have a correct description of the material behavior in order to have an accurate finite element model. The Young's modulus, thermal expansion and stress-strain relationship in function of temperature are defined. The shear modulus is equal to 80 GPa and

the Poisson ratio is 0.3 in every direction of the material. These properties are the same for all temperatures.

Initially, the longitudinal stiffener and the deck plate are not connected. Before the welding process starts, the bridge deck is inverted and the stiffener is positioned on top of it. To ensure the exact location of the stiffener relative to the deck and to facilitate the welding process, two tack welds with a length of 25mm are provided at the beginning and ending of the considered cross section. The deck plate is positioned on top of a series of springs to simulate a rigid surface. The own weight is also assigned to the model by specifying the gravitational acceleration.

The boundary constraints must result in static equilibrium without introducing additional stresses. However, some additional constraints must be defined in order to prevent free body movement of the model. Therefore, symmetry points are defined and considered fixed in two directions in such a way that the deck plate and the stiffener cannot move in a horizontal direction during the welding process.

The load for the mechanical analysis consists out of the temperatures calculated during the thermal analysis and are implemented in the model as a time-dependent loading.

The residual stresses at the final time step, when the test piece is cooled down to environmental temperature after the welding, are of importance since these are the welding residual stresses that are present for the entire lifetime. To evaluate the residual stresses, the results in a zone at 25mm distance from the edges will be neglected to avoid the influence of the boundary conditions.

Residual stress results

The residual stresses for the orthotropic bridge deck will be discussed for the transverse direction only (perpendicular to the welding direction). A general overview of the residual stresses for the top of the deck is given in Fig. 4. The residual stress results for the middle section for the top and bottom of the deck plate are shown in Graph 1. The results for the longitudinal stiffener are not shown but will be discussed.

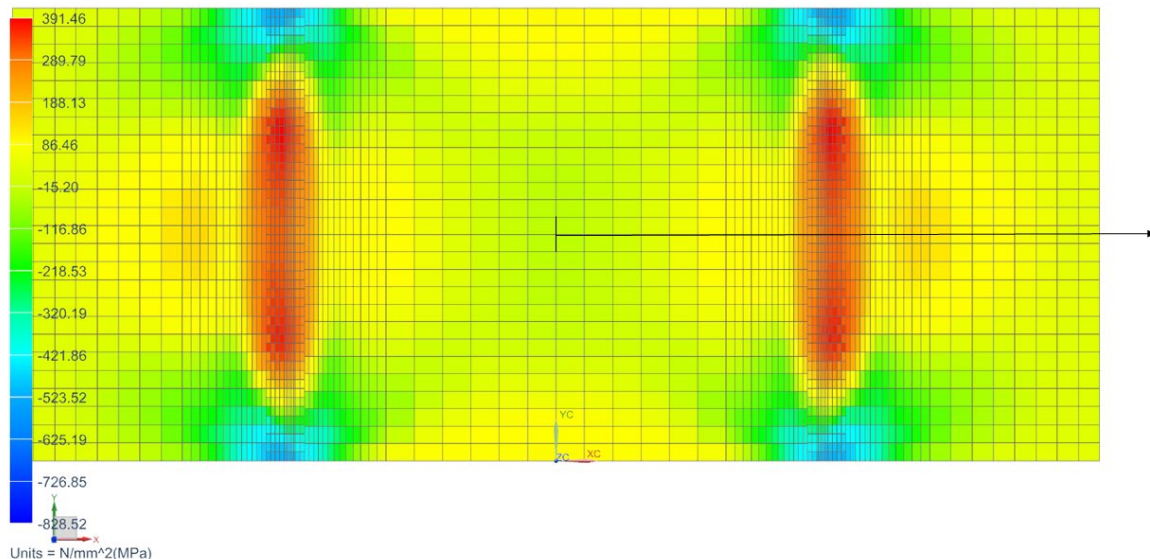
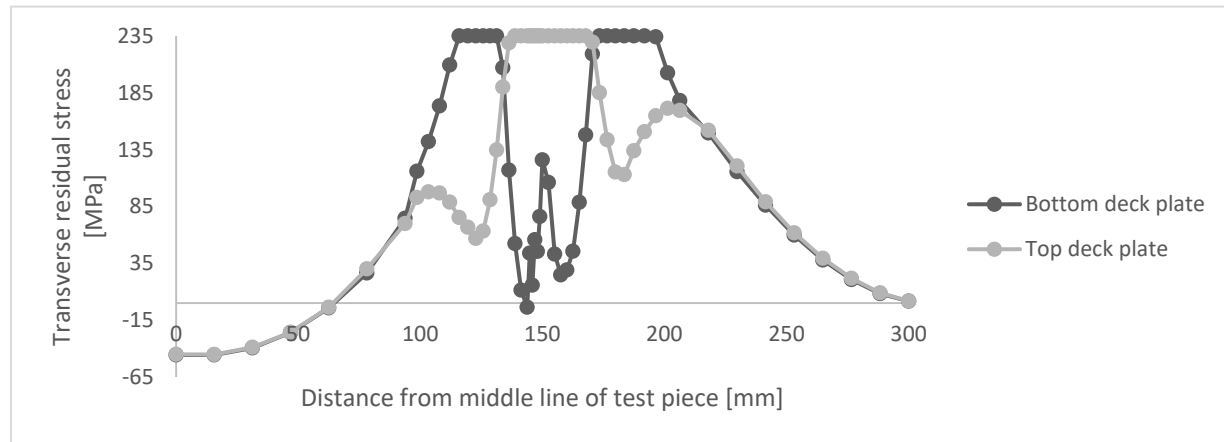


Fig. 4 Transverse residual stresses on top of the deck plate

There are tensile yield stresses present on top of the deck plate in the region near the weld seam which diminish at a greater distance from the welding line. On the bottom of the deck plate, the tensile yield stresses are at a distance of 20mm symmetrical around the weld toe, near the weld seam there are smaller tensile residual stresses present. The residual stress at the side of the test piece is close to zero while there is a compressive residual stress of about 45MPa in the

middle. The longitudinal stiffener has a small tensile residual stress peak of 200MPa near the weld toe root while the residual stress at the weld toe is already close to zero. Along the stiffener there are small tensile residual stresses present and at the lower soffit, there is no longer residual stress present. These results agreed well with experimental data obtained with incremental hole-drilling.



Graph. 1 Transverse residual stresses on for top and bottom of the deck plate

Summary

The residual welding stresses of the longitudinal stiffener-to-deck plate weld from an orthotropic bridge deck plate are determined. These residual stress results can be used in future research to increase the accuracy of fatigue life prediction of the welded orthotropic bridge component.

References

- [1] S.W. Wen, P. Hilton and D.C.J. Farrugia, Finite element modelling of a submerged arc welding process, *Journal of Materials Processing Technology*, 2001, Vol. 119, pp. 203-209. [https://doi.org/10.1016/S0924-0136\(01\)00945-1](https://doi.org/10.1016/S0924-0136(01)00945-1)
- [2] Z. Barsoum and A. Lundbäck, Simplified FE welding simulation of fillet welds – 3D effects on the formation residual stresses, *Engineering Failure Analysis*, 2009, Vol. 16, pp. 2281-2289. <https://doi.org/10.1016/j.engfailanal.2009.03.018>
- [3] X. Shan, C.M. Davies, T. Wangsdan, N.P. O’Dowd and K.M. Nikbin, Thermo-mechanical modelling of a single-bead-on-plate weld using the finite element method, *International Journal of Pressure Vessels and Piping*, 2009, Vol. 86, pp. 110-121. <https://doi.org/10.1016/j.ijpvp.2008.11.005>
- [4] L.-E. Lindgren, Numerical modelling of welding, *Computer methods in applied mechanics and engineering*, 2006, Vol. 195, pp. 6710-6736. <https://doi.org/10.1016/j.cma.2005.08.018>
- [5] J.A. Goldak, A.P. Chakravarti and M. Bibby, A new finite element model for welding heat sources, *Metallurgical Transactions 15B*, 1984, pp. 299-305. <https://doi.org/10.1007/BF02667333>
- [6] E.A. Bonifaz, Finite element analysis of heat flow in single-pass arc welds, *Welding research supplement*, 2000, pp. 121s-225s
- [7] L.-E. Lindgren, Finite element modelling and simulation of welding part 2: Improved material modelling, *Journal of Thermal stresses*, 2001, Vol. 24, pp. 195-231. <https://doi.org/10.1080/014957301300006380>