Analysis on Coupled Forming-Welding Process Using FEM Simulation and Experimental Verification

Alexander Bauer¹, Y HP Manurung², *M R M Redza², Joeran³, Marcel Graf¹ and Birgit¹
¹Professorship of Virtual Production Engineering, University of Technology, Germany
²Faculty of Mechanical Engineering, Universiti Teknologi MARA, 23200 Bukit Besi, Terengganu, Malaysia
³University of Applied Sciences, Germany

*ridhwanredza@uitm.edu.my

Abstract. In this paper, a coupled forming-welding process is to be analyzed. Therefore, the process chain of a tube u-/o-bending process was combined with the longitudinal welding of slotted tubes. Both processes were conducted and evaluated experimentally as well as numerically. To verify the process in a wide range of experimental technologies, different analyzing methods were used, which include geometric measurements, microsections (microstructure), X-ray-diffraction and material phase analyses. A good agreement was found for the numerical simulation of the welded tube in accordance with the experimental processes, based on the different evaluation methods. For this reason, it can be concluded that the developed setup can be used as a good forecast for metal inert gas welding processes of tubular products.

1. Introduction
The manufacturing of tubes is one of the main processing technologies within the metal industries with regard to high importance and wide application area of its products. There are different production routes which can be applied to produce tube manufacturing such as forming, casting and welding etc. Because of the permanently increasing demands on quality and quantity, the duration of development, trial and start-up periods of products are supposed to be shortened. Furthermore, processes keep constantly changing due to new research and development strategies. This leads numerical simulation of processes and technologies to be an irreplaceable tool within the fabrication industry. Because of the mentioned facts, the demands on the quality forecast within these numerical simulations are increasing too. The tendency shows, that the simulation of single process steps is more often replaced by analyzing process chains, to consider the process history too. Since the finite element method (FEM) is the state of the art for the calculation of forming processes, the numerical simulation of welding processes is less popular and a recent research area. The main topics of actual welding simulations are the prediction of thermal residual stresses, distortions and sometimes the microstructure [2]. Typical commercial FE tools for the calculation of welding processes are Simufact.welding, SYSWELD or SimWeld which offer different opportunity and options for the computation of welding processes. As reason for the increasing opportunities with the calculation of welding processes, actual research is involved in parameter studies, calibration or a deeper understanding of the process itself. For example, Jiang carried out a study of different laser parameters...
followed by an optimization [3]. Another research topic deals with the distortions caused by multi-layer welding of T-joints [13] or the numerical multi-pass welding compared to the reality [8]. Prajadhiana studied the distortion of welded T-joints, using different elements and experimental verification [10] and Sulaiman analysed the welding induced distortion of butt-joints [12]. Furthermore, multiple research projects deal with the topic of laser welding with special materials or supporting structures [5] [4] [9]. As a reason for the limitation of numerical welding simulations mostly for research projects, the coupled processes of forming and welding are a very new field of interest. One of the most common problems within the calculation of coupled processes is the connection between the results of the different specific simulation tools. However, there are still a few research projects which deal with the topic of the coupled forming and welding technologies. For example, Loose successfully developed a numerical process chain for the deep drawing of a previously laser welded two part circular blank [6] [7]. Another multi-stage coupled forming and welding setup was presented by Schafstall, with the numerical modelling of a crash-box. Therefore, the connection between a forming and a laser-welding process was finally followed by a crash simulation of the previously welded structure [11]. Furthermore, Adams and Härtel investigated the coupling of an arc welding process followed by flat rolling with the aim to improve the microstructure of the weld seam in-line without intermediate cooling [1]. For the reason of the actuality of this topic, this paper deals with the numerical simulation and experimental validation of a coupled forming and welding process for the manufacturing of longitudinal welded tubes. Furthermore, different parameters and influences on the numerical simulation were studied, which leads to restrictions that have to be considered for a correct simulation.

2. Experimental setup dan procedure
Within the first forming step, a three-step U-/O-bending process was established for the manufacturing of the longitudinal slotted tubes. For this purpose, commercial DC04 steel sheets, with the measurements 300 mm x 63 mm x 1.5 mm were used. The result was a tube with an outer diameter of 20 mm. For the conduction of these forming processes a multi-servo press MSP 200-250 of H&T Productions Technology GmbH, with a maximum press force of 2000 kN was used. Within this press, the tools for the U- and O-bending processes were installed Figure 1.

Figure 1. Left: Forming tools for U-bending, Right: Forming tools for O-bending

Within the first step the U-bending process formed the initial sheet in the shape of the die. A spring controlled counterholder leads to an even surface at the bottom of the workpiece without uncontrolled bending during the forming motion of the punch. The second and third step was performed by the semicircular tools (Figure 1 - Right). Within the second step, a distance holder between the two dies ensures, that the first O-bending step leads to an oval shape of the tube, while the second step, without distance holder, finishes the forming process to the longitudinal slotted tube with a slight gap between both ends of the metal sheet. The three bending steps can be seen in Figure 2 from left to right.
Figure 2. Three forming steps of U-/O-bending, Left: U-Bending, Middle: O-Bending (Pre-bending); Right: O-Bending (Finishing)

Figure 3. Left: Welding robot, Right: Clamping device for tube welding with measurements

The filler material used was a G2Si1 (OK AristoRod 12.57) and the shielding gas was a mixture of argon (82%) and carbon dioxide (18%). During the welding process, a current of 88 A and a voltage of 18 V were applied for a welding speed of 9 mm/s.

In addition to the forming and welding technologies, different methods were used to evaluate and examine the given results. The general measurements of the length were conducted with a conventional digital calliper, while exact measurements of the diameter and the circular shape of the cross section were performed with a 3D microscope VHX-600 by Keyence. With the connected software VHX-H2MK 3D, a three point measurement was used to determine the different diameters, at different sections, on the in- and outside as well as close to the weld seam. Additionally, the residual stresses of the specimen were detected within the different steps of the forming technology and after the welding process. For the X-ray detection of residual stresses, a Bruker AXS D8 Discover diffractometer was used with a cobalt source of radiation and a radiant power of 40 kV and 40 mA. The evaluation of the results was conducted within the related program LEPTOS, which is also provided by Bruker AXS. Both technologies (X-ray diffraction (XRD) and 3D microscope) were applied for the forming products (longitudinal slotted tube and intermediate stages) and the welded tubes. In addition to the mentioned technologies, the welded tube was subjected to some enhanced analyzing technologies. First of all, it was very important to measure the distortion after the welding process, which is caused by the thermal induced residual stresses which affect the tubes while welding and cooling. For the geometrical measurement the tubes were measured before and after welding, so that the differences are clearly visible. For this reason, an individual designed clamping system was established within the Mitutoyo Beyond 707 coordinate measurement system (Figure 4 - Left). Within the clamping device a mechanical stop and a mark on the specimen were implemented to make sure the measuring position stays the same (Figure 4 - Left). For the measurements, six measuring sections
were defined at different positions over the length of the tube (next to the weld seam (1-2), at the side (3-4)) and at the bottom (5-6)), which include eleven measuring points each (Figure 4 -Right).

**Figure 4.** Left: Coordinate measurement system Beyond 707 and Clamping device for tube measurement, Right: Schematic illustration of the arrangements of measurement points

3. Numerical setup

The numerical simulations were divided in the parts forming and welding. The forming simulations were conducted within the FEM tool Simufact.forming 14.0, while the welding simulations were done within Simufact.welding 6.0. Both programs were published by the MSC Software Company. Due to the differences between the processes, there are different special settings in each program which have to be considered. The numerical simulation of the U-/O-bending was setup as a 2D planar simulation to save computational time Figure 5.

**Figure 5.** Numerical setup, Left: U-Bending, Right: O-Bending (Pre-Bending)

The needed geometries were rebuild from the experimental tools within the computer aided design (CAD) tool Siemens NX 10.0. The mesh was build up with an advancing front quad mesher and the total element of the geometry were 56,700 with edge length of 0.5 mm respectively. Quad mesher has been selected due to workpiece have meant to be drawn in a rectangular shape basically. Thus, quad mesher are more suitable to obtain an accuracy result compare with another meshing scheme. A combined friction model with $\mu = 0.1$ und $m = 0.2$ was applied. The temperatures for workpiece, tools and environment were set to 20 °C like in the real process. For the first computation, the general material data (e.g. Youngs module, Poisson ratio) were taken from the database, while the flow curve was determined from a flat compression test of the used material. Later the results of the simulation with the self-generated flow curve and a simulation with a general DC04 and flow curves in a temperature range from 25 °C to 1500 °C were compared. It was proven that the results have almost no differences (geometry, plastic strain, residual stresses), so the material with the higher range of
temperatures was used because it has to be used within the forming and the welding simulation, where the heat influences and temperature changes affect the workpiece. The press stroke was given with 1 mm/s. After the forming motion, the press stroke included the return of the punch / the dies into the initial position, like in the experimental processes. To create a coupled simulation of forming and welding processes, the geometry with all of the results has to be transferred from the forming into the welding simulation. Within the first step, the 2D geometry was expanded to a 3D geometry with a length of 100 mm, like the pipe sections which were used for the real welding processes. Afterwards the geometry was transferred as SPR-data, under consideration of specific settings like the matching of the plasticity models for both programs, to additive plasticity. As a calculation of a MIG-welding was conducted, a special weld seam had to be constructed within Simufact.forming GP 14.0 (Figure 6 - Left). The weld seam was rebuilt after the geometry of the experimental weld seam. The elements edge length of the tube remains at approx. 0.5 mm. Within the area of the weld seam at around a distance of 3 mm, a refinement box was added which causes element edge lengths of 2.5 mm in these areas. The weld seam itself consists of elements with an edge length of 0.2 mm. The setup is calculated as a thermomechanical process because distortions play a major role in the comparison with the real process. The whole process time was set to 250 s. This consists of 30 s welding and holding within the clamping device, followed by 220 s of cooling released from the clamping device, like it was done in the experimental procedure. The velocity of the welding robot was 9 mm/s and the heat source model were the Goldak double ellipsoid with a front length of 2 mm, a rear end of 3 mm and a width and depth of 2 mm. The settings for the welding parameters are 90 A of current and 18 V voltage. An idealization was made due to the clamping of the tube. While in the real process, the used clamping causes a line contact on the workpiece, a small surface contact was used as reason for numerical instabilities which appeared for the use of line contacts. All other geometrical properties were taken from the experimental process including the use of a spring controlled clamping. After finishing the welding process, the tube was released from the dies, to enable free springback Figure 6 – Right.

**Figure 6.** Left: Constructed weld seam, Right: Numerical setup

### 4. Numerical setup

Within the first step of calibration, the numerical forming model of the bending process was compared with the results of the experimental studies. Prior to the decision of using the 2D model, a comparison between 2D and 3D simulation was done for the forming process. Therefore, the result was almost identical with some slight deviations of stresses and the formation of the gap for the weld seam. As reason of the significant higher computational time of the 3D model and a worse ability for the connection with the numerical welding simulation, the 2D simulation was used for further analysis. Additionally, it could be seen, that the accordance between the geometrical properties with special focus on the weld seam, was even better for the 2D simulation. The comparison of the geometry showed a very good agreement between experiment and FEM. Hereby FEM and the evaluation software of the 3D microscope used the same evaluation method to calculate the diameter from a three point section placed on the tubular geometry Figure 7 - Upper. Therefore, different sections lead to different diameters and it was ensured, that always the same points were used in FEM and in the microscopy. With regards to this fact, the following measurements were done.
The values for these measurements can be seen in Table 1. Furthermore, it can be seen that the geometrical shape of the metal sheet ends is the same within the experimental formed tubes and the simulated ones.

**Table 1. Comparison of the measurements between FEM and experiment**

| Measurement            | FEM (mm) | Experiment (mm) | Deviation (%) |
|------------------------|----------|-----------------|---------------|
| Inner diameter 1       | 16,7     | 17,17           | 3             |
| Outer diameter 1       | 19,6     | 19,50           | 1             |
| Inner diameter 2       | 16,74    | 16,88           | 1             |
| Outer diameter 2       | 19,38    | 19,56           | 1             |
| Top of the gap (1)     | 0,64     | 0,67            | 5             |
| Bottom of the gap (2)  | 0,35     | 0,34            | 3             |

With the base of the good agreement between experiment and FEM, the simulated geometry was used for the preparation of the numerical welding process. Therefore, the geometry was transferred like described in the chapter numerical setup. After preparing the simulation with all numerical parameters and constraints, a various amount of numerical calculations was performed, to examine special influences of parameter or property changes on the results and the computational procedure, which should not be discussed further within this paper. However, after finishing the comparisons, one numerical setup was chosen as the most effective and accurate one. Therefore, the comparison was done with different methods as described in the chapter experimental setup. First of all, the geometrical values are of great importance again. Due to the thermal induced influences on the residual stresses, a deformation can be seen after conducting the welding experiments and after releasing the tube from the clamping device. Two different methods (coordinate measurement and 3D microscopy) were applied for the geometrical measurements. The coordinate measurement performs the measurements of the shape along the longitudinal axis, over the length of the specimen. The 3D microscopy focuses on the measurements of the cross section shape of the tube.
The coordinate measurement was conducted as described in the experimental setup. For the comparison, the same measurement points which were applied with the coordinate measurement were analyzed in the FEM. It was visible, that there is a gap between the experimental values and the FEM. Due to the fact, that the shape of the curves are almost completely identical, it could be assumed, that there was a slight difference in the structure of the clamping device within the measurement system, so all experimental values lie slightly below the ones form the FEM in Z-direction. The results can be seen in Figure 8. The picture from the FEA (Figure 8: Upper – Left) shows the measured values for section 2 (Figure 8: Lower – Left).

![Figure 8. Comparison of the distortion after welding between experiment and FEM](image)

It can be seen, that there is a noticeable distortion in Z-direction over the length of the specimen, while the distortion in X is rather low. Like described before there is also a good agreement between FEM and experiment with a maximum deviation of 0.34 mm in Z-direction and 0.33 mm in X-direction from all measurements. The percentage of error for experiment and simulation result were shown less than 20% for overall distortion result in X and Z direction respectively. Besides the measurements of the distortion along the whole specimen, again the cross section of the welded tube was measured with 3D microscopy. Like already described for the formed tubes, the measurement software was used to calculate the diameter from three points which were applied at the same location of the cross section in the microscopic measurements as well as in the FEM. Therefore, the shape of the cross section showed a good agreement between experiment and numerical simulation too. The maximum deviation of the average of eight measuring sections, performed on four tubes, was 6 %. Thereby, it can be seen, that the shape of the cross-section develops in the same way for the experiments and the numerical calculations. Another very important parameter for the calibration of the numerical welding simulation is the development of the weld seam. Thereby, it can be seen if the heat influence is calculated correctly or if some adjustments have to be made. Different micro sections were made (Figure 9 - Left) and compared with the numerical results (Figure 9 - Right).
Within Figure 9, it is visible, that all areas of the numerical simulation have sharp borders while the transitions in the reality are mixed and smoother. Within the transition zone, a mixed area of weld metal and rough grain can be seen, while the numerical simulation shows a uniform field of weld metal (1). The detection of a straight cut (2) for the transition from weld metal (1) to the heat affected base material (3) is not that clear, whereby the experimental weld gives a hint for this. Next to the heat affected base material, the unaffected base material can be seen clearly in Figure 9 – Left but also at the edges of Figure 9 - Right. Furthermore, the measurement of the width of the weld seam was determined with 3.32 mm for the experimental approach and 3.41 mm for the numerical simulation. The extension of the heat influenced zone starting at (2) up to the base material (4) was measured with 1.5 mm for the microscope view and 1.68 mm for the FEM, which makes a maximum difference of 0.18 mm. To sum up, this comparison shows that there is a good agreement for the heat influenced zone and the weld seam too, even though the FEM is not able to show a more detailed structure with fluent limits. Finally a measurement of the residual stresses was done too, because these are the most influencing factor behind the distortion which are caused by the thermal influences during the welding process. As reason for the scatter of the x-ray beam in an area of approx. 5 mm2, certain areas have to be defined for the measurements (FIG).

For the comparison, these areas were measured within the FEM too. While the numerical calculation always provides three residual stresses (σMax, σInt, σMin) the XRD is just able to evaluate two stress components (σ1, σ2). Within the comparison of the results, no values for σMax can be found for the evaluation of the measured XRD values. This could be caused by the tensor orientation because the XRD is not able to measure stresses, which are directed perpendicular to the surface. However, as a reason for this, the stresses of σInt and σMin from the FEM were compared with σ1 and σ2 from the experiments. The experimental values were gained from two different tubes and averaged afterwards for the comparison with the FEM. Furthermore, it has to be considered that the XRD measurements can show a variance of approx. +/- 20 MPa for each of the measured points. Figure 11 shows a summary of the measured values for each of the measured sections from 1-6, like described in Figure 10.
It can be seen that there is a good agreement between the evaluated stresses except for one value in section 4, which shows a strong deviation compared to the other values. The complicated measuring device for the XRD, with a maximum measuring depth of 10 µm, combined with the fact that the welding process can lead to strong influences around the weld seam especially at the surface of the specimen, could cause a deviation as it can be seen here. As a reason for this various kind of influences it cannot be clearly sort out, why area 4 shows this effect.

Generally, all of the used methods for the calibration show a good agreement between the experimental procedure and the numerical calculations. Therefore, it was proven that the modelling of the process chain between forming and welding was successful, even though it is a high effort. With the gained knowledge about the coupled processes and the generated models it is easier to calculate similar processes for other geometries and the developed setup can be used for further computations.

As an additional research focus, the temperature development within the numerical simulation was analyzed concerning the influence on phase transformation processes at different positions around the welded seam of the tube. Therefore, zone 1 (close to the weld seam), 2 (transition zone) and 3 (base material) were selected within the numerical model of the welded tube (Figure 12 – Left).

![Figure 11. Comparison of residual stresses between FEM (blue) and XRD (red)](image)

![Figure 12. Left: Temperature development within FEA with highlighted areas 1 – 3, Right: Microsections of area 1 – 3](image)
Afterwards the temperature development during the welding process was evaluated from the results of the FEA. Furthermore, microsections were generated from the experimental welded tubes at the same positions, the temperature development was analyzed. These microsections were examined due to the different material phases (Figure 12 – Right). Additionally, the cooling curves for the positions 1 – 3 were compared with the time temperature transformation (TTT) diagram, which was given by the simufact material database of the used DC04 within the numerical simulation (Figure 13).

Figure 13. Cooling curves generated from FEA (1 – 3) integrated within the TTT diagram of DC04

By the means of the analysis of microsections from the welded tube, it was recognized that the phase fraction directly within the weld seam consists almost completely of bainite. Within area 1, the microsection shows a combination of bainite and ferrite. Due to the fact, that the given TTT-diagram cannot show an exact percentage of the phase fractions after the cooling process, it can be seen that the tendency is correct. The areas 2 and 3 are showing ferrite as the only phase fraction within the microsections. This is in a good agreement with the TTT-diagram too, which shows that the temperatures of area 2 and 3 are not reaching the temperature-limit (Ac3 of DC04: approx. 860 °C) which is needed to activate the energy which is required to start a phase transformation. Overall it can be seen that the analysis of the microstructure shows a good agreement between the temperature development of the FEA and the phase transformations within the experimental process.

5. Conclusion

To sum it up, it is visible that a good agreement between experimental realization and numerical results was found, for all of the evaluation methods, which were used within this work. Some slight deviations were detected for the geometrical measurements over the length of the tube. This could might be caused by some problems with the consistent positioning, related to the fact, that the welded tubes were measured before and after welding and therefore have to be positioned twice. Furthermore, the tendency and the shape of the geometric development is still the same after all. With special focus on the microstructural analyses of the weld seam and the surrounding material as well as the phase analyses, consistent results can be seen between the numerical simulation and the experimental processes. All in all a good forecast is possible for the combination of forming and welding processes with special focus on the tube production. As a result, the numerical simulation of the forming to welding process chain can be used for parameter variations or adjustments of the real process settings, to reduce experimental effort and shorten tryout times.

References
[1] Bauer A, Manurung YHP, Sprunk J, Graf M, Awisgus B, and Prajadhiana K 2019. The International Journal of Advance and Manufacturing Technology, 102(508), pp. 2399-2408
[2] Gebhardt, C 2014. Auf ewig zusammen- Numerische Simulation in der Verbindungstechnik-Schweißsen. CADFEM Journal 2, pp 24-25.
[3] Jiang P, Wang C, Zhuo Q, Shao X, Shu L, and Li X. 2016. Optimization of laser welding process parameters of stainless steel 316L using FEM,Kriging and NSGA-11. Advances in Engineering Software 99 pp.146-160.
[4] Koudri-Henni, A., Seang, C, Malard, B, and Klosek V 2017. Residual stress induced by laser welding process in the case of a dual-phase steel DP600: Simulation and experimental approaches. Material and Design 123 pp. 89-102
[5] Li Y, Zhao Li, Q Wu A, Zhu R, and Wang G 2017. Effects of welding condition on weld shape and distortion in electron beam welded Ti2AlNb Alloy Joints. Materials and Design 114, pp. 226-233.
[6] Loose T and Kloeppe T 2015. An LS-DYNA material model for the consistent simulation of welding forming and heat treatment. International Seminar: Numerical Analysis of Weldability, pp. 1-22
[7] Loose T 2015. Einbindung der Schweißsimulation in die Fertigungssimulation mit Simweld und DynaWeld: Umformen Schweißen Wärmebehandeln. Präsentation: DVS Congress – Workshop: Anwendungsnahe Schweißsimulation, Nürnberg.
[8] Manurung YHP, Lidam RN, Rahim MR, Zakaria MY, Redza MR., Sulaiman MS, Tham G, and Abas SK 2013. Welding distortion alaysis of multipass joint combination with different sequences using 3D FEM and experiment. International Journal of Pressure Vessels and Piping pp. 89-98.
[9] Piekarska W, Goszczynska-Kroliszewska D, Domanski T, and Bokota A 2017. Analytical and Numerical Model of Laser Welding Phenomena with the Initial Preheating. Procedia Engineering 177, 149-154.
[10] Prajadhiiana KP, Manurung YHP, Zainir MZ, Saedon J, Shahriman M, Omar AR, Kuntijoro W 2019. Comparative Distortion Analysis of Welded T-Joint between 2D-Shell and 3D-Solid Element using FEA with Experimental Verification. Journal of Mechanical Engineering S15(2) 99-115.
[11] Schafstall H, Neubauer I, Litzkow J. 2015. Einführung in die Prozesskettensimulation mit Simufact am Beispiel einer Crashbox-Von der Umformsimulation über das thermische Fügen zum Crash, Simulation pp. 158-161
[12] Sulaiman MS, Manurung YHP, Graf M, and Bauer A 2019. Analysis of weld induced distortion of butt joint using simulation and experimental study (in press).
[13] Vetriselvan R, Devakumaran K, Sathiya P, and Ravichandran G 2016. Transient out-of-plane distortion of multi-pass fillet welded tube to pipe T-joints. Defense Technology (in press)

Acknowledgments
The authors would like to express their gratitude to staff member of Welding Laboratory, Advanced Manufacturing Laboratory and Advanced Manufacturing Technology Excellence Centre at Faculty of Mechanical Engineering, Universiti Teknologi MARA (UiTM) for encouraging this research. The simulation and further evaluation technologies were carried out at our partner university Chemnitz University of Technology in Germany under international research grant of DAAD (Ref. Nr.: 57347629). This research is also financially supported by Geran Inisiatif Penyeliaan (GIP) from Phase 1/2016 with Project Code: 600-IRMI/GIP 5/3 (0019/2016).