Analysis of the influence of the pipe's wall thickness on the corrugation of the surface during inductive bending of pipes

Milan Sága[*,1], *Marián* Handrik[1], *Peter* Kopas[1], *Maroš* Tropp[2]

[1]University of Zilina, Faculty of Mechanical Engineering, Department of Applied Mechanics, Univerzitná 1, 010 26, Zilina, Slovak Republic
[2]Triple D Bending, 4707 Glenmore Trail SE, Calgary, Canada

Abstract. The paper presents a calculation model for simulation the technological process of inductive bending of pipes. In the case of an incorrect combination of geometric bending parameters (pipe diameter, bend radius and pipe wall thickness) occurs the corrugation of the inner part in the pipe bend. The proposed calculation model for the ADINA FE software allows for a relatively short time and at low cost to predict the emergence of corrugation.

Keywords: finite element analysis, induction bending, pipes, wall thickness, surface corrugation

1 Heat induction tube bending process analysis

The technological process of the hot heat induction tube bending is based on the principle of local heating of a tube and thus the change of mechanical properties of the material due to changed temperature. By means of a local increase in the temperature of the tube material, the mechanical properties are changed by plasticizing the material at lower stress values [1-3].

The first end of the tube is fixed in the bending arm, which rotates freely around the axis of rotation. The state of stress corresponding to the bend is induced in the tube by pushing the other end of the tube in the direction of the axis. The local change of mechanical properties in the point at the increased temperature allows the appearance of plastic deformations in the point of the increased temperature. The stresses in the area of elastic deformations are in other parts of the tube. Forming the boundary of the place with the local change of temperature is determined by a gradual displacement of the cold parts of the tube to the heating point, and there is a water spray on the other side of the inductor which cools the plastic deformed material. Two ways of guiding the tubes are used when bending the tubes using the induction heating: the use of guide pulleys, or the constraint at the feed point [4]. The schematic illustration of the tube bending process using the guide pulleys is shown in Figure 1.

* Corresponding author: milan.saga@fstroj.uniza.sk

Reviewers: *Jozef* Bocko, *Juraj* Gerlici

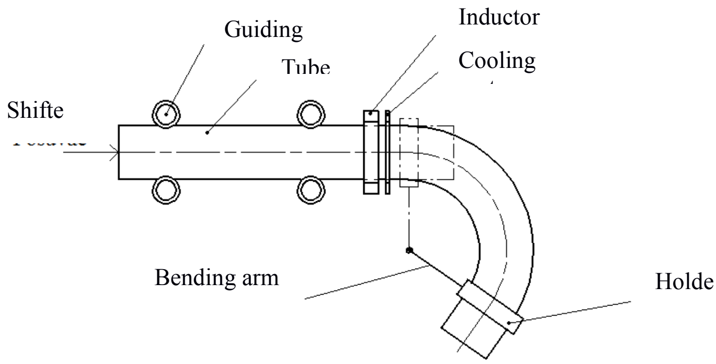

Fig. 1. The scheme of heat induction tube bending

A modern design of electrical power supplies for induction heating allows a non-contact temperature measurement at the heating point and subsequent control of the electrical quantities of the supply so as to maintain a constant temperature of the material at the heating point. A heat induction tube bending device is shown in Figure 2.

2 A simplified calculation model of heat induction tube bending

If we want to create a physically accurate model of the heat induction tube bending process, the computational model must contain the following physical fields:

- Electromagnetism – induction heating.
- Thermomechanical fields - change of mechanical properties due to temperature, plasticization of the material and contact of bodies.
- Interaction of fluid with the structure - coolant flow, heat transfer from the structure, free surface.

Such a computational analysis model becomes too complex when using the mentioned multiphysical model. The complexity of the problem lies in the large number of unknown variables that need to be solved and, at the same time, in the non-linearity of the problem solved. At present, it is possible to solve these complex physical models mainly by the finite element method (FEM). When solving the problem of the electromagnetism, fluid flow and contact, the finite element method requites a fine mesh of finite elements to achieve the convergence. The resulting models can have approximately hundreds of millions of equations and the solution is time-consuming.

In a more in-depth study of the hot heat induction tube bending process we find that it is possible to eliminate the electromagnetic fields and fluid flow from the model. The resulting model will deal with the thermomechanical analysis with the plasticity of the material:

- The purpose of the induction heating is to heat the tube at a defined constant temperature at point of inductor. In the induction heating, the heat is induced in the surface layers of the material and progressively spreads from the surface to the material. During the heating, the power of the inductor's electrical source is changed so that the temperature at the tube surface is constant.
- The purpose of the water spray is to cool the surface of the tube and to remove heat. In practice, the cooling water flow is set so that the heat removal is sufficient and the tube has cooled.

Fig. 2. Device for heat induction tube bending

There are two zones on the tube, one with the heating temperature and the other with the cooling temperature. Such a zone can be defined in the finite element method by applying appropriate limit conditions. During the bending, the tube is moved and the inductor with the water spray is not moved, which is a problem in defining the limit conditions that would also have to be moved around the surface.

The second way is to use heat transfer contact. At the inductor site, the bodies with the heating temperature are formed. Their contact with the tube as well as heat transfer in the respective contact are defined. The bodies which have coolant temperature are also formed at the point of the spray, and contact with the surface of the tube is defined [5].

3 Finite element method calculation model

If we replace the induction heating and the cooling spray with the heat transfer contact then the physical model of the heat induction tube bending is reduced to a thermomechanical analysis with heat transfer in the contact. All degrees of freedom apart from the radial movement are taken away from the formed spare bodies for inductor and spray. For all the bodies formed, the force of a small value is defined which causes that the bodies are pushed against the surface of the tube in the radial direction. The pushing the bodies against each other in the radial direction is necessary to ensure the contact lasting during the whole calculation and also when the shape of the tube is changed in the point of the plastic deformation - tube bending point, see Figure 3.

This type of task has a relatively good convergence and allows a significant reduction of the solved final equation system, in our case the final model had about 500 000 equations for the mechanical analysis and 150 000 equations for the heat transfer. An iterative solver was used in solving the thermomechanical analysis, where the structural analysis with the contact and the transfer of heat in a given time step were solved separately. Two to three iterations are needed to solve one time step.

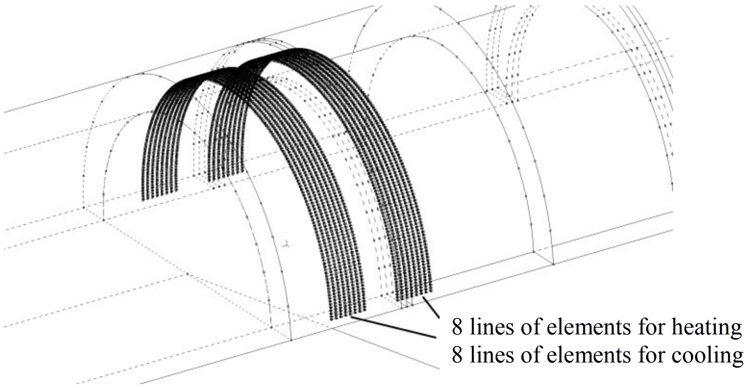

Fig. 3. Auxiliary bodies for heating and cooling the surface of the tube

On the other hand, there are separate matrices for structural analysis and temperature analysis with a smaller difference in value of elements on the main diagonal, which significantly improves the convergence of the contact problem and reduces the number of iterations required. Another simplification of the problem is the use of symmetry to create a geometric model and limit conditions. Due to the low feed rate of 2.4 m / hour, the task is solved as a quasi-static task considering large displacements and distortions. We use the temperature-dependent material bilinear model in the solution, see Figure 4 [6]. The geometric model of the tube is made of volume bodies; the need for the use of the volume bodies is given by the heat transfer from the surface of the tube in the direction of the wall thickness of the tube. If the shell elements are used, it would not be possible to determine the temperature gradient in the direction of the thickness of the tube. 5 elements for the wall thickness of the tube, 18 elements in the direction of the circumference of the tube on the half model, 10 mm in the longitudinal direction are used when generating a mesh. The finite element mesh is shown in Figure 5. The model is supplemented by the volume elements at the beginning of the tube and the beam elements that model the bending arm and its fixing to the tube [7].

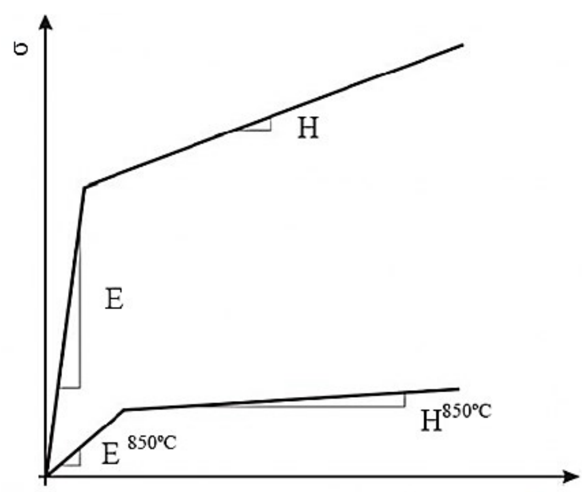

Fig. 4. Thermal-dependent bilinear material model

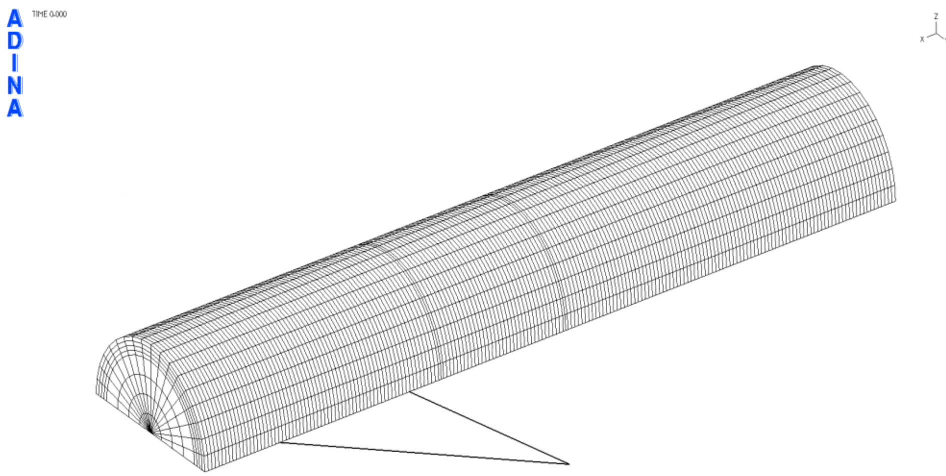

Fig. 5. Finite element mesh

4 Experimental verification of the calculation model

The proposed calculation model for simulating the heat induction hot tube bending process was experimentally verified and the results obtained by the calculation revealed a sufficient compliance with the experimental sample [8-12]. The investigated changes of forms (simulation vs. experiment) showed an error of less than 7%. The shape of the bend and change of the wall thickness of the tube were evaluated when comparing the obtained experimental results and FE model. The final shape of the tube after bending is shown in Figures 6 and 7.

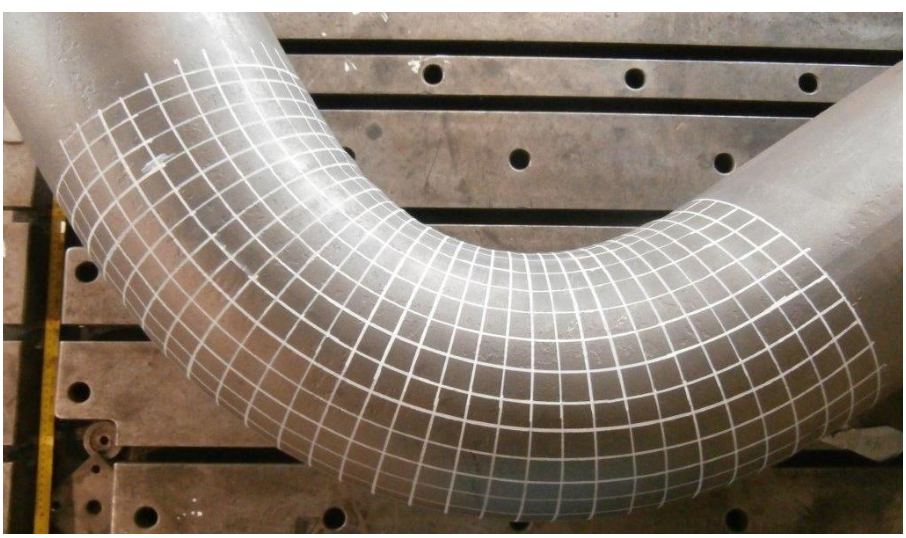

Fig. 6. Final shape of the tube after bending

Fig. 7. Final shape of the tube after bending, sectional view of the tube

5 Analysis of the effect of the wall thickness of the tube on the final shape after bending

The parameters describing the geometry of the tube were observed when studying the effect of the geometrical dimensions of the tube on the final shape of the tube after bending:

- Wall thickness of the tube: 10, 15, 20, 25, 30 [mm].
- Diameter of the tube: 245 [mm].
- Bending radius: 508 [mm].
- Angle of bend: 30 [°].
- Inductor and water spray width: 30 [mm].
- Inductor and water spray distance: 30 [mm].
- Heating temperature: 900 [°].
- Cooling temperature: 20 [°].

The final shapes of the tubes after bending for the respective wall thickness are shown in Figures 8 to Figures 12. A slight waviness can be observed on the inner side of the bent tube when the wall thickness of the tube was 10 and 15 mm. An increase in the wall thickness of the tube was found in case of a higher diameter of tubes.

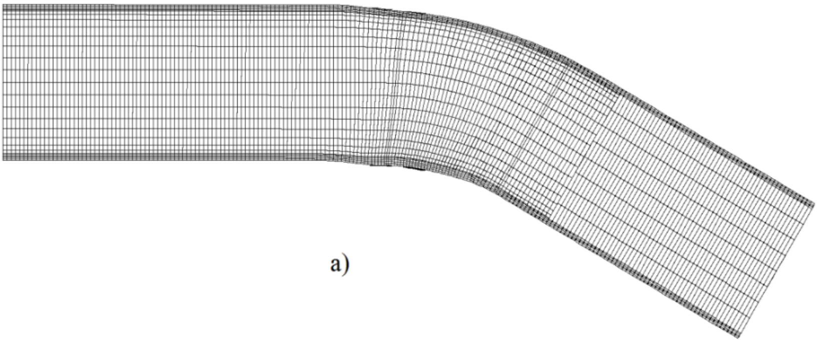

Fig. 8. Calculated bend form, wall thickness 10 [mm]

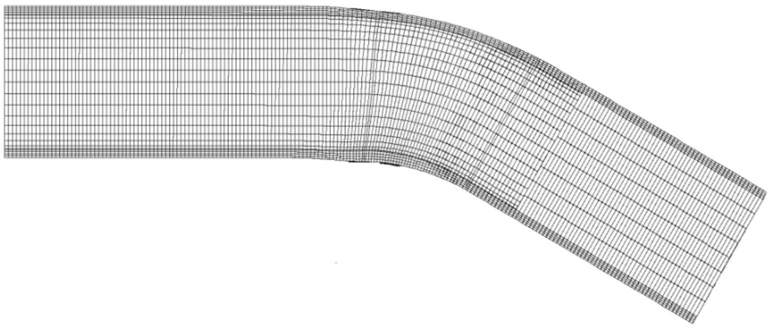

Fig. 9. Calculated bend form, wall thickness 15 [mm]

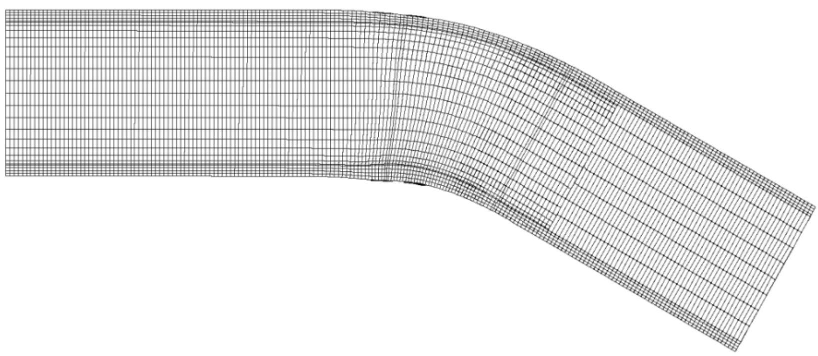

Fig. 10. Calculated bend form, wall thickness 20 [mm]

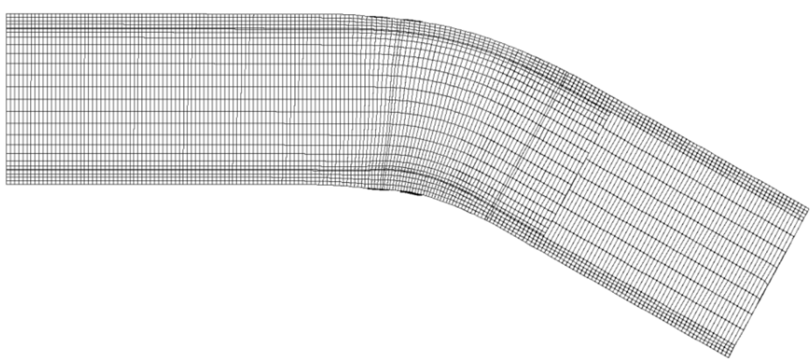

Fig. 11. Calculated bend form, wall thickness 25 [mm]

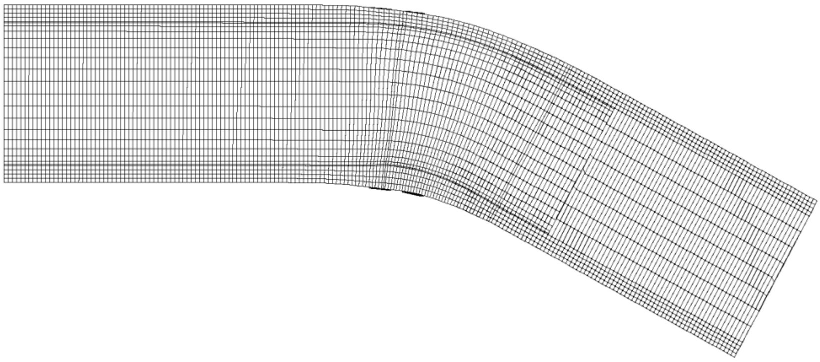

Fig. 12. Calculated bend form, wall thickness 30 [mm] is defined.

Conclusion

The created computational analysis model models the hot heat induction tube bending with the sufficient accuracy. The speed of the calculation model is acceptable for a given type of task. The calculation takes about 2 times longer than the actual bending process in the real production. The use of calculation models is effective despite the prolonged time of calculation, which leads, for example, to lower power consumption, fewer occurrences of production defects and the machine may be used for the production itself and not for the trial-and-error method experiments.

By using a computational calculation model, we can predict the shape of the tube after bending and so avoid the experiments, in which we know in advance that the final shape will not meet the required tolerances for the shape of the bend.

This work is supported by KEGA 015ŽU-4/2017 Digital technologies for machine building study programs as a part of strategy "Internet of Things".
This work was supported by the Slovak Research and Development Agency under the contract No.APVV-14-0096.

References

1. M. Handrik, P. Kopas, V. Baniari, M. Vasko, M. Saga, *Analysis of stress and strain of fatigue specimens localised in the cross-sectional area of the gauge section testing on bi-axial fatigue machine loaded in the high-cycle fatigue region*. Procedia Engineering **177**, 516-519 (2017)
2. P. Kopas, M. Saga, V. Baniari, M. Vasko, M. Handrik, *A plastic strain and stress analysis of bending and torsion fatigue specimens in the low-cycle fatigue region using the finite element methods*. Procedia Engineering **177**, 526-531 (2017)
3. M. Tropp, M. Handrik, P. Kopas, M. Saga, *Computer simulation of induction bending process*. Pipeline and Gas Journal **239**, (2012)
4. W. Piekarska, M. Kubiak, M. Vaško, *Numerical estimation of the shape of weld and heat affected zone in laser-arc hybrid welded joints*. Procedia Engineering **177**, 114-120 (2017)
5. L. Jakubovičová, B. Ftorek, V. Baniari, A. Sapietová, T. Potoček, M. Vaško, *Engineering design of a test device*. Procedia Engineering **177**, 520-525 (2017)

6. A. Vasko, M. Vasko, J. Belan, E. Tillova, *Comparison of fatigue properties of nodular cast iron at low and high frequency cyclic loading*. Procedia Engineering **177**, 576-581 (2017)
7. P. Kovacikova, R. Bezdedova, J. Vavro jr., J. Vavro, *Comparison of numerical analysis of stress-strain states of cast iron with vermicular graphite shape and globular graphite shape*. Procedia Engineering **136**, 28-32 (2016)
8. M. Nad, L. Rolnik, L. Icmancova, *Prediction of Changes in Modal Properties of the Euler-Bernoulli Beam Structures Due to the Modification of Its Spatial Properties*. International journal of structural stability and dynamics **17**, (2017)
9. R. Halama, P. Macura, L. Pecenka, F. Fojtik, M. Sofer, *Experimental Analysis of Residual Stresses in Backup Roll and FE solution*. Proceeding of 49th International Scientific on Experimental Stress Analysis, 85-90 (2011)
10. P. Zidlik, P. Ferfecki, B. Strnadel, *New computational procedures for evaluation of fracture surfaces of the drop weight tear test of the piping steel*. Proceedings of the ASME pressure vessels and piping conference Vol 1: Codes and standards, 1055-1060 (2012)
11. R. Grega, J. Krajnak, L. Zulova, G. Fedorko, V. Molnar, *Failure analysis of driveshaft of truck body caused by vibrations*. Engineering Failure Analysis **79**, 208-215 (2017)
12. M. Bruna, D. Bolibruchova, R. Pastircak, *Numerical simulation of porosity for Al based alloys*. Procedia Engineering **177**, 488-495 (2017)