DOI: 10.21276/sjet

Scholars Journal of Engineering and Technology (SJET)
Sch. J. Eng. Tech., 2017; 5(5):191-196
ISSN 2321-435X (Online)
ISSN 2347-9523 (Print)

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

Research Article

Based on Finite Element Simulation and Affected Analysis of Pipeline Thermal Stress under Wall Thickness

LV Chao*, ZU HongYu
Northeast university of petroleum, School of Civil Engineering and Architecture, Daqing Heilongjiang 163318, China

***Corresponding author**
LV Chao
Email: 963270758@qq.com

Abstract: When gas transmission pipelines are working, due to various reasons such as being heated or cold and expansion and contraction, the pipeline will produce deformation, and then the deformation is limited by displacement constraints or the opposite pressure. As a result, the pipeline will produce thermal stress, thus posing a threat to pipeline working. Therefore, this article will adopt the method of model simulation and establish a channel space finite element model, then the ANSYS finite element analysis software will be applied to the calculation of thermal stress of the pipelines. Finally, we will do some research and analysis of the factors affecting the thermal stress in pipeline. The results show that the temperature had a greater influence on the thermal stress of pipeline, when the larger internal and external temperature difference of the pipeline get, the greater thermal stress the pipes produce. Secondly, the wall thickness of pipelines also affects thermal stress. Under the condition of the constant pipe diameter, thermal stress produced by pipes becomes small with the increase of pipe wall thickness.
Keywords: pipeline finite element temperature wall thickness thermal stress.

INTRODUCTION

In recent years, the global energy consumption is increasing, especially for some of the clean energy needs such as natural gas gradually increased, the world's long-distance gas pipeline project is also increasing year by year. By the end of 2012, has reached 55 thousands Km, China's natural gas trunk pipeline transport in 2014, China and Russia signed a gas purchase and sale agreement [1, 2] billion, the future China will build a comprehensive natural gas pipeline network, so the gas pipeline stress changes and distribution of research is very necessary. Scholars at home and abroad have done a lot of research on the damage of pipeline leakage, built the gas diffusion model for the jet generated by the leakage, and analyzed the influence factors of leakage diffusion [3, 4]. There is also a scholars carried out relevant research on pipeline temperature field [5], the heat pipe is less stress influence factors were analyzed, in this paper, the thermal structure coupling direction stress and influencing factors of the size distribution of the research and analysis of the heat pipe, hoping to provide help for future design and construction of pipeline Study.

FINITE ELEMENT ANALYSIS THEORY

ANSYS thermal analysis finite element theory

There are three basic ways of heat transfer in thermodynamics: conduction, convection and radiation. Depending on the thermal motion of molecules and atoms and free electrons, the high temperature transmits from high temperature part of objects or objects to the low temperature part of objects or objects, which is producing heat transfer, called heat conduction.

Thermal analysis follows the first law of thermodynamics, the law of conservation of energy:

$$Q-W=\Delta U+\Delta KE+\Delta PE \qquad (1)$$

In the formula, Q :the quantity of working of system; W : heat for the system; ΔU :the internal energy of the system; ΔKE :the kinetic energy of the system; ΔPE :the potential energy of the system.

Heat transfer is mainly based on Fu Liye's law

$$Q=-\lambda grad\phi=-\lambda\vec{n}\frac{\partial\varphi}{\partial n} \qquad (2)$$

Among them, Q: the heat flux. Thermal conductivity:

$$\lambda = -\frac{\vec{q}}{\vec{n}\frac{\partial \phi}{\partial n}} \quad (3)$$

For steady-state thermal analysis, the influence of time on temperature is not considered

$$\frac{\partial}{\partial x}\left(\lambda_x \frac{\partial T}{\partial t}\right) + \frac{\partial}{\partial y}\left(\lambda_y \frac{\partial T}{\partial t}\right) + \frac{\partial}{\partial z}\left(\lambda_z \frac{\partial T}{\partial t}\right) + \rho q = 0 \quad (4)$$

To solve the problem of heat conduction, the essence is solution to the differential equation of heat conduction. In order to obtain the temperature distribution of a specific heat conduction problem, some additional conditions must be given to characterize the particular problem. For steady state heat conduction problem, there is no initial condition and only boundary condition.

The boundary condition is the law of heat exchange between the surface of the object and the surrounding medium, which can be divided into three types, i.e., the temperature boundary condition, the heat flux boundary condition and the convection boundary condition.

The first kind of boundary condition gives the temperature distribution at any time on the boundary of the object;

The second kind of boundary condition gives the heat flux at any time on the boundary of the object;

The third kind of boundary condition gives the convective heat transfer coefficient and the temperature of the surrounding fluid.

ANSYS thermal structure coupling analysis

Thermal structure coupled finite element equation

By the generalized Hooke's law:

$$\begin{cases} \varepsilon_{11} = \frac{1}{E}\left[\sigma_{11} - \upsilon(\sigma_{22} + \sigma_{33})\right] \\ \varepsilon_{22} = \frac{1}{E}\left[\sigma_{22} - \upsilon(\sigma_{11} + \sigma_{33})\right] \\ \varepsilon_{33} = \frac{1}{E}\left[\sigma_{33} - \upsilon(\sigma_{11} + \sigma_{22})\right] \\ \varepsilon_{23} = \frac{1}{2G}\sigma_{23} \\ \varepsilon_{31} = \frac{1}{2G}\sigma_{31} \\ \varepsilon_{12} = \frac{1}{2G}\sigma_{12} \end{cases} \quad (5)$$

Temperature induced strain:

$$\varepsilon^e = B^e U^e = \nabla N_u^e U^e \quad (6)$$

Among those $B^e = \nabla N_u^e$, ∇ :differential operator.

The element stiffness matrix K^e,unit node load P^e,and element node heat load P_T^e are calculated with the nodal temperature obtained from the thermal analysis. Based on the assumed displacement model, the global stiffness matrix K and load vector P are derived by using the equilibrium condition and the variation principle:

$$\begin{cases} K = \sum K^e \\ P = \sum P^e + \sum P_T^e \end{cases} \quad (7)$$

In the formula, $P_T^e = \int_{\Omega_0} B^e D^e \varepsilon_0 d\Omega$ B^e :stiffness matrix; D^e :geometric matrix.

Thermal structure coupling method

The coupling field analysis is the process of finite element analysis considering the interaction and the interaction of two or more physical fields. The analysis of thermal structure coupling field can be summed up in two ways: indirect method and direct method. The coupling field can be obtained by solving only one solution directly, and it is suitable for the case of multiple physical fields. Direct coupled field analysis is the huge amount of computation cost of the machine, indirect coupling field analysis is related to two or more times in accordance with the order, the first field analysis as a result of the second field analysis of the load to achieve coupling two midfield, analysis the indirect coupling method can obtain multiple independent and the ratio of direct coupled field, send a small amount of calculation, high efficiency.

The research content of this paper is in accordance with the thermal structure coupling field, and the indirect coupling method and advanced nonlinear pipeline thermal analysis are used to analyze the static structure in order to achieve the purpose of final analysis.

FINITE ELEMENT MODEL ANALYSIS

For the stress analysis of the influencing factors of the heat pipe, a three-dimensional pipeline model model in the space for the Z axis symmetry axis cylinder in the radial direction as y axis symmetry axis of the ring, to establish 3D model using thermal analysis unit SOLID70, first on the thermal analysis model, and then converted to SOLID185 structure unit structure analysis on the structural model. Reference [6-

8], specific parameters: material of pipeline steel X80, diameter 1219mm, thickness 16mm~30mm, the elastic modulus of 2.06×10^5MPa, Poisson's ratio of 0.3, the heat transfer coefficient of 14.7W/ (M - K), the temperature in the tube is 27 ℃ at 10 ℃ for 50 ℃, the external temperature of the pipeline 10 ℃to 50 ℃, the coefficient of thermal expansion 1.2x10-5, yield strength of 520~620MPa. In this paper, the displacement of X direction in the Y axis direction of the pipe is restrained, and the displacement of the Y direction in the X axis direction of the pipe is restrained. Mesh model is using hexahedral element mesh, mapping method, the radial grid length 0.0025mm, axial circle grid row length 5rad, torus and mesh number is 2, the model is divided into 28800 units, the final mesh structure results in Figure 1, shown in figure 2.

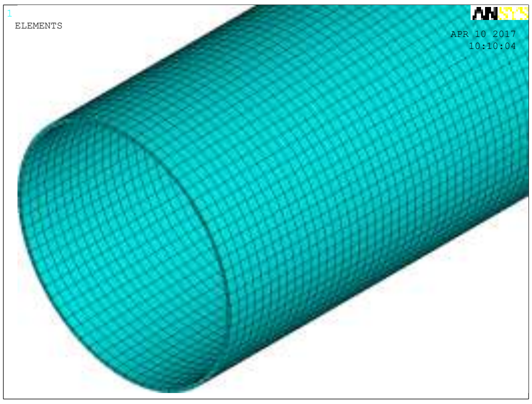

Fig-1: Local mesh partition

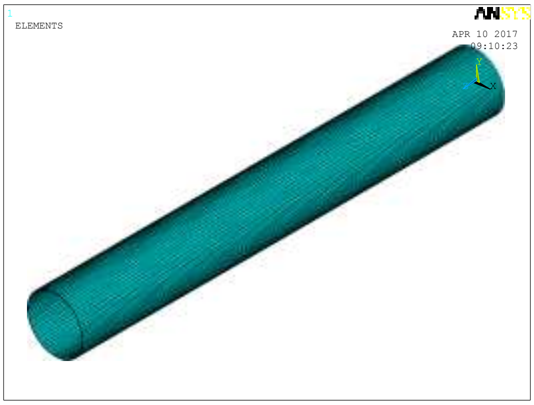

Fig-2: Global mesh partition

According to the theoretical analysis of the literature [9-11], it is known that the internal pressure is 8~12 MPa of the gas pressure, but the distribution of the internal pressure in the pipeline is usually distributed. External pressure including soil pressure and weight pressure, most of the buried pipeline depth is 3~5 m, most of them are sand, calculation of pipeline steel and covered by soil density external pressure is less than 1 MPa, even in the outer wall of the pipe are non uniformly distributed, but far less than the yield strength of pipeline 520~620MPa, ignoring the effect of pressure on the outer wall of pipeline can be caused in the process of gas transmission pipeline is. Therefore, it is considered that the main factors that affect the size and distribution of thermal stress are the thickness of the pipe wall thickness and the temperature.

According to the above analysis, for the analysis of temperature effects on the pipe size and the effect of thermal stress distribution, the pipe wall thickness 20mm, establish the finite element model, pipe length 10m, setting the tube temperature is 27℃, as changes in variables outside the tube temperature, the pipeline temperature gradually increased from 10℃ to 50℃.Finally, simulation analysis, the maximum thermal stress results as shown in figure 3.

From the analysis of the calculation results, when the pipeline temperature than the pipe outside with the increase of temperature of maximal thermal stress decreases, the maximum thermal stress appears in the outer wall of the pipes; when the pipeline temperature is less than the pipeline outside the pipeline with the increase of temperature the maximum thermal stress increases gradually, the maximum thermal stress values appear in the pipeline; when the temperature difference between inside and outside the pipeline, the pipeline thermal stress generated by the greater the value, indicating that the effect of temperature difference on the pipeline thermal stress generated by the more obvious, the main factors of the stress change of the heat pipe. As shown in Figure 5, Figure 6. 20℃ and 40℃ Celsius pipeline thermal stress distribution nephogram is as follows.

In addition, in the case of 16mm and 30mm, the calculated results of thermal stress of pipeline under different temperature values are shown in figure 4. At the same time, the minimum value of thermal stress occurs at 30℃, which is due to the minimum temperature difference between the inside and outside of the pipe, and the temperature difference will affect the thermal stress of the pipeline.

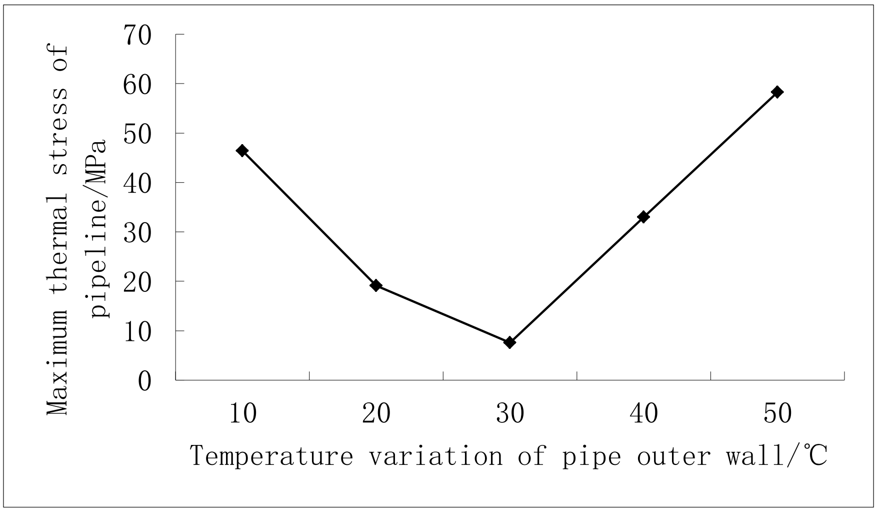

Fig-3: Calculation results of thermal stress of pipeline under different temperature

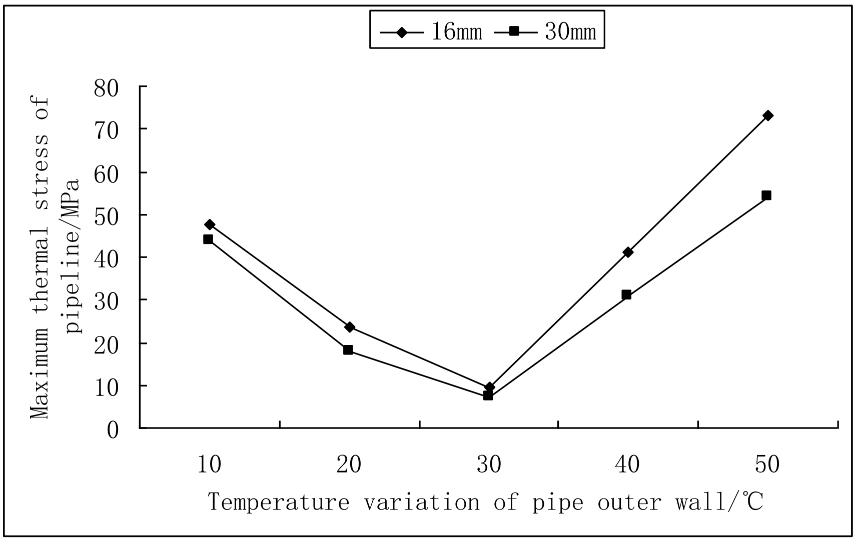

Fig-4: Calculation results of thermal stress of pipeline under different temperature in 16mm and 30mm

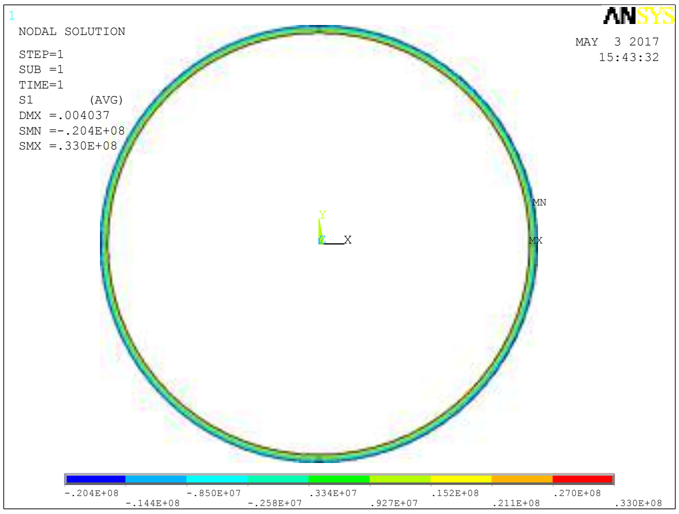

Fig-5: The thermal stress distribution of the pipe with 20℃, wall thickness of 20mm

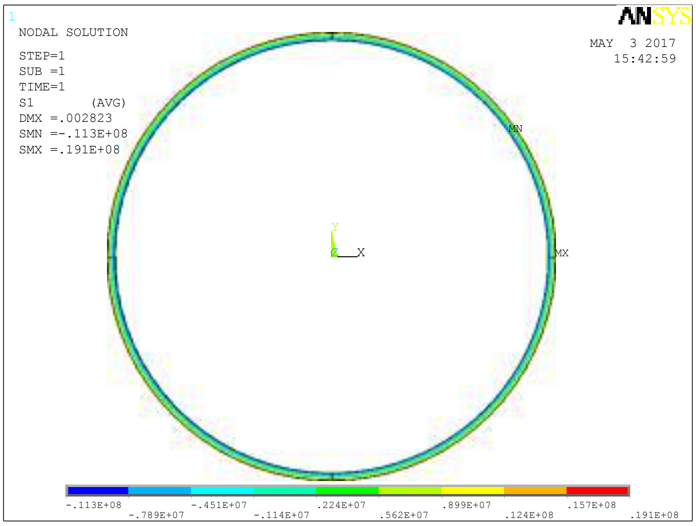

Fig-6: The thermal stress distribution of the pipe with 40℃, wall thickness of 20mm

For the analysis of heat pipe wall thickness under different stress distribution, different pipe wall thickness, different wall thickness of 16mm, 20mm, 30mm, finite element model is established, 10m pipeline tube, set the temperature at 27℃, as the variable changing tube temperature, the pipeline temperature by 10℃ gradually increased to 50℃, fixed constraints on both ends of the pipeline, the pipeline maximum thermal stress are shown in Figure 6.

From the analysis of the calculation results, the comparison results of three lines in the figure can be found with the increase of the pipeline wall thickness of pipeline, the maximum thermal stress value decreases, but decreased slightly, and with the pipe wall thickness reached 20mm after the influence of wall thickness of pipeline thermal stress decreases, even wall thickness from 20mm increase to 30mm, increase the pipeline wall thickness of 10mm is not 16mm or 20mm to increase the 4mm pipeline stress effect on heat; as shown in table 1, the following is a group of fixed end restraint data table also obtained the above conclusion. The results show that the temperature effect is still the main factor of the change of the thermal stress of the pipeline, no matter how the supporting conditions change.

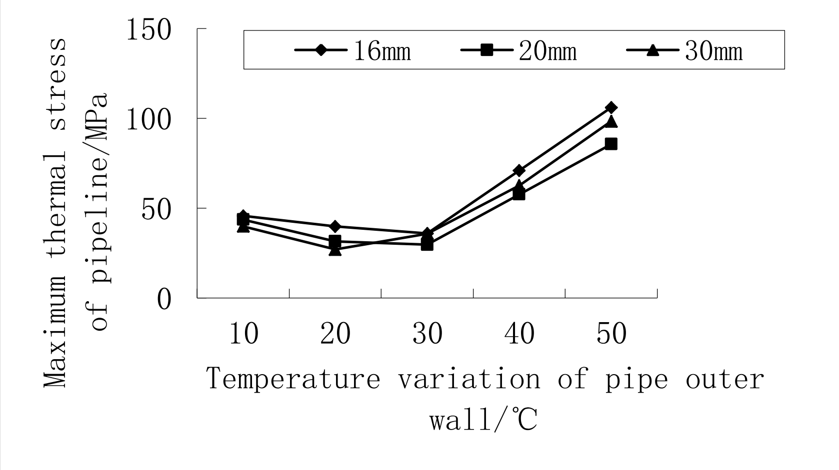

Fig-7: Thermal stress calculation results of different wall thickness

Table 1: The maximum thermal stress on different wall thickness pipelines with fixed constraints on two ends

temperature/°C	(16mm)stress/MPa	(20mm)stress/MPa	(30mm)stress/MPa
10	62.7	55.5	46.9
20	50.3	36.8	44.4
30	86.5	71	78.5
40	130	106	113
50	174	141	147

CONCLUSIONS

1) The temperature change affects the size and distribution of stress were heat, the temperature difference between inside and outside the pipeline, pipeline thermal stress more greatly; and when the temperature in the tube is greater than the tube temperature, the maximum thermal stress distribution on the outside of the pipeline, when the temperature in the tube is smaller than that of the tube when the maximum thermal stress the internal stress distribution in pipeline.

2) Different wall thickness also influence the stress of pipeline heat, with the increase of the pipeline wall thickness, pipe thermal stress decrease, but the change is not obvious, and the wall thickness is more than increase significantly smaller thermal stress after 20mm, so that the pipe wall thickness from 18 to 20mm is reasonable.

REFERENCES

1. Baoqun W, Zhongliang I, Yanhong L. Status and development direction of China's natural gas pipelines. International Petroleum Economics. 2013; 08:76-79+109-110.
2. Quezhi Z, Peixia D, Hongju W, Qiuyang L, Chuanxi Z, Ning S, Yanping Z. Current situations and future development of oil and gas pipelines in the world . Oil & Gas Storage and Transportation. 2015; 12:1262-1266.
3. Jianlan X, Baohe L, Mingxian W, Jiabei Z. Study Progress on Leakage Model for Gas Pipeline. Gas & Heat. 2006; 02:7-9.
4. Qingmin H. Research on Natural Gas Leakage and Diffusion Numerical Simulation. Harbin Institute of Technology, 2009.
5. Tong Z, Lei Y, Jiazheng W, Yongkang S, Kemin Z. Analysis and Calculation of Gas Pipeline Temperature Field. Gas & Heat. 2004; 24(1):5-8.
6. Maohua L, Leibin S, Wei Z, Jianfeng G. The finite element analysis and calculation of pipeline stress and strain under the effect of heat stress and internal or external pressure in the West-East China Gas Pipeline. Natural Gas Industry. 2013; 08:119-124
7. Jinghong X. Analysis for Seismic Isolation Pipelines Across Fault. Institute of Engineering Mechanics, China Earthquake Administration, 2008.
8. Xincheng Z. Research on Influence on Adjacent Gas/Oil Pipeline by Fire Resulted from Natural Gas Pipeline Failure.Harbin Institute of Technology, 2013.
9. Ru T. Application of ANSYS finite element software analysis in the design of large diameter and thin-walled pipes. Ningxia Electric Power. 2015; 04:54-56+68
10. Jian L. Criteria for piping stress analysis and flexibility design. Petroleum Refinery Engineering. 2010; 01:61-64.
11. Lisong Z, Xiangzheng Y, Xiujuan Y. Analysis of Pressure Change in Oil Product Pipeline during Shut-down Period, OGST Oil & Gas Storage and Transportation. 2009; 12:25-27.