TELKOMNIKA, Vol. 11, No. 7, July 2013, pp. 3863 ~ 3873
e-ISSN: 2087-278X

Contact Analysis on The Whole Frame of 32.8MN PRESS

Zhicheng Huang, Nanxing Wu, Xingguo Wang
College of Mechanical and Electronic Engineering, Jingdezhen Ceramic Institute
Jingdezhen, 333001, China
*Corresponding author, e-mail: huangwu555@sina.com

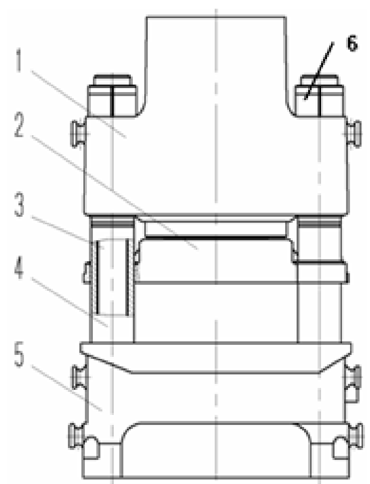
Abstract

Automatic hydraulic press is one of the key Ceramic machinery and equipments for pressing ceramic wall and floor tiles, and its frame withstands all of the work force when it works, so the work performances of the frame have an important impact on the structural performance of the entire automatic hydraulic press. Took the frame of 32.8MN automatic hydraulic press as research object, established its three-dimensional model through Pro/Engineer software and carried on contact finite element analysis for it by using the large commercial finite element software ANSYS to gain the stress and strain distribution of the frame, and then carried out a detailed analysis of the results. The work done can help to fully understand the structure properties of existing automatic hydraulic press and serve as guidance to further improve its performance, and at the same time provide a theoretical basis for the structural design of the same type of automatic hydraulic press frame.

Keywords: *automatic hydraulic press, frame, contact, finite element analysis, static analysis*

1. Introduction

Automatic hydraulic press (Hereinafter referred to as PRESS) is one of the pressure processing equipment combining with mechanical, electrical, hydraulic and computer control technology, and it widely used for pressing ceramic wall and floor tiles [1]. There are many forms of its frame, but the most common form is three- beam four-column structure [2], and the 32.8MN PRESS studied in this paper is a typical representative of this form. Figure 1 is its structure schematic diagram. Figure 2 is its three -dimensional model.



1-upper beam; 2-active beam 3-column; 4-sleeve; 5-lower beam; 6-big nut

Figure 1. Schematic diagram of PRESS frame

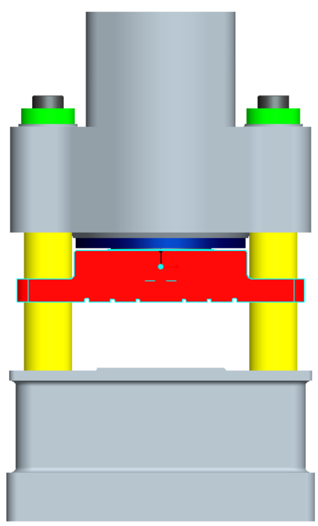


Figure 2. Three-dimensional model of PRESS frame

Received January 31, 2013; Revised April 20, 2013; Accepted May 8, 2013

The “three-beam” refers to the upper beam, active beam and lower beam; the “four-column” refers to the four upright columns with screw threads at both ends. They and the four sleeves and eight nuts together constitute the whole closed frame. When assembly, apply preload to the column, so the columns are stretched in full-length and the sleeves withstand pre-pressure. That the retightening force of the columns enables the upper and lower beams, sleeves and big nuts form the frame.

The frame is one of the important parts of the PRESS. It weighs about 50% - 60% of the whole weight, and when the PRESS works, it bearing almost all of the work loads, so its strength and stiffness play a decisive role in the entire PRESS security and the quality of products. Therefore, need to understand the stress and strain distribution of the frame and ensure sufficient strength and stiffness.

In traditional design, the structure parameters of frame are derived mostly from the experience of the designer or analogy with similar products, the design calculations fact only play a role in checking. The finite element analysis method is a kind of important assistant means of computer engineering, and it has been widely applied in engineering practice [3-4].For these reasons, the finite element analysis has become an important method in designing of large PRESS. Many scholars carried on finite element analysis to the frame of PRESS [5-9]. This paper took the frame of 32.8MN automatic hydraulic PRESS as research object, established its three-dimensional model through Pro/Engineer software and carried on contact finite element analysis for it by using the large commercial finite element software ANSYS. The results can provide guidance to further improve frame’s performance, and at the same time provide a theoretical basis for the structural design of the same type of automatic hydraulic PRESS frame.

2. Finite Element Model Establishment of the 32.8MN PRESS Frame

There are two general ways for establishing the finite element model: the first one is directly creating method, which can be created directly in ANSYS or read directly into the node unit data generated by other finite element program; the second one is automatically meshing creating method, which is the first to establish the entity model, and then define elements properties, mesh and generate nodes and elements data [10-11]. The former applies to the mechanical structure system with simple geometry shape; the latter is very useful for complex systems, especially complex three-dimensional system. The frame of the PRESS has bulky volume and complex structure, so use the automatically meshing method.

2.1. Simplification of the Frame Model

Before finite element analysis, first need to convert the object model into structural analysis model or mechanical model to facilitate the analysis. In this conversion, it is very important to simplify the analysis object, and ensure that the main structure mechanical properties of the original analysis object do not change. So according to the frame structural features and working conditions, its structure model was simplified as follows:

(1) Assume that the PRESS frame is a constant linear system, and ignore the influence of damping;

(2) The materials of the frame are isotropic material with uniform density, and the frame is a fully elastic body;

(3) The Materials, loads and physical shape of the frame have symmetry, in order to save computer resources, the study object can be taken as 1/4 of the entire model;

(4) Because obviously does not affect the frame overall strength and stiffness, the characteristics such as threaded holes, pin holes, some mesas of the upper beam and the lower beam can be simplified.

(5) Assuming there is no movement interference between the active beam and columns, so in the process of analysis, the active beam and piston can be removed.

2.2. Solid Modeling

The entity model of the PRESS frame is established by Pro / ENGINEER (showing in figure 2), after simplified (remove active beam and take 1/4 of the entire model), input it to ANSYS.

2.3. Element Choosing

When establish the finite element model, need to select the appropriate finite elements to simulate the actual structure, and how to select the appropriate elements type is one of the key issues of the finite element analysis. In this paper, the SOLID92 is used to simulate the entity structure; TARGE170 and CONTA174 simulate the contacts, and PRETENSION179 simulate pretightening force [12-14].

2.4. The Material Properties of Each Portion of the Frame

The materials of the upper beam and lower beam are steel castings- ZG270~500, its mechanical properties are as follows: elastic modulus $E = 175GPa$, Poisson's ratio $\mu = 0.28$, density $\rho = 7.8\,g/cm^3$, $\sigma_s = 265MPa$, $\sigma_b = 485MPa$, $\sigma_{-1} = 155MPa$. The sleeve material is 45 steel and its mechanical properties are as follows: elastic modulus $E = 194GPa$, Poisson's ratio $\mu = 0.28$, density $\rho = 7.85g/cm^3$, $\sigma_s = 355MPa$, $\sigma_b = 600MPa$. The material of the columns and nuts are 35CrMo and its mechanical properties are as follows: elastic modulus $E = 207GPa$, Poisson's ratio $\mu = 0.29$, $\sigma_s \geq 835MPa$, $\sigma_b = 924MPa$, $\sigma_{-1} = 423MPa$, density $\rho = 7.9\,g/cm^3$.

2.5. Meshing of Solid Model

Meshing is another important part of finite element analysis, and the number, density and quality of the mesh should be all taking into account. The number of the mesh will affect the accuracy and size of the computing. In general, if increase the mesh number, the calculation accuracy will be improved, but simultaneously the calculating size will increase accordingly. So both of the two factors should be considered when determine the number of mesh. This paper used the automatic meshing function to mesh, and manually set the mesh size of the column, sleeve, upper beam, lower beam and nuts respectively. The upper beam was divided into 20833 nodes, 19284 elements; the lower beam was divided into 37848 nodes, 33,920 elements, the sleeve was divided into 1306 nodes, 2028 elements; the column and nuts were divided into 7623 nodes, 7906 elements. Figure 3 is the mesh model of the 1/4 frame.

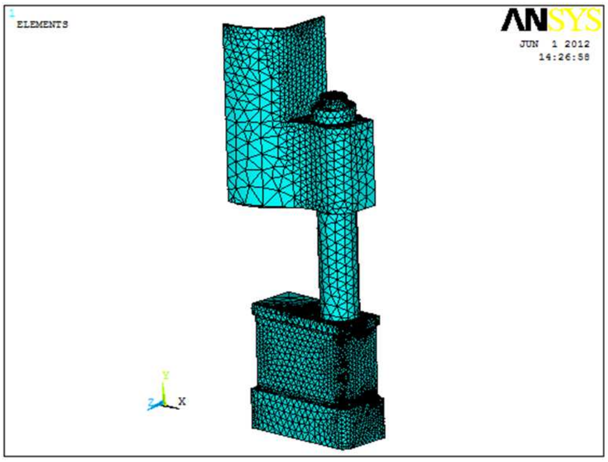


Figure 3. The mesh model of the 1/4 frame

2.6. Contact Problems in Components of the Frame

After meshing, the handling of the connecting parts is much more trouble. Because in the deformation process, some of the corresponding surfaces of the nut and the upper beam, the nut and lower beam, the nuts and the column, the sleeve and the two beams as well as the column and sleeve may be contact, while else are separate, so need to use the finite element contact analysis technology to analyze. Because the upper and lower beams, column, sleeve and nuts have similar stiffness, it can be considered that the contacts between them as flexible body-flexible body contacts, and all of them are face -- face contact problems.

According to the structure and the working state of the frame, defined four contact pairs for it and respectively gave four different real constant numbers for each contact pair. The four contact pairs respectively are: the contact of upper beam and nut, the contact of upper beam and sleeve, the contact of sleeve and lower beam, and the contact of the lower beam and nut. The thread connections of the column and nuts were handled by bonding in the course of the study, and it is reasonable to do so for the analysis of the entire frame. (The contact analysis of the thread connections of the column and nuts is very complex, and the author will discuss it in another paper in detail). The contacts of the column and sleeve, upper beam and lower beam did not use contact pairs, because there are a certain gaps between them, even if establish contact pairs for them, it would be very difficult to converge, and the X, Y, Z direction displacements of them are limited by the other members, there will not occur rigid displacement. Taking into account the actual work process, the horizontal displacements between the two beams and column as well as nuts are limited, so simply setting the coefficient of friction can not limit its displacement, and so in the friction model choose “rough” contact type, which considers that the frictional resistance is infinite. In the contact simulation process, use TARGE170 to simulate 3-D target surface, and CONTA174 to simulate the3-D contact surface.

2.7. Create Preload Force Element

The frame is a sleeve-tie rod type structure, and the tie rod (column) is preload in full-length. Under the preload force, the column is in the state of tension, and the upper beam, lower beam and sleeve is under pressure. ANSYS provides a preload force element PRETS179 and “PTSMESH” pre-stretch meshing operating to simulate column preload force. Use 3-D physical structure to descript each of the frame components, and the preload force element PRETS179 simulates a pre-tensile-section in the axial center surface of the column. The easiest method for applying preload force element on the column is using PSMESH command to definite preload force-section, and generate preload force element. It automatically divides the meshed column into two parts, and inserts the preload force elements.

3. Applying Boundary Conditions

The bottom of the lower beam is fixed on a rigid foundation, so constrain the underside of the lower beam, that is to say, its three directions displacements are all constrained to zero. In addition, based on the symmetry of the frame, took 1/4 of the entire model as study object, so it is necessary to constraint the normal direction displacements of the symmetry planes. The model applied boundary conditions is shown in Figure 4.

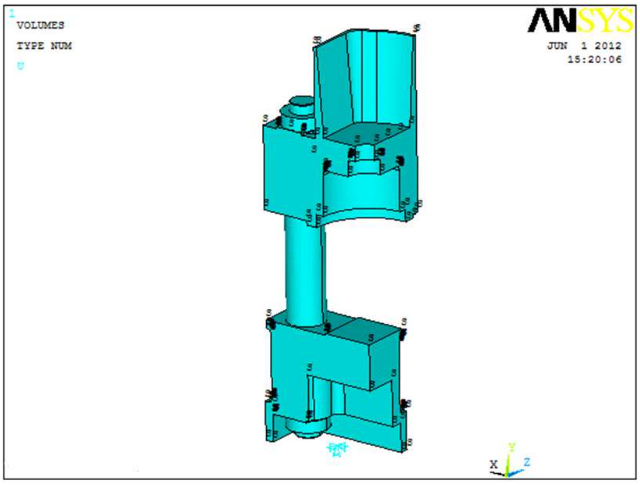


Figure 4. The model applied boundary conditions

4. Loading

4.1. Applying Pretension Load

The preload application method of this type frame is respectively stretching both ends of the column by the tension jack, so that the mating portion length of the upper beam, lower beam, sleeve and column will produce trace reduction, and then tighten them with nuts. When the tension jack uninstall, the column will produce a certain amount of preload, which can ensure that the contact surfaces will not separate due to force when the PRESS works. The total preload is generally taken as 1.2-2 times of nominal pressure, and for this 32.8MN PRESS, the preload is taken as 1.37 times of the nominal pressure. Because the model studied is 1/4 of the total frame model, so the preload is $1.37 \times 32.8 \div 4 = 11.25MN$. The pretension load is applied to the column through the preload force element.

4.2. Applying Working Load

When the PRESS works, the high-pressure oil goes into the cylinder and generates a high pressure on the cylinder wall, upper flange and the piston. The high pressure pushes the piston and active beam downward to suppress the tiles. Therefore, there are three categories loads on the upper beam: the first category is the high-pressure oil of the master cylinder acting evenly in the cylinder wall, and the pressure act on the side wall of the upper beam through the cylinder wall; the second category is the high-pressure oil of the master cylinder acting evenly on the upper flange, and then act on the surface of the upper beam; the third category is the gravity. This paper omitted the influence of gravity and in the process of study, removed the upper flange, cylinder and lower flange, which were replaced by the according surface loads acting on the sidewalls and the surface of the upper beam. In addition, there are two categories loads on the lower beam: the first category load is the gravity; the second category load comes from the piston, which is transmitted to the active beam through the piston, and then transmitted to the mold on the upper surface of the lower beam, finally acts on the upper surface of the lower beams through the mold bottom plane. So omitted the influence of gravity too, and in the process of study, removed the piston, active beam and mould, which were replaced by the according surface loads acting on the part of upper surface of the lower beam. The surface loads above were evolved from the nominal pressing force (32.8MN).

5. Solving

Use directly solving method (“current LS”) to solve.

6. Analysis of Results

In order to ensure the correctness of the contact finite element analysis results, it is necessary to first verify the reliability of pretension element method and the reasonable of the pretension coefficient.

6.1. The Reliability Verification of the Pretension Element Method

There are three methods to verify the reliability of the pretension element method.

In the ANSYS post-treatment process, selected any one of the engagement surface, and then summed the axial forces of all the nodes on the surface, the result was exactly the preload 11.25MN, which proved that the pretension element method can analog the preload well.

The column tensile stress σ caused by the preload force can be calculated by the formula (1),

$$\sigma = \frac{4Q_p}{\pi d} \quad (1)$$

In the formula, Q_p is preload, d is the column diameter. Substituting the known conditions of $Q_p = 11.25MN$, $d = 300mm$ to formula (1), and got $\sigma = 169MPa$. Figure 5 shows the axial stress of the column at preload condition. It can be seen from Figure 5 that in preload conditions the maximum axial stress of the column is $208.688MPa$, which distributes at the two ends where contacting with the nuts, but the other parts of the column belong to the same color area where the nodes axial stress values are almost all in constant 169MPa plus or minus, which are consistent with the calculated tensile stress value. This proved once again that the preload element method simulation preload is effective.

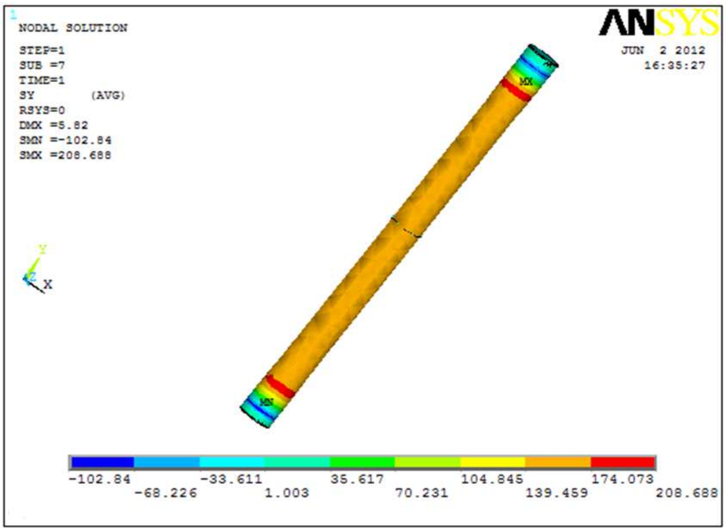


Figure 5. The axial stress of the column at preload condition

The sleeve compressive stress σ' caused by the preload force can be calculated by the formula (2).

$$\sigma' = \frac{4Q_p}{\pi d_2^2 - \pi d_1^2} \tag{2}$$

In the formula, Q_p is preload, d_1 is sleeve inner diameter and d_2 is sleeve outside diameter. Substituting the known conditions of $Q_p = 11.25MN$, $d_1 = 300mm$ and $d_2 = 400mm$ to formula (2), and got $\sigma' = 204.7MPa$. Figure 6 shows the axial stress of the sleeve at preload condition.

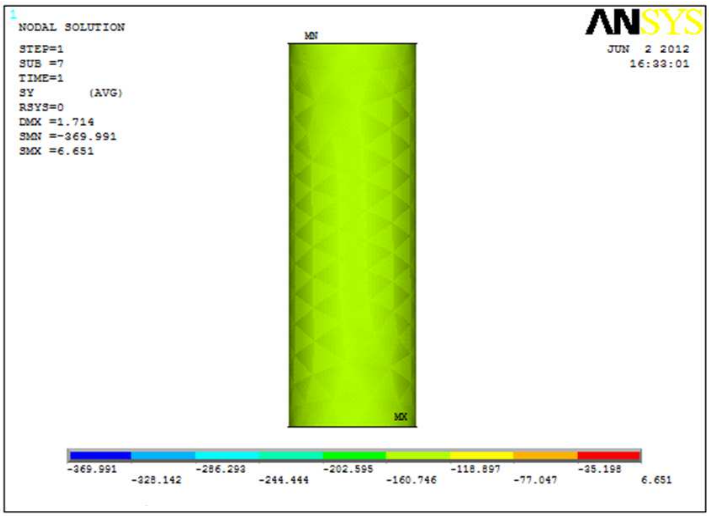


Figure 6. The axial stress of the sleeve at preload condition

It can be seen from Figure 6 that in preload conditions the maximum axial stress of the sleeve is $369.991MPa$, which distributes at the two ends where contacting with the upper beam and lower beam, but the other parts of the sleeve belong to the same color area where the nodes axial stress values are almost all in constant 202MPa plus or minus, which are basically the same with the calculated tensile stress value. The third time proved that the preload element method simulation preload is effective.

The three analysis methods above proved that the preload element method can simulate the preload well, this because in preload element method the preloads were directly loaded in the column, so it was less affected by the factors such as the number of elements.

6.2. Reasonable Verification of Pretension Coefficient

In the installation process, the columns and the upper beam, the sleeves and the lower beam are preloaded to ensures that they will not separated when the PRESS works, in addition, because of the presence of the preload, it will offset part of the work load, which can increase the carrying capacity and fatigue strength of the frame. Therefore, must apply sufficient preload force to the columns so that the contact surfaces of the frame members will not produce slits. Under normal circumstances, the preload is set by giving certain of preload coefficient. As mentioned in the previous that the PRESS total preload is generally taken to be 1.2-2 times of its nominal pressure, and the nominal pressure of the PRESS studied this article is 32.8MN, its total preload is 45MN, so the pretension coefficient is 1.37.

The pretension coefficient is reasonable? It can be verified by the axial stress on the contact surfaces between the sleeve and the beams. Figure 7 shows the axial stress of the sleeve at working condition, and from it we can see that the axial direction stress in the entire sleeve is compressive stress in the working conditions, and then the stress of the contact surface is compressive stress too, which indicates that the contact surface betweens the column and beams is not slit and respective contact surface is at good contact state. That is, the preload factor of 1.37 is reasonable.

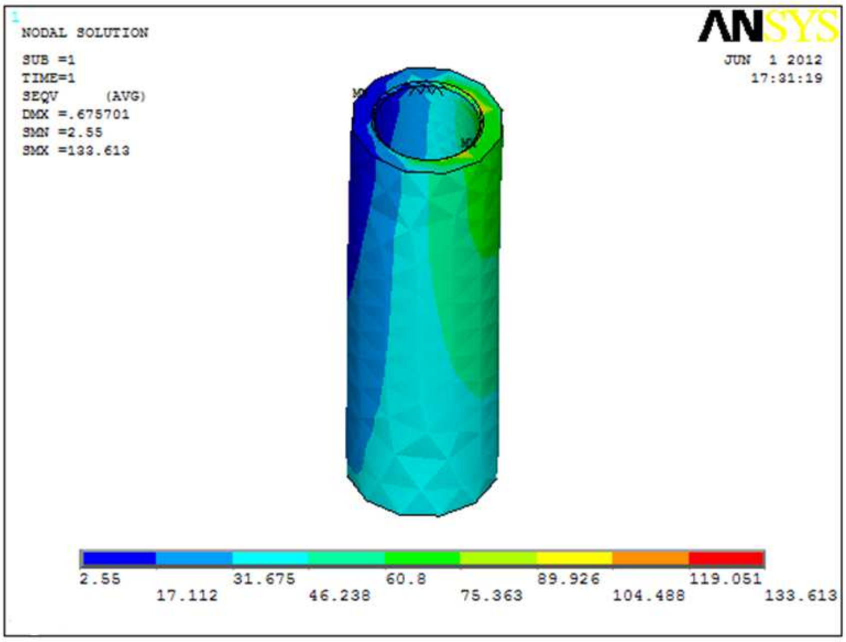


Figure 7. The axial stress of the sleeve in working condition

6.3. The Finite Element Results of the Frame

6.3.1. The Deformation of the Fame

In the working state, the deformation of the PRESS fame directly affects the performance, life and tile molding accuracy. Figure 8 shows the total displacement deformation of the frame in working condition. As can be seen from Figure 8, the maximum displacement of the frame occurs at the node in the middle of the column. After further examination, it is found that the node is the pretension node of the pretension element and the displacement of the node in the three directions of X, Y, Z are 4.14mm, 0mm and 0mm, and the values are the same in the preload conditions and working conditions. According to the basic knowledge of the pretension element, the pretension node has only UX direction displacement. The displacement values of the other nodes on the frame are far smaller than this node, and this node has not any contact with the displacement change of its surrounding nodes. So in the analysis of the displacement deformation, this node should be removed. After removal of the pre-tension node, the maximum displacement of the frame is still near the pretension element and its value is about 2.2mm.

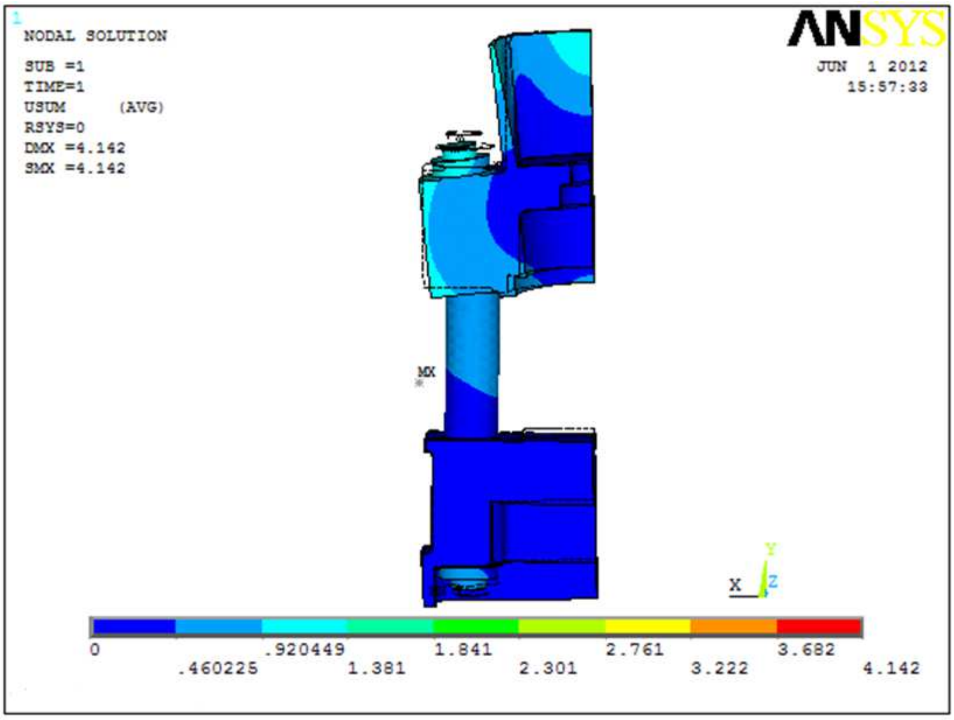


Figure 8. The total displacement deformation of the frame in working condition

Figure 9, Figure 10 and Figure 11 respectively show the X, Y, Z three directions of displacement deformation of the frame in the working condition. By comparing them, can found that except the pretension node has big X direction displacement, the other nodes' X and Z directions displacements are quite small, and they have little effect of the total displacement. The frame mainly has Y direction (axial) displacement and where the maximum axial displacement occurs is where the total displacement occurs.

Currently, the deformation allowable values for the various PRESS frame have no unified standard. Formula (3) is one of an empirical formula for calculating the deformation allowable value of the frame this article studied.

$$[f] \le (0.3 \quad 0.5) \times P(mm) \tag{3}$$

In the formula, $[f]$ is preload, P is the nominal pressure (MN) of the PRESS. Substituting P = 32.8MN, can get $[f] \le 9.84 \quad 16.4(mm)$, which is far greater than the frame deformation. Therefore, the stiffness of the entire frame can meet the design requirements.

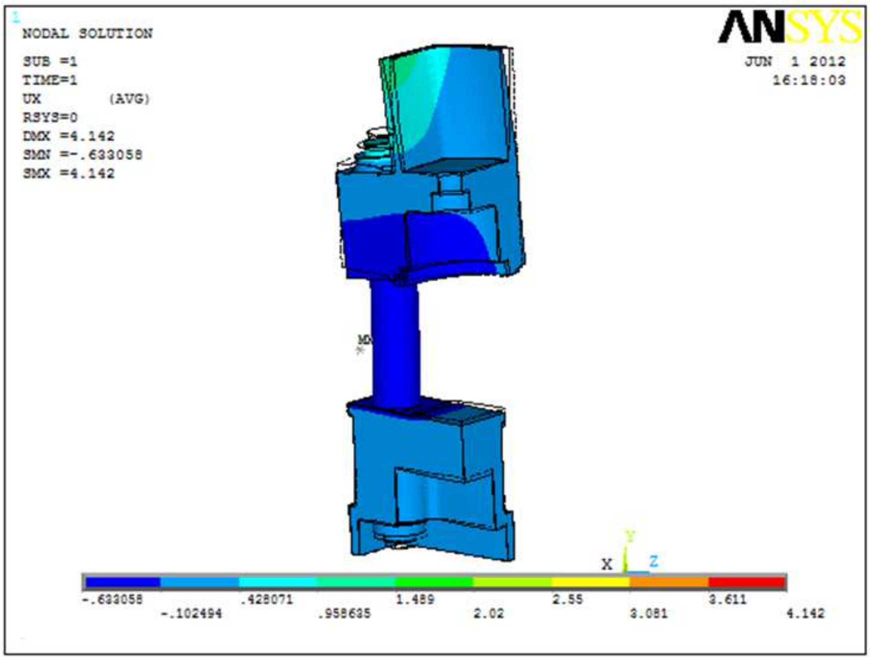


Figure 9. the X direction displacement deformation

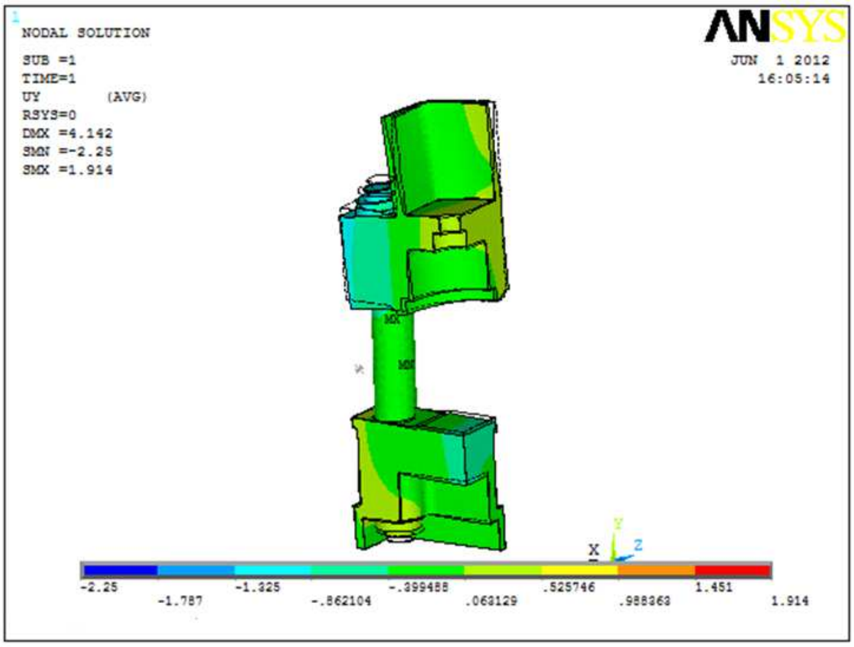


Figure 10. the Y direction displacement deformation

6.3.2. The Stress Distribution of the Fame

Figure 12 shows the von mises stress nephogram of the frame in working condition. It can be seen from Figure 12 that the maximum stress is 490.692MPa, which locates at the transition arc where the upper beam and nut contact. The stress distribution of the frame is substantially uniform and most of them are in the scope between 79MPa 150MPa. A handful of regional presence stress concentration. The transition arc of the upper beam and the contact surfaces between the nuts and beams are high-stress areas.

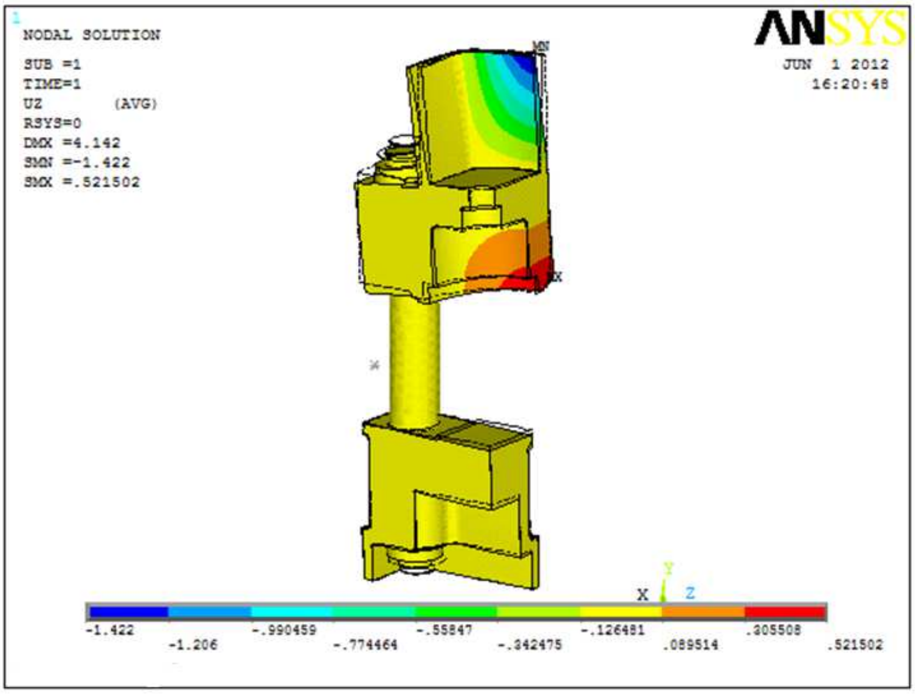


Figure 11. the Z direction displacement deformation

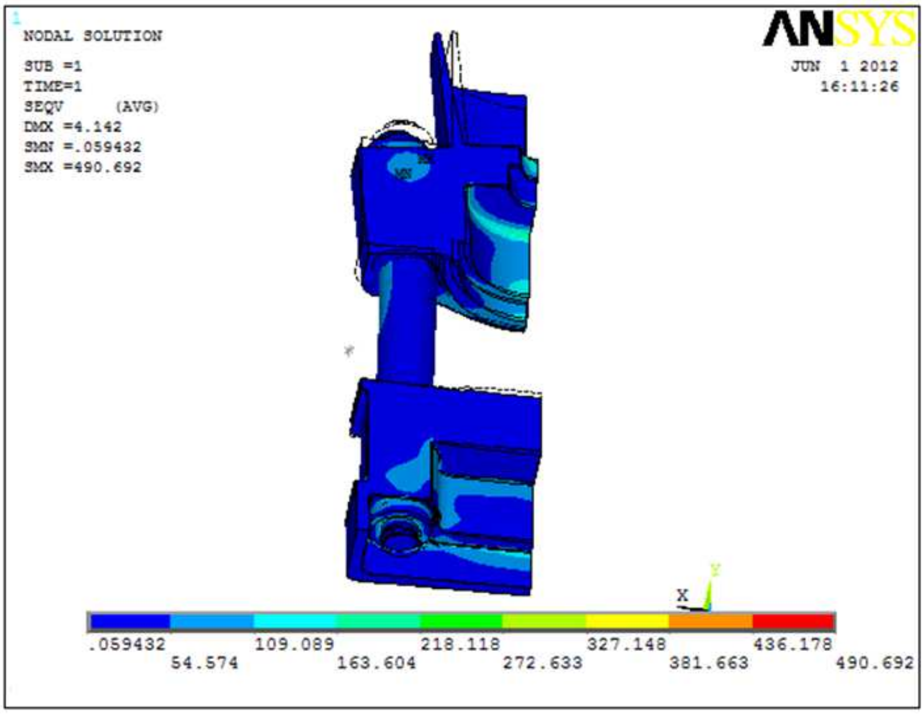


Figure 12. the von mises stress nephogram

4. Conclusion

(1) The pretension coefficient of the frame is reasonable;
(2) The preload element method can simulate the preload well;

(3) The maximum deformations of the frame are fare less than the allowable value, so the stiffness of the entire frame can meet the design requirements;
(4) The max von mises stress of the frame are far less than the materials yield limits in the preload condition and working condition, and there is a lot of optimization design space, can consider reducing materials and reduce costs.

Acknowledgements

The authors are grateful to the Natural Science Foundation of Jiangxi Province (NO. 20122BAB216006) and Science & Technology Pillar Program of Jiangxi Province (NO. 2011ZBBE50021) for supporting the research.

References

[1] Zhang BQ. *The ceramic industrial machinery and equipment*. Beijing: China Light Industry Press. 2012.
[2] Zhang BQ. *Automatic hydraulic press*. Nanchang: Jiangxi Science and Technology Press. 2001.
[3] Wang XW, Yang ZJ, Liu HJ. Finite Element Analysis on Double-Telescopic Prop of Hydraulic Support. *TELKOMNIKA.* 2013; 11(1): 270-279.
[4] Zhang GG, Sheng J, Wang F. A Finite Element Analysis of the Rectangle Spline Broach. *TELKOMNIKA*. 2012; 10(8): 2125-2130.
[5] Wang XA, He JS, Tan JC. The Structure Finite Element Modal Analysis of Full-automation Hydraulic Pressure Brick Machine Body. *Journal of Hunan Institute of Engineering*. 2007; 4: 34-37.
[6] Xiao RX, Han W, Xu ZH. Modal analysis of automatic hydraulic press of steel wire-wound. *Journal of china ceramic industry*. 2003; 10(2): 9-12.
[7] Huang ZC, Hu WW. The finite element analysis on the upper beam of automatic hydraulic press. *Journal of mechanical design and manufacturing*. 2007; 9: 116-118.
[8] Huang ZC, Duan JH, Hong C. The fatigue life research on the crossbeam of hydraulic automatic tile press. *Journal of China ceramic*. 2007; 43(6): 45-47.
[9] Wu NX, Cheng CQ, Huang ZC. The finite element study on the fatigue life of the ceramic hydropress top beam. *Journal of china ceramic industry*. 2011; 18(1): 20-22.
[10] Xian LY. ANSYS engineering analysis software application examples. Beijing: Tsinghua University Press, 2004
[11] NSYS.INC. Basic Analysis Procedures Guide. Release5.7
[12] NSYS.INC. Modeling and Meshing Guide. Release5.7
[13] SYS.INC. ANSYS Element Reference. Release5.7
[14] NSYS, Inc. ANSYS Advanced Technical Analysis Guide. 1998.