DOI: 10.21276/sjet

Scholars Journal of Engineering and Technology (SJET)
Sch. J. Eng. Tech., 2017; 5(5):182-190

(An International Publisher for Academic and Scientific Resources)
www.saspublisher.com

ISSN 2321-435X (Online)
ISSN 2347-9523 (Print)

Research Article

Thermodynamic Analysis of Variable - Diameter Pipes under Temperature

Xue Jinghong, Chu Yanghua
School of Civil Engineering and Architecture, Northeast Petroleum University, Daqing 163318, China

***Corresponding author**
Xue Jinghong
Email: 963270758@qq.com

Abstract: In the event of thermal expansion and contraction of the pipeline will produce thermal stress, due to the existence of thermal stress to reduce the diameter of the pipeline extrusion caused by deformation of the pipeline. In this paper, the finite element software ANSYS is used for modeling and analysis, and the eight-node solid element model is used to simulate the variable pipe. In this paper, we study the weak position of the variable pipe under the action of temperature, the influence of the temperature on the variable pipe, the influence of the temperature difference on the variable pipe, and the stress analysis of the variable pipe under the fire. The results show that the weak position of the variable pipe is located around the change of the pipeline under the action of temperature. The limit stress of the variable pipe is related to the temperature and temperature of the pipeline. The study provides a reference for the design of thermal pipelines, making the heat transfer variable diameter pipeline design reasonable energy saving.
Keywords: variable pipe; finite element software; Temperature, maximum strain position; thermosetting coupling.

INTRODUCTION

Under the influence of economic globalization, the global economy and population have undergone tremendous changes. Pipeline engineering has become a livelihood project in various countries and regions, and has become a life engineering [1]. In many places have been inseparable from the application of the pipeline, such as heating projects, submarine oil transport, natural gas transportation, municipal engineering, pipeline more than the advantages of its decision as the national economic development and residents of the important project [2-4]. However, due to the differences in the production process of the pipeline itself and the normal operation of environmental protection and energy has brought challenges, people pay more and more attention to this issue [5].

Through the analysis of the energy saving and environmental protection of the pipeline, many scholars have made a lot of theoretical research, experimental research and finite element simulation analysis in the energy saving of the pipeline, and have obtained many useful research results [6]. Many of the research results on the pipeline are the result of the establishment of a straight pipe, but in engineering practice, many pipes can't be avoided [7-16]. In this paper, based on the previous scholars' research, the finite element simulation software is used to simulate the variable pipe model, which can provide a good reference for the design of the pipeline. The influence of the temperature on the thermal stress of the pipe, the influence of the temperature difference on the thermal stress of the pipe, and the thermal stress analysis of the variable diameter pipe under the fire are studied.

FINITE ELEMENT MODEL WAS ESTABLISHED

Design of variable pipe fittings

In the engineering design, in order to make the variable pipe in the operation in a stable state, for the variable pipe design has a certain requirement. In this paper, we use the API5LX60 imported steel to design the variable diameter pipeline. In this paper, we use the 8-node solid unit to establish the concentric variable diameter pipeline model to analyze the variable angle of the variable diameter device. Take the two ends fixed at both ends of the variable pipe, The welding device is connected at the variable diameter device.

Geometric models and meshing

In this paper, API5LX60 imported steel as a research material, the stress-strain relationship of the material for the oil and gas pipeline design specifications of the three-fold simplified model, see Figure 1. The length of the pipeline is 32 meters, and the 8-point solid unit is used to divide the variable pipe. The element is divided by the automatic division grid function of ANSYS finite element software, 0.5m in the axial direction along the pipeline, Divide 32 units. The finite element model of the modified pipe is shown in Fig.

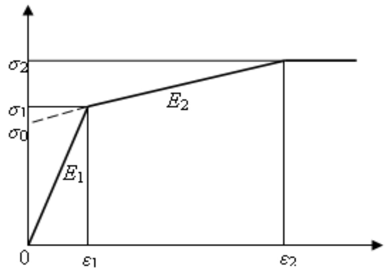

Fig-1: stress-strain relationship three line model

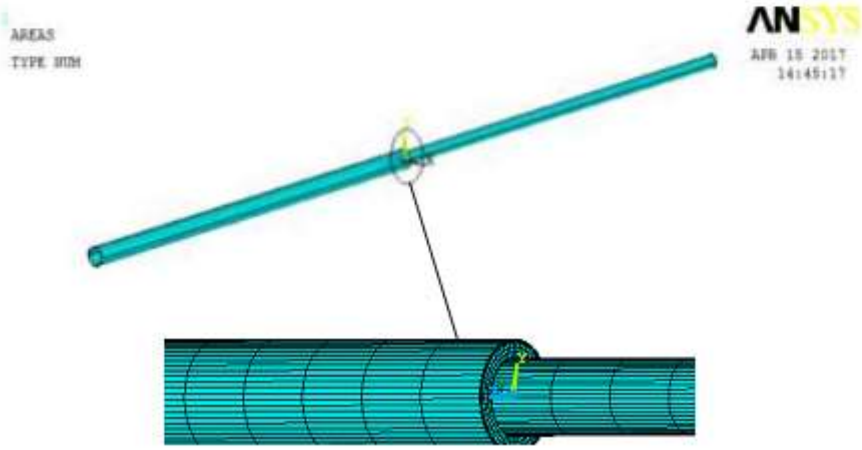

Fig-2 variable diameter pipe finite element model

Analysis of Heat - Structure Coupling in Pipeline

In this paper, the temperature loading method is used to load the temperature on the inner and outer surfaces of the variable diameter pipe by means of indirect loading. The thermodynamics is used to convert the temperature into stress and then applied to the structure (equivalent to the external load) The corresponding thermodynamic units are replaced by structural mechanics units and the corresponding internal forces (stresses) are obtained. In this paper, the API5LX60 steel is used for modeling and analysis. The large diameter D1 = 0.762m, wall thickness = 0.0238m; small diameter D2 = 0.508m, wall thickness = 0.0238m; elastic modulus E = $2.1e^{10}$Pa, Poisson's ratio = 0.3; pipe embedding depth H = 2.5m; heat transfer coefficient 21W / (m • K), pipeline external temperature -40 ℃ ~ 500 ℃, tube temperature -40 ℃ ~ 400 ℃, thermal expansion coefficient 1.25×10^{-4}, yield Strength of 520 ~ 620MPa.

ANALYSIS OF NUMERICAL SIMULATION RESULTS

Variable pipe strain analysis

Through the finite element software ANSYS to establish and analyze the variable diameter pipe in the sand field, select the API5LX60 imported steel, large diameter D1 = 0.762m, wall thickness = 0.0238m; small diameter D2 = 0.508m, wall thickness = 0.0238m; Modulus E = 2.1e10Pa, Poisson's ratio = 0.3; pipe embedding depth H = 2m; heat transfer coefficient 21W / (m • K).Thermal expansion coefficient of 1.25 × 10-5, yield strength of 520 ~ 620MPa. Respectively, to simulate the external temperature of the diversion pipe is 20 ℃, the corresponding variable pipe outside the temperature of 70 ℃ under the stress and strain analysis.

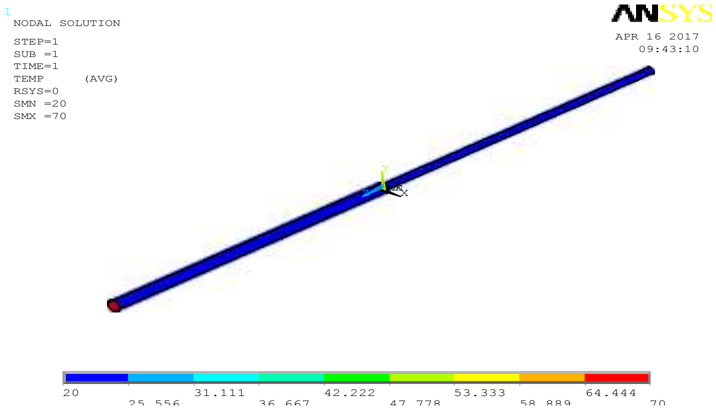

Fig-3: variable diameter pipe along the axial temperature distribution

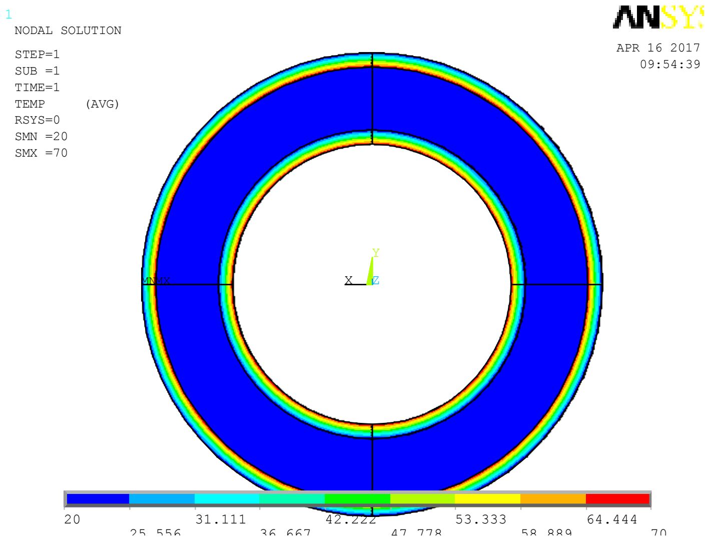

Fig-4: variable diameter pipe along the radial temperature distribution

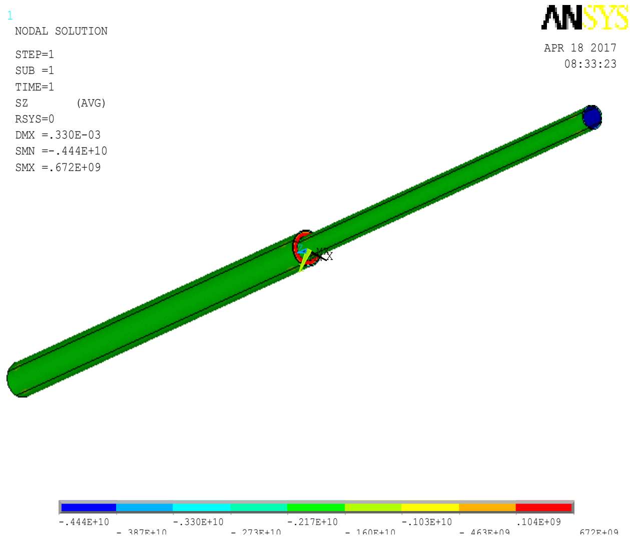

Fig-5: variable diameter pipeline axial tensile stress nephogram (pressure)

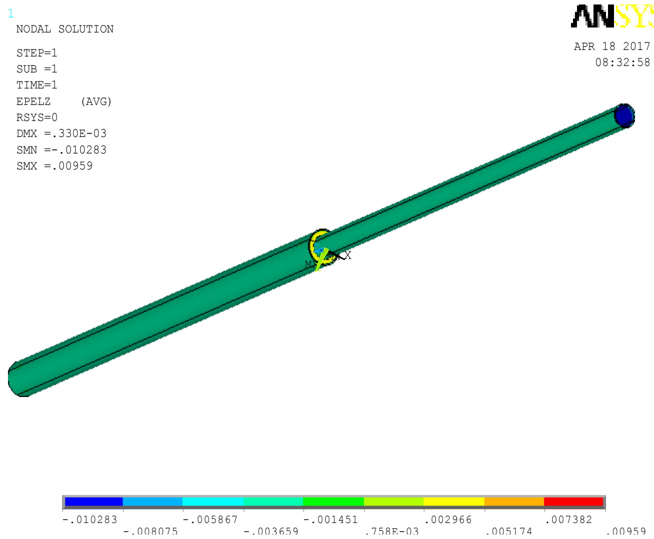

Fig-6: variable diameter pipeline axial tensile strain contours (pressure)

Through the simulation of finite element software, it is concluded that the maximum tensile stress point and the maximum tension and strain point of the variable pipe are located near the pipe diameter. The maximum tensile strain is 0.959e-2, the maximum compressive pressure is -1.0283e^{-2}, the maximum tensile stress is 0.672e9, and the maximum compressive stress is -0.444e^{10}, which occurs at the position of the variable diameter device.

The effect of temperature on the reduced pipe

Through the finite element software ANSYS to establish and analyze the sand in the ground on the variable diameter pipeline, select the API5LX60 imported steel, Large diameter D_1 = 0.762m, wall thickness = 0.0238m; diameter D_2 = 0.508m, wall thickness = 0.0238m; elastic modulus E = 2.1e^{10}Pa, Poisson's ratio = 0.3; pipe embedding depth H = 2m; Thermal coefficient of 21W / (m • K), thermal expansion coefficient of 1.2×10^{-4}, yield strength of 520 ~ 620MPa. Respectively, the simulation of the variable pipe in the temperature difference of 5 degrees, adjustable pipe outside the temperature of 20 ℃, 25 ℃, 30 ℃, the corresponding pipe internal temperature of 25 ℃, 30 ℃, 35 ℃ under the variable diameter pipe under the variable pipe Stress and strain analysis.

Through ANSYS finite element simulation study, it is found that the position of the maximum tensile pressure point is about the distance around the variable diameter device with the increase of temperature when the temperature difference is constant.

The maximum axial compressive strain of the variable pipe under the influence of temperature is shown in Fig. 7-9.

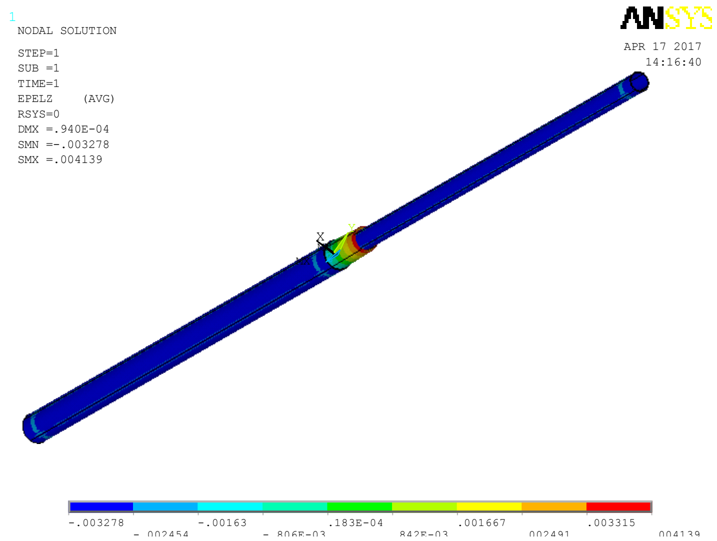

Fig-7: 20 ℃ under variable diameter pipeline axial strain contours

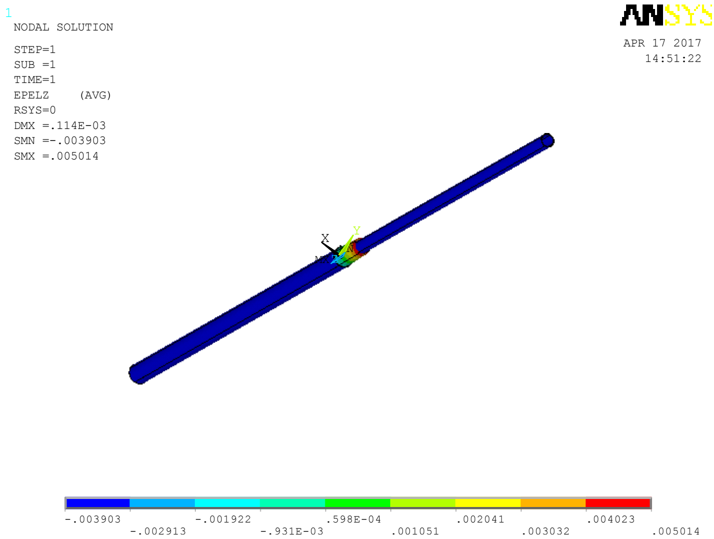

Fig-8: 25 ℃ under variable diameter pipeline axial strain contours

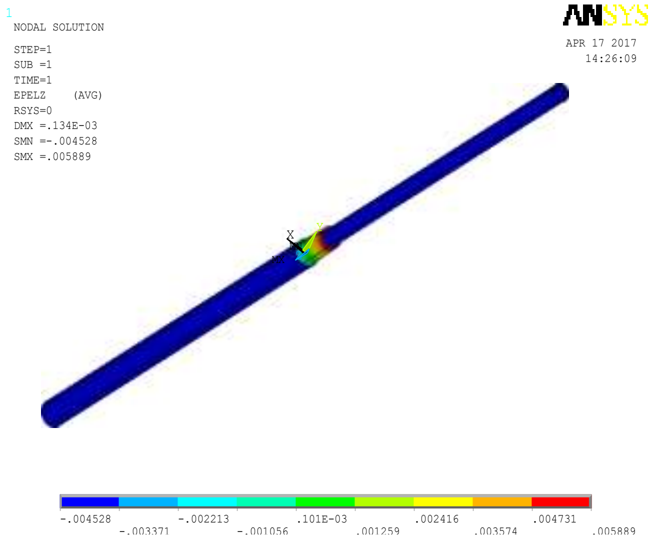

Fig-9: 30 ℃ under variable diameter pipeline axial strain contours

The maximum axial compressive strain of the variable pipe under the influence of temperature is shown in Table 1 below.

Table 1: the maximal compressive strain values under the influence of temperature

temperature	$\varepsilon_t /10^{-2}$	$\varepsilon_e /10^{-2}$
20	0.4139	-0.3278
25	0.5014	-0.3903
30	0.5889	-0.4528

According to the finite element simulation results, the internal and external temperature of the variable pipe has a great influence on the variable diameter pipe under the condition of constant temperature difference between inside and outside, and the maximum tensile pressure increases with the increase of temperature The strain value of the pipe is larger than that of the pipe strain.

The axial ultimate tensile stress under the influence of the variable pipe is shown in Figs. 10 to 12

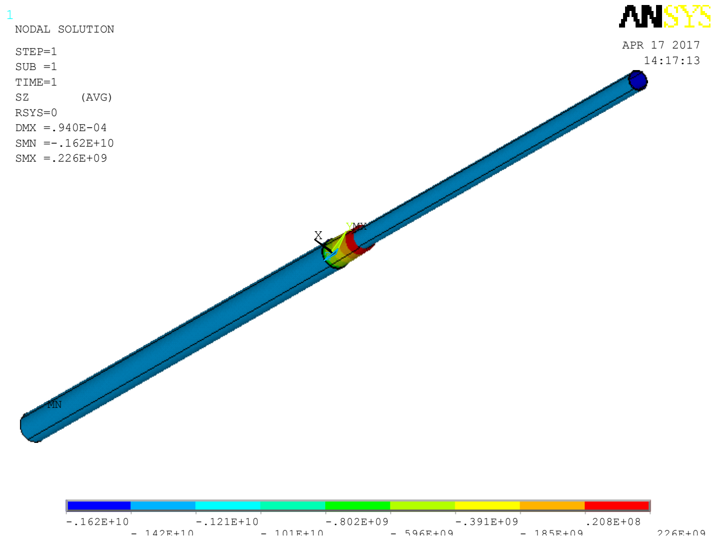

Fig-10: 20 ℃ variable diameter pipeline axial tensile stress (pressure)

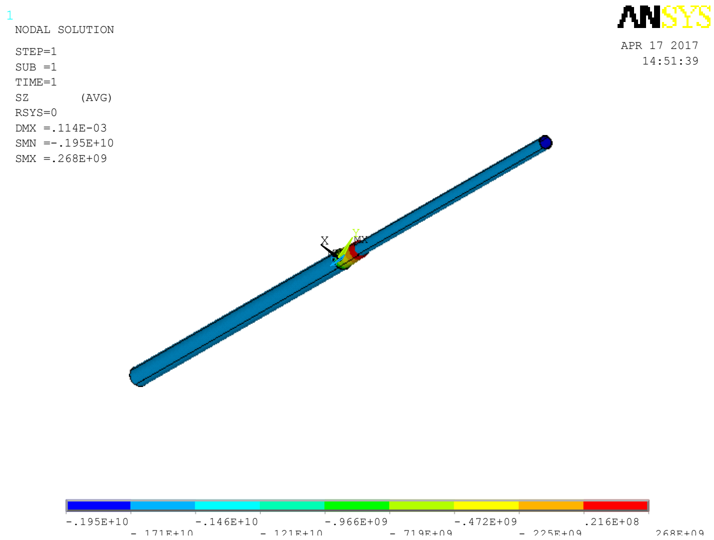

Fig-11: 25 ℃ variable diameter pipeline axial tensile stress (pressure)

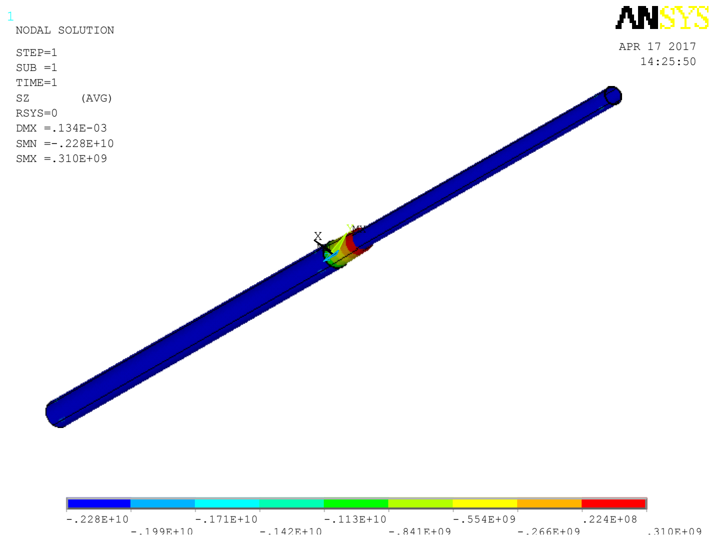

Fig-12: 30 ℃ variable diameter pipeline axial tensile stress (pressure)

The maximum axial compressive strain of the variable pipe under the influence of temperature is shown in Table 1 below.

Table 2: temperature under the influence of the maximal compressive stress value

temperature	$\delta_t \times 10^{10}$	$\delta_e \times 10^{9}$
20	-0.165	0.226
25	-0.195	0.268
30	-0.228	0.310

According to the finite element simulation results, the internal and external temperature of the variable pipe is within the same temperature difference between the inside and outside, and the compressive stress near the variable diameter device reaches the maximum value, and the temperature increases. Compressive stress increases.

Impact of Fire on Reduced Pipeline

Through the finite element software ANSYS to establish and analyze the sand in the ground on the variable diameter pipeline, select the API5LX60 imported steel, Large diameter D1 = 0.762m, wall thickness δ = 0.0238m; diameter D_2 = 0.508m, wall thickness δ = 0.0238m; elastic modulus E = 2.1e^{10}Pa, Poisson's ratio = 0.3; pipe embedding depth H = 2m; Thermal coefficient of 21W / (m • K), thermal expansion coefficient of 1.2 × 10^{-4}, yield strength of 520 ~ 620MPa.

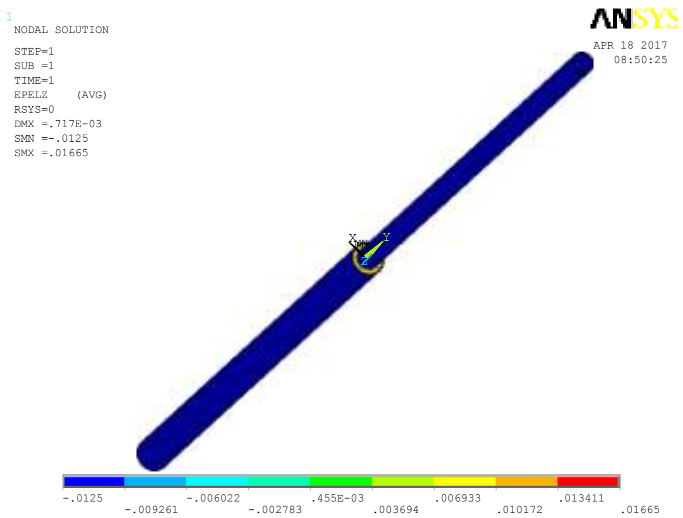

Fig-13: 100 ℃ under variable diameter pipeline axial strain contours

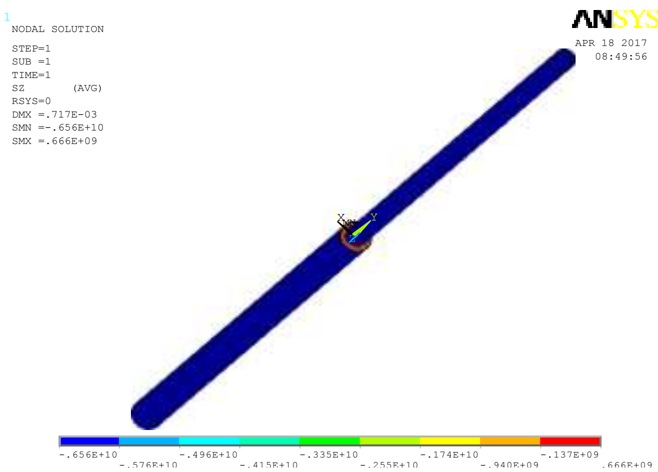

Fig-14: 100 ℃ under variable diameter pipeline axial stress nephogram

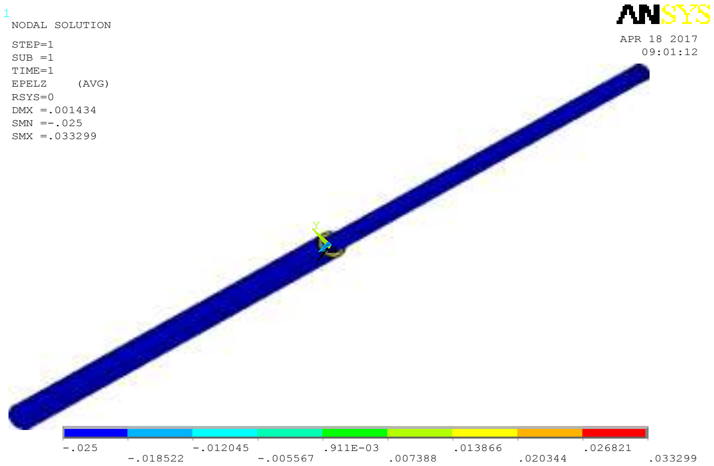

Fig-15: 200 ℃ under variable diameter pipeline axial strain contours

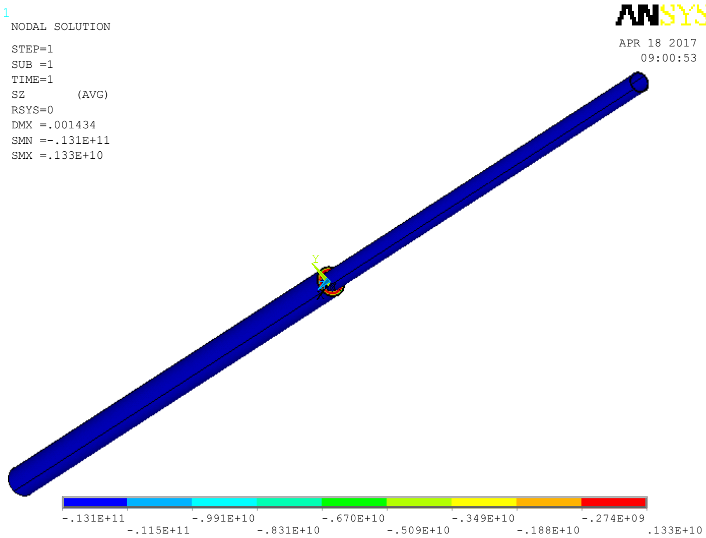

Fig-16: 200 ℃ under variable diameter pipeline axial stress nephogram

The results of finite element simulation show that the tensile strain and the compressive strain of the variable diameter pipe are different under the high temperature, but the limit value of the tension and strain is relatively large. The ultimate tensile pressure strain of the variable diameter pipe at high temperature and The ultimate tensile stress occurs at the variable diameter device.

CONCLUDING REMARKS

The results show that the weak position of the variable pipe is in the vicinity of the connection between the large diameter pipe and the small diameter pipe. The change of the temperature has a great influence on the variable pipe. When the temperature difference is constant, the strain of the variable pipe increases with the temperature increasing. The temperature difference has a certain influence on the variable pipe. The tension and strain of the variable-diameter pipe increases with the increase of the temperature difference. At high temperature, the variable-diameter pipe will produce a large stress strain at the variable diameter device. In practical engineering, it is necessary to strengthen the design of the anti-tension pressure at the variable diameter device. This paper has a guiding significance for the design of pipeline variable diameter devices in engineering.

REFERENCES

1. Yu W. Variable diameter pipe to the influence of cold and hot crude oil transportation mix oil. Journal of shandong industrial technology. 2017; (5) : 266.
2. Xue JH,, Louyanpeng. Buried variable diameter pipe across the fault seismic analysis. Learn to heilongjiang university of science and technology. 2016; (03) : 272-276.
3. Zhang SC, Jun F. directly buried hot water piping design theory and the development direction of. Journal of gas and thermal power. 2015; (9) : 20 to 22.
4. Huang K, Jinchuan G, Cherly. Variable diameter pipe pressure regulating effect test. Journal of guangxi water conservancy and hydropower. 2014;

(02) : 6-8.
5. Huiquan, Jian M, Wu ZP, Zhaoxu Z, Wild ZD. The west-east gas pipeline integrity management practices and performance analysis. Journal of natural gas industry. 2013; (12) : 124-130.
6. Wu ZP, Jiang HY, Chang LC. Oil and gas pipeline integrity management efficiency evaluation technology research. Journal of natural gas industry. 2013; (12) : 131-137.
7. Xu ZY, Han Y. Variable diameter long distance gas transmission pipeline pig research progress. Journal of pipeline technology and equipment. 2013; (6) : 45-48.
8. Li MH, Wei SLZ, Jianfeng G. Internal and external pressure heat stress under the influence of the deformation of the second line of west-east gas transmission pipeline of limited element analysis. Journal of natural gas industry. 2013; (8) : 119-124.
9. Tao LPYL, Jian Z. Based on the strain design of pipeline buckling strain limit calculation. Journal of natural gas industry. 2013; (7) : 101-107.
10. Luqinli, Du MJ, Ma GY, Wang HF, Deming C. Variable diameter pipe in cold and hot crude oil transportation mix oil. The influence of oil and gas storage and transportation. 2011; (3) : 183-186 + 4.
11. Shaopeng L. It Stretched, Han Haicang Sun Wei. ANSYS software application in the chemical pipeline heat preservation. Journal of liaoning province Workers. 2010; (4) : 2010-440.
12. Yong L, Xiuping Z. Variable diameter pipe in pneumatic conveying system design research. Journal of lifting the transport machinery. 2008; (10) : 25 to 27.
13. Xue JH. Isolation pipelines across the fault analysis. Institute of engineering mechanics. China seismological bureau. 2008.
14. Meng ZH. Hot crude oil pipeline operation process analysis. Journal of pipeline technology and equipment. 2005;(6) : 5-8.
15. Meng ZH, Yizhong C. Crude oil pipeline in the process of thermal analysis. Jiangsu polytechnic institute study. 2005; (3) : 19-23.
16. Shirakawa. Pipeline thermal stress analysis examples. Journal of chemical industry in Guangzhou. 2005; (02) : 49-51.