Research on Pipe Vibration Analysis and Optimal Supporter Arrangement

***J.J. Liu[1,2,3], Ding Yu[1,2,3], and †Long Yu[1,2,3]**

[1]State Key Laboratory of Ocean Engineering, Shanghai Jiaotong University, Shanghai 200240, China;
[2]Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration(CISSE), Shanghai 200240, P.R. China;
[3]School of Naval Architecture, Ocean & Civil Engineering, Shanghai Jiaotong University, Shanghai 200240, China;
*Presenting author: 15855702880@163.com
†Corresponding author: yulone@sjtu.edu.cn

Abstract

The discharge pipe line is one of the important devices for cutter suction dredgers, while the pipes connecting to cutter, pump and valves guarantee the high dredging and transportation efficiency. The flow mixed with water and soil with different density is highly turbulent and unsteady, which induces vibrations along the pipe and also cause abrasion inside the pipe especially at bends and T branches. The flow exciting force induce the pipe stress and deformation while the pipe deformation will influence the flow pattern. Due to the large diameter of discharge pipe and lots of bends and T branches, the Fluid Structure Interaction (FSI) should be introduced for pipe dynamic analysis and supporter design. The uncertainty of the mixed flow makes it not possible to design or test all the working conditions. Therefore, the CFD technique has been employed for numerical analysis. The typical pipe lines and their supporters with specified exciting source are investigated. Modal analysis and stress results give proof for optimal supporter design and pipe line.

Keywords: discharge pipe line, FSI, modal analysis, CFD

1.Introduction

The pipe line is a convenient way of transport. With the rapid development of economic, discharge pipe line has been used in more and more fields, such as biomedical, aerospace engineering, automobile manufacturing, civil engineering, electronics industry, and especially in the naval architecture and marine engineering(Levitan et al., 1991). The discharge pipe line plays a very important role in the ships, which are always concerned as large and complex systems. For example, the hydraulic pipe systems have great effects on the starting, reversing, transmission and other action of the ships, which can reduce the pressure on the ship power station(Dai Xueliang et al.,2000). In addition to the hydraulic pipes, ships also contain a variety of discharge pipe lines, such as the pipe lines of cold water, hot water, sewage, black water and oil. They are all hidden in the corner to maintain the ship's normal operation.

There are very complex pipe line systems in the cabin of the large ships. In the process of voyage, the various pipe lines take the task to discharge the oil, water, gas and other fluids of the whole ship. Working in a high temperature and high pressure environment, once the pipe line systems are damaged, they will pose a serious threat to the security of the ships. For example, if the pipe lines supplying fuel to the main engine leaks, it will not only cause fire and explosion, but also make the main engine break down. In some serious cases, ships may extremely lose power, which will even lead to serious shipwrecks(Liang Chunyu,2013). Therefore, reasonable and effective measures must be taken to reduce the pipe vibration
To analyze vibration and stability of the pipe lines, people mainly study the influence of the flow velocity on the dynamic characteristics of the pipeline system. Ignoring the fluid compressibility, people usually pay attention to the interaction between the flow and the

pipeline. At the end of the nineteenth century, people began to study the problem of pipeline vibration, but the study was once paused for a long time. From the fifties of the twentieth century, people began to systematically study the vibration and stability of the pipeline, and since then, the articles around this field is endless(Liu Zhongqun,2001). In the field of vibration and stability of the pipeline, Chen and Paidoussis (1984) have done a representative research work. When the flow is non-stick, incompressible and steady, if neglecting the gravity, structural damping and external force on pipe, equations of the bending and free vibration of the straight pipe line is given by:

$$EI\frac{\partial^4 u_y}{\partial z^4} + MW^2\frac{\partial^2 u_y}{\partial z^2} + 2MW\frac{\partial^2 u_y}{\partial z \partial t} + (M+m)\frac{\partial^2 u_y}{\partial t^2} = 0 \tag{1}$$

In which EI is the bending stiffness of the pipe, M is the linear density of the fluid, m is the line density of the pipe, W is the average flow rate of the fluid, u_y is the displacement of the transverse vibration of the pipe, z is the axial coordinate of the pipe, and t is the time variable.

Paidoussis and Issid(1980) proposed a more general equation based on the above equation, which takes into account the axial load of tension and compression of the pipe, the gravity and the damping of the pipe. The form of the equation is given by:

$$\begin{aligned} &E^*I\frac{\partial^5 u_y}{\partial z^4 \partial t} + EI\frac{\partial^4 u_y}{\partial z^4} + \{MW^2 - T + PA(1-2\mu\delta) - [(M+m)g - M\frac{\partial W}{\partial t}] \\ &-(L-z)\}\frac{\partial^2 u_y}{\partial z^2} + 2MW\frac{\partial^2 u_y}{\partial z \partial t} + (M+m)g\frac{\partial u_y}{\partial z} + C\frac{\partial u_y}{\partial t} + (M+m)\frac{\partial^2 u_y}{\partial t^2} = 0 \end{aligned} \tag{2}$$

In which E^*I is the internal resistance coefficient of the material, C is the viscous damping coefficient of the support, δ (0 or 1) is the factor that indicates whether the end of the pipe can be moved, μ is the Poisson ratio, P is the average pressure in the pipe, T is the axially load on the end of the pipe.

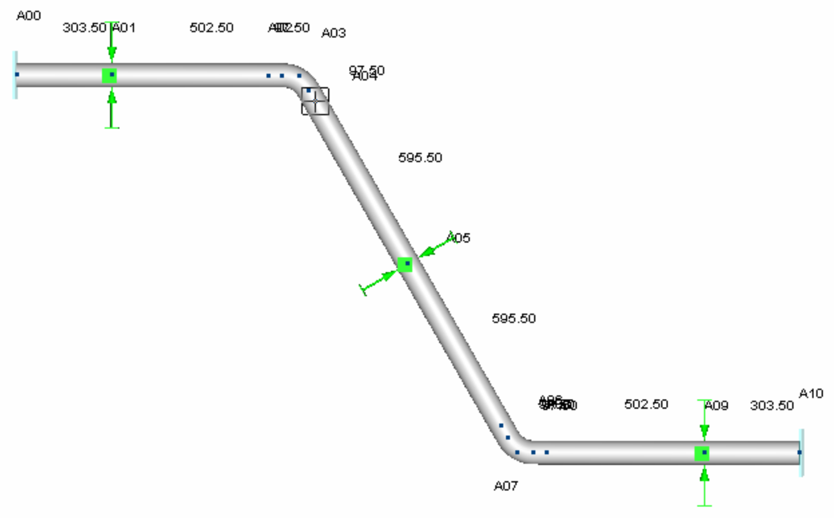
Paidoussis(1987) gave an overview of the vibration of the discharge pipeline system, and introduced the bifurcation behavior of the discharge pipeline system and showed many obtained research results. He stated that there were two kinds of instabilities: divergence instability and flutter instability. The kinds of instabilities that firstly appeared depended on the supporting situation. His review was mainly for the linear problem. And over the past decades, people had much promising work done in the nonlinear vibration of discharge pipeline system, finding new phenomenon never found in the linear range.

In this paper, the finite element analysis method is used to study the static structure and natural vibration characteristics of the pipeline. Models of pipeline are established both in Autopipe and Ansys for numerical simulation. After calculation by the softwares, the stress, strain and deformation of the pipeline can be obtained and compared. And in the dynamic simulation, the vibration response of the pipeline can also be calculated. Finally, various damping measures and the most optimal supporter arrangement can be taken to ensure the safety of the discharge pipeline according to the simulation results

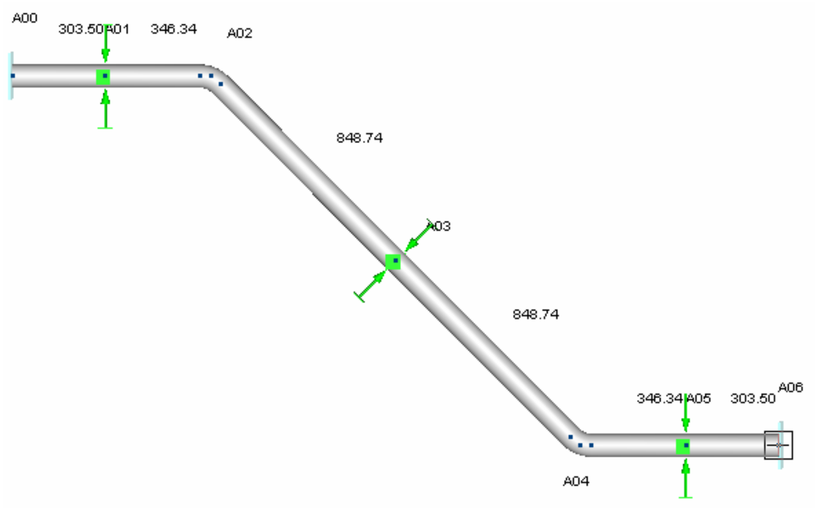
2.Analysis by Autopipe

2.1 Model of pipeline

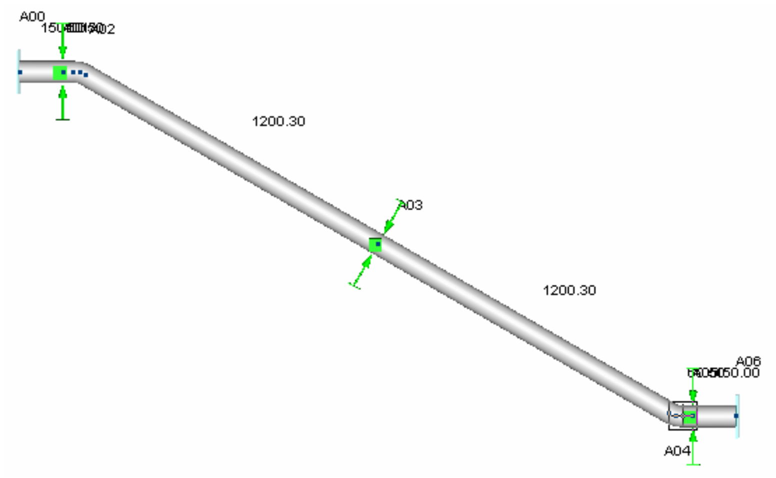
The establishment of pipeline model in Autopipe consists of three steps: defining pipe properties, connecting pipes and adding pumps, valves and supports. The outside diameter of the pipeline is 76mm, the wall thickness is 2.5mm, and the density is 7850 kg/m^3, Young's modulus is 200 GPa, Poisson's ratio is 0.3.Both ends of the pipeline are pumps, so both ends are set to fixed. Guide brackets are installed in the support position and all the gaps of the guide brackets are set to 0 to ensure a better support. Different models are established with the angle of the corner A= 120°, 135°, 150°, and in each degree the distance between two brackets also varies from L=1600mm, 1900mm to L=2200mm. (Difference L means difference distance between the fixed ends and the brackets).Some of the models is shown in Figure 1.



（a）A=120°, L=1600mm



（b）A=135°, L=1600mm

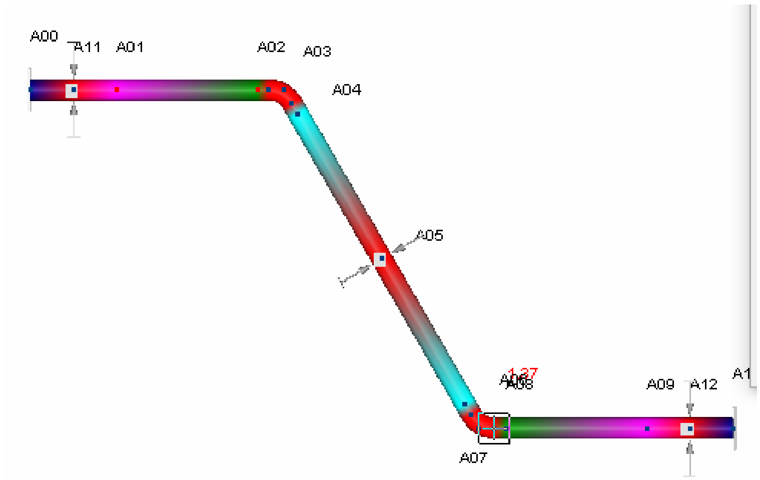


（c）A=150°, L=1900mm

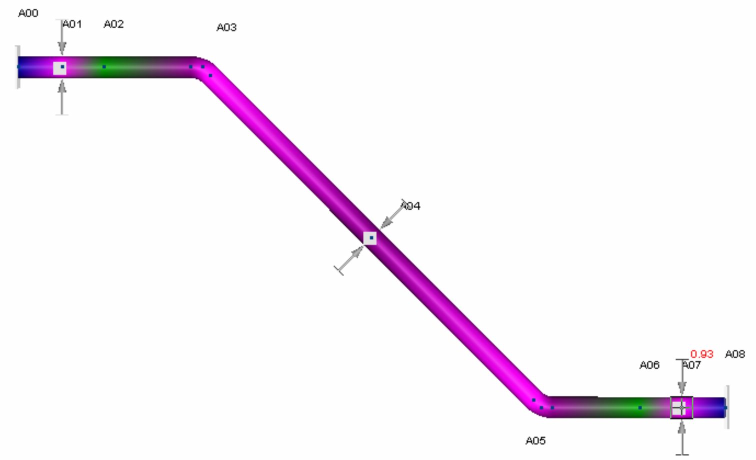
Figure 1. Models established by Autopipe

2.2 Static Analysis

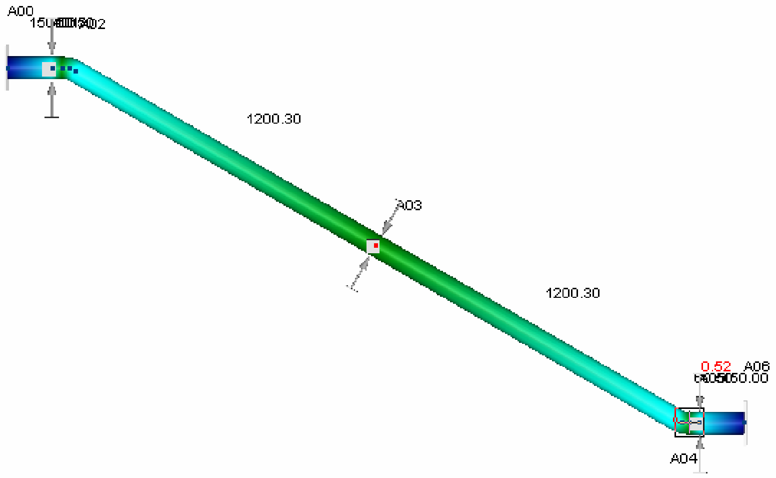
After simulation and calculation in Autopipe, the static stress of the pipeline will be obtained. Figure 2 are the static stress of some models.



（a）A=120°, L=1600mm



（b）A=135°, L=1600mm

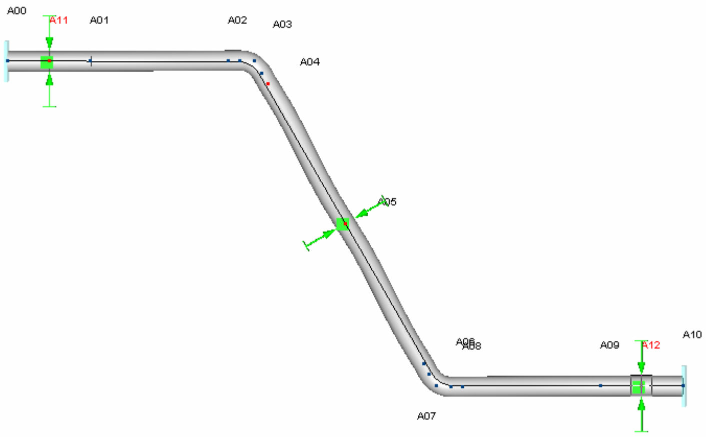


（c）A=150°, L=1900mm

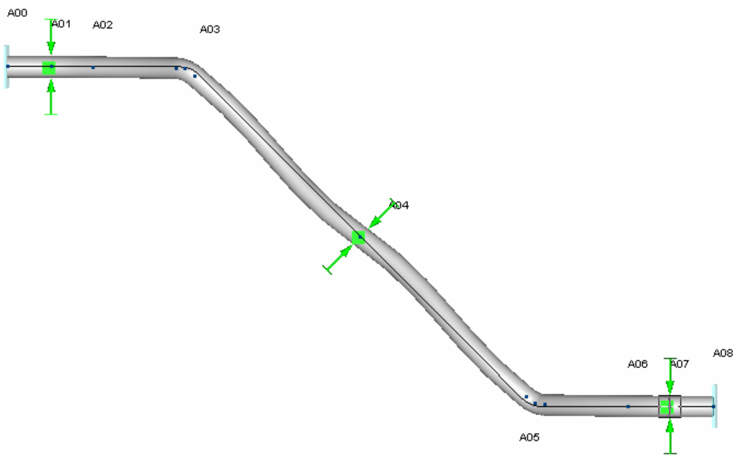
Figure 2. Static stress

2.3 Modal analysis

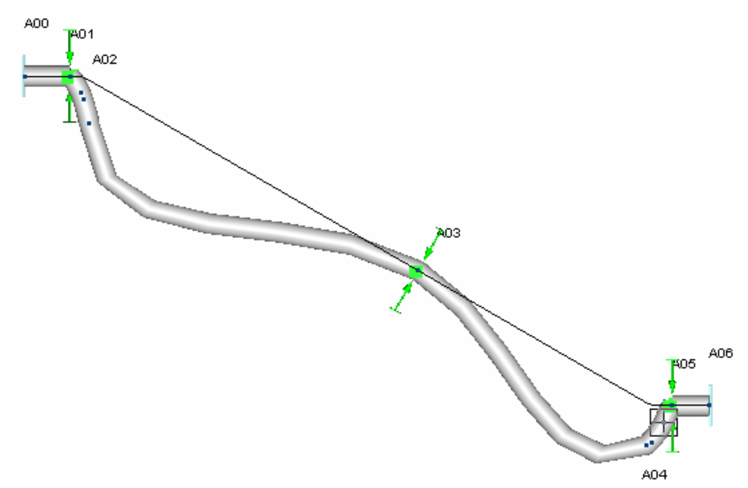
The modal of vibration of some modals is shown in Figure 3.



（a）A=120°, L=1600mm



（b）A=135°, L=1600mm



（c）A=150°, L=1900mm

Figure 3. Modal of vibration

2.4 brief summary

From the static results, it can be concluded as follows.

(1)There is only sustained stress rather than swelling stress in all different cases.

(2)In the case of A=120 degrees and 135 degrees.

a. The max stress appears at the brackets.

b. The closer the brackets to the corner, the smaller the sustained stress and the sustained stress ratio.

(3)In the case of A=150 degrees

a. The max stress appears at the both fixed end when no brackets are set.

b. When brackets are 50mm or 100mm to the fixed end, the max stress appears at the midpoint. And the closer the brackets to the corner, the smaller the sustained stress and the sustained stress ratio.

c. When brackets are 150mm to the fixed end, the max stress appears at the corner.

From the dynamic results, it can be concluded as follows.

(1)As the angle increases, the modal frequency also increases;

(2)The first-order modal frequency are far greater than 50Hz;

(3)If concerning about the sustained stress only, A=120 degrees is better than others.

3.Analysis by Ansys

3.1 model of pipeline

The model built in Ansys is similar to the model in Autopipe. The outside diameter of the pipeline is 76mm, the wall thickness is 2.5mm, and both ends of the pipeline are set to fixed. The angle of the corner is initially set as 120 degrees, and the length of the three straight pipeline is 0.806m, 1.191m and 0.806m. Considering of meshing, the radius of the corner can not be too small, so it’s set to 0.0931m. And after meshing, the total number of the elements are 4448 and the nodes are 8960.

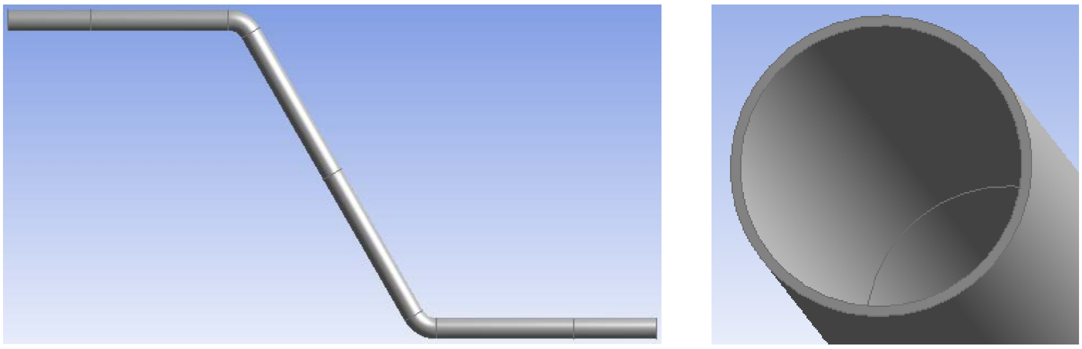


Figure 4. Model of pipeline in Ansys

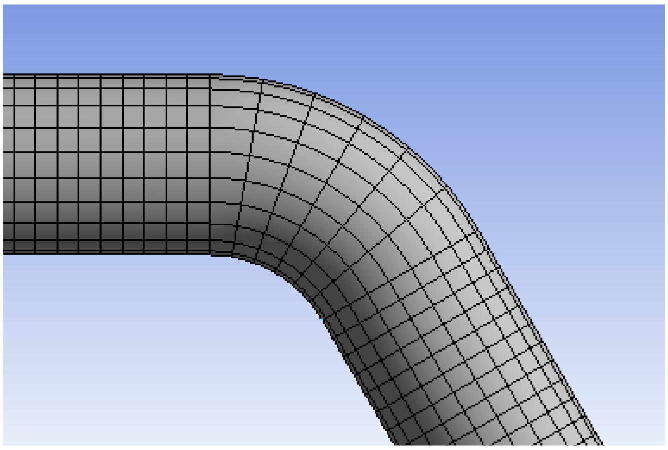
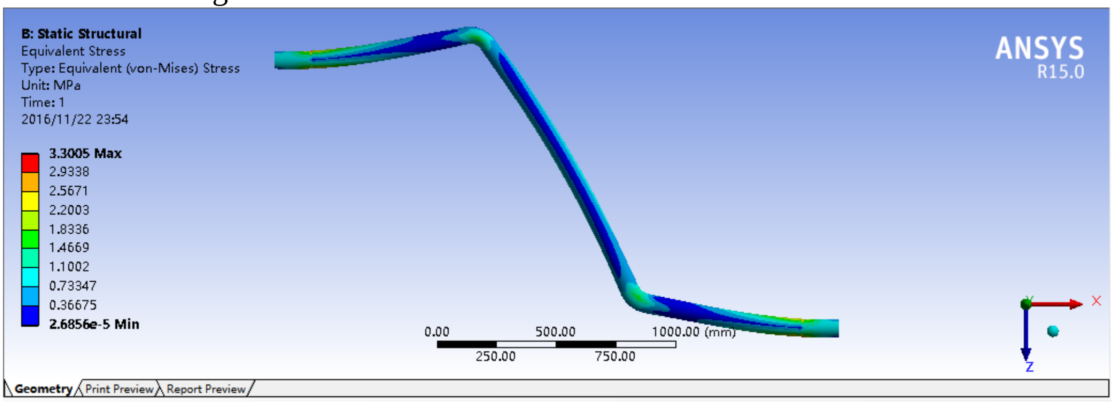


Figure 5 Mesh of the corner

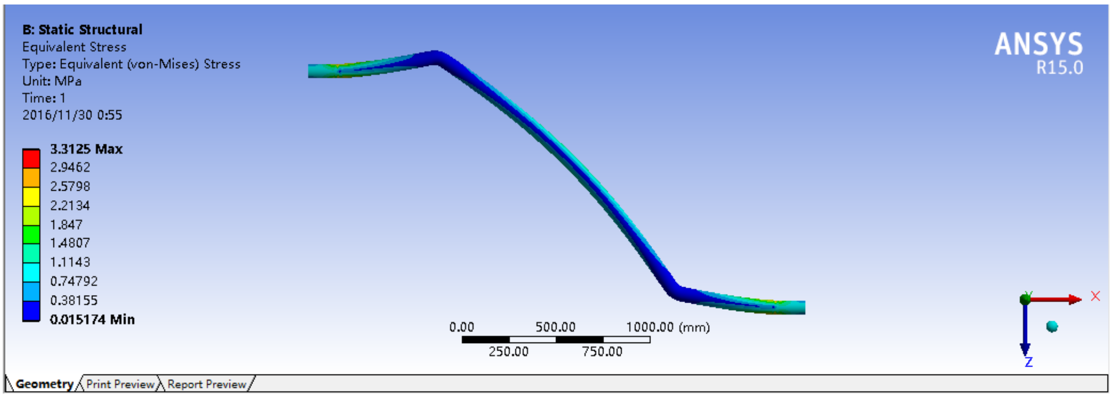
3.2 Static analysis

3.2.1 Influence of angle changing

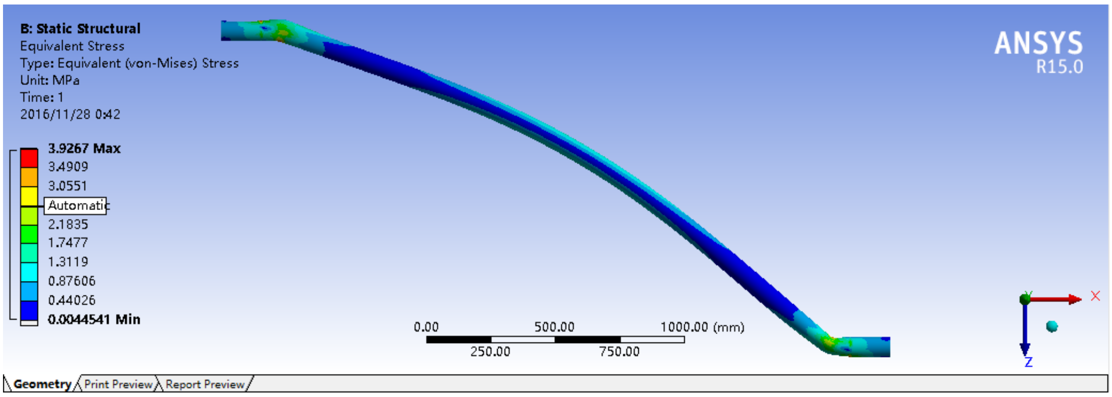
Keep the distance between the brackets and fixed endings to 155.50mm, change the angle of the corner, and obtain the stress of the pipeline in different angles. Figure 6 shows the pipeline stress in different angles.



（a）A=120°



（b）A=135°



（c）A=150°

Figure 6. Static stress in different angles

The max stress, strain and deformation in different cases are showed in the Table 1.

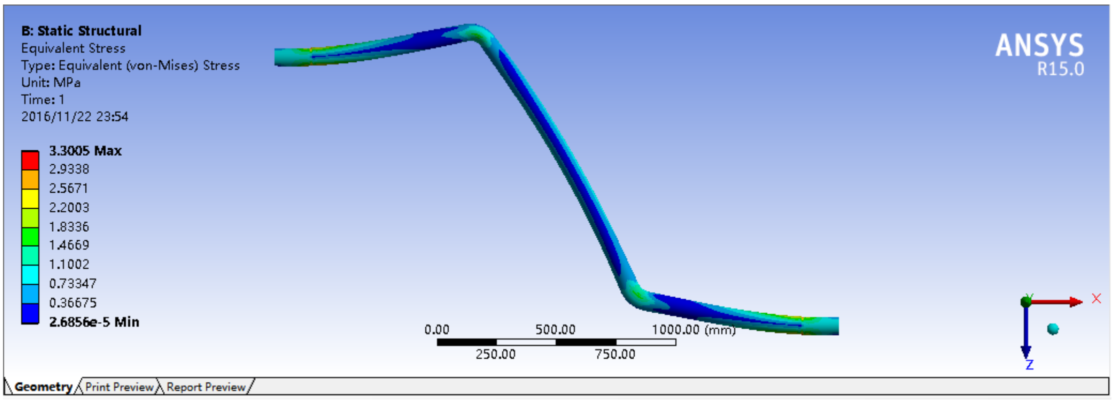
Table 1. Max stress, strain and deformation of the pipeline

Angle of the corner/°	120	135	150
Stress/MPa	3.3005	3.3125	3.9267
Strain/$\times 10^{-5}$	1.668	1.6723	1.9641
Deformation/$\times 10^{-2}$mm	7.7511	9.2919	9.0469

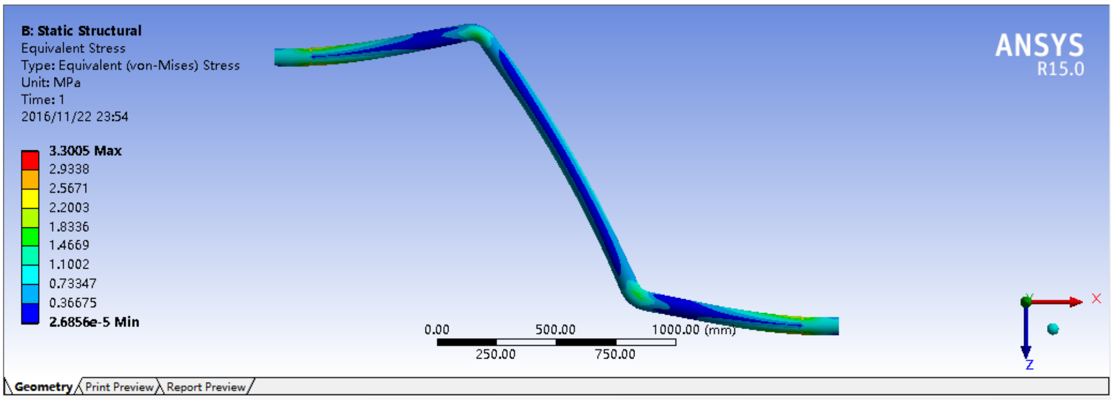
From the Figure 6 and Table 1, it can be concluded that when the angle of the corner increases, the max stress and strain also increase. So the angle of the corner can not be too large so that the structure would not be damaged by the large stress and strain. In addition, when the angle increases 15 degrees (from 120 to 135), the stress increases 0.012MPa, the strain increases 0.0043e-5; when the angle increases 15 degrees (from 135 to 150), the stress increases 0.6142MPa, the strain increases 0.2918e-5. So it can be concluded that when the angle becomes larger, the effect posed by the angle changing on strain changing will be more obvious.

3.2.2 Influence of the brackets position

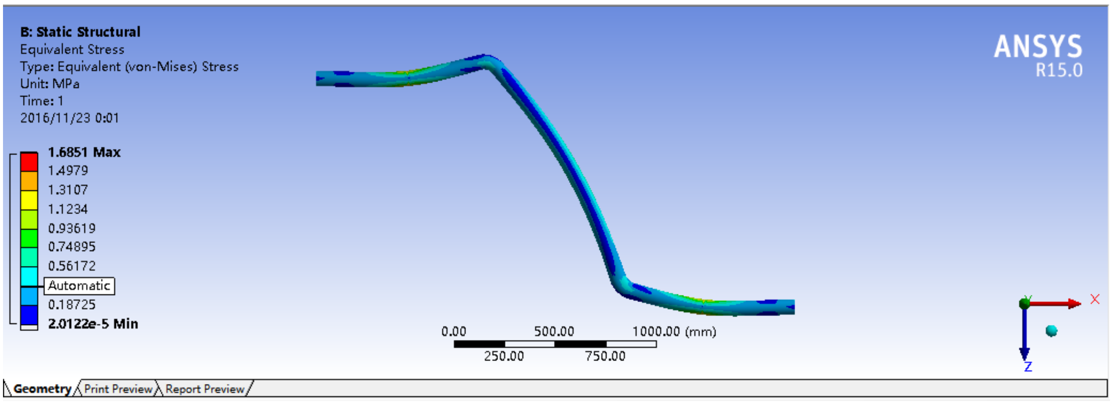
Change the distance between the brackets and the fixed ends, keep the angle of the corner A=120 degrees (the performance of the pipeline is better when A=120 degrees according to above analysis), and analyze the dynamic characteristics. The pipeline stress is shown as in Figure 7.



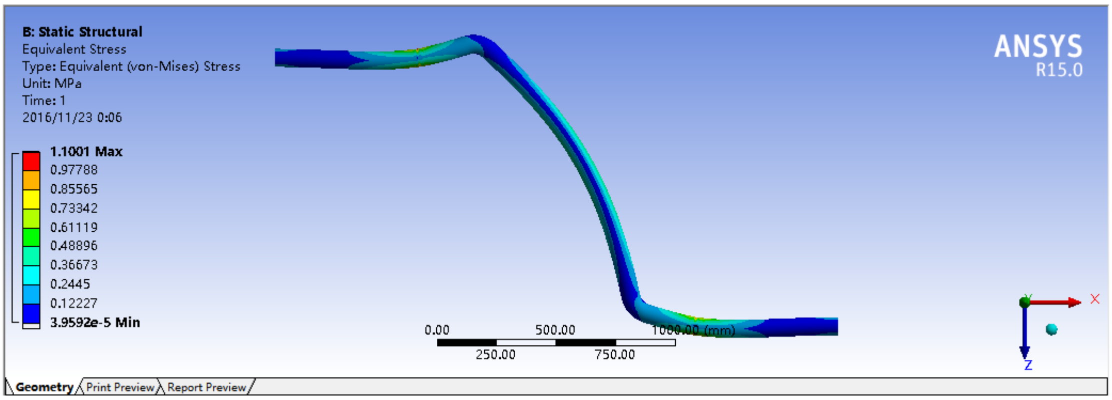
（a）155.50mm to the fixed end



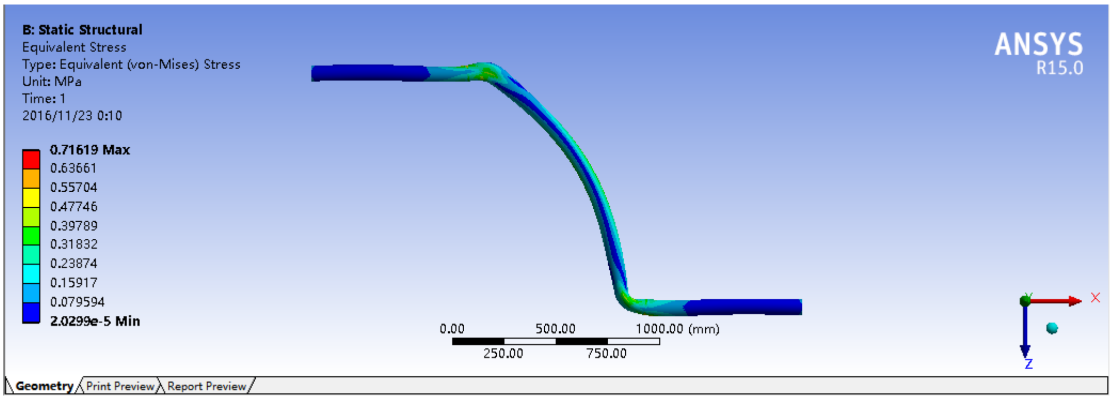
（b）305.50mm to the fixed end



（c）455.50mm to the fixed end



（d）605.50mm to the fixed end



（e）755.50mm to the fixed end

Figure 6. static stress in different distance

The max stress, strain and deformation in different cases are showed in the table4-2.

Table 2. Max stress, strain and deformation of the pipeline

Distance to the fixed end/mm	155.50	305.50	455.50	605.50	755.50
Stress/MPa	3.3005	2.4304	1.6851	1.1001	0.71619
Strain/$\times10^{-6}$	16.68	12.277	8.514e	5.5015	3.6249
Deformation/$\times10^{-2}$mm	7.7511	4.8394	2.8164	1.5596	0.85126

From the Figure 6 and Table 2 above, it can be concluded that when the distance between the guide brackets and the fixed ends increases (the guide brackets become closer to the corner), the max stress and strain of the structure decreases. As a result, it would be safer for the pipeline structure. So in the actual project, it's a good choice to shorten the distance between the guide brackets and the corners to ensure the safety of the pipeline structure.

3.3 modal analysis

3.3.1 Influence of angle changing

Change the angle of the corner, keep the distance between the brackets and the fixed ends, and analyze the dynamic characteristics. The results of the first six modals is shown in Table 3 and Table 4.

Table 3. Frequency of vibration

Angle of the corner/°	120	135	150
first-order frequency/Hz	60.781	55.079	55.79
second-order frequency/Hz	61.78	56.168	56.758
third-order frequency/Hz	105.08	124.63	161.48
forth-order frequency/Hz	183.15	169.86	162.93
fifth-order frequency/Hz	249.32	224.79	319.32
sixth-order frequency/Hz	484.08	279.27	321.2

Table 4. Modal of vibration

Angle of the corner/°	120	135	150
first-order modal/mm	13.426	13.809	14.043
second-order modal/mm	13.002	13.762	14.077
third-order modal/mm	16.062	13.847	13.736
forth-order modal/mm	17.114	12.802	13.549
fifth-order modal/mm	14.555	16.224	13.508
sixth-order modal/mm	16.71	15.662	13.8

From the modal analysis, it can be concluded that the vibration frequency of each modal has no fixed relationship with the angle. The vibration modal changes with the angle, but the difference is not so significant. So there is little meaning to make the angle a criterion for program evaluation. In addition, the external input frequency is 50Hz, so the chosen frequency should be away from 50Hz to avoid resonance. If only concerning about the first and second modal, the model of 120 degrees it the best.

3.3.2 Influence of the brackets position

Change the distance between the brackets, and still keep the angle A=120 degrees, the results of the first six modals is shown in Table 5 and Table 6.

Table 5. Frequency of vibration

Angle of the corner/°	120	135	150
first-order frequency/Hz	60.781	75.436	95.568
second-order frequency/Hz	61.78	77.341	98.298
third-order frequency/Hz	105.08	135.16	185.11
forth-order frequency/Hz	183.15	197.99	227.64
fifth-order frequency/Hz	249.32	285.21	345.65
sixth-order frequency/Hz	484.08	495.2	497.89

Table 6. Modal of vibration

Angle of the corner/°	120	135	150
first-order modal/mm	13.426	14.344	15.574
second-order modal/mm	13.002	13.923	15.373
third-order modal/mm	16.062	16.582	16.965
forth-order modal/mm	17.114	16.768	15.86
fifth-order modal/mm	14.555	16.484	18.817
sixth-order modal/mm	16.71	18.076	18.124

From the modal analysis, it can be concluded that the vibration frequency of each modal would increase when the distance between brackets and fixed ends increases. In this cases, the external input frequency is 50Hz, so the chosen frequency should be away from 50Hz to avoid resonance. If only concerning about the first and second modal, it’s best to set the distance between brackets and fixed ends to 755.50mm. If the input frequency changes, the optimal distance may also change.

4.Comparison of calculation results in Autopipe and Ansys

Since there are two different calculation software, there may be deviations in the calculation results. In order to learn more about the deviations, two sets of models are chosen to compare the results.

Model 1: angle A=120 degrees, both ends are fixed, no guide brackets;

Model 2: angle A=120 degrees, both ends are fixed, guide brackets are 303.5mm to the fixed ends.

Calculation results of the two models are shown in Table 7 and Table 8.

Table 7. Comparison of the Model 1

	Autopipe	Ansys
Stress/MPa	3.9	3.7003
first-order frequency/Hz	46.1753	52.827
second-order frequency/Hz	46.3340	53.471
third-order frequency/Hz	70.6685	89.773

forth-order frequency/Hz	148.2002	175.56
fifth-order frequency/Hz	177.5147	228.88
sixth-order frequency/Hz	465.9268	422.14

Table 8. Comparison of the Model 2

	Autopipe	Ansys
Stress/MPa	2.3	2.4304
first-order frequency/Hz	64.5740	75.436
second-order frequency/Hz	65.8979	77.341
third-order frequency/Hz	98.9831	135.16
forth-order frequency/Hz	159.1330	197.99
fifth-order frequency/Hz	194.4324	285.27
sixth-order frequency/Hz	678.3102	495.2

From the two tables above , it can be concluded that:

(1) The calculation results of the static analysis are similar in Autopipe and Ansys. If based on the Autopipe results, the error between them is as follows:

Model 1:Δ1=（3.9-3.7003）/3.9=5.12%

Model 2:Δ2=（2.4304-2.3）/2.3=5.67%.

The error stabilized at around 5%, which is acceptable in the engineering practices, so the static results are ideally matched.

(2) The calculation results of modal analysis are different in Autopipe and Ansys. In the first five modal, all the results in Ansys is significantly greater than those in Autopipe. But in the sixth modal, the results in Autopipe increase rapidly, and become greater than that in Ansys. If based on the Autopipe results, the error between them is as follows:

First modal, Model 1: Δ11=（52.827-46.1753）/46.1753=14.41%

First modal, Model 2:Δ12=（75.436-64.574）/64.574=16.82%

Sixth modal, Model 1: Δ61=(465.9268-422.14)/465.9268=9.40%

Sixth modal, Model 2: Δ62=（678.3102-495.2）/678.3102=27.00%

The error in first modal is around 15%, and the error in varies with models. Both results are not good.

During the modal analysis, both Autopipe and Ansys use the same equation:

$$([K]-w_i^2[M])\{\phi_i\}=0 \tag{3}$$

In which K is the stiffness matrix, ϕ_{i} modal matrix, W_i is the vibration frequency matrix, and M is the mass matrix.

There is no difference between the two softwares in the calculation principle, and the density Young's modulus, Poisson's ratio and shear modulus of the pipeline are the same. So if only concerning the finite element method, the two results should be consistent, but in fact the results have large difference. The reason for the large errors may be as follows.

(1) In the calculation, Autopipe uses a prestressed modal analysis, while Ansys does not consider prestressing, which results in a different stiffness matrix.

(2) The two models established in Autopipe and Ansys may not be exactly the same. Maybe there are some difference in the corner or the fixed ends

5.Initial analysis with FSI method

When the fluid flows in the pipeline, it induces pipe stress and deformation while the pipe deformation will influence the flow pattern at the same time(Yu Meng,2007), which is solved by FSI method. It is used in this paper to get the fluid pressure in the mid-section of the pipeline, the shear stress in the pipeline wall and the normal stress in the pipeline wall. The simulation results are as shown in Figure 7, Figure 8 and Figure 9.

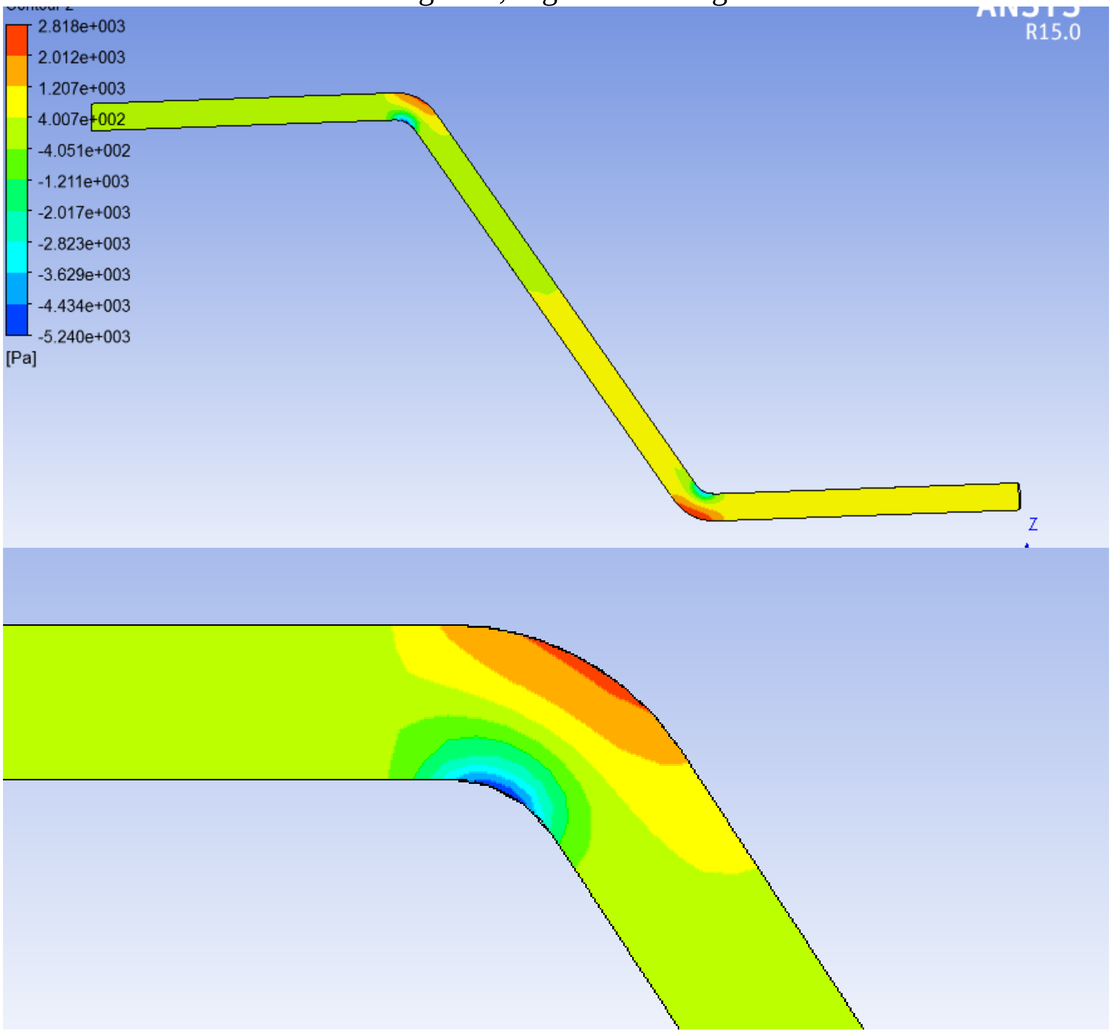
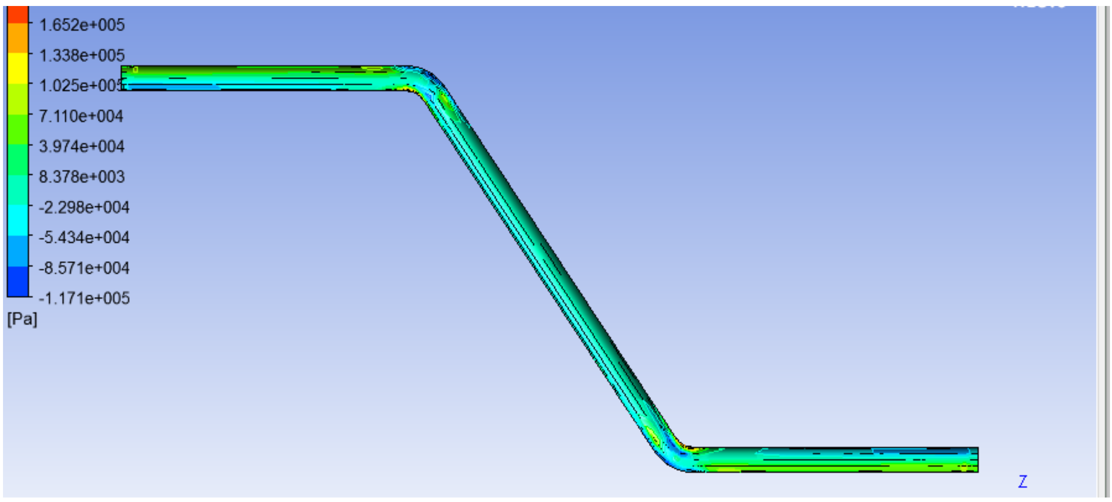


Figure 7. Fluid pressure in the mid-section of the pipeline



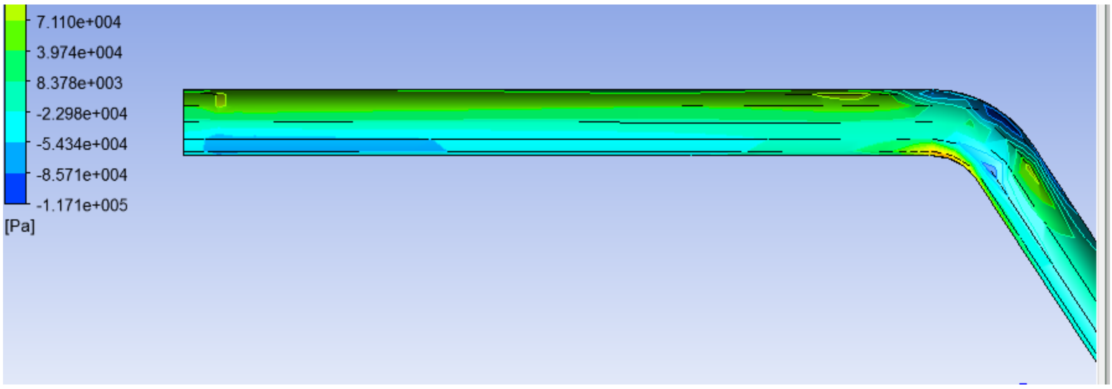


Figure 8. Shear stress in the pipeline wall

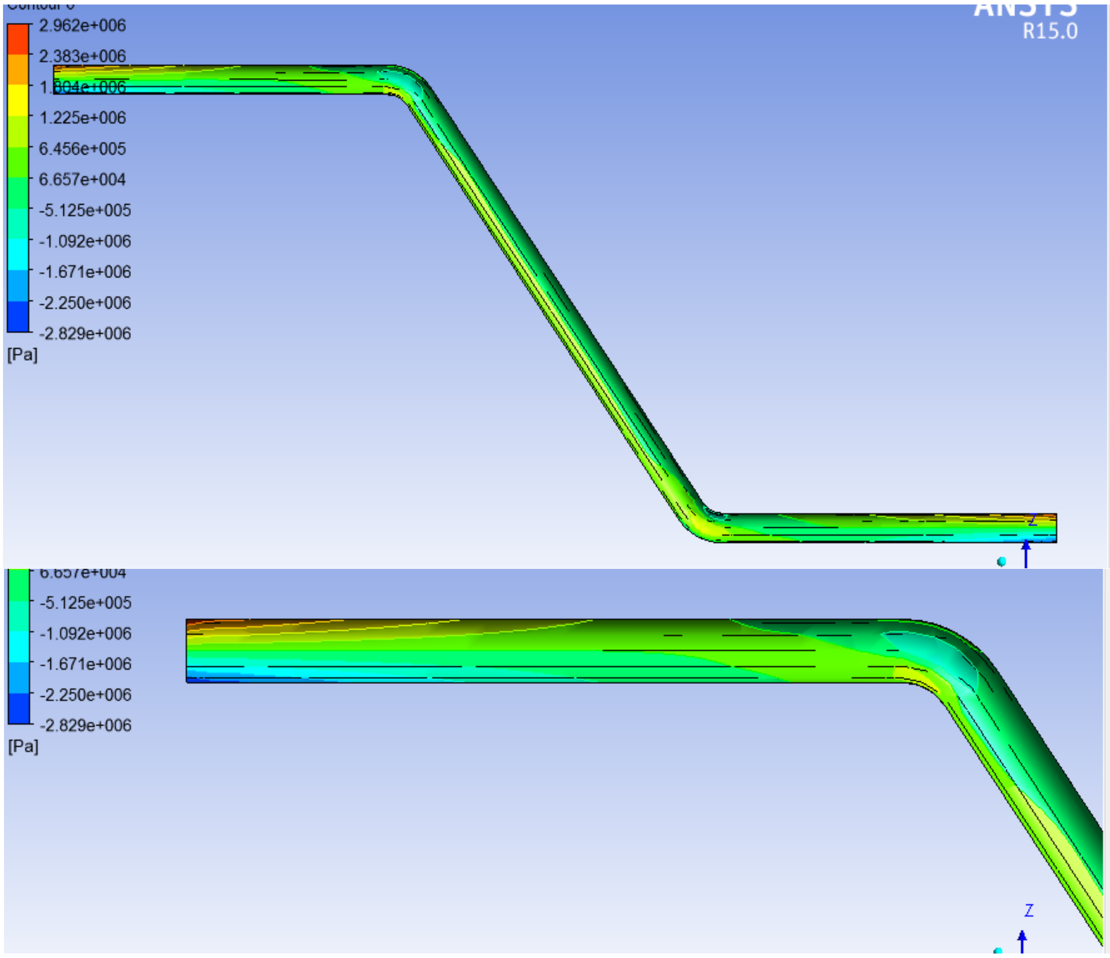


Figure 9. Normal stress in the pipeline wall

6.Conclution

Base on the simulation in Autopipe and Ansys, there are some suggestions:

(1) When the angle of the bend increases, the max stress and strain of the structure will also increase. The stress and strain will be more sensitive to the angle change. Large angle of the corner should be avoid so that the structure would not be damaged.

(2) When the distance between the guide brackets and the fixed ends increases (the guides bracket become closer to the corner), the max stress and strain of the structure decreases. As a result, it would be safer for the pipeline structure. So it’s a good choice to shorten the distance between the guide brackets and the corners to decrease the stress and strain to ensure the safety of the pipeline structure.

(3) When the distance between the guide brackets and the fixed ends increases (the guides brackets become closer to the corner), the vibration frequency of each modal also increase. When the frequency of external excitation is close to the resonance area, it's a good idea to adjust the position of the brackets to change the natural frequency to avoid resonance

Of course there are still many deficiencies**:**

(1) The result in this paper is only the simulation results. The comparison with the corresponding experiment results or on-site measured data should be added in further research.
(2) There may be some problems with the results of Fluid and Structure Interaction method. Maybe when using the software, some parameters are not consistent with the reality. Those parameters need to be further corrected.
(3) Further analyses are needed to explain the different results in Autopipe and Ansys.

Reference

[1] M.L.Levitan,K.C.Mehta and W.P.Vann et al. Field measurement of pressure on the Texas Tech Building, Journal of Wind Engineering and Industrial Aerodynamics, 1991, 38(4): 227-234.

[2]Dai Xueliang, Liu Ping. Application and Precautions of Hydraulic Pipeline in Ship. Jiangsu Ship. 2000 (3) :27-29

[3] Liang Chunyu.Study of Vibration of the Ship Engine Fuel Oil Pipeline System. Power Machinery and engineering. Harbin Engineering University.2013.05.

[4] Liu Zhongqun, Sun Yudong, Wu Yousheng. Current Situation and Trends on the Study of Coupled Fluid - Structure Vibration and Sound Propagation of Pipeline Systems. China Ship Scientific Research Center,Wuxi 214082,China. Ship Mechanics,2001.

[5] Chen.S.S, Paidoussis.M.P, Au - Yang.M.K, et al. Flow - Induced Vibration of Circular Cylindrical Structures by S.S. Chen, M.P. Paidoussis, and M.K. Au - Yang[J]. Journal of the Acoustical Society of America, 1984, 75(1):299-299.

[6] Paidoussis.M.P, Issid.N.T, M.Tsui. Parametric Resonance Oscillations of Flexible Slender Cylinders in Harmonically Perturbed Axial Flow. Department of Mechanical Engineering, Journal of Applied Mechanics, 1980.47:709-714.

[7] Paidoussis MP. Flow-induced Instabilities of Cylindrical Structures. ASME. Appl. Mech. Rev. 1987.40(2):163-175. doi:10.1115/1.3149530.

[8] Song Xueguan, Cai Lin, Zhang Hua. Ansys Fluid-Solid Coupling Analysis and Engineering Examples [M]. Beijing: China Water Resources and Hydropower Press, 2012.

[9] Xie Longhan, Zhao Xinyu. Ansys Cfx Fuid Analysis and Aimulation [M]. Beijing: Electronic Industry Press, 2013.

[10] Yu Meng. Analvsis on Characteristics of Fluid-structure Interaction for Fluid Conveying Pipes by ANSYS. China Ship Development and Design Center, Wuhan 430064,China.2007.10.