Finite Element Simulation of Partial Pipeline Suspended Completely

Zhixiang Li, Peng Zhang, Guizhi Li, Zunhai Ke, Gengxin Wu

College of Civil Engineering and Architecture, Southwest Petroleum University, Chengdu, Sichuan 610500, China

Abstract

Oil and Gas Pipelines will pass through some areas with complex and adverse geological conditions which are vulnerable to disasters such as washout of flood inevitably. Soil around the pipe will be washed away, and pipe shall be gradually exposed to completely vacant. Result in large deformation, and very likely to damage, which will affect the safe operation of the pipeline greatly. In view of this, finite element models of buried pipeline partly suspended completely are established to calculate the mechanical response of pipe, and compared with the test results. Further calculations consider pipelines with angle. Finite element simulation results show good agreement with experiment, and angle will affect internal forces and deformation of the pipe significantly.

Keywords

Oil and gas pipeline; suspended; finite element simulation; angle.

1. Introduction

Oil and gas pipelines, known as the artery of the national economy, are responsible for the task of conveying the uneven distribution of oil and gas resources in China. Because of the vast area, the distribution of oil and gas pipelines show characteristics of long-distance and cross-regional. Pipelines therefore will pass through some areas with complex terrain and geological conditions inevitably, and these areas are usually have development of geological disaster. Typical geological disasters include landslide, mudslide, washout, collapse, soil subsidence and earthquake. Affected by these disasters, the soil around the buried pipeline will be taken away or collapsed, resulting in pipelines partial suspended. Pipelines will bending and deformation, may even rupture destroyed when suspended too long. Safe operation of pipelines are under severe threat. Disasters are associated with randomness. Most research of suspended pipeline now are ex post approach, namely governance and analysis after disaster. And parameters of pipelines and buried conditions are varied. Specific and targeted research is difficult to implement, while finite element simulation is a convenient way.

On considering this, finite element models of buried horizontal pipelines with no angle and partly suspended completely are established at first, and calculated results are compared with results from experiments. Continuous debugging of model parameters is made to find a better model to analyze the mechanical response. For some pipelines in engineering practice are buried at a certain angle along hillside, further analysis based on verified model are made to study the impact of angle. All of these studies of suspended pipeline are clear and intuitive.

2. Establishment of finite element model

Finite element model of buried horizontal pipeline partly suspended is built. Soil is modeled by SOLID element, and constitutive model of soil is Drucker-Prager (D-P). After series of debugging, the density of soil of the model in good agreement with the experimental results is 2000kg / m^3. And the elastic modulus is 19MPa, Poisson's ratio 0.4, cohesion 50000Pa, internal friction angle of 20°. It is found during the calculation that settlement of soil under the action of deadweight has adverse effect on results when the length of the suspended section is over 80m. More ideal results can obtained at this time when treating soil as elastomer.

SHELL element is adopted to simulate pipe, which parameters are consistent with that in the experiment. External diameter is 508mm, wall thickness of 7.9mm, and buried in 2.5m depth [1]. Pipe material is X52, has a density of 7850kg / m^3. The elastic modulus is 207GPa, lower yield strength limit of 360MPa, Poisson's ratio of 0.3. Trilinear model [2] and Ramberg-Osgood model [3] are two kinds of pipe material used most in finite element simulation of oil and gas pipelines. Two kinds of models are built using these two constitutive relationship respectively. And results of each are compared with the experiment to select a closer model for next calculation. Two kinds of constitutive relation of data points are shown in Table 1 and Table 2. Transformation of real stress - strain is needed for engineering strain is adopted in Ramberg-Osgood model. Two kinds of constitutive relation curve are plotted in Figure 1.

Table 1 Data points in trilinear constitutive relationship model of X52 pipe

Stress(MPa)	0	405.72	455
Total strain	0	0.00196	0.069

Table 2 Data points in Ramberg-Osgood model of X52 pipe

Real stress(MPa)	Calculated total engineering strain	Calculated total real strain	Data points in Ramberg-Osgood model	
			Real plastic strain	Real stress(MPa)
			0	360
380	0.004414884	0.004405167	0.002569418	380
400	0.00646671	0.006445891	0.004513524	400
420	0.009784251	0.009736695	0.00770771	420
440	0.015062614	0.014950299	0.012824695	440
460	0.023317761	0.023050055	0.020827833	460

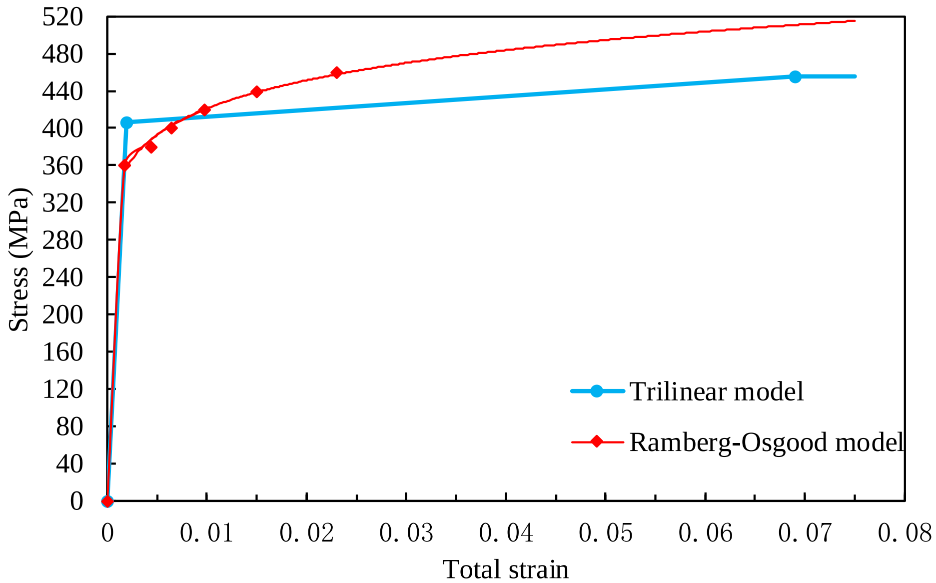

Figure 1 Constitutive relation curve of X52 pipe

Nonlinear face to face contact is used to model the interaction of pipe and soil. Weight of the product oil transported is equivalent to pipe density [4]. The working pressure is 4.3MPa, which is applied as normal pressure to the inner surface of SHELL elements. Finite element model is shown in Figure 2. It's a half model here on considering the symmetry. Upper and free surface of soil is free and the bottom is fixed, while other surfaces are all constrained except displacement in vertical. Symmetry boundaries are applied to pipe at symmetry end, while axial displacement and rotation at three direction are constrained.

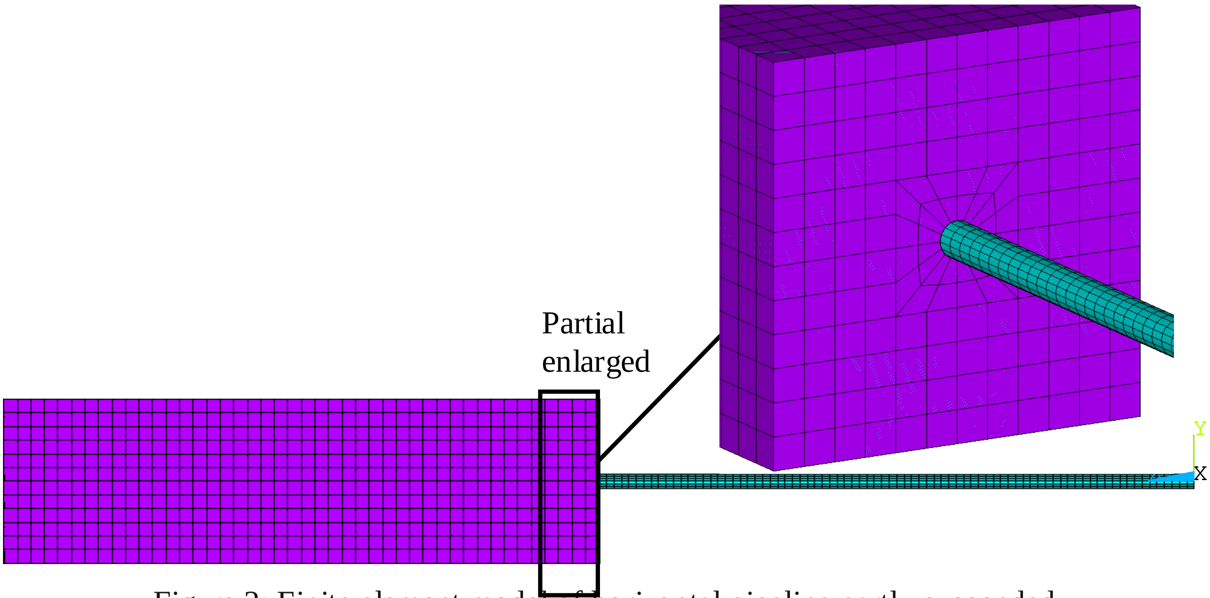

Figure 2: Finite element model of horizontal pipeline partly suspended

3. Results Verification and Analysis

Results of two kinds of constitutive relation model (every relevant parameter is same except pipe constitutive relation) which are in good agreement with the experimental results are listed in Table 3.

Table 3 Results comparison between finite element simulation and experiment

Suspended length(m)	Trilinear model			Ramberg-Osgood model			Test results in references [1]		
	Stress (MPa)	Strain	Maximum displacement (m)	Stress (MPa)	Strain	Maximum displacement (m)	Stress (MPa)	Strain	Maximum displacement (m)
20	227	0.001143	0.221078	224	0.00113	0.221169	231.4	0.001022	0.21
40	263	0.00129	0.64879	263	0.00129	0.648818	268.5	0.002156	0.42
80	405	0.002329	1.369045	365	0.002829	1.376827	341.1	0.003609	0.82
100	409	0.003971	1.896591	375	0.004237	1.914061	376.2	0.005124	1.35
120	411	0.006124	2.444546	388	0.00566	2.462846	395.3	0.008155	2.06
140	417	0.008592	3.015195	399	0.007071	3.02857	401.3	0.010255	2.72
160	415	0.011151	3.59925	408	0.00852	3.61458	406.2	0.014336	3.48
180	412	0.013782	4.208021	415	0.009842	4.213042	410.5	0.017882	4.09
200	423	0.01689	4.829508	421	0.011201	4.83884	414.7	0.022016	4.71

From data in the table, we can see results from the two models are close to that from experiment. And the pipe stress in Ramberg-Osgood model is more uniform, more stable, and more consistent with the experimental results (see Figure 3), which is decide by the characteristic of pipe constitutive relation model.

Diagram of pipe displacement, stress and strain of Ramberg-Osgood model when pipeline is suspended 20m long is shown in Figure 4, 5 and 6. As the model is half, left section of the pipeline is buried, while the right part is suspended. It is seen from the displacement map that buried pipeline also have displacement. This is because soil will settle under the action of deadweight, and drive pipe moves together. While in engineering practice, soil settlement has been completed before the disaster. So actual maximum displacement of pipe caused by partly suspended pipeline is 0.489974-0.268805 = 0.221169m.

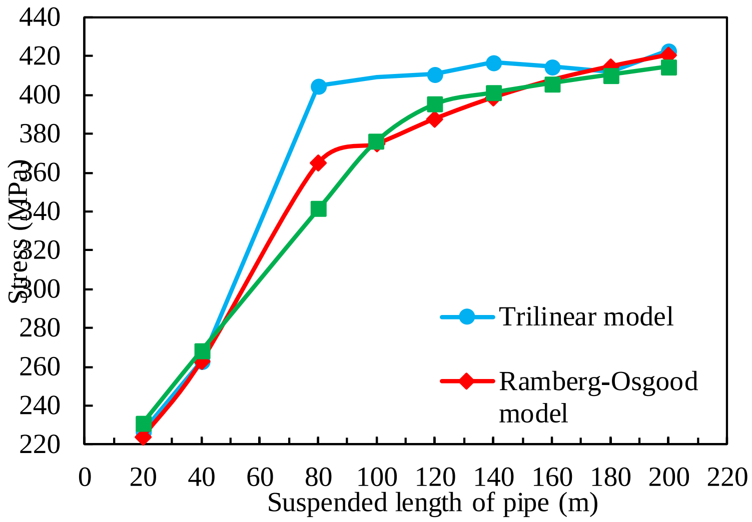

Figure 3 Comparison of pipe stress curve

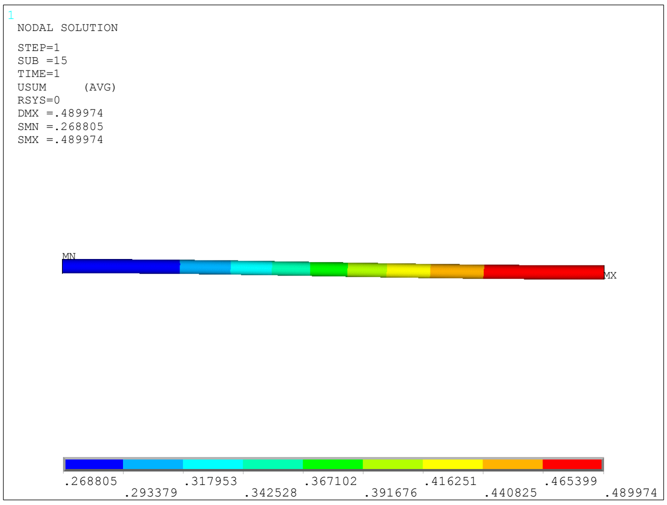

Figure 4 Displacement of pipe at suspended length of 20m

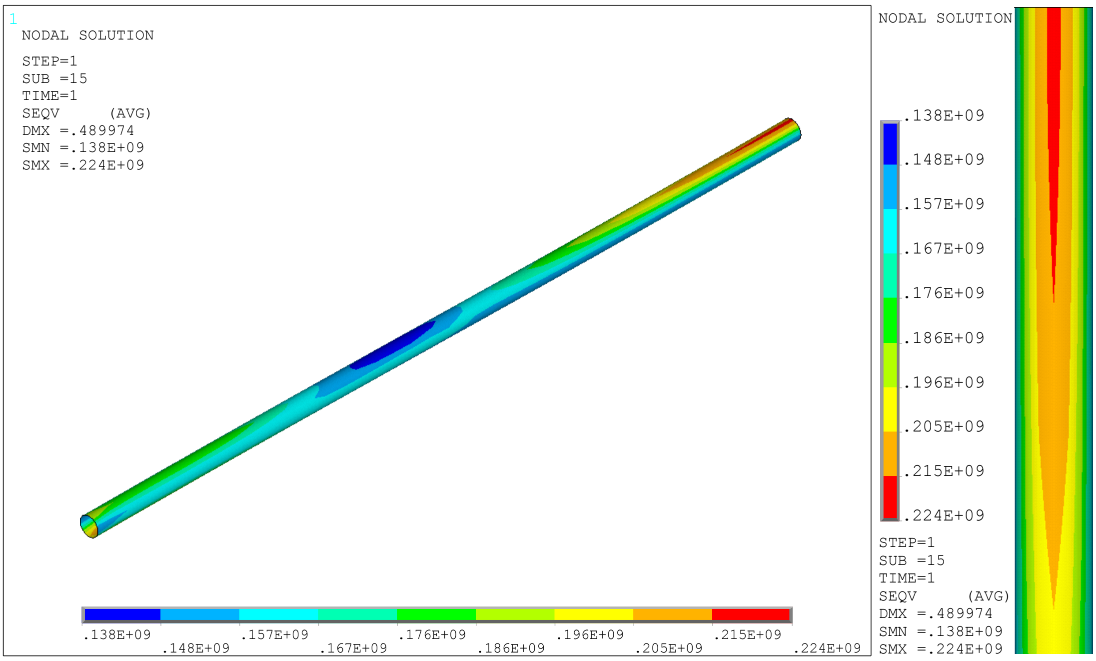

Figure 5 Pipe stress at suspended length of 20m

It can be seen clearly from the pipe stress and strain diagram that, for this pipeline at suspended length of 20m, maximum stress and maximum strain is located on upper at the middle of suspended section, while the minimum is located on the top surface of pipe near the junction of buried and suspended. This is because pipe the near the junction is pulled by the action of suspension. Upper surface of pipe at central suspended section is pressed for bending deformation. The suspended length at this time is short, pipe deformation is small. And soil is relatively soft, stress concentration of the pipeline at the junction will not occur. Tensile properties of steel is good, and the cover earth is good to pipe bearing too. Suspension action strengthening as the suspended length increase, which will result in larger bending deflection. Maximum stress and maximum strain of pipe will move to the top surface of the pipe at the junction.

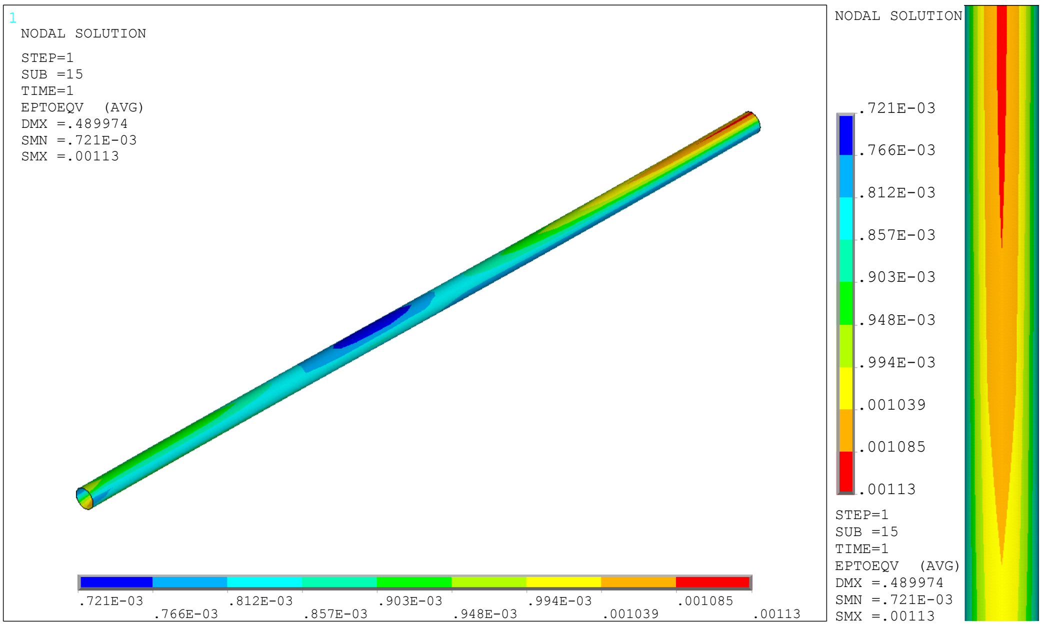

Figure 6 Pipe strain at suspended length of 20m

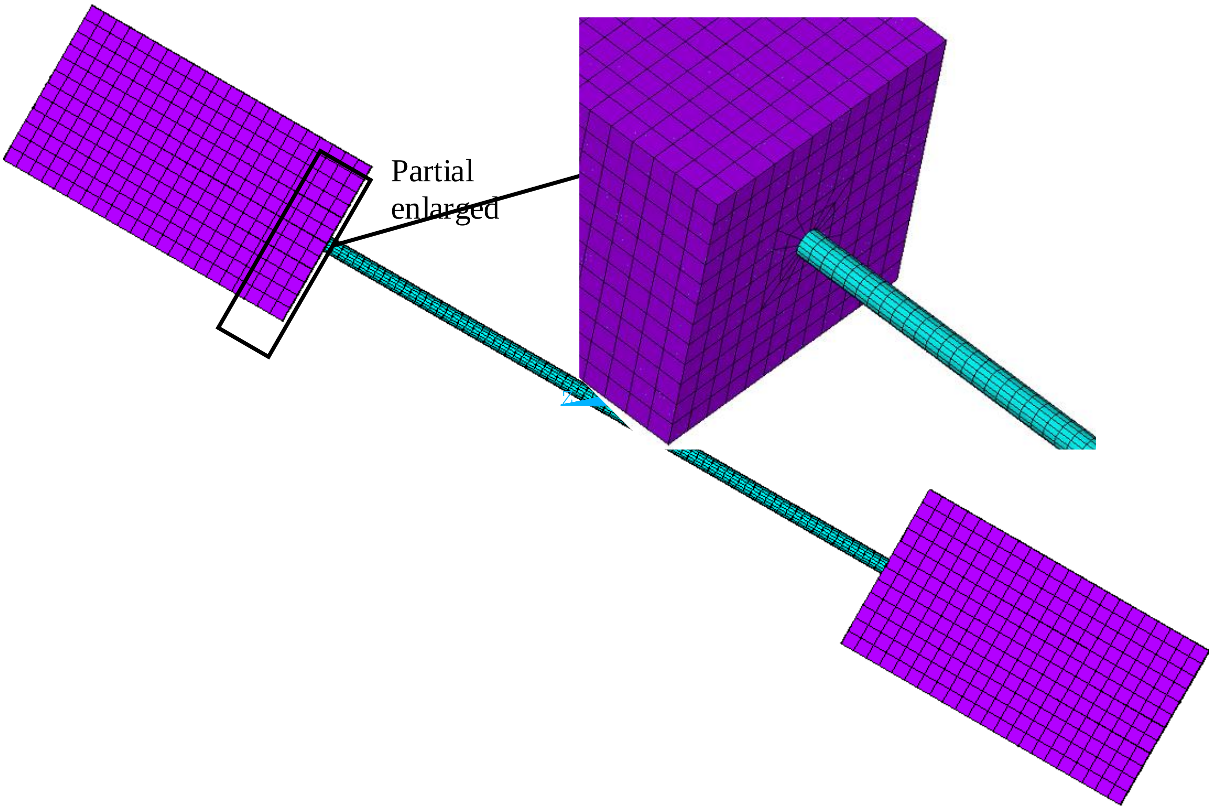

Figure 7 Finite element model of partly suspended pipeline with angle

4. Effect of dip angle

Remain the suspended length 20m and other parameters of the model unchanged. Build finite element models of buried pipelines with certain angle and partly suspended completely to analyze the effect of dip angel. From previous simulation we can see results from Ramberg-Osgood model is better. So pipe constitutive relation adopted this model in this section. For loads of the model with angle are no longer has that symmetry in the previous section. Full model is established here as Figure 7 shows. And because the soil is inclined, there is a certain angle with the acceleration of gravity. It will not consistent with realities if continues to let the soil subsidence due to its own weight. So soil density are not considered here, soil self-weight stress of buried section is calculated as $\sigma_{cz} = \gamma z$ and applied on pipe.

The dip angle of pipe changed from 5 °to 45 °in every 5 °. Results are listed in Table 4.

Table 4 Results of partly suspended pipe with different angle

Angle(°)	Displacement(m)	Stress (MPa)	Strain	Angle(°)	Displacement(m)	Stress(MPa)	Strain
5	0.171851	194	0.000986	30	0.151398	189	0.00096
10	0.170063	194	0.000983	35	0.143926	187	0.000951
15	0.16712	193	0.00098	40	0.135457	184	0.000941
20	0.163029	192	0.000975	45	0.119138	181	0.00093
25	0.157778	190	0.000968				

From the results we can see that when pipe is inclined at certain angle, for the weight component perpendicular to the pipe axis is smaller than the full weight when pipe is horizontal and have no dip angle, pipe stress and deformation are also smaller than that of horizontal suspended pipeline. And the larger the angle, vertical pipeline weight component is smaller, suspension action would weaken. Pipe maximum displacement, maximum stress and maximum strain will gradually decrease. Displacement, stress and strain diagram of pipe with angle of 30 °is shown in Figure 8, 9, 10.

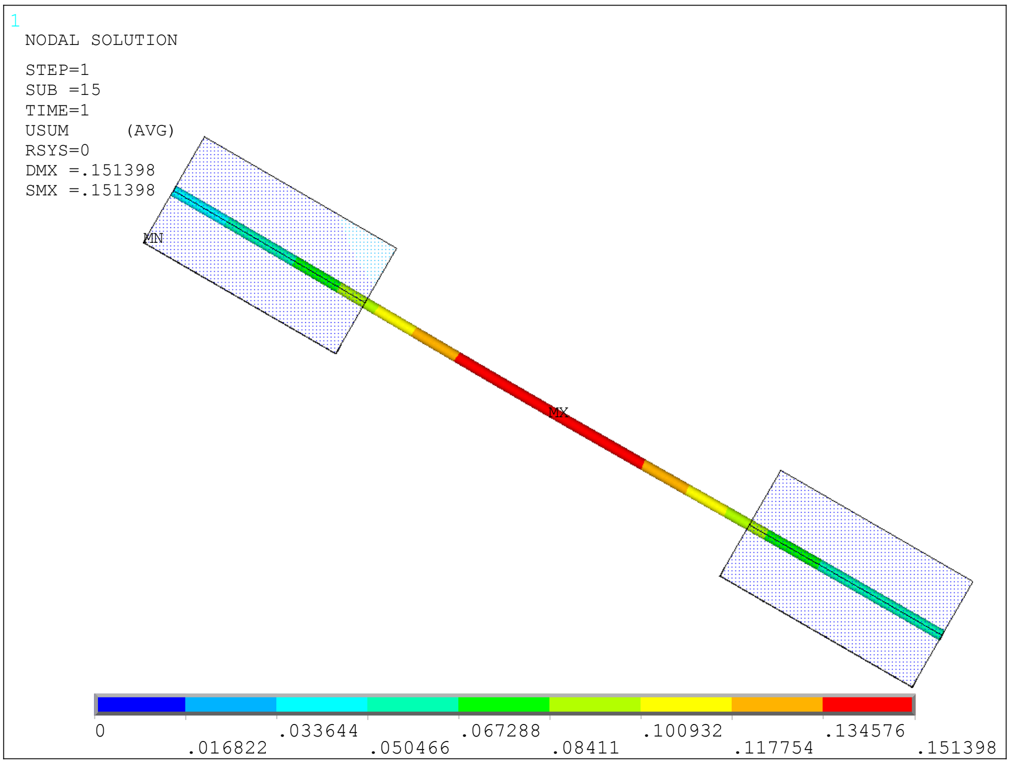

Figure 8 Pipe displacement with angle of 30 °and at suspended length of 20m

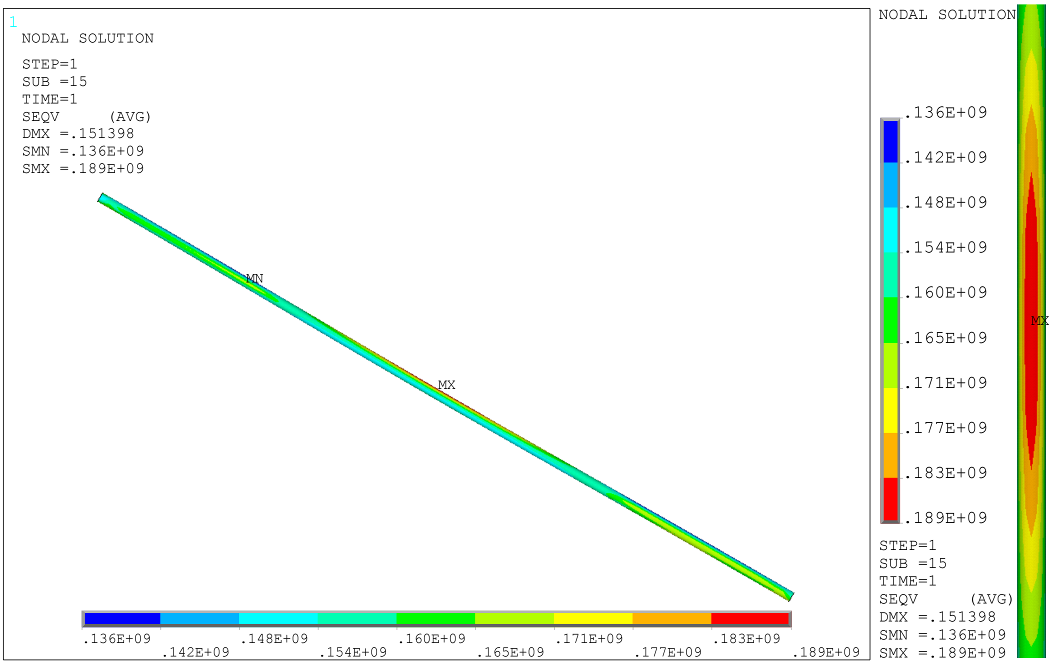

Figure 9 Pipe stress with angle of 30 °and at suspended length of 20m

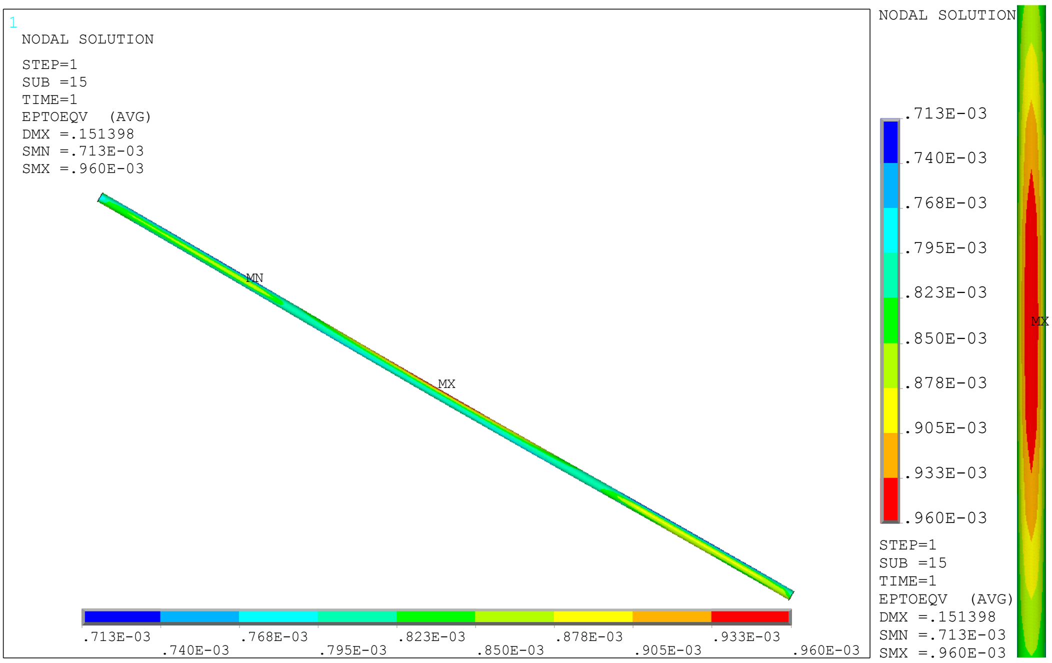

Figure 8 Pipe strain with angle of 30 °and at suspended length of 20m

We can see from these figures that, as is same to suspended pipeline horizontal with no angle, the maximum value of the results are still at the middle of suspended section. The maximum displacement is mainly perpendicular to the axial of the pipe. But when pipeline has bigger angle (greater than 45 °), due to the limited nature of the soil pipe friction constraints, the weight component along pipeline axial will have significant effect on pipe. Lower part of the pipeline will be compressed, which need special attention.

5. Conclusions

Finite element models of buried horizontal pipeline or with angles and partly suspended completely are established to calculate and analyze the effect of suspension as well as incline. Relevant results are compared with the results of experiment. Several conclusions can be made according to this study:

1 Results from finite element simulation show good agreement with the experimental results. Especially when the constitutive relation of pipe material adopts Ramberg-Osgood model. And finite element simulation is more convenient, more intuitive.

2 The suspended of pipeline will produce bending deformation, result in maximum stress and strain on the upper surface of pipe at middle of suspended section when the suspended length is short. And these maximums move to the top surface of the pipe at the junction of pipeline been buried and suspended.

3 Mechanical responses of pipe weaken when pipeline has angle as the action of gravity differs from horizontal pipeline. Weight component along pipeline axial will make lower part of the pipeline compressed.

References

[1] Ma Tingxia， Wu Jinqiang， Tang Yu， et al. Maximum Suspended Length of Production Pipeline [J]. Journal of Southwest Petroleum University: Science & Technology Edition, 2012, 34 (4):165-173.

[2] GB 50470-2008. Seismic technical code for oil and gas transmission pipeline engineering [S], 2008.

[3] Indian Institute of Technology Kanpur, Gujarat State Disaster Management Authority. IITK-GSDMA Guidelines for Seismic Design of Buried Pipelines: Provisions with Commentary and Explanatory Examples [S]//National Information Center of Earthquake Engineering, 2007.

[4]Wang enqing. STATIC, PIGGING AND SEISMIC ANALYSIS OF THE PIPELINE SUSPENSION BRIDGE [D]. Beijing: Beijing University of Chemical Technology, 2007.