Journal of Achievements in Materials and Manufacturing Engineering
VOLUME 17
ISSUE 1-2
July-August
2006

Shell element simulation of the push method of tube bending

Y. Zhang, D. Redekop*
Department of Mechanical Engineering, University of Ottawa, Ottawa, K1N 6N5, Canada
* Corresponding author: E-mail address: dredekop@uottawa.ca

Received 15.03.2006; accepted in revised form 30.04.2006

Analysis and modelling

ABSTRACT

Purpose: In this paper the new push bending process for the forming of curved tubes is simulated using the finite element method. It is demonstrated that the results obtained using shell elements compare closely with those obtained earlier using three-dimensional elements. A parametric study is carried out which gives on indication of the effect of changes in geometry and material properties on results.

Design/methodology/approach: A non-linear finite element analysis is carried out using the program LS-DYNA. A bilinear elastic plastic material is assumed, and both aluminum and steel are modelled.

Findings: It is found that the radius of the bend is significant with respect to potential wrinkling. The inner pressure can be increased to suppress possible wrinkling. Lubrication is shown to be significant with regard to final results for wall thickness.

Research limitations/implications: The present work is restricted to quasi-static behavior, and thermal effects are not considered.

Practical implications: Some limitation on thickness variation in the finished product is possible through choice of lubricant.

Originality/value: This paper gives original simulated results for tube push bending relating to new geometries and different materials.

Keywords: Computational mechanics; Finite Element Method; Simulation; Tube bending

1. Introduction

The use of modern methods to fabricate curved tubes (elbow sections) has become critical for success in the pipe forming industry. There is a need to decrease production costs and to improve component properties. The costs can be decreased by increasing work rates and reliability, while properties can be improved by offering better process characteristics. These requirements lead to the need of developing new, innovative manufacturing techniques.

Zeng and Li [1] have carried out experimentation of a new push bending process, while Baudin et al. [2] have described a finite element simulation based on the use of 3D elements. The process features the use of an urethane rod as a mandrel that serves to prevent wrinkling of the blank during forming (Figs. 1-2), The new method offers several advantages over some existing methods. The tube itself is placed inside a die and a plunger located at the end of the tube exerts an axial load on the tube and rod. This results in the tube and rod being pushed into the die, and the generation of an internal pressure in the tube. This pressure serves to inhibit wrinkling of the tube, and thus leads to a product with improved characteristics.

The objective of this study is to provide data concerning the new process. A new finite element method (FEM) simulation of the process is carried out, as analytical approaches to tube bending [3] give limited results. The intention in the present study is to generate further data to understand the deformation mechanisms in the tube, and to determine the effect of varying some process parameters. A non-linear explicit FEM software is used in the analysis. The current method of analysis is validated by comparing results with previous work. A parametric study is then carried out, giving results for a number of variations in the forming process, including geometry, material properties, and lubrication parameter, and finally conclusions are drawn.

2. Background to the approach

2.1. Tube bending studies

The topic of tube bending and its analytical and numerical simulation continues to be a very active one [1-13]. In addition to the new push bending process [1-2] a number of other processes have been proposed in recent years. An early work by Hu and Li [4] concerned the local induction heating method for small bending radius. An FEM and an analytic approach using finite strain elasto-plastic theory were developed. Hydroforming has been discussed by Yang and Jeon [5], Yuan et al [6] and Wang et al [12], mainly for large radius bends. Both experimentation and FEM simulation are mentioned in these studies.

The laser tube bending process have been discussed by Hao and Li [8], and Hsieh and Lin [10]. Goodarzi et al [11] performed a deformation analysis for the shear bending process. Most work has concerned the formation of circular tubes, but Lee et al [13] have dealt with oval tubes. FEM simulation of tube bending process has been highlighted in a number of papers including those of Zhan et al [7] who considered the NC bending process, Gao and Strano [9] who modeled pre-bending and hydroforming.

2.2. Finite element method

The explicit non-linear FEM software LS-DYNA [14] was used in this study to perform the simulation. The explicit method provides fast solutions for a large deformation dynamics and complex contact/impact problem. The geometry was preprocessed by ETA FEMB, the explicit dynamic solution was obtained using LS-DYNA3D, and results were archived using the LS-DYNA post-processing software.

Exploiting symmetry, one half of the geometry was modelled (Figs. 3-4). Shell elements rather than 3D elements were employed. A non-linear elastic-plastic material model [15] was used to describe the tube material behavior. In this case the overall duration of the forming process is relatively long so a strain-rate independent material model is acceptable. In LS-DYNA there are two strain rate independent plasticity models available: the classical bilinear kinematics hardening model and the classical bilinear isotropic hardening model, of which the latter was used.

The geometry of the model was developed using the SolidWorks software. The basic model consisted of a 90^{o} bend in a straight tube of inner diameter 37.6 mm, outer diameter 40mm, and length 140 mm. The die had an inner radius of 40mm and a die bend radius of 70 mm. A meshing of the die, tube and rod is shown in Figs. 3-4. An automatic surface to surface contact algorithm was employed at the interface between the die and the tube. The coefficient of friction of 0.1 was initially assigned for die-tube contact.

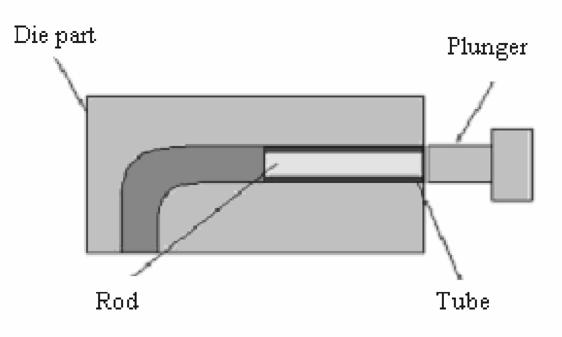


Fig. 1 Symbolic representation of experimental apparatus – before forming

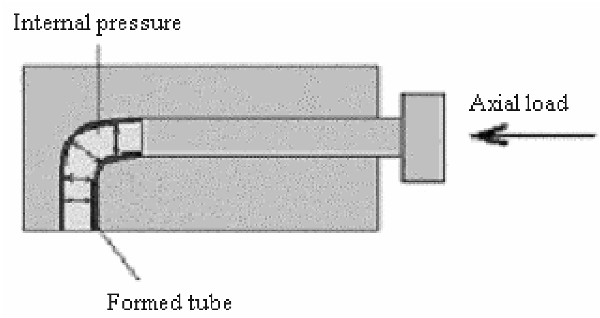


Fig. 2 Symbolic representation – after forming

The die in this case was modelled as a rigid body and the nodes in the symmetry plane of the tube were constrained in the appropriate direction. A displacement versus time loading curve was applied to the nodes at the free end of the tube and rod in order to model the punch force. An internal pressure of 5 MPa pressure in the tube was employed to simulate the effect of the urethane rod on the tube.

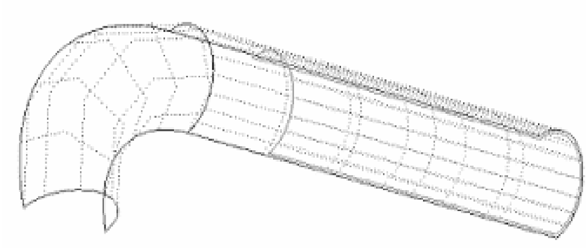


Fig. 3. FEM model of die and blank

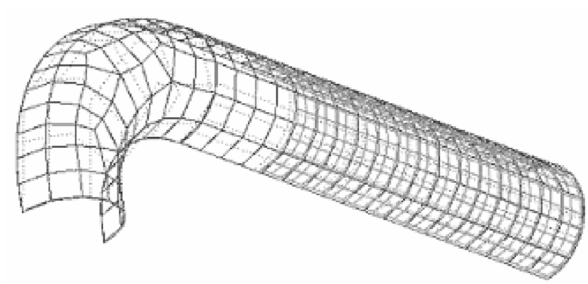


Fig. 4. FEM mesh of die and blank

3. Validation and results

3.1. Comparison with previous work

To validate the current work comparisons were made with results from the previous study of Baudin et al [2]. In the previous study, 3D elements were used for both the tube and urethane rod. In the present study, thin shell elements were used to mesh the tube part, and inner pressure was applied on the inner surface of the tube to represent the rod effect. For the validation a tube bending radius of 70mm was used. The material was assumed to be aluminum alloy AA6063 with the following properties ρ=2670 kg/mm^3, E=65 GPa, ν=0.38, η=181.09MPa (hardening modulus), σ_y=135MPa (yield stress), σ_u=152MPa (ultimate stress).

A comparison of the resuls from the previous [2] and present study is given in Table 1. Results are given for the final wall thickness at the extrados and intrados sides of the bend for three cross-sectional positions, A, B, C (Fig 5). Two sets of results from the present study are given, one corresponding to a regular mesh, and the other to a fine mesh. Although a different element type and a different simulation method were employed in the present study there is close agreement in the results at all locations.

Table 1.
Comparison of results using regular and fine mesh (model for radius = 70mm, aluminium material)

Cross section	Previous results [2] (mm)		Current results (mm)	
	Extr.	Intr.	Extr.	Intr.
A	1.34	1.63	1.46^r	1.75^r
			1.34*	1.58*
B	1.05	1.51	1.12^r	1.52^r
			1.14*	1.50*
C	1.22	1.24	1.19^r	1.29^r
			1.19*	1.32*

^{r} results from regular mesh, * results from fine mesh

Table 2.
Simulation result for medium bending radius (model for radius = 55mm, aluminium material)

Cross section	Thickness (mm)		Stress (max. von-Mises) (GPa)	
	Extr.	Intr.	Extr.	Intr.
A	1.44	1.58	0.10	0.19
B	1.11	1.63	0.14	0.17
C	1.20	1.26	0.09	0.13

Table 3.
Simulation results for smallest bending radius (model for radius = 40mm, aluminium material)

Cross section	Thickness (mm)		Stress (max. von-Mises) (GPa)	
	Extr.	Intr.	Extr.	Intr.
A	1.44	1.45	0.13	0.17
B	1.11	1.73	0.15	0.18
C	1.19	1.27	0.13	0.13

3.2. Parametric study

Push bending is suitable for tubes with small bend radius. To further explore the use of the method two additional simulations, involving smaller bend radius, were carried out. The additional simulations were for radii of 55mm and 40mm respectively. Results from the present study are given in Tables 2 and 3. In these tables results are also given for the von Mises stress at the various locations.

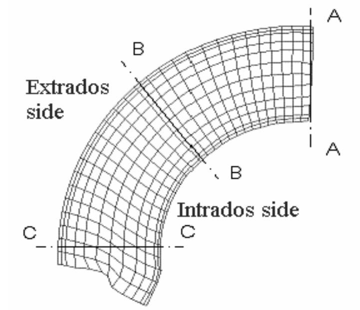


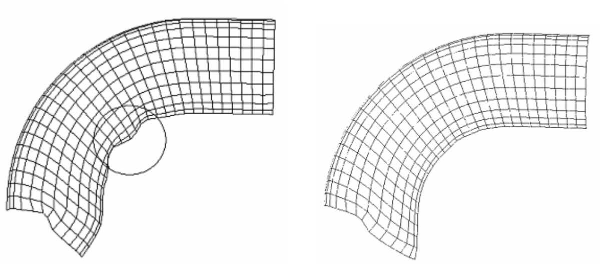
Fig. 5. Position of cross-sections A, B, and C for Tables 1-5

As the bending radius decreases, the maximum change in thickness increases. The largest effect is at cross-section B. As the bend radius decreases, the thickness at the extrados (tensiled part) decreases and the thickness at the intrados (compressed part) increases. Furthermore, the maximum von Mises stress increases as the bend radius is decreased. It is clear that there is a difficulty in bending of a very small radius tube. For a low value of wall thickness there are obvious problems: there is easier breakage and erosion, and even the potential for buckling. Furthermore, the stress is important in the process; in the tensile part if the stress exceeds the ultimate stress of the material the tube will fail.

To investigate the effect of material properties a simulation was carried out for a steel material. The material was assumed to have the following properties ρ=7900 kg/mm^3, E=200 GPa, ν=0.3, ˜ =300MPa (hardening modulus), σ_y =250MPa (yield stress), σ_u=400MPa (ultimate stress). Results for this simulation are given in Table 4. The von Mises stress is clearly larger than for the aluminium material. This result is explained as due to the larger Young's modulus and yield stress of steel. However, the lowest value of wall thickness for the steel tube is smaller than for the aluminium one, also the largest value of thickness exceeds that for aluminium (both of the models were based on a bending radius of 70mm).

The lubrication has an effect on the geometry of the bent tube, and thus a simulation was carried with a different value of this parameter. This simution is for a case of lower lubrication characterized by a friction coefficient of 0.15 (rather than 0.10 used earlier). The results are given in Table 5. It is observed that there is a decrease in the thickness, and that the von Mises stress increases. The thinnest part of the tube decreased by 3.7% relative to the model with a friction coefficient of 0.1. Thus use of good lubrication condition is clearly favorable for this bending process.
It was found also that as the bend radius is decreased there is an increased potential for wrinkling. The wrinkling of course is a problem at the intrados in the compressed part, and Fig. 6(a)

shows a simulation where wrinkling occured. To control the wrinkling problem a higher internal pressure could be applied, or a longer forming time used. A higher internal pressure would imply a stiffer mandrel, with a larger Young's modulus. The slower the forming, the lower is possibility of wrinkling. Fig. 6(b) shows a bent tube shape for a simulation with a higher inner pressure (10MPa) and a longer processing time (one minute). No wrinkling ocurred for these choices of forming parameters.



a) b)

Fig. 6. Results for simulation of forming, a) with wrinkling, b) without wrinkling

Table 4.
Simulation results for steel tube (model for radius = 70mm)

Cross section	Thickness (mm)		Stress (max. von Mises) (GPa)	
	Extr.	Intr.	Extr.	Intr.
A	1.36	1.9	0.13	0.33
B	1.07	1.56	0.17	0.25
C	1.25	1.23	0.12	0.13

Table 5 Simulation results for friction coefficient of 0.15 (model for radius = 70mm, aluminium material)

Cross section	Thickness (mm)		Stress (max. von Mises) (GPa)	
	Extr.	Intr.	Extr.	Intr.
A	1.33	1.54	0.14	0.25
B	1.10	1.48	0.10	0.15
C	1.19	1.31	0.07	0.12

4. Conclusions

From the results for the simulation it was concluded that LS-DYNA is a powerful tool in analysing tube bending. Results obtained using a shell element give results for wall thickness results that are very close to previously published data. A number of parameters affect the forming results, including bend radius, internal pressure, material, friction coefficient, and process time. The bend radius is an essential parameter, as it is decreased difficulties arise. Inner pressure is important in maintaining the shape of the tube, and specifically to reduce winkling. Material properties are also important parameters for the forming. When steel is used, a lower wall thickness is predicted. The friction coefficient is less important, but when it is largerr than say 0.2, an undesirable shape containing wrinkles is generated. Limiting the coefficient to 0.15, seems advisable, suggesting the need for good lubrication. A final method of controlling wrinkling is to vary the forming time.

References

[1] Y.S. Zeng, Z.Q. Li, Experimental research on the tube push-bending process, Journal of Materials Processing Technology 122 (2002) 237–240.
[2] S. Baudin, P. Ray, B.J. Mac Donald, M.S.J. Hashmib, Development of a novel method of tube bending using finite element simulation, Journal of Materials Processing Technology 153–154 (2004) 128–133.
[3] H.A. Al-Qureshi, Elastic-plastic analysis of tube bending, International Journal of Machine Tools & Manufacture 39 (1999) 87–104.
[4] Z. Hu, J.Q. Li, Computer simulation of pipe-bending processes with small bending radius using local induction heating, Journal of Materials Processing Technology 91 (1999) 75-79.
[5] J.B. Yang, B.H. Jeon, S.I. Oh, the tube bending technology of a hydroforming process for an automotive part, Journal of Materials Processing Technology 111 (2001) 175-181.
[6] S.J. Yuan, B.G. Teng, Z.R. Wang, A new hydroforming process for large elbow pipes, Journal of Materials Processing Technology 117 (2001) 28–31.
[7] M. Zhan, H. Yang, Z.Q. Jiang, Z.S. Zhao, Y. Lin, A study on a 3D FE simulation of the NC bending process of thin-walled tube, Journal of Materials Processing Technology 129 (2002) 273-276.
[8] N. Hao, L. Li, Finite element analysis of laser tube bending process,Applied Surface Science 208-209 (2003) 437-441.
[9] L. Gao, M. Strano, FEM analysis of tube pre-bending and hydroforming, Journal of Materials Processing Technology 151 (2004) 294-297.
[10] H.S. Hsieh, J.M. Lin, Study of the buckling mechanism in laser tube forming, Optics & Laster Technology 37 (2005) 402-409.
[11] M. Goodarzi, T. Kuboki, M. Murata, Deformation analysis for the shear bending process of circular tubes, Journal of Materials Processing Technology 162-163 (2005) 492-497.
[12] Z.R. Wang, G. Liu, S.J. Yuan, B.G. Teng, Z.B. He, Progress in shell hydroforming, Journal of Materials Processing Technology 167 (2005) 230-236.
[13] H.K. Lee, C.J. Van Tyne, D. Field, Finite element bending analysis of oval tubes using rotary draw bender for hydroforming applications, Journal of Materials Processing Technology 168 (2005) 327-335.
[14] LS-DYNA Theoretical Manual. Livermore Software Technology Corporation, 1998.
[15] W. Johnson, P.B. Mellor, Engineering Plasticity, Wiley & Sons, New York, 1983.