Research Journal of Applied Sciences, Engineering and Technology 4(18): 3415-3424, 2012
ISSN: 2040-7467

Submitted: April 16, 2012 Accepted: May 18, 2012 Published: September 15, 2012

Finite Element Analysis on the Main Frame of Horizontal Type High-pressure Grouting Machine

Zhicheng Huang
College of Mechanical and Electronic Engineering, Jingdezhen Ceramic Institute,
Jingdezhen 333000, China

Abstract: For understanding the stress, strain and dynamic characteristics and improving the stability of the main frame of the horizontal type high-pressure grouting machine, took a type of ceramics for daily use horizontal high pressure grouting machine's main frame as the object of study, established its CAD model and then conducted finite element static analysis and modal analysis on it. Got the values and distribution of the stress and strain, obtained the first six order natural frequencies and the corresponding vibration modes of the main frame based on the theory of finite element and vibration mechanics and the influence of each vibration mode to the working condition of the main frame was analyzed. The analysis results can provide a reference for some reference to improve and enhance the design of the main frame of the main frame of horizontal type high-pressure grouting machine and have a large general engineering and practical value.

Keywords: Ceramics for daily use, finite clement analysis, frame, high-pressure grouting machine, horizontal type, modal analysis, modeling

INTRODUCTION

High-pressure grouting machine is widely used in molding sanitary ceramics and ceramics for daily use products with complex shape such as fish plates. Nowadays, most of the frame structure of the high-pressure grouting machine is horizontal type and it is the installed base of the other components. The front and back module plates open and locking model mechanism and fine-tuning mechanism and other components are installed on the main frame. The main frame works under a high grouting pressure and associated with the vibration. As one of the main forced components of the high-pressure grouting machine, its force and deformation must be considered in the design and if the vibration is serious, it will weaken the work ability of high-pressure grouting machine; impact the accuracy of products and the life of the mold and at the same time it will have adverse effects on the environment and human health. Therefore, to accurately predict the overall performance of the main frame, Kinetic analysis must be done on the main frame of the high-pressure grouting machine. Xiao (1982) have a research of the introduction of high-pressure grouting technology and equipment. Yinnan *et al*. (2006) have a research of the local and parallel finite element algorithms for the Navier-Stokes problem. He *et al*. (2006) and Ma *et al*. (2007) make a research of the local and parallel finite element algorithms based on two-grid discretization for steady Navier-Stokes equations. Xun and Yongsheng (2004) analyze the stiffness and modal finite element analysis and optimization on frame. Ye (2004) and Wang (2007) have a study of the ANSYS engineering analysis software application examples. Wensheng (2002) study the application of the sub-model meth-od in the engine structure strength analysis. Pradlwarter and Schueller (2005) study the consistent Concept for High and Low Frequency Dynamics Based on Stochastic Modal Analysis.

In this study, the finite element analysis software ANSYS is applied to the static analysis and modal analysis of the main frame, the values and distribution of the stress and strain and first six order natural frequencies and the corresponding vibration modes of the main frame are obtained. The influence of each vibration mode to the working condition of the main frame is analyzed and provides some reference to improve and enhance the design of the main frame of horizontal type high-pressure grouting machine. At the same time it laid the foundation for researching main frame vibration, noise and other issues.

MATERIALS AND METHODS

The finite element static analysis on the main frame of high-pressure grouting machine:

The technical conditions of the main frame: The material of the main frame is 45 steel and its characteristic coefficients and mechanical properties are as follows: mass density: 7800 kg/m^3; elastic modulus: 2 e+11 Pa; Poisson's ratio: 0.32; yield limit: 355 MPa; limit strength: 600 MPa. The maximum working pressure is 800 kN.

Modeling: This study established the 3D model of the main frame by suing CAD software Pro/Engineer and then imports it into ANSYS program to analysis through the ANSYS Data Access module (DDA). Figure 1 is the three-dimensional model of the main frame.

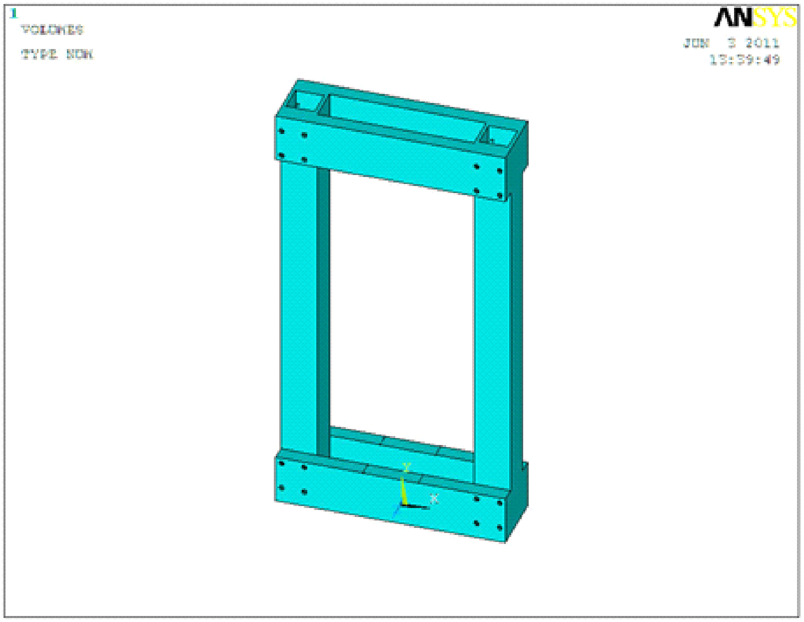

Fig. 1: Three-dimensional model

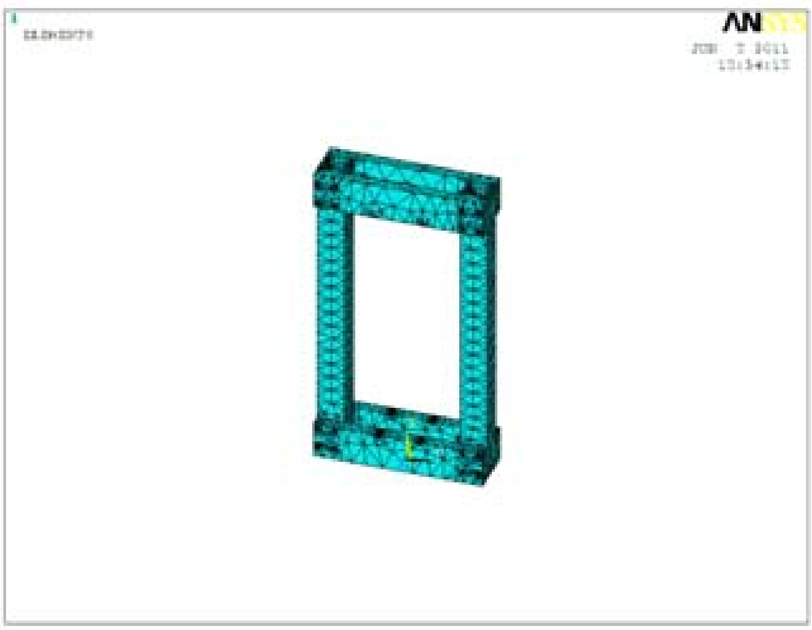

Fig. 2: The finite element mesh model

Meshing: When meshing, the unit selected is Solid92 unit and the method used is free mesh method. The size of the mesh is controlled by "Smart Sizing" command, which can automatically generate reasonable unit shapes and the meshes where stress concentration may be produced are meticulously meshed. Figure 2 is the finite element mesh model of the main frame.

Constraints imposed: The main frame is placed on the support rack and the left end is fixed in the horizontal direction, the right end is free in the horizontal direction. Therefore, on the left contact surface of the main frame and the support rack, its all freedom is constrained, but on the right contact surface, its freedom in the vertical direction is constrained and in the horizontal direction is free.

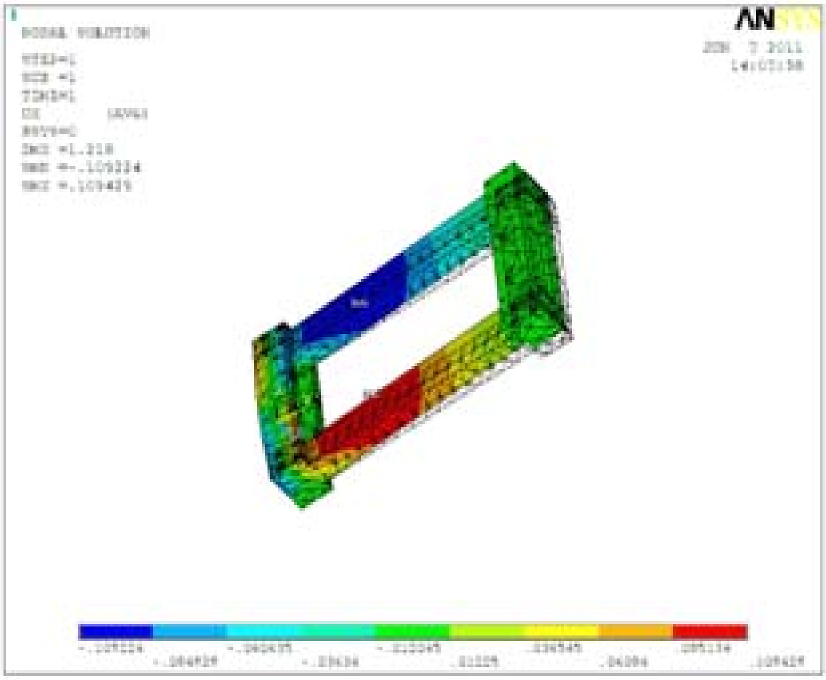

Fig. 3: The X direction displacement nephogram

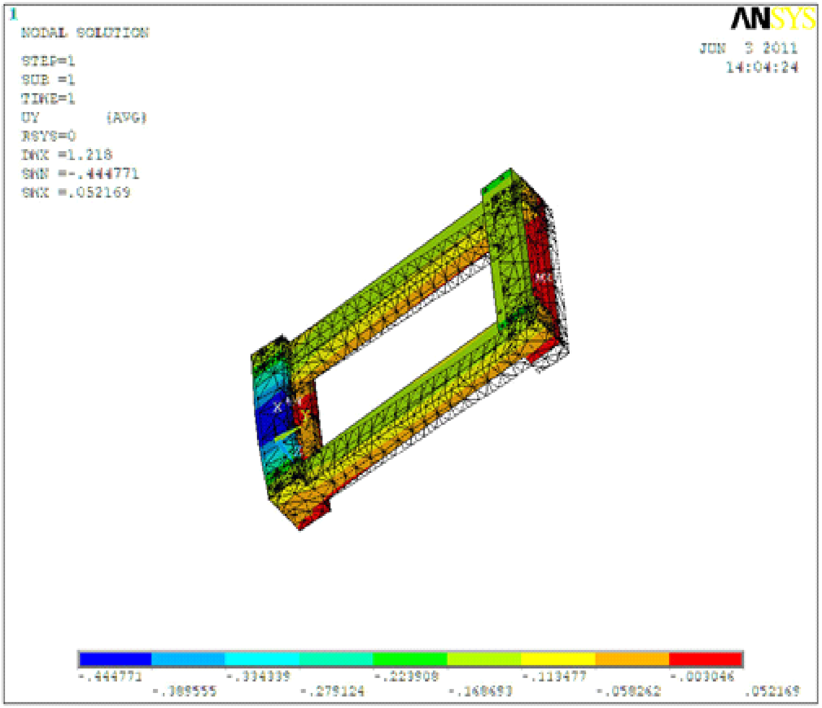

Fig. 4: The Y direction displacement nephogram

Loading: When the high-pressure grouting machine works, the maximum load is 800KN. This load is imposed on both columns of the main frame through two Bearing boxes. Therefore, according to the contact area of the two bearing seats and columns, 800KN force is transformed into surface loads and imposed on the contact surfaces of the bearing boxes and the columns.

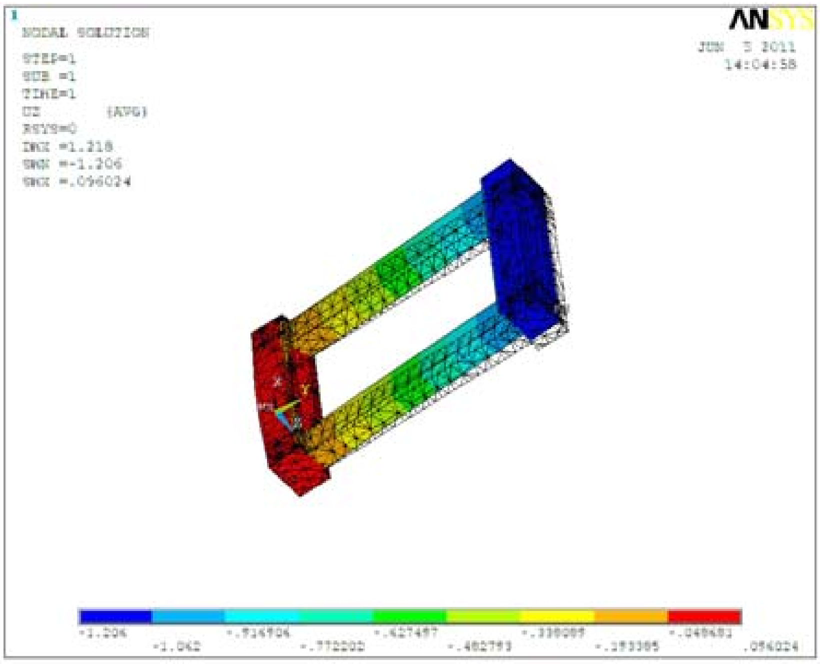

Fig. 5: The Z direction displacement nephogram

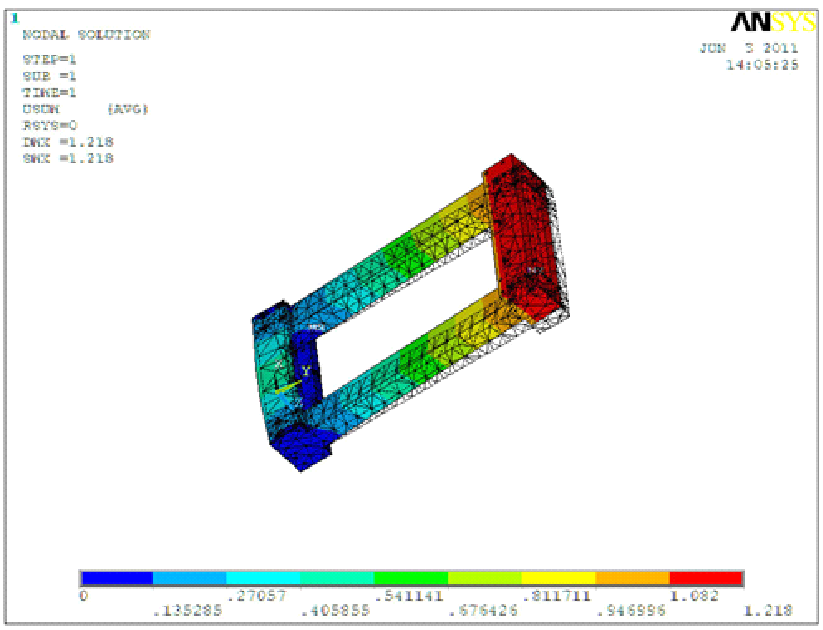

Fig. 6: The total displacement nephogram

Solving: To obtain very accurate solutions, we select "the Frontal Solver" method.

STATIC ANALYSIS RESULTS

Deformation distribution: Figure 3, 4, 5 and 6 are respectively showed the X direction, Y direction, Z direction and total displacement nephogram of the main frame when it withstands the maximum load. The maximum deformation of the main frame is in the middle of the columns where there the bearing houses contact with the columns and the value is 1.218 mm. In addition, the displacements of X direction, Y direction and Z direction are respectively 0.109, 0.05 and 0.09 mm and

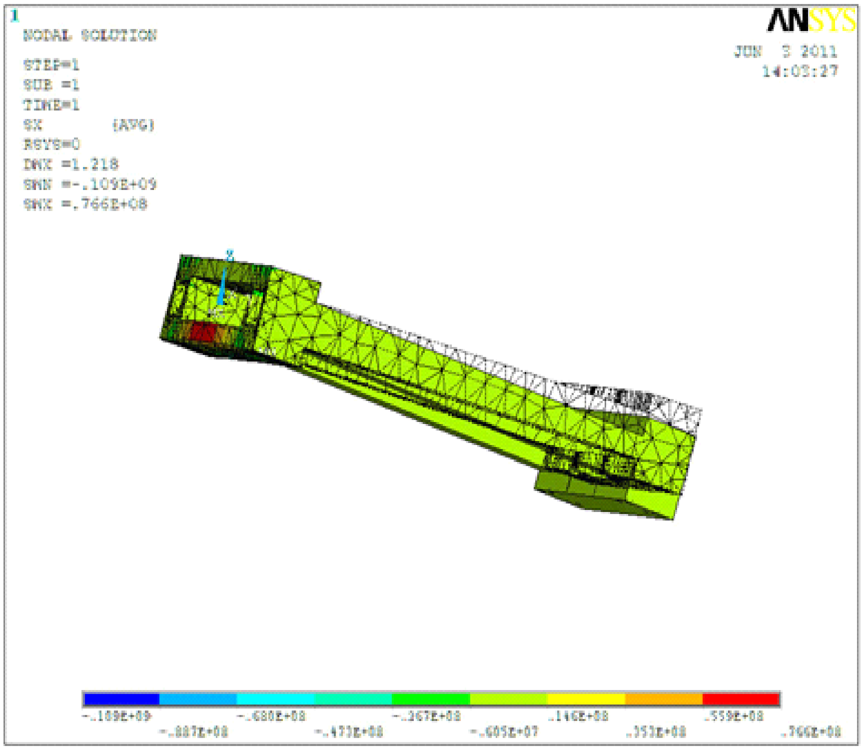

Fig. 7: The X direction stress nephogram

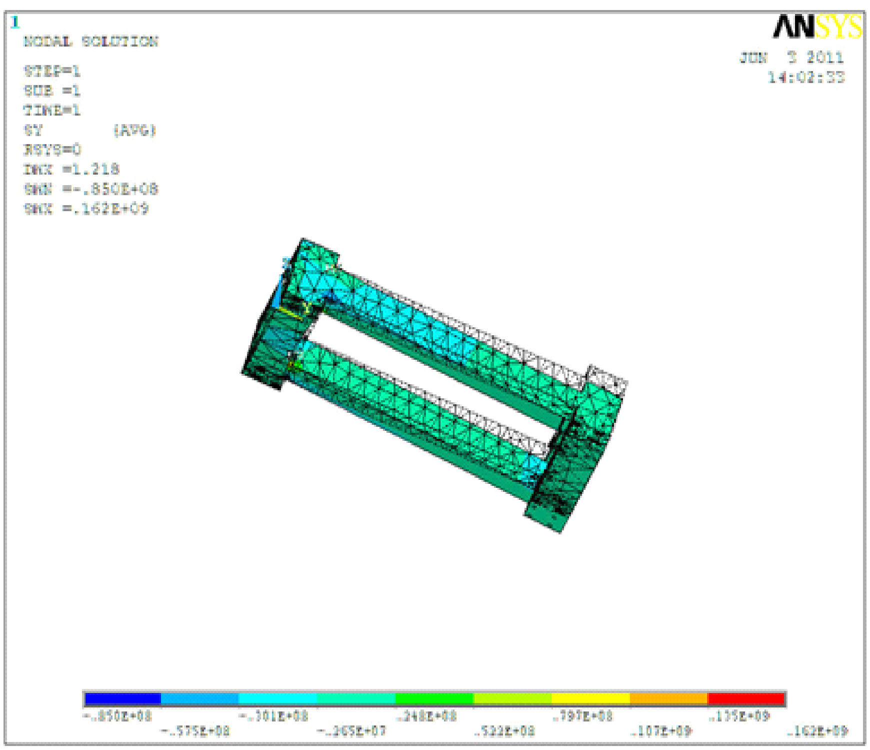

Fig. 8: The Y direction stress nephogram

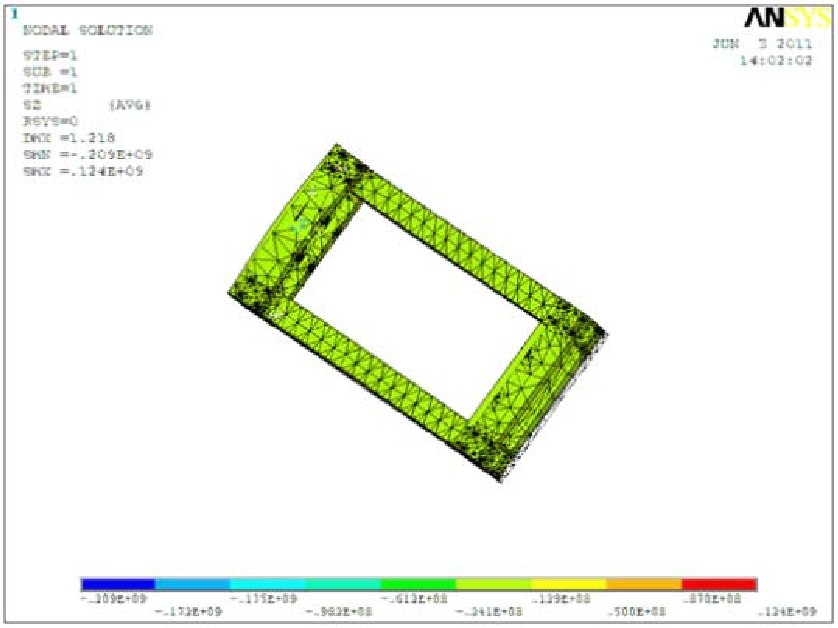

Fig. 9: The Z direction stress nephogram

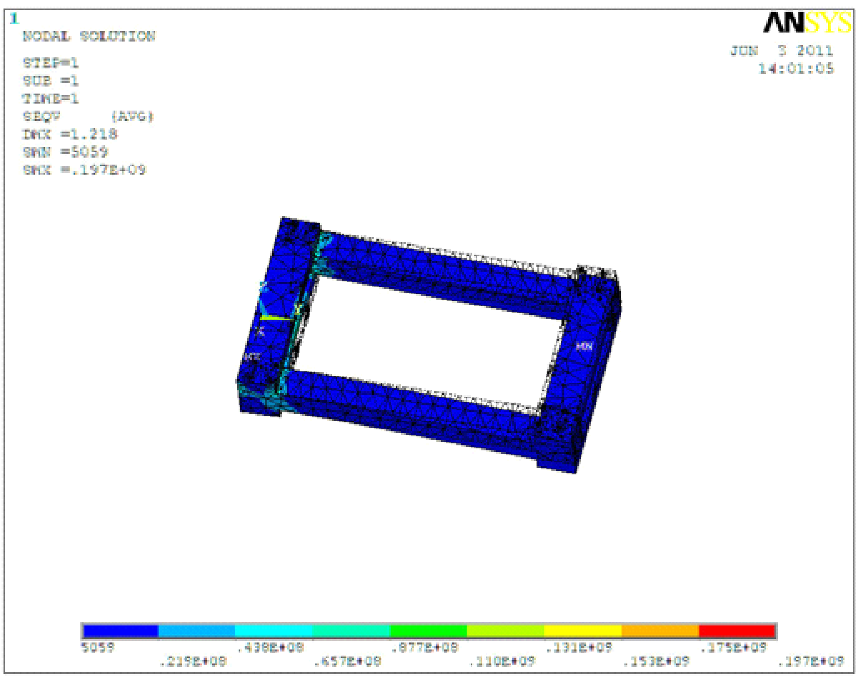

Fig. 10: Thevon-mises stress nephogram

they are all located in the midpoint of the column. All displacements are all very small and it shows that the stiffness of the main frame meet the design requirements.

Stress distribution: Figure 7, 8, 9 and 10 are respectively showed the X direction, Y direction, Z direction and the von-mises nephogram of the main frame when it withstands the maximum load. The von-mises stress of the main frame is in the middle of the columns where there the bearing houses contact with the columns too and the value is 197 MPa. In addition, the stresses of three directions are respectively 76.6, 162 and 124 MPa,

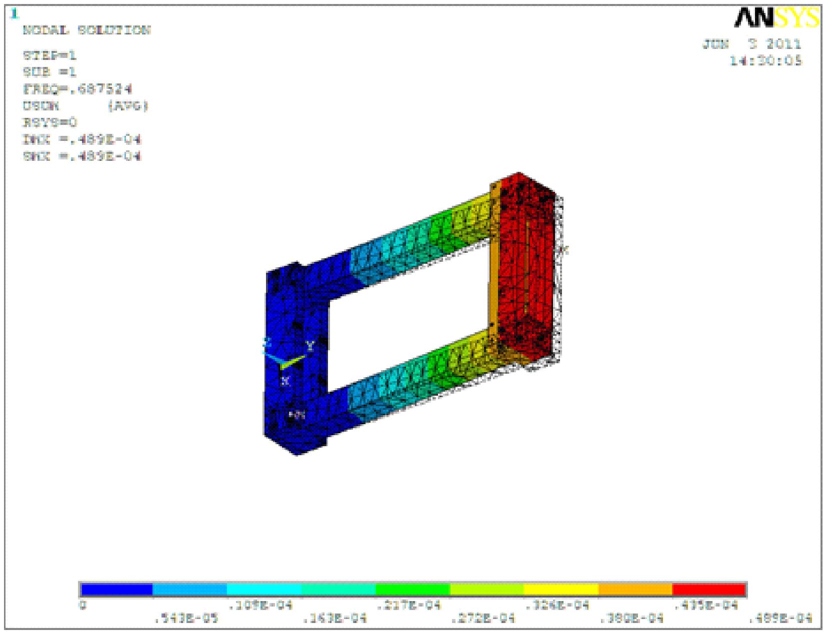

Fig. 11: The firstorder vibration mode

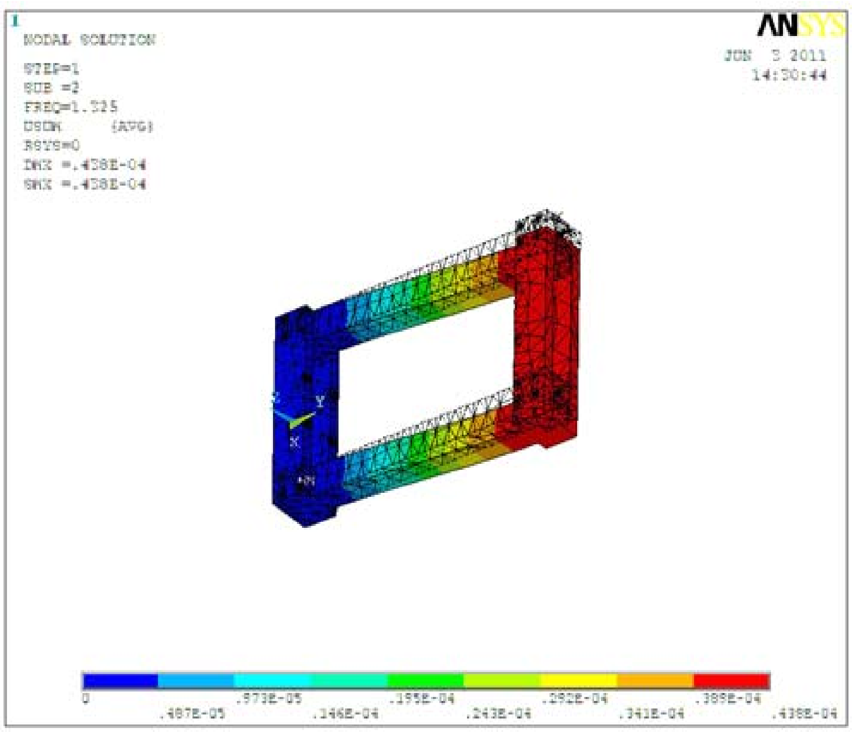

Fig. 12: The second order vibration mode

respectively they are all less than the value of von-mises stress. This shows that the strength of the main frame meet the design requirements.

Finite element modeling of high-pressure grouting machine main frame:

The determination of the calculated frequency: The

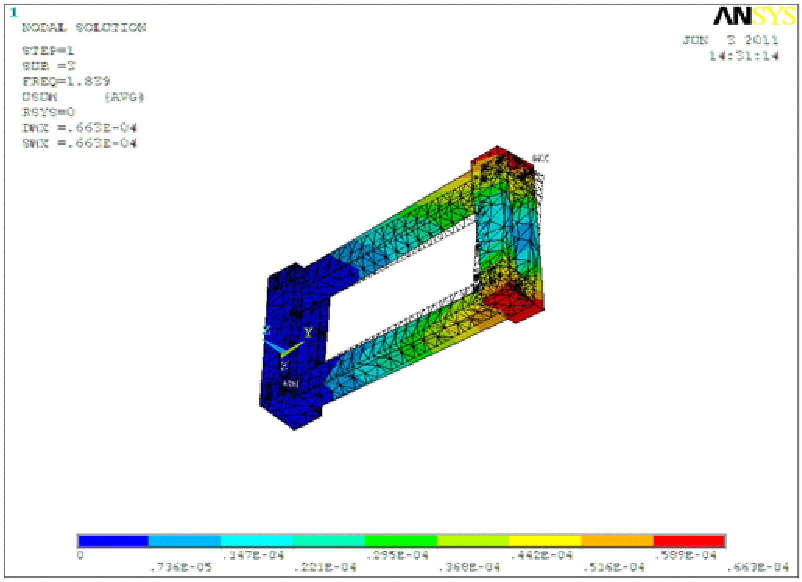

Fig. 13: The third order vibration mode

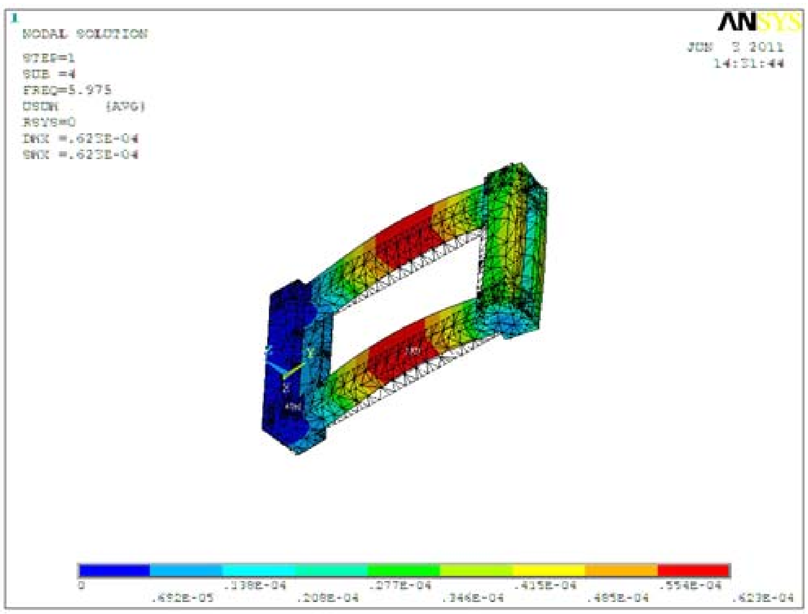

Fig. 14: The forth order vibration mode

selection of the calculate frequency range refers to in which band to solve and extract mode when calculate. In the structural dynamic analysis, the weighting factor size of each mode is proportional to the reciprocal of the modal frequencies, that is, the lower the frequency the greater the weight, which means that the lower modes the basic characteristics determine the structure of dynamic performance. For these reasons, when research the response of the system, often only need to know a few lower Characteristic value and corresponding characteristic vector. According to the principle of the highest frequency of the modal extraction is at least two times of the analysisfrequency, in the range of 0 ~ 100 Hz, extract the first six modes.

Solving the intrinsic mode: In this study, Lanczos method is selected to solve. After calculation, we get the first six natural frequencies and corresponding vibration modes. The first six natural frequencies are shown in Table 1 and each vibration type shown in Fig. 11-16.

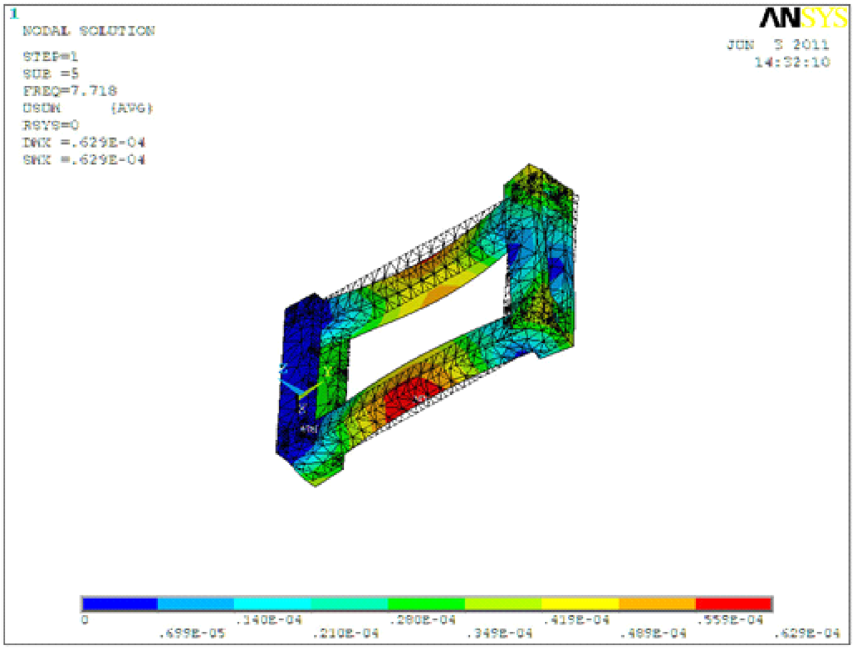

Fig. 15: The fifth vibration mode

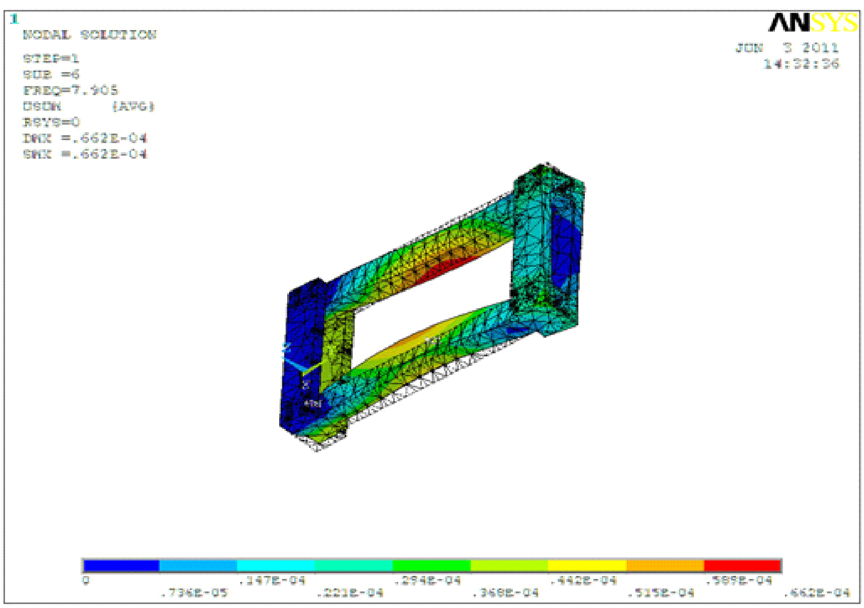

Fig. 16: The sixth order vibration mode

Table 1: The natural frequency of the frame

Order	Natural frequency (Hz)	Corresponding vibration modes
1	0.68752	Front and rear swing of the frame
2	1.32510	Top and bottom swing of the frame
3	1.83900	Torsion of the frame
4	5.97410	Bending of the frame
5	7.71770	Bending plus torsion of the frame
6	7.90540	Bending plus torsion of the frame

CONCLUSION

- The maximum deformation of the main frame is in the middle of the columns where there the bearing houses contact with the columns and the value is 1.218 mm. In addition, the displacement of three directions are all less than 1mm. This shows that the stiffness of the main frame meet the design requirements.

- The von-mises stress of the main frame is in the middle of the columns where there the bearing houses contact with the columns too and the value is197 MPa. In addition, the stresses of three directions are all less than the value of von-mises stress. This shows that the strength of the main frame meet the design requirements.
- All the stress and strain are in the design range, so the design of the main frame meets the work requirements.
- The minimum natural frequency of the main frame is 0.68752 Hz, which is much larger than the operating frequency (0.3 Hz) of the high-pressure grouting machine; therefore, the frame will not produce resonance.
- The minimum natural frequency of the main frame is less than 20 Hz, which is in the scope of the infrasound (Less than 20 Hz), therefore, The main frame will produce noise pollution.
- The main frame not only has front and back, up and down, left and right bending vibration, but also distort vibration. These vibrations will affect the strength and stiffness of the main frame and then affect the life of the ceramics for daily use horizontal type high-pressure grouting machine. Therefore, in the design process, shall appropriately increase the local stiffness and damping to suppress the effects of these vibrations.

REFERENCES

He, Y.N., J.C. Xu and A.H. Zhou, 2006. Local and parallel finite element algorithms for the Navier-Stokes problem. J. Comput. Math., 24(3): 227-238.

Ma, F.Y., Y.C. Ma and W.F. Wo, 2007. Local and parallel finite element algorithms based on two-grid discretization for steady Navier-Stokes equations. Appl. Math. Mech. Engl. Ed., 28(1): 27-35.

Pradlwarter, H.J. and G.I Schueller, 2005. A consistent concept for High and low frequency dynamics based on stochastic modal analysis[J]. J. sound and Vib. 88(3): 653-667.

Tinnan, H., X. Jinchao and Z. Aihui, 2006. Local and parallel finite element algorithms for the Navier-stokes problem. J. Comput. Math, 24(3): 227-238.

Wang, H., 2007. Modal analysis on Light truck frame based on ANSYS. Coal Min. Mach., 3: 59-61.

Wensheng, Z., 2002. Application of the sub-model method in the engine structure strength analysis [J]. Arms Plant Trans., 23(2): 270-272.

Xiao, Y., 1982. Introduction of High-Pressure Grouting Technology and Equipment. Jingdezhen Ceramic Design Institute, China.

Xun, M. and S. Yongsheng, 2004. Stiffness and modal finite element analysis and optimization on frame. Bus Technol. Res., 4: 8-11.

Ye, X., 2004. ANSYS Engineering Analysis Software Application Examples. Tsinghua University Press, 4.

Yinnan, H., X. Jinchao and Z. Aihui, 2006. Local and parallel finite element algorithms for the Navier-stokes problem. J.Comput. Math, 24(3): 227-238.