Yield strength mapping in the cross section of ERW pipes considering kinematic hardening and residual stress

Dongwook Kim¹, Luca Quagliato², Wontaek Lee¹ and Naksoo Kim¹

¹ Department of Mechanical Engineering, Sogang University, Seoul, 04107, Republic of Korea
² Department of Management and Engineering, University of Padua, Vicenza, 36100, Italy

E-mail: nskim@sogang.ac.kr

Abstract. In the ERW (electric resistance welding) pipe manufacturing, material properties, process conditions and settings strongly influences the mechanical performances of the final product, as well as they can make them to be not uniform and to change from point to point in the pipe. The present research work proposes an integrated numerical model for the study of the whole ERW process, considering roll forming, welding and sizing stations, allowing to infer the influence of the process parameters on the final quality of the pipe, in terms of final shape and residual stress. The developed numerical model has been initially validated comparing the dimensions of the pipe derived from the simulation results with those of industrial production, proving the reliability of the approach. Afterwards, by varying the process parameters in the numerical simulation, namely the roll speed, the sizing ratio and the friction factor, the influence on the residual stress in the pipe, at the end of the process and after each station, is studied and discussed along the paper.

1. Introduction

The electric resistance welding process is a well-established joining technology largely used in the industry and it is one of the key stages of the pipe forming process, in between the forming and the sizing operations. At the beginning of the pipe manufacturing process, a steel sheet is uncoiled and fed into the forming station, where the pipe-shape is obtained, leaving a small gap between the two edges of the original sheet. Afterwards, by utilizing the ERW process, the two edges are welded together, resulting in a continuous structure for the pipe. At the end of the process, in order to calibrate the diameter of the pipe, a sizing operation is performed, allowing to obtain the final product.

In the roll forming station of the pipe manufacturing process, pair of rolls are arranged in tandem and progressively deforms a metal sheet which is put in motion by the rolls themselves, allowing to obtain a desired final pipe shape [1]. Concerning the pipes roll forming process, Han et al. [2] utilized the updated-Lagrangian method to study the influence of the process parameters on the deformation flow during the process as well as on the opening value and the springback. The result of their research highlighted how the strain field is not constant along the circumferential direction of the pipe as well as the springback value is small, thus negligible. Jiang et al. [3] studied the metal strip deformation during the cage roll-forming process of pipes by means of numerical simulation, analyzing the non-bending-are in the strip as well as deriving the evolution of the relative curvature throughout the process.
In relation to the welding, the second operation in the ERW pipe manufacturing process, Sattari-Far et al. [4] developed an analytical model for the estimation of the distortion in pipes, due residual stress, at the end of the welding. The results of the developed model have been compared with those of laboratory experiments showing how that the model is able to properly predict the residual stress after the welding operation as well as they can be reduced by choosing an appropriate welding sequence.

In the work of Deng et al. [5], the circumferential front-to-end pipe welding process has been studied by means of both 3D and 2D numerical simulation and the results have been compared with those of experiments. Although the 2D simulation allows a considerable spare in computational time, it does not allow a proper estimation of the residual stress hence, if a precise analysis is meant to be carried out, the three-dimensional explicit analysis represents the right choice.

In this paper, a multi-stage numerical model is implemented ABAQUS/Explicit in order to simulate forming, welding and sizing operations on the pipe. Since the welding simulation must be implemented in a different ABAQUS module, a mapping strategy has been utilized to export the results of the forming operation and to import them into the welding module as well as for the successive results export at the end of the welding and input, as initial state, in the sizing simulation. The mapping strategy allows to successfully exchange information between different modules of ABAQUS while committing negligible errors. In addition, at the end of the sizing process, same by means of mapping, the results have been mapped on a simple test specimen shape, for different circumferential positions of the pipe, allowing to estimate the position dependent yield strength and, according, the residual stresses.

For the model validation, the results of the numerical simulation have been compared, in terms of pipe thickness and diameter, with those of the real production after both roll forming and sizing station, allowing to conclude that the developed model is able to properly replicate the real process conditions. In order to investigate the influence of both forming and sizing process parameters on the yield strength of the pipe, three different roll forming angular velocities for the rolls and three different sizing ratios have been tested. This final analysis allows understanding how, for a fixed product and roll forming machine, the process settings can actively influence the final mechanical properties of the pipe.

2. Kinematic hardening model
Especially during the forming station of the process, the initial metal strip undergoes a sequence of tension and compression deformations which makes the Bauschinger effect likely to arise. For this reason, in order to properly account for the yield locus of the material, the non-linear kinematic hardening (NLKH) model is utilized in the numerical model and a brief introduction is given in this paragraph.

The NKLH model includes two different hardening effects: the first one, related to the back stress tensor \( \alpha \) and to the stress tensor \( \sigma \), which makes the yield locus to translate in the space and the latter one which accounts for the expansion of the yield surface, depending equivalent plastic strain \( \varepsilon_{eq}^{p} \), as shown in Figure 1.

\[ F = f(\sigma - \alpha) - \sigma_{y}(\varepsilon_{eq}^{p}) = 0 \]  

The yield function of Eq.(1) is the combination of a kinematic and an isotropic part; the first one is detailed in Eq.(2), where the symbol \( \bullet \) represents the tensor inner product whereas \( \sigma^{dev} \) and \( \alpha^{dev} \) the deviatoric stress tensor and the deviatoric back stress tensor, respectively.

\[ f(\sigma - \alpha) = \frac{3}{2}(\sigma^{dev} - \alpha^{dev}) \bullet (\sigma^{dev} - \alpha^{dev}) \]  

The original formulation for the incremental variation of the back stress tensor is due to Choboche [6] but the recent reformulation due to Fu et al. [7], Eq.(3), allows an immediate implementation in the
ABAQUS material property definition section thus, for this reason, has been utilized for the setting of the numerical model.

$$\dot{\alpha}_i = \frac{C_i}{\sigma_s} (\sigma - \alpha) \dot{e}_{eq}^p - \gamma_i \alpha_i \dot{e}_{eq}^p$$

(3)

The size of the back stress tensor is chosen in order to minimize the differences between the fitted curve and the results of the material characterization but also the computational time relevant for a more complicated formulation should be taken into account. The summation of all the $\alpha_i$ components of the back stress tensor allows calculating $\alpha$. In Eq.(3), $C_i$ and $\gamma_i$ are the kinematic hardening model constant whereas $\sigma_s$ the yield strength of the material.

![Isotropic, Kinematic, Non-linear kinematic hardening](image)

**Figure 1.** Yield surface evolution according to (a) isotropic, (b) kinematic and (c) NLKH models.

The part of the yield surface of Eq.(1) can be expressed by an isotropic hardening model and, in this paper, the Swift power has been utilized, as shown in Eq.(4), where $A$ and $b$ are the model parameters and $\sigma_s^0$ the initial yield strength of the material.

$$\sigma_s (\dot{e}_{eq}^p) = \sigma_s^0 + A \left(1 - e^{b \dot{e}_{eq}^p}\right)$$

(4)

The material properties to be utilized in both the kinematic hardening model and in the isotropic model are derived from the material characterization, which is detailed in the following section.

### 3. Material characterization

For the determination of the material properties of the K55 steel as well as the model constants to be utilized in the isotropic and kinematic hardening models, cyclic loading test have been carried utilizing the cylindrical specimen shown in Figure 2.

![Cyclic loading test specimen dimensions](image)

**Figure 2.** Cyclic loading test specimen dimensions (ASTM E8).
In order to inversely calibrate the material property of the material, a simple tension test simulation has been implemented in ABAQUS. The results of both laboratory cyclic loading test as well as the curve representing the NLKH behavior of the material, after the inverse calibration process, are shown in Figure 3. In the considered research, the number of terms forming the backstress tensor has been chosen as three and, according to this choice, the difference of the area between the experimental and model curve, after the inverse calibration, is calculated in 9.12%, showing the reliability of the choice. Finally, the K55 steel material properties and model constants are summarized in Table 1 whereas the Poisson ratio has been obtained from the material datasheet and is 0.33.

![Figure 3. K55 steel cyclic loading results and NLKH model curve.](image1)

![Figure 4. Specific heat capacity and conductivity of K55 steel.](image2)

Table 1. Material properties and model constants.

|         |         | A   | b   | C_1 |
|---------|---------|-----|-----|-----|
| E [GPa] | σ_y^0 [MPa] | 160 | 8   | 9650 |
| γ_1     | C_2     | 25  | 20  | 35  |

For the proper setting of the welding simulation, the specific heat capacity and the conductivity of the K55 steel have been obtained from the online material database Matilda (Material Information Link and Database Service) and are reported in Figure 4.

4. Model implementation

As previously anticipated, the proposed numerical model includes four different simulations, three relevant for the forming, welding and sizing stations of the ERW pipe manufacturing process whereas the last one is a simple tension test simulation, for the estimation of the yield strength of the material after the whole process and for different circumferential positions of the pipe. The flow chart of the implementation is shown in Figure 5.

In order to export the data from one of the simulation of the model and to import them in the following one, a data mapping strategy has been utilized and is here below detailed. In the considered case, the overall dimensions of the initial strip are 13,170mm for the length, 670mm for the width and 8.83mm for the thickness but, in order to reduce the computational time, only a 500mm length portion of the pipe has been considered for the calculation of the result. This particular section of the pipe will be afterwards referred as “fine mesh portion” whereas the remaining part of the pipe will be named as “coarse mesh portion”.

4
Figure 5. Numerical model implementation flowchart.

Considering the fine mesh portion of the pipe, in order to operate the data mapping, the information concerning stress, strain, temperature, etc. have been exported as a list and have been used to build the input file of the following simulation. The result of the data export is a point cloud where, for each point, the relevant material properties, at the end of the simulation, are saved and used as input.

In all the four simulations, the models have been meshed utilizing the C3D8R hexahedron element, those allow a good calculation while reducing the computational time, as demonstrated in the work of Zou et al. [8]. The element dimensions of the fine mesh portion of the pipe are 2.25 mm width, 10.0 mm height and 6.0 mm length whereas those of the coarse mesh portion are 4.5 mm width, 10.0 mm height and 30 mm length, respectively, for a total count of 72,000 elements. Four elements are located in mesh along the thickness of the pipe. The reliability of this choice has been checked by running different simulations with three, four and five elements along the thickness direction; since no significant differences in the results have been observed in case of utilizing either four or five elements, authors have chosen to utilize only four elements, in order to optimize the calculation time.

The roll forming numerical model is composed of 20 roll passes through couples of rolls with the same dimensions whose relative distance, between adjacent couples of rolls, is shown in Figure 6(a). The sizing simulation instead, it is composed of five couples of rolls, all having the same dimensions and whose relative distance is reported in Figure 6(b).

Figure 6. Rolls relative distance in the (a) roll forming and (b) sizing stations.

Concerning the welding station, the ABAQUS welding interface (AWI) has been utilized, allowing to add elements to the mesh in the positions where the welding arc passes. In the ERW welding
process the material composing the edges of the pipe is molten and fills the gap, creating a continuous structure. In the numerical model, this operation is simulated by adding elements in the mesh between the two edges of the pipe, as shown in Figure 7.

In all the simulations, in accordance to the real process subsequently used for the model validation, the environment temperature has been set to 20°C and the Coulomb friction factor to 0.2, thanks to the continuous good lubrication supplied during the manufacturing.

![Initial state to Final state](image)

**Figure 7.** Mesh addition in the welding simulation.

In order to estimate the influence of the main process parameters in the final yield strength of the pipe, in the roll forming operation, three different roll velocities for the rolls have been tested, namely 2 rad/s, 4 rad/s and 6 rad/s. In the sizing simulation instead, since the key parameter is represented by the sizing ratio, 0.2, 1.0 and 1.5 have been tested and the results are presented in the following section.

In addition to the material data reported in the previous section 3, the set-up parameters of the welding simulation are reported in following Table 2.

| Welding speed | Welding Temperature | Emissivity | Heat transfer coefficient |
|---------------|---------------------|------------|--------------------------|
| 50 mm/s       | 1450 °C             | 0.4        | 25 W/m²K                 |

Both forming and sizing simulations have been run utilizing the ABAQUS/Explicit module whereas the welding and the testing simulation by utilizing the ABAQUS/Static module. In all the simulation, the contact has been defined considering the so-called “balanced master-slave” contact, which allows a good representation of the contact constraints hence more accurate results in the calculation.

5. **Results and discussion**

In order to validate the proposed combined numerical approach, the result of thickness and diameter of the pipe, resulting from the numerical simulation, for the combination of 4 rad/s roll forming rolls speed and 0.2% sizing ratio, have been compared with those of the real process and the results are summarized in Table 3, where a good agreement among the results can be evinced.

| Dimension                  | Production [mm] | Numerical Simulation [mm] | Error [%] |
|----------------------------|-----------------|---------------------------|-----------|
| Diameter (after forming)   | 220.7±1.0       | 219.94±1.0                | 1.25      |
| Thickness (after forming)  | 8.99±0.001      | 8.87                      | 1.35      |
| Diameter (after sizing)    | 217.65±1.0      | 216.63±1.0                | 1.29      |
As previously mentioned, in order to export the results from one ABAQUS module and to import in the following one of the simulation chain, a mapping strategy has been utilized. In order to prove that no relevant loss of information arise due to this choice, the comparison of the von Mises equivalent stress field is presented after the forming simulation (pre-mapping) and at the beginning of the welding simulation (post-mapping), for the fine mesh portion of the pipe, are shown in Figure 8. Following the same rationale, the comparison between the von Mises equivalent stress field after the welding simulation and, after the mapping, at the beginning of the sizing analysis, same for the fine mesh portion of the pipe, are shown in Figure 9. The result shown in Figure 8(a) are related to the roll forming simulation with a speed of 4 rad/s whereas that of Figure 9(a) to the 0.2% sizing ratio.

![Figure 8](image1.png)

**Figure 8.** (a) Result of the roll forming simulation and (b) beginning of the welding simulation.

![Figure 9](image2.png)

**Figure 9.** (a) Result of the welding simulation and (b) beginning of the sizing simulation.

The comparison shown in Figure 8 and Figure 9 show a good agreement both in terms of maximum value of the von Mises equivalent stress as well as in terms of stress distribution of the pipe surface, proving that the adopted strategy is reliable for the transfer of data between different modules.

Finally, in order to show the influence of the variation of both roll forming speed and sizing ratio on the residual stress in the pipe, hence on its yield strength, the results of the simple tension test numerical simulations, after the relevant mapping, are shown in following Figure 10, Figure 11 and Figure 12.
6. Conclusion
In the research presented in this paper, a combined numerical model for the simulation of the whole ERW pipe manufacturing process is presented and validated by comparing the results of the geometry of the pipe, after roll forming and sizing, with those of the real production. In addition, after the model validation, three different roll speeds, for the rolling station, as well as three different sizing ratios have been tested with the developed numerical model and the results have shown how higher speed always causes higher residual stresses in the pipe, with a consequent higher yield strength. Moreover, as understandable, also the sizing process influences the final stress state of the pipe, and the higher the sizing ratio the higher the residual stress in the pipe, as highlighted in the last comparison charts.

In addition that, the simulation mapping procedure proposed in this paper is an useful methodology which might be extended also to different simulations and different manufacturing processes, since it allows creating a link between different modules of a software, widening the possibility for a full understanding of the overall manufacturing sequence. In conclusion, the proposed methodology allows estimating the yield strength in different radial positions of the pipe, highlighting how the manufacturing process influences, in different ways, the residual stress, thus the final yield strength.

References
[1] Kiuchi M, Abe K and Onodera R 1995 CIRP Annals 44/1 239-242
[2] Han ZW, Liu C, Lu WP, Ren LQ and Tong J 2004 J. Mat. Proc. Tech. 145 311-316
[3] Jiang J, Li D, Peng Y and Li J 2009 J. Mat. Proc. Tech. 209 4850-4856
[4] Sattari-Far I and Javadi Y 2008 I. J. Press. Ves. and Pip. 85 265-274
[5] Deng D and Murakawa H 2006 Comp. Mat. Sc. 37 269-277
[6] Chaboche JL 1986 I. J. Plast. 2 149-188
[7] Fu J, Barlat F, Kim JH and Pierron F 2016 I. J. Sol. and Struct. 102-103 30–43
[8] Zou T, Zhou N, Peng Y, Tang D and Li D 2016 J. Ph. Conf. Proc. 734 032016

Figure 10. Yield strength after roll forming.

Figure 11. Yield strength after welding.

Figure 12. Yield strength after (a) 0.2%, (b) 1.0% and (c) 1.5% sizing ratios.