TELKOMNIKA, Vol. 11, No. 7, July 2013, pp. 3964 ~ 3972
e-ISSN: 2087-278X

Contact Analysis on Thread Connections of 32.8MN PRESS Columns

Zhicheng Huang*, Nanxing Wu, Xingguo Wang
College of Mechanical and Electronic Engineering, Jingdezhen Ceramic Institute
Jingdezhen333001, China
*Corresponding author, e-mail: huangwu555@sina.com

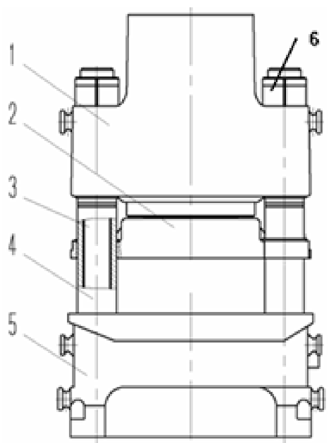
Abstract

Automatic hydraulic press is widely used for pressing ceramic wall and floor tiles, the four columns of its frame withstand the all work force when it works, so the columns are the key compartments of the frame, and its work performance has an important impact on the structural performance of the entire automatic hydraulic press. Took the column thread connection of 32.8MN automatic hydraulic press as research object, established axisymmetric finite element model and carried on contact finite element analysis for it by using the large commercial finite element software ANSYS to gain the stress distribution of the column's thread connection. The results and discussion can provide reference for designing automatic hydraulic press columns.

Keywords: *Automatic hydraulic press; column; contact; finite element; thread connection*

1. Introduction

Automatic hydraulic press (Hereinafter referred to as PRESS) is one of the essential equipment of ceramic industry for pressing ceramic wall and floor tiles [1]. It is generally composed by three parts: frame, power systems and hydraulic control system. Most of the PRESS frame is made up of three- beam and four-column structure [2], and the 32.8MN press studied in this paper is this structure too, Figure 1 shows its structure. Figure 2 is its three-dimensional model. It is a sleeve- tie rod retightening type closed frame composed by upper beam, lower beam, active beam, four columns (called tie rods here), four sleeves and eight big nuts. The frame withstands the entire work load. The columns are locked with big nuts and stretch in full-length, the sleeves withstand preloading. The tensile preload of the columns enable the upper and lower beams and sleeves form a closed frame.



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

Figure 1. Schematic diagram of PRESS

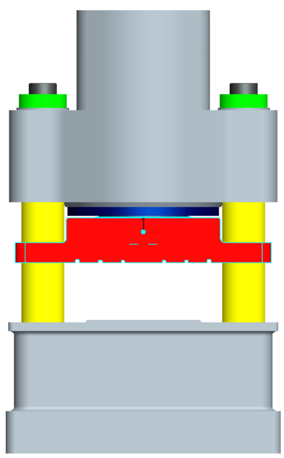


Figure 2. Three-dimensional model of PRESS

Received February 7, 2013; Revised April 24, 2013; Accepted May 10, 2013

As a major force component, the column is one of the dangerous parts of the entire frame, and there have occurred column fracture accidents. Broken column analysis showed that the fatigue fracture of the column occurs mostly at the thread root near the contact surface of nut and beam, That is, the dangerous parts of the entire column are the contact area of the column and nuts. So only study the threaded parts of the column while ignoring other parts to seize the main part of the problem, avoid too large model and save computing resources.

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]. Many scholars carried on finite element analysis to PRESS [5-9]. In the past analysis of the PRESS frame, most of them consider the columns and nuts as a whole to carry on finite element analysis. The analysis of the columns was considered as simple stretching to calculate. Of course, as the rigidity of the whole analysis, this assumption is feasible; however, as the stress-strain analysis of the columns, this assumption is clearly not in line with the actual. Because the columns thread roots will produce local stress concentration, and the actual stress condition is much more complicated than simple stretching. Therefore, in order to improve the stress analysis accuracy of the PRESS column threaded coupling, finite element contact analysis method should be used, rather than consider the columns and nuts as a whole to carry on the finite element analysis. Of course, in order to facilitate comparison, both of the contact analysis and simple tensile assumptions were discussed in this paper.

2. The Contact Problem Analysis of 32.8MN PRESS Column Threaded Connections

The contact analysis capabilities of ANSYS are very excellent. In recent years, ANSYS developed a series of contact elements to describe the two objects in contact with each other or sliding interface. Contact problem is a highly non-linear behavior, which requires a large amount of computing resources. In order to effectively calculation, understanding the problem characteristics and establishing a reasonable model are very important.

Contact problem is divided into two basic types: rigid body - flexible body contact and soft body-soft body contact. In rigid body – flexible body contact problem, one or more of the contact surface is treated as a rigid body (compared to the deformable body in contact with it, it has a much greater stiffness), under normal circumstances, the contact of a soft material and a hard material can be assumed to be rigid body – flexible body contact, and many metal forming problems can be classified as such contact; Another type of soft body - soft body contact is a more common type, in this case, two of the contact body are deformable body (have approximation stiffness).

ANSYS supports three kinds of contacts: point - point, point - surface and surface-surface, and the contact elements used for each contact mode is suitable for different contact problems [9-12]. Obviously, the 32.8MN PRESS columns thread connection contact problem is a soft body - soft body surface-surface contact problem.

3. The Finite Element Contact Model

3.1. Geometric Modeling.

The geometry shape of the column threaded connection is very complex, the contact surface of the column and nuts is a spatial spiral surface, and the forces between them is a general space force system.

A large number of bolts three-dimensional finite element analysis showed that when the helix angle is less than 4 °, the load distribution along the thread teeth is almost independent of the impact of the helix angle, so under axial loads, the bolt can be simplified into axisymmetric problem.

The relations of helix angle and pitch are as follows:

$$tg\alpha = \frac{h}{2\pi r} \qquad (1)$$

Here "α" is helix angle, "h" is pitch, "r" is the circle radius of the points on the spiral line.

The major radius of the PRESS column thread is 150 mm, pitch is 8㎜submitting them into equation (1) can get the helix angle of 0.49°, which is far less than 4°. Therefore, in the process of model building can ignore the impact of the helix angle, using a series of circumferential annular flanges with standard tooth type instead of the continuous thread; at the same time simplify the split nut to cylindrical shape with the same stiffness; In addition, as the load and constraint condition of the column threaded connection are also axisymmetric, so the column threaded connection contact problem can be simplified plane axisymmetric problems, and the stress analysis is greatly simplified.

The basic parameters of the column thread teeth are as follows: nominal diameter d is 300 mm, pitch P is 8 mm, middle diameter d2 is 294.804㎜, minor diameter d1is 291.34 mm; using common thread. According to the above parameters, can establish the axisymmetric finite element model of the column thread connection in ANSYS through the order of point → line → face.

3.2. Element Choice

Taking into account the model is axisymmetric, and the shape of the screw thread is irregular, so the element selected must simultaneously satisfy the requirements of axisymmetric analysis and adapt to the irregular shape. Taken together, plan2 element is a good chose. Plan2 is a six-node triangular element and it can compatible with the 8-node PLANE82 element. The element has a secondary displacement function and can be used to adapt to irregular model grid (for example, the models produced by different CAD / CAM systems); the element has six nodes and each node has two degrees of freedom, which respectively are the x and y directions parallel movement, and it can be used either as a plane element (plane stress or plane strain) or axisymmetric element; The element has the capacity of plasticity, creep, radiation expansion, stress stiffening, large deformation and large strain [13-14]. Figure 3 and Figure 4 respectively show the geometric diagram and stress output results diagram of the plan2.

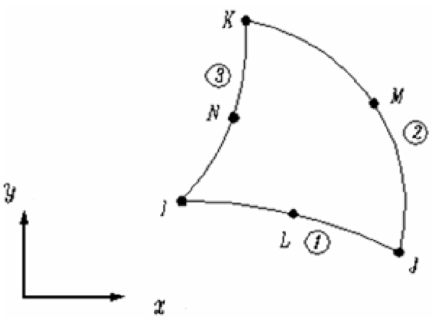


Figure 3. Geometric diagram of plan2

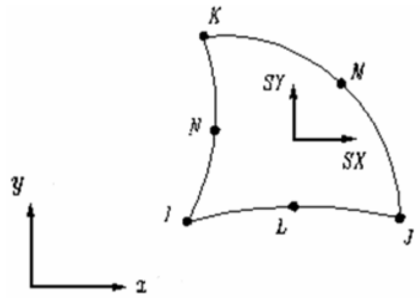


Figure 4. Stress output of plan2

3.3. Meshing

ANSYS provides intelligent meshing function; it can mesh automatically by considering curvature of the geometry and the close degree of lines and lines. Smart mesh control only applies to the free mesh, and it has the accuracy level from 1 to 10. Take into account that the finite element model established in the first step is relatively simple plane model and the element shape has good adaptability, so the meshing of the model use the intelligent automatic meshing, and the meshing accuracy level is set to 3 grade. After screening, the mesh quality is good, so it can be considered that intelligent meshing is reasonable.

3.4. Exerting Constraint and Load

According to the actual work situation of the column and the characteristics of the axisymmetric model, on the column axis (Y axis), constraint the X direction displacement; on the column thread section (x axis), constraint the Y direction displacement. The pretightening force of each of the column is 11.25MN, which is exerted on the contact surface of the big nut and the upper beam. Figure 5 is the axisymmetric finite element model of the column and big nut, Figure 6 is its partial enlargement.

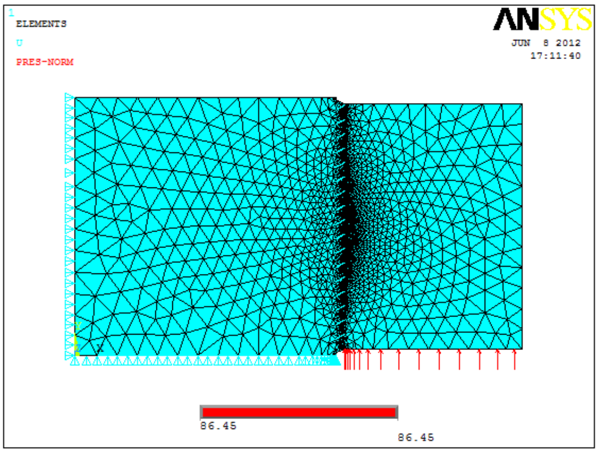


Figure 5. The axisymmetric finite element model

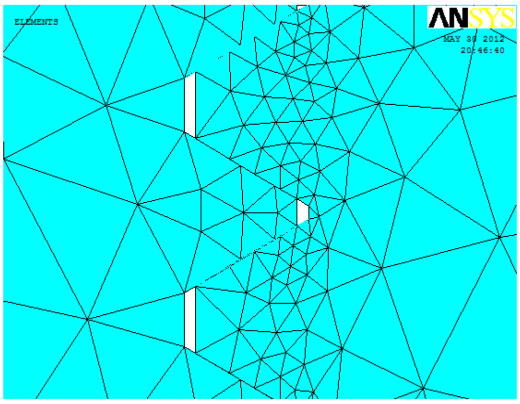


Figure 6. The partial enlargement

4. Pre-Processing

4.1. The Model Material Definition

Need to define the elastic modulus and Poisson's ratio, etc. The material of the column and big nuts is 35CrMo, the elastic modulus $E = 207GPa$, Poisson's ratio $\mu = 0.29$, $\rho = 7.9\, g/cm^3$, σ_s ≥835 MPa, σ_b ≥980 MPa.

4.2. Definition of the MISO Data Sheet and Enter the Corresponding Value

Need to input the material's stress and strain values, which are showed in Table 1.

Table 1. The stress and strain values of 35CrMo

strain	0.000482	0.00102	0.0058	0.03	0.04	0.045	0.05
stress	100	200	800	900	930	950	970

In addition, need to definite “Analysis Type “as “static” and “Analysis option” as “Large deform effects option”; set “output control” and “Load step opts”, finally, using contact wizard to generate the surface-to-surface contact element, these contents are not elaborated In detail here.

5. Solving

After pre-processing the work is completed, enter the solver to solve. Directly select “current LS” option to solve.

6. Analysis of Results

6.1. Simple Stretching Analysis Results

For comparison purposes, first analyzed the column stress in accordance with the assumption of simple stretching. Convert the stress acting on the large nut to the tensile force on the end surface of the column. When the column is pretended and works, the pulling forces are respectively 169MPa and 157MPa. Figure 7 and Figure 8 respectively shows the von mises stress nephogram of the column when it is pretended and works under the assumption of simple stretching. It can be seen from Figure 7 and figure 8 that when the column is pretended and works, the total stresses (von Mises) respectively are 261.244MPa and 242.668MPa, and the stress amplitude $\Delta\sigma$ = 18Mpa

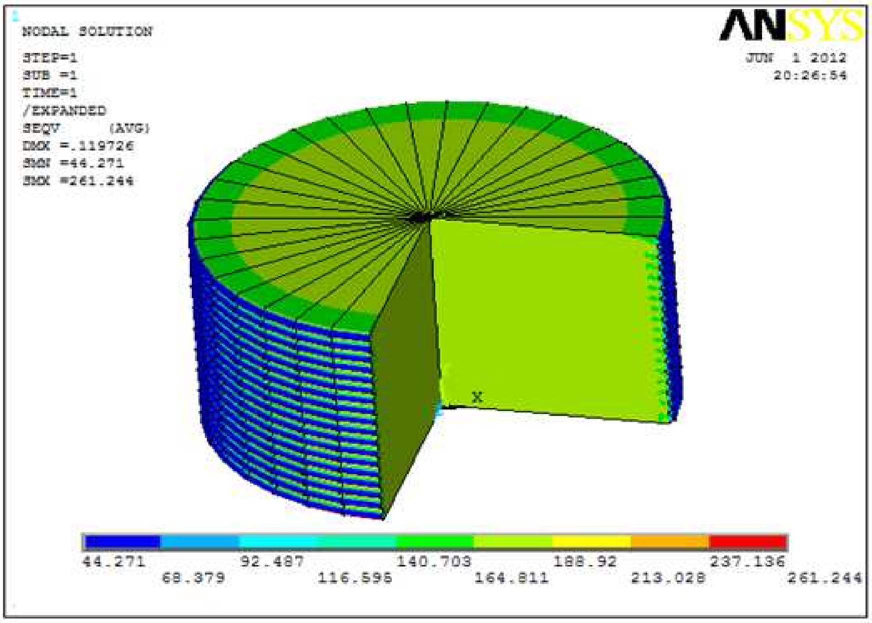


Figure 7. The von mises stress nephogram in the preload condition (simple stretching)

6.2. Contact Analysis Results

Figure 9 is the von mises stress nephogram of the column in the preload condition based on contact analysis, and In order to facilitate the observation, Figure 10 shows the expand three-quarters of the stress nephogram; Figure 11 is the von mises stress nephogram of the column in the work condition based on contact analysis, and In order to facilitate the observation, Figure 12 shows the expand three-quarters of the stress nephogram, too. It can be seen from Figure 9 and Figure 10 that in the preload condition, the max von mises stress of the column is 501.812MPa; it can be seen from Figure 11 and Figure 12 that in the work condition, the max von mises stress of the column is 463.749MPa; so the stress amplitude is 38MPa.

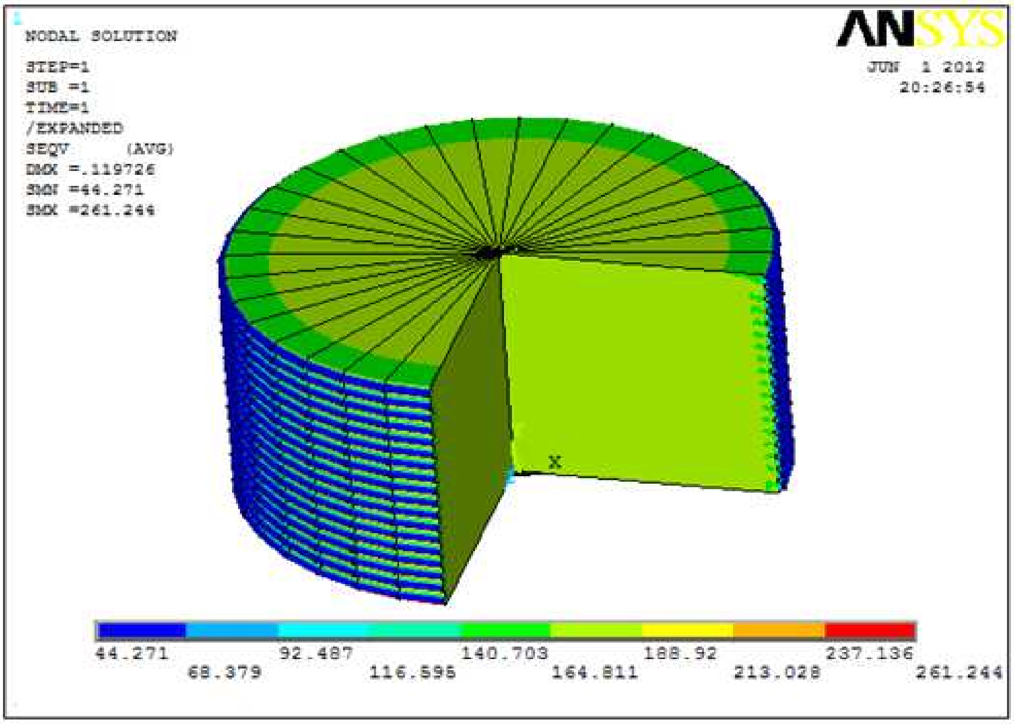


Figure 8. The von mises stress nephogram in the work condition (simple stretching)

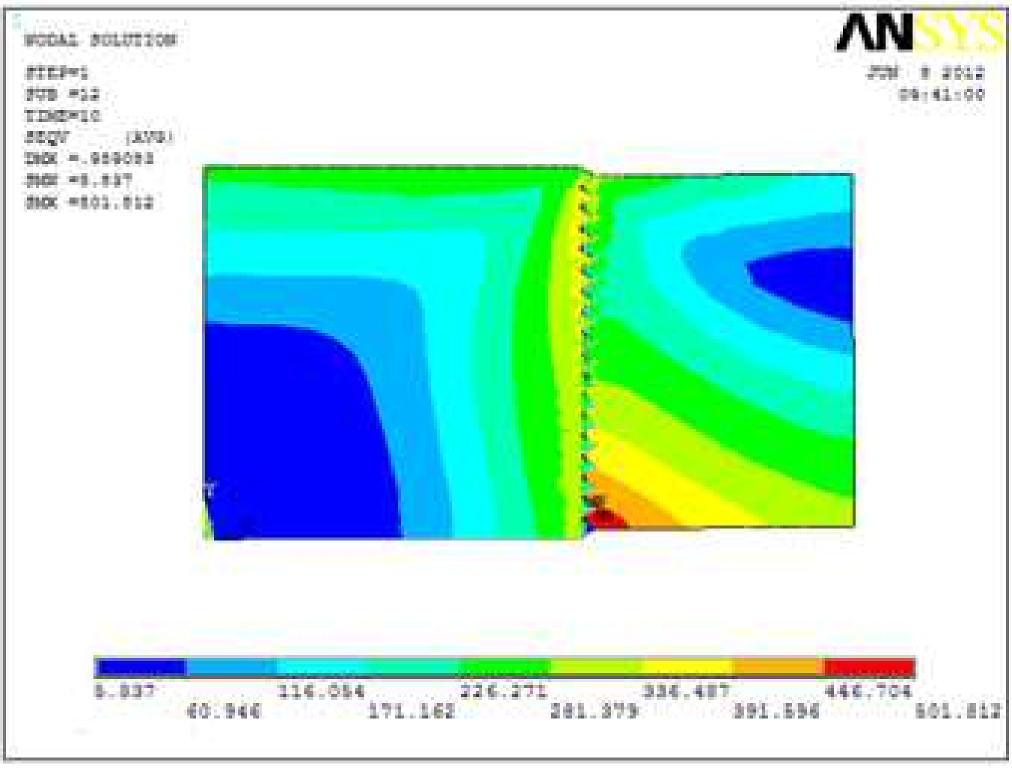


Figure 9. The von mises stress nephogram in the preload condition (contact analysis)

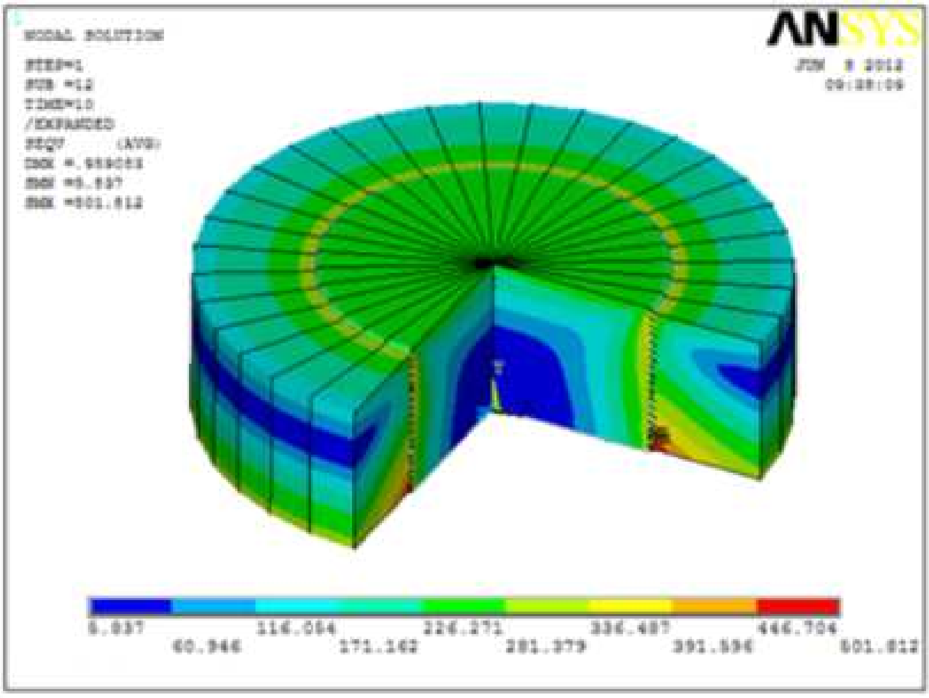


Figure 10. The expand three-quarters of the stress nephogram of Figure 9

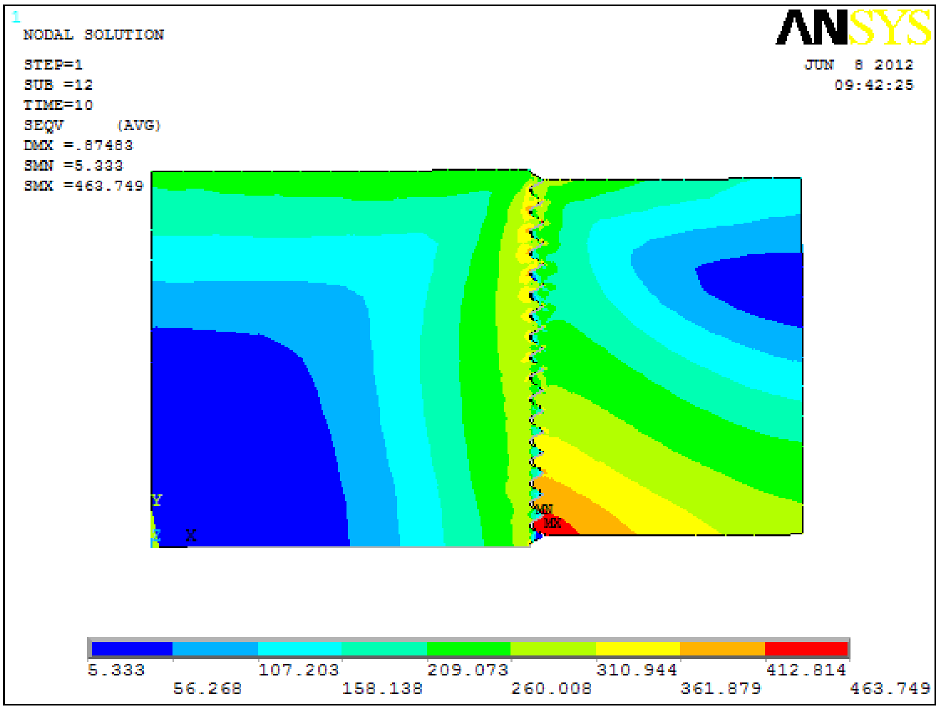


Figure 11. The von mises stress nephogram in the work condition (contact analysis)

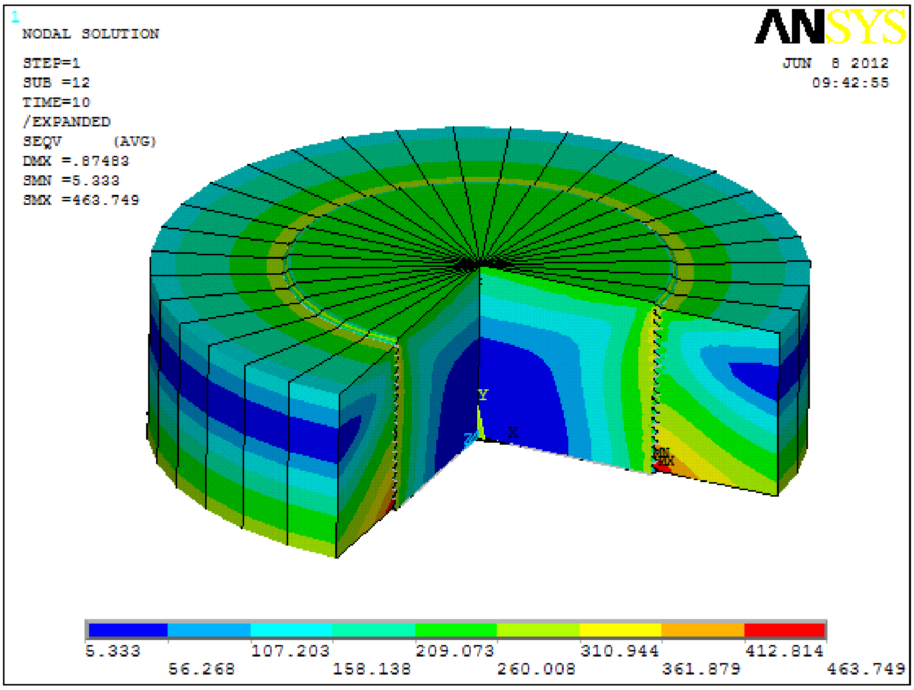


Figure 12. The expand three-quarters of the stress nephogram of Figure 11

7. Conclusion

(1) Contact analysis results compared with simple tensile results are very different, and apparently contact analysis simulation is closer to the actual situation, therefore, in the analysis of column thread connection, the contact analysis method should be used rather than the simple tensile analysis.
(2) The max von mises stress of the column is 501.812MPa, which is less than the yield limit (835 MPa) of the column material, so it can meet the strength requirement.
(3) The stress of the column thread portion is decreasing from the first meshing teeth, and there are almost no forces on the thread tooth after the eighth lap. This shows that the increasing thread screwing length is not able to enhance the strength of threaded connection.
(4) Apparent stress concentration phenomenon exists on the column thread, and they are all at the thread root where geometrical shape mutation exists.

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. Automatic hydraulic press. Nanchang: Jiangxi Science and Technology Press. 2001.
[2] Zhang BQ. The ceramic industrial machinery and equipment. Beijing: China Light Industry Press. 2012.
[3] Guangguo Zhang, Jing Sheng, Feng Wang. A Finite Element Analysis of The Rectangle Spline Broach. *TELKOMNIKA*. 2012; 10(8): 2125-2130.
[4] WANG Xuewen, YANG Zhaojian, Liu Hunju. Finite Element Analysis on Double-Telescopic Prop of Hydraulic Support. *TELKOMNIKA.* 2013; 11(1): 270-279.
[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, HanW, 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. Bei jing: Tsinghua University Press. 2004.
[11] NSYS.INC. Basic Analysis Procedures Guide. Release 5.7.
[12] NSYS.INC. Modeling and Meshing Guide. Release 5.7.
[13] SYS.INC. ANSYS Element Reference. Release 5.7.
[14] NSYS, Inc. ANSYS Advanced Technical Analysis Guide. 1998.