Analysis of Dynamic Response of Seismic Load in Medium Pressure Vertical Pipeline

Xinyu Hu[1, a], Hehui Wang[2,b], Ye Huang[1,c], Ge Cui[1,d], Peng Zeng[2,e]

[1]General Construction of CCTEB Group Co.,Ltd, Wuhan, 430064, Chian;

[2]School of Mechanical and Power Engineering, East China University of Scinece and Technology, Shanghai, 200237, China

[a]18086639570@189.cn, [b]hhwang@ecust.edu.cn, [c]hy0982@vip.163.com, [d]cuige@126.com, [e]244745955@qq.com

Keywords: ANSYS, Medium pressure vertical pipe, Structural transient dynamic analysis.

Abstract. Based on ANSYS finite element analysis software, the finite element numerical analysis model of the medium pressure vertical pipe was established. The seismic response of pipeline was analyzed by using the structural transient dynamic analysis, which provides a theoretical basis for the seismic design of the pipeline.

Introduction

In the high-rise building, the design of pipeline system has become an essential link. When the pipeline working in high pressure, big drop, full of water, the gravity of the water and the pipes produce huge energy, which bring a lot of problems to the inside standpipe design, manufacture and installation method. There are many vertical pipes, when the earthquake happens with horizontal acceleration and vertical acceleration, seismic horizontal acceleration gives pipeline horizontal load, the pipeline has a great moment, and causing vibration [1, 2]. Because the piping system of vertical pipe is very high in the high rise building, and the pipe is long, the mechanical deformation produces easily under earthquake load. In the pipeline design, the corrugated compensator will be set at intervals of several layers of the pipe. Therefore, the dynamic characteristics of the medium pressure pipeline, especially the transient dynamics analysis, the dynamic stress and displacement of the structure change with time, and the maximum value is found to be the basis for the design, analysis and calculation.

The Basis of Transient Dynamic Analysis

Transient dynamic analysis [3], also known as the time history analysis, is used to calculate the dynamic response of the structure under the action of load changed with time. The aim is to obtain the solution of the time varying of the displacement, strain, stress, etc.

Transient dynamics is a general equation of motion, which is as follows:

$$[M]\{\ddot{X}\}+[C]\{\dot{X}\}+[K]\{X\}=\{F(t)\} \quad (1)$$

In this equation:

$[M]$——Mass matrix,

$[C]$——Damping matrix,

$[K]$——Stiffness matrix,

$\{\ddot{X}\}$——Node acceleration vector,

$\{\dot{X}\}$——Node velocity vector,

$\{X\}$——Node displacement vector,

$\{F(t)\}$——Time varying load vector.

At any given time t, the equation can be considered as a series of static equations, at the same time, the inertia force and damping force are considered. In the ANSYS program, the time integration method is used to solve the equation of the time points. The time increment is called the time integration step [4].

Transient Dynamic Analysis

Pipeline Structure and Parameters. Piping system mainly includes the pipe part and the restriction part: the fixed frame, the guide frame, the bearing beam, and the ear type support. There pipes locate between the third layer (building elevation is 14.6m) and the thirteenth layer (building elevation is 70.7m) of the building. Pipe thickness is 16mm, and material is 20# seamless steel tube. Considering the weight of the full water, the working pressure of the 2.5MPa is under the pressure of the tube. The upper end of the pipe is provided with a corrugated pipe expansion joint. That is used to compensate for the thermal expansion and migration of the pipeline, and to prevent the deformation and failure of the pipeline due to the thermal stress caused by the medium temperature. At the same time, in order to restrain the vibration of the pipes, the fixed brackets are located between the third layer and sixth layer of the pipeline, and the remaining layers are provided with the axial guide brackets. Figure 1 is a pipes supports position diagram. Under the action of earthquake, the reasonable layout and design of the pipe support system can effectively exert its function to limit the displacement and vibration of the pipeline.

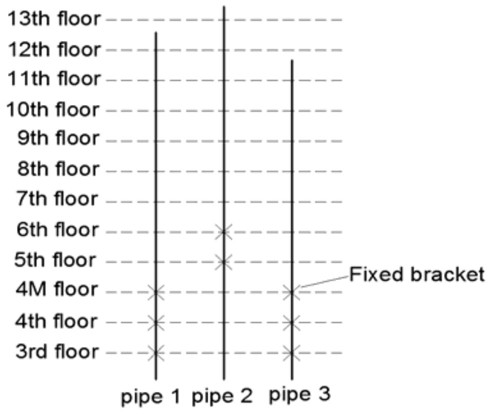

Fig. 1 Pipe support position

The Establishment of Finite Element Model of Pipeline. The model consists of three vertical pipes, supporting beams, lug supports, reinforcing ribs and bottom plates. Figure 2 is the overall finite element model. The three vertical pipe main body part adopts the shell63 element, each node has six degrees of freedom. It not only has ability to bend but also has membrane force, and can bear the plane load and normal load; Each layer of support beam using beam189 element, each node has six to seven degrees of freedom, it can be very good for linear, large rotation, and nonlinear large strain; Lug support, strengthen rib plate and the bottom plate adopt element shell63. The position of each layer support is shown in Figure 3 and Figure 4.

Fig. 2 The overall finite element model

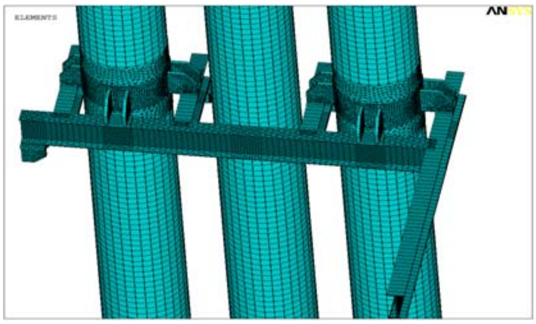

Fig. 3 The finite element model of fixed supports on both sides

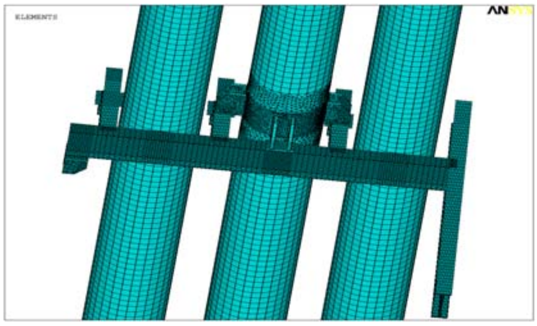

Fig. 4 The finite element model of middle fixed brackets

The beams and lug supports connect through the corresponding node with all degrees of freedom coupling. The degrees of freedom of lug supports (one side welding with the wall side) all restrain, The ribbed plate and the supporting beams connect through the corresponding nodes with all degrees of freedom coupling, and the end part of the pipe body connects with expansion joint, releasing of degrees of freedom without constraint handling. Considering the influence of the overall weight of the model, the axial acceleration of gravity is defined. The working medium of the interior body is water, the three pipes is applied inward pressure along the height direction of the linear distribution method to replace the effect of water on the pipeline, at the same time, the pipe bottom and horizontal pipe are connected by an elbow. When the vertical pipe is full of water, tremendous pressure will appear in the elbow. And the elbow gives the vertical pipe an axial direction of the tensile force. The top of the pipe is expansion joint, and the tensile force of the pipe is generated along the axial direction when the temperature changes. The overall constraints are shown in Figure 5. The Figure 6 is the fixed supports constrain.

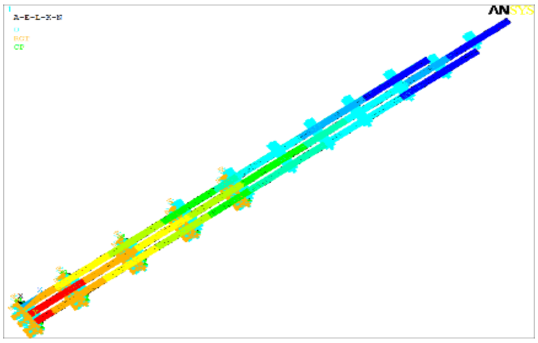

Fig. 5 The overall model constraints

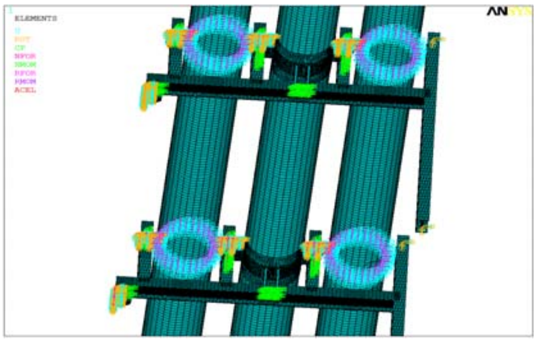

Fig. 6 The fixed supports constraints

Three pipes have the same fundamental parameters. Elastic modulus $E=2.2*105N/mm^2$, density p $=7.85*10-9t/mm^3$ and Poisson's ratio $\mu=0.3$, the length of three pipes is respectively 56171mm, 51750mm, 47430mm, outside diameter D1 is 711mm, inside diameter D2 is 679mm.

Solve of Transient Analysis. There are three ways to solve the problem of transient analysis: Full, Reduced and Mode Superposition. The Full method is used to analyze the seismic response. The seismic wave is recorded in Tianjin Ninghe earthquake [5], and the vertical direction and the direction of the north and the south are recorded. The time of the record is 19.11 seconds. Take a value from the record for each 0.1s, a total of 190. In order to read the data file easily for ANSYS. The seismic wave data is made into two text files. The file ACELX.txt is used to store the time and horizontal acceleration, and the file ACELZ.txt is used to store time and the vertical acceleration. In these two text files, the first column is for time data, the second column is for the acceleration data, if it is positive, there are two blank spaces between the first column and the second column; if negative, there is one blank space. At last, these two files are stored in the working directory of the ANSYS. Because circulatory function of APDL is used in the solution, the command flow method is used to load and solve the problem.

Transient Dynamic Analysis Results

Displacement Calculation Results. Through the analysis and calculation, the pipeline displacement image (Fig. 7) is obtained. As is shown in the picture, the maximum displacement occurs at the top of the pipes, and the maximum displacement is 3.13229mm.

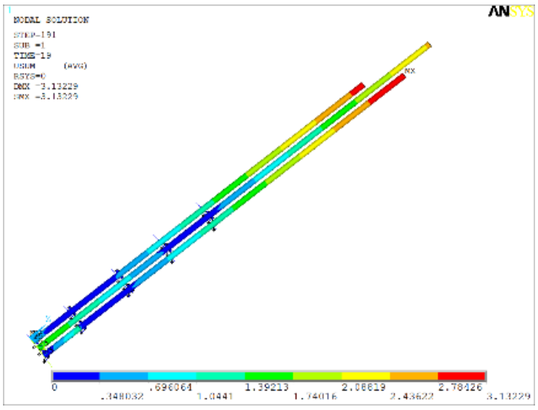

Fig. 7 The pipeline integral displacement image

Three nodes are selected respectively from the maximum displacements of the top of the pipes, node number was 52114, 62354, 36883. After the time course of post-processing, the time varying curves of X direction displacement of these three nodes are obtained, as shown below:

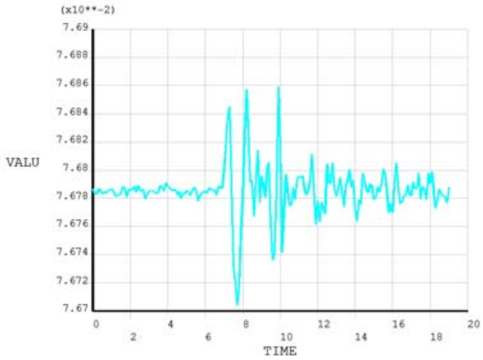

Fig. 8 The time variation curve of X direction displacement of node 52114

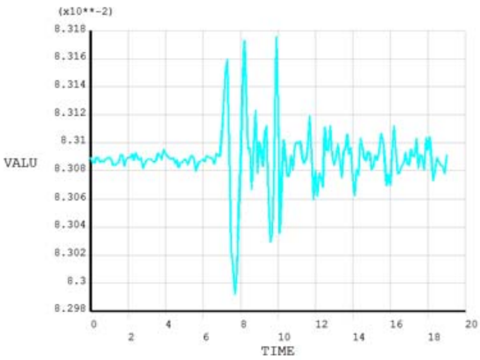

Fig. 9 The time variation curve of X direction displacement of node 62354

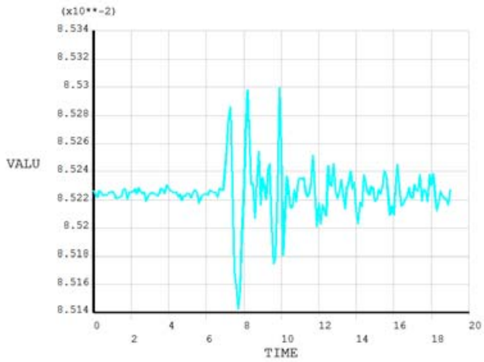

Fig. 10 The time variation curve of X direction displacement of node 36883

Stress Calculation Results. The overall stress of the pipeline is shown in Figure 11. It can be seen that the stress distribution in the pipe itself is relatively balanced, the main stress is concentrated at the fixed support of each layer, and the maximum stress occurs at the fixed position of the middle of the third layer, as shown in Figure 12.

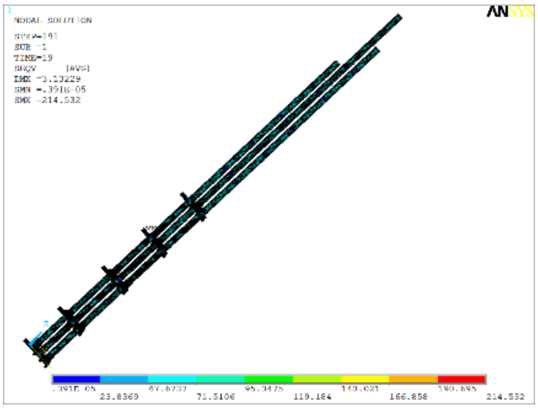

Fig. 11The overall stress of the pipeline

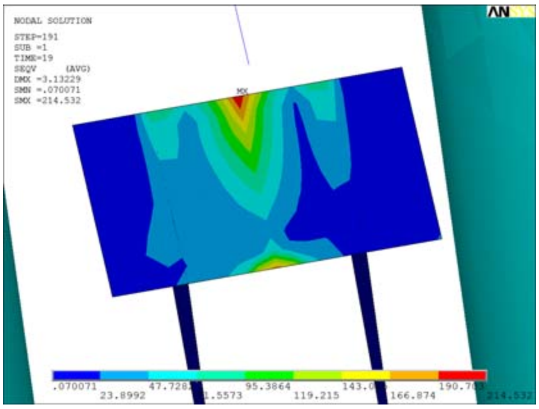

Fig. 12 The position of maximum stress

Conclusions

Through the finite element modeling of the medium pressure pipeline structure, the time history response of pipeline under seismic load is calculated by using transient dynamic analysis. The displacement and stress results and the time history curves are obtained. The maximum vibration of the pipe occurs at the top of the pipe. Maximum stress occurs at the fixed stent in the middle of the third layer. In the seismic design of the pipeline, the design of the allowance at the top of the pipe and the strength of the bracket is to be taken into account.

References

[1] ARIMAN T, MUI, ESKI G.E. A Review of the Response of Buried Pipelines Under Seismic Excitations. International Journal of Earthquake Engineering and Structure Dynamics, 1981.9(2):133-152(in Chinese).

[2] HINDY A, NOVAK M. Earthquake Response of Underground Pipelines. International Journal of Earthquake Engineering and Structure Dynamics, 1979,7(5):451-476(in Chinese).

[3] X.J. Shang, F. Qiu, H.F. Zhao. ANSYS Structure Finite Element Advanced Analysis Method and Case Application. Beijing: China Water Conservancy and Hydropower Press, 2008, 219-220(in Chinese).

[4] Y.F. Huang, J. Zhang, X.X. Ding. Transient Dynamics Analysis of Different Types of Welded Metal Bellows Based on ANSYS. Chemical machinery, 2011, 562(in Chinese).

[5] W.W. Yu, H.J. Chen, Y.P. Wang, B.J. Gao. Application of ANSYS in Mechanical and Chemical Equipment. Beijing: China Water Conservancy and Hydropower Press, 2007, 226(in Chinese).