Study on forming and laying process of high strength pipeline steel

Lian Yeda*, Ma Qiang, Zhang Bing, Zhang Yuqiang and Li Hao
School of Mechanics and Civil Engineering & Architecture, Northwestern Polytechnical University, 710072, Xian, China

*Email: lianyeda@qq.com

Abstract. In order to analyze the simulation results of UOE molding and solve the buckling problem of high strength pipeline steel in the process of ocean laying, taking X80 pipeline steel sample as the object of study, a better mechanical model of kinematic hardening was adopted. With the aid of ABAQUS finite element analysis software, the finite element analysis of UOE simulation results, the loading of UOE simulation results and buckling in ocean laying are carried out. The results show that: Through the extraction of the load at the end of the welded tube and the strain at the buckling position, draw it into curves and observe the position of the strain corresponding to the inflection point of the curve. The critical strain of welded pipe subjected to buckling under different working conditions is analyzed, so it is found that the critical strain increases with the increase of internal pressure and the critical strain becomes smaller with the increase of the external pressure. The analysis method and results can also provide reference for the actual production, laying and installation of oil and gas pipelines.

1. Introduction

High-strength pipeline steel is one of the popular research fields in the pipeline engineering industry at home and abroad, which is characterized by high strength and high toughness. The UOE molding process in the pipeline steel forming process is a commonly used molding method in the world today. In the process of laying the pipeline in the ocean, the pipeline is prone to buckling for environmental factors, so the problem of buckling failure of pipeline steel is one of the problems to be solved in the pipeline laying process.

Xing et al. [1] studied the buckling analysis of the submarine pipeline under axial load based on the Euler-Bernoulli theory, and obtained the failure criterion in this buckling condition. Wang Xiaoxiu et al. [2] studied the three-dimensional finite element simulation of UOE pipe forming based on elasto-plastic finite element method, verified the accuracy of the model, and studied the variation of sheet size in UOE forming process. Edited et al. [3] simulated the UOE molding of high-strength pipeline steel to verify the correctness of the mechanical model selection. At the same time, the load applied to the mold in each molding step is also analyzed, and it is verified that the load applied by the process machine in the molding process is within the planned range. Liang Zhenting [4] studied the various pipelining methods in the ocean, analyzed the buckling problem in marine laying and proposed the solution to the buckling problem.

In this paper, X80 pipeline steel is selected as the sample, and the ABAQUS finite element analysis software is used to establish the kinetic hardening mechanical model which is most consistent with
actual situation. It is applied to the finite element simulation of buckling analysis of UOE forming and marine pipe laying.

2. Establishment of the kinetic hardening model of Pipeline Steel

In recent years, the research on the kinetic hardening model of steel under large deformation conditions has been studied in the literature [5]. Rudolph et al. [6] used ABAQUS finite element analysis software to analyze the constitutive model of metallic materials. In this paper, the unidirectional tensile test and tensile-compression test of the round bar intercepted in X80 sheet are carried out, and the basic mechanical parameters of the material are obtained. The ABAQUS finite element analysis software was used to establish the same simulation scheme as the experiment. The kinematic hardening model of ABAQUS material library and the kinematic hardening model defined by UMAT are used to calculate and compare the stress-strain curves and the experimental curves respectively. The results show that the effect of the kinematic hardening model defined by the UMAT subroutine is in accordance with the experimental results.

3. UOE finite element modeling simulation

U molding process is to use the vertical die to load the sheet until the ends of the horizontal angle of about 90° to stop loading, then use both sides of the roll to load the plate horizontally to the sheet metal contact, and then unload to make the sheet rebound into "U"; O molding process is from the U mold out of the sheet into the fixed O mold lower part and the upper part of the O die is loaded downward to move the ends of the sheet to both sides. When the O die is loaded down to the maximum displacement, the sheet is essentially O-type, and the upper part of the O-die is withdrawn along the loaded reverse path; After leaving the O molding die, the sheet contacts the expanding die, the eight molds in the middle of the sheet load to the sheet metal along the circular radial outward to complete the final expansion process of UOE molding.

3.1. Analysis of UOE simulation results

In the process of bending edges, the mold exerts a load on the sheet, the stress distribution of the sheet is relatively concentrated, as shown in figure 1 (a). The stress value produced by the outer layer of the sheet shown in figure 1 (b) is larger, the stress value produced by the intermediate layer is smaller and the stress range is spread. As shown in figure 1 (c), the degree of loading of the sheet is gradually increased, and the degree of bending produced by the sheet is also increased. The change of the equivalent stress is larger and the equivalent stress concentration also spreads gradually to the edge of the sheet metal. When the crimping process is substantially completed, the position of the maximum sheet stress is located near the middle layer, as shown in figure 1 (d).

In the bending process of U forming, when the die moves down vertically, both sides of the sheet are bent. At this time, the maximum stress is located at the joint of the plate and the die. As the die is loaded gradually, the stress changes gradually to both ends, which makes the stress on the outer surface of the sheet larger and the stress on the inner surface smaller, as shown in figure 2 (a). Figure 2 (b) shows the die exerts a horizontal load on the sheet, so that the sheet is drawn closer to the interior. The maximum position of the sheet metal is derived from the edge of the sheet, and the stress in the other positions of the sheet is not significantly changed. In this molding process, the sheet is made "U" by pressing down the mold, and then the horizontal roller mold continues to load the sheet. The plastic deformation of the sheet is completed. Finally, the horizontal loading die is unloaded to make the plate rebound, from figure 2 (c) can be seen in the middle part of the sheet stress is greater than the external surface stress.

As can be seen from figure 3 (a), the sheet is deformed at the bottom of the forming process in the O molding process, the inner layer is compressed to produce compressive stress, and the outer layer is stretched to produce tensile stress. When the applied load is larger, the pressure of the sheet is also increased, the two ends of the sheet and the central part began to protrude outward, gradually forming as shown in figure 3 (b). As can be seen from the figure 3 (b), the stress of the sheet is gradually
increased, and spread to the center and both ends of the sheet and the final shape is O type. As can be seen from figure 3 (c), the sheet is basically compressed and molded, the compression position is the greater stress, and the stress on the outer surface is greater than the inner surface. The upper end of the mold moves upwards, the O molding of the sheet material is unloaded, and the sheet material generates local rebound, as shown in figure 3 (d).

The expansion process is shown in figure 4. As shown by figure 4 (a), due to the impact of incomplete contact with the mold, the plastic deformation first appears near the edge of the sheet, and because the curvature is small, the position of the maximum stress is also located, as shown in figure 4 (b). With the expansion of the volume gradually increased, local plastic deformation gradually extended to the inner wall of the sheet. The sheet material gradually enters the plastic stage, from inside to outside, from the center to the sides, where the internal stress state of the sheet is mostly tensile stress, as shown in figure 4 (c). When the mold is unloaded, the sheet is spring back. As shown in figure 4 (d), after the mold is completely unloaded, the residual stress will be greatly reduced before the expansion due to the expansion process. It can be seen from the figure 4 (d) that the residual stress is mainly distributed in the area where the sheet is subjected to the maximum diameter.

3.2. Load Analysis of UOE simulation results
In order to visually understand the stress changes in the molding process, the sheet on the mold by the loading capacity of a large number of units were removed alone. The main method of selecting the points is to divide the sheet into two parts and half to be the object of study, and then select the three
units in the upper and lower layers respectively, and select the position as shown in figure 5. The stress change of the unit in the whole process is extracted, and the stress forming step curve is plotted, as shown in figure 6.

As can be seen from figure 6, the forming process of maximum load is the crimping, O forming and expanding, respectively. In other molding processes, the load applied to the sheet is small, and the stress curve can also be used to determine the UOE sheet strength.

![Diagram](image)

**Figure 5.** Unit location selection.

![Graphs](image)

**Figure 6.** Stress molding step diagram.
4. Finite element simulation analysis of buckling in marine laying

YAO Baoheng et al. [7] established the vibration equation in the longitudinal direction for the submarine pipeline, and solved the influence of the wave frequency on the stress distribution and amplitude of the pipeline in different marine environments by Galerkin truncation method. Fabian [8] studied the application of joint load to the pipeline in the elastic deformation range to cause buckling, the results were analyzed and studied. As shown in figure 7, the kinetic hardening model defined by UMAT is loaded without the application of internal pressure until buckling occurs and the buckling position is in the middle of the pipe compression surface. Under the ideal condition, the finite element simulation is carried out. Compared with a large number of tests, it can be seen that the buckling location occurs in the middle of the model under the kinetic hardening model, which is more in line with the actual situation. At the same time, the data of the cloud image are extracted, and the relationship curve between load and strain is drawn, and then compared with the experimental results of Fathi [9].

When the critical strain of the simulation results is 1.41%, the pipe is buckled and the load on the pipe is 1034 kN. And the critical strain is 1.63% when the pipe is buckled on the Fathi test curve, the load of the pipeline is 1051 kN. By comparison, it is found that the results of the two are basically consistent, and the accuracy of the model is verified as shown in figure 9.

In the process of oil and gas transportation, the internal pressure is required to be applied to the pipeline. Therefore, when the finite element simulation is carried out, the relationship between the critical stress and the critical strain is observed by applying 8Mpa, 10Mpa and 12Mpa pressure respectively. Figure 8 shows the buckling simulation cloud diagram with internal pressure applied to the pipe and figure 10 is the strain-load curve under three internal pressures.

As shown in table 1, the critical strain of the buckling increases with the increase of the internal pressure of the pipeline within a certain range, and the critical load decreases.

| Pipe internal pressure (MPa) | Pipe external pressure (MPa) | Critical strain (%) | Critical load (kN) |
|-----------------------------|------------------------------|--------------------|-------------------|
| 8                           | 0                            | 1.07               | 925               |
| 10                          | 0                            | 1.13               | 868               |
| 12                          | 0                            | 1.21               | 798               |

When the pipeline has not yet run, there is no internal pressure, so in the laying process of marine pipelines, the pipeline is mainly affected by bending moment, axial force and the role of external pressure. Figure 11 shows the buckling simulation of the pipe when subjected to external pressure. Figure 13 shows the strain-load curves under three external pressures.

As shown in table 2, the critical strain of the buckling decreases with the increase of the external pressure of the pipeline within a certain range, and the critical load decreases.

| Pipe internal pressure (MPa) | Pipe external pressure (MPa) | Critical strain (%) | Critical load (kN) |
|-----------------------------|------------------------------|--------------------|-------------------|
| 0                           | 0.5                          | 1.23               | 1030              |
| 0                           | 1                            | 1.18               | 1017              |
| 0                           | 2                            | 0.98               | 991               |

When the pipeline in the submarine officially operates, not only the internal internal pressure, the sea also caused external pressure on the pipeline, the pipeline is mainly subject to bending moment, axial force, internal pressure and the role of sea pressure. Figure 12 shows the buckling simulation of
the pipe when subjected to internal and external pressure. Figure 14 shows the strain-load curves under three external pressures.

Table 3. Analysis of pipe buckling results.

| Pipe internal pressure (MPa) | Pipe external pressure (MPa) | Critical strain (%) | Critical load (kN) |
|-----------------------------|-------------------------------|---------------------|-------------------|
| 8                           | 2                             | 1.07                | 972               |
| 10                          | 2                             | 1.08                | 925               |
| 12                          | 2                             | 1.11                | 871               |

As shown in table 3, the external pressure does not change and the internal pressure increases within a certain range, and the critical strain is increased and the critical load is reduced when the pipe is buckled.

Figure 7. Buckling simulation of noninner pressure welded pipe.

Figure 8. Buckling simulation under the condition of internal pressure.

Figure 9. Load strain curve without internal and external pressure.

Figure 10. Load strain curves under three internal pressure conditions.

Figure 11. Buckling simulation under external pressure.

Figure 12. Buckling simulation under the condition of internal and external pressure.

Figure 13. Load strain curves under three external pressures.

Figure 14. Load strain curves of three kinds of internal and external pressure.
5. Conclusion
Based on the basic theory of elasto-plasticity and the X80 round bar test, this paper establishes the kinetic hardening mechanical model which is the most consistent with the actual situation. For the UOE forming process and the buckling problem in the pipeline laying, ABAQUS finite element software is used to simulate the simulation. As follows:

1) Establish the same simulation scheme as the test. The kinematic hardening model of the ABAQUS material library and the kinematic hardening model defined by UMAT are calculated respectively. The data obtained from the two methods are compared with the experimental data. It can be seen that the effect of the kinematic hardening model defined by the UMAT subroutine is consistent with the experimental results.

2) A detailed stress-strain analysis is performed for each step of UOE molding. The three-dimensional model of ABAQUS can be used to understand the deformation of the sheet during UOE molding. In the simulation process, some important positions on the sheet and the stress, strain and load of the mold are output, so that the weak trend of the structure and the stress change inside the plate can be understood in advance.

3) By using ABAQUS finite element simulation, a simplified pipeline model is established and loaded. For different working conditions, according to the load and strain data to draw the curve, the critical load and critical strain are obtained when the pipeline is buckled. The results show that if the internal pressure increases, the critical strain will increase; if the external pressure increases, the critical strain will decrease. The critical load will be reduced regardless of the increase of internal and external pressure.

Acknowledgement
This work is supported by National Natural Science Foundation of China Under Grant No. 51308458.

References
[1] Xing Jingzhong, Liu Chuntu and Xu Yongjun. Buckling analysis of buried spanning submarine pipeline[J]. Engineering Mechanics, 2006, 23 (2) pp 173–76(in Chinese)
[2] Wang Xiaoxiu, Zou Tianxia and Li Xinwen, et al. Simulation and analysis of UOE pipe production processing by three-dimensional finite element method[J]. Baosteel Technology, 2014(1) pp 40–46(in Chinese)
[3] Edited, By T, Cui-Yan L I. Numerical Simulation Analysis on High Strength UOE Steel Pipes[J]. Welded Pipe & Tube, 2010
[4] Liang Zhenting. Analysis on mechanical performance for deepwater pipe-laying[D]. Hangzhou: Zhejiang University, 2008(in Chinese)
[5] Wallin M, Ristimaa M and Ottosen N S. Kinematic hardening in large strain plasticity[J]. European Journal of Mechanics - A/Solids, 2003, 22 (3) pp 341–56
[6] Rudolph J, Gilman T, Weitze B, et al. Using Nonlinear Kinematic Hardening Material Models for Elastic-Plastic Ratcheting Analysis[C]// ASME 2015 Pressure Vessels and Piping Conference. American Society of Mechanical Engineers, 2016
[7] Yao Baoheng, Ren Ping and Li Zhigang, et al. Dynamic analysis of pipeline laying in deep ocean based on Galerkin truncation method[J]. Journal of Vibration and Shock, 2008, 27 (10) pp 43–45(in Chinese)
[8] O, Fabian. Collapse of cylindrical elastic tubes under combined bending pressure and axial loads[J]. International Journal of Solids and Structures. 1977(13) pp 1257–70
[9] Fathi A, Cheng J J R, Adeeb S, et al. Critical Buckling Strain in High Strength Steel Pipes Using Isotropic-Kinematic Hardening[C]//International Pipeline Conference. 2010 pp 39–47