Design and Analysis for EPB Shield Bracket Based on Ansys

Sun An'bo[1], Zhang Yongliang[1], Min Rui[2], Ji Changqin[1] & Zhuo Pan[1]

[1] School of Mechanical Engineering, University of Shanghai for Science and Technology, Shanghai, China

[2] Shanghai Tunnel Engineering Co., Ltd, Shanghai, China

Correspondence: Sun An'bo, School of Mechanical Engineering, University of Shanghai for Science and Technology, Shanghai, China. Tel: 86-188-1758-1557. E-mail: sab19870216@yeah.net

Abstract

According to the structural characteristics of EPB shield bracket, the author establishes 3D solid model by Solidworks and corresponding finite element model of the bracket portion which is connected to the cutterhead by ANSYS WORKBENCH software. Through the static analysis of bracket's stress characteristics under extreme conditions, we get its stress, deformation and safety coefficient distribution law under the maximum constraint conditions. After getting the maximum equivalent stress, the analysis of the calculation results shows that this kind of bracket with good static characteristics can meet the design strength requirement. This paper points out the weak position of bracket's strength, and provides some reference data for the structural optimization design, as well as some basic data for both the structural design of bracket and the construction maintenance. Moreover, the structure analysis in the process of the grid selection and the key technology of the post-processing method are discussed in detail. The design example shows the effectiveness of the method.

Keywords: finite element analysis, equivalent stress, static analysis, shield bracket

1. Introduction

Shield tunneling machine is a special machine for tunneling, and the shield bracket is one of the main working partsof shield tunneling equipment. The structure of shield bracket is directly related to the quality of the tunnel project and the costs and service life of cutting tool. Because the overall size of shield bracket is larger while its local design size is relatively small and design installation and testing are very complex with high cost and complex on-site maintenance, high requirements on bracket design are put forward.

With complex cutterhead design structure and great construction intensity, the bracketwhich transfers power to the cutterheadis required to be with high strength. Therefore, it is needed to check the bracket's strength so as to ensure the project needs. Because of the restrictions of the shield tunneling machine itself and construction environment, both the costs and conditions of multiple tests in physical environment are difficult to achieve the requirements. As a result, under the condition of simulation, using 3D modeling software and finite element analysis software to make mechanical analysis and intensity checking can quickly optimize the cutterhead bracket design. Reasonable structure, as one of the key technologies of the shield, can not only ensure the high-efficiency operation of the shield construction machine, but also improve the service life of cutterheadand many other parts of the machine and narrow tool cost.

The work to research shield bracket has not been done clearly before. This paper puts forward a simulation design method specialized for the strength of shield bracket, and it combinesthe analysis of the stress and strain fields of the structure element with the shield bracketwhich isdesigned and checked by Ansys, one kind of powerful analysis system, the targeted physical experiment strength analysiscan be realized so as to finish the rapid and effective design.

2. Build Calculation Model for Bracket

2.1 Build the Finite Element Model

According to the actual size requirement of bracket, 3D model is established in Solidworks while physical model is established in the imported Ansys workbench. In order to avoid the situation "efficiency without accuracy" or "accuracy without efficiency" appearing, and model transformation failing because of too many detailsinthe conversion process, the 3D model of the cutter bracket established inthe DM shall be simplified properly, the geometrical characteristics (such as round angle and chamfer angle) whichhave little effect on finite element analysis shall be deleted, and matrix part shall be keptfor reducing the difficulty and complexity of the mesh of finite element model, so as to speed up the analysis. As a result, it ensures both the solving efficiency and the operational precision of finite element analysis. The following Figure 1 and Figure 2 are separately for bracket's design model and finite element model:
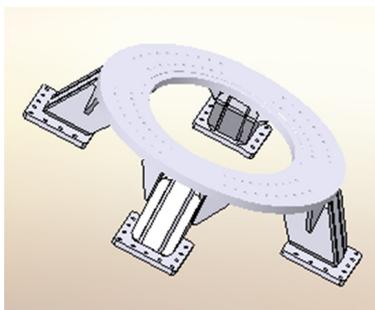
Figure 1. Bracket's design model   Figure 2. Bracket's finite element model
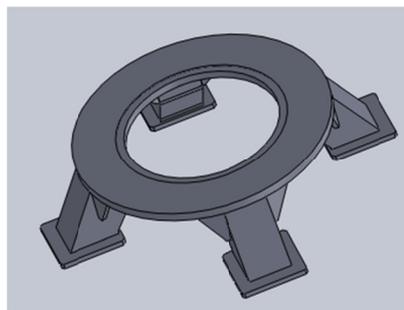
2.2 Settlement of Boundary Condition

According to the installation and movement request of shield cutterhead bracket, constraints are exerted to it. The cutterheadmainly makes rotary motion and axial movement of the propulsion during the process of shield tunneling, and because both the rotation speed and the propulsion speed are slow, shield bracket strength can be approximately analyzed as static analysis. This paper mainlyanalyzes the strength under limit situation, so the cutterheadcan be equivalent to a rigid disc with same shape and dimension, and a full constraint of bracket flange is implemented by exerting the thrust and torque, which are gained from the experiment, on the equivalent surface. According to the experiment, the thrust on the cutterhead is 3600 t and the torquemoment is 5669 $KN \cdot m$, and after the calculation of a quarter of the whole model which is cut under the symmetrical principle, the stress and constraintof the cut bracket are shown in Figure 3:
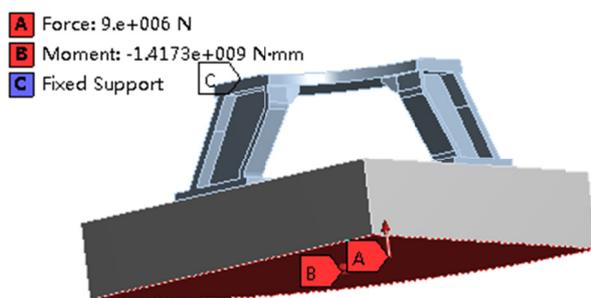
Figure 3. Bracket constraint and stress

2.3 Design of Finite Element Calculation of Model

For complex geometric structure model analysis, finite element calculation, based on the numerical solution of differential equations, has obvious advantages: it can discretely process themodel, the classificationof the grid is not be required as regular as that in finite difference method, and all grid element modules can be used simultaneously. Under same grid conditions, when the grid number increases, finite element analysis will generally become more precise, followed by more elements and meshes formed, as well as more equation time

will be spent. So reasonable choice of elements is necessary, and the optimization of finite element model, based on the structure characteristics, shall be adopted.

The processing material for designing cutterhead bracket is steel Q345B whose density is 7850 kg/m^3, elastic modulus is 206 GPa, and Poisson ratio is 0.28. Based on the characteristics and complex interface structure of the finite element model, it meshes mainly with hexahedrons, mixed with the combination of high order tetrahedrons and hexahedrons. Because the model structureis symmetric, it is enough to mesh a quarterof the cut model in which joins the boundary constraints. The finite element model consists of 40685 elements and 163934 nodes, as shown in Figure 4:
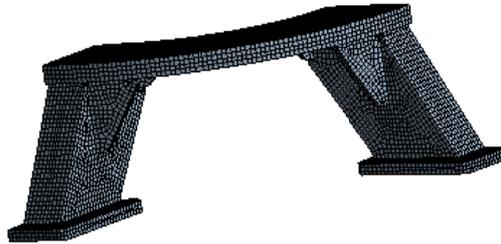
Figure 4. Mesh model for the bracket

3. Strength Analysis of Cutterhead Bracket

3.1 Analysis of Element Types Selection

The purpose of the finite element calculation is to completely obtain the accurate internal mechanics information of any deformable body with complex shapes, under the condition of complex external force. On the basis of mechanical analysis, designers can check and judge the strength, stiffness, and other aspects of the design objects, so as to modify unreasonable design parameters, and then obtain more optimized design scheme. In calculation, the accuracy of tetrahedron elements is relatively a little lessthan that of hexahedron, but its applicability is the most widely, since it can be used for grid partition of any model. Therefore, Tet10 element and 20 node hexahedron element are used as examples in this design model: with the application of high order tetrahedron elements and hexahedron, and the advantages of the two elements, according to the geometry characteristics of 3D tetrahedron elements and hexahedron, natural coordinate system is adoptedas body coordinate system, each difference functionoftetrahedron elements appears, as well asthe displacement, stress and strain parameters of each element node.
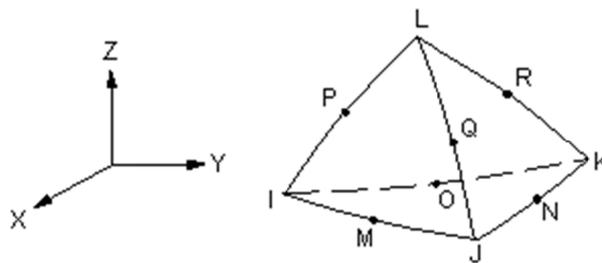
Figure 5. 10 node tetrahedron element

The node displacement of element has 30 degree of freedom, the node displacement array of elementis q^e and nodal force array is P^e.

Displacement mode:

$$u = u_i(2l_1-1)l_1 + u_j(2l_2-1)l_2 + u_k(2l_3-1)l_3 + u_l(2l_4-1)l_4 + 4u_m l_1 l_2 + u_n l_2 l_3 + u_o l_1 l_3 + u_p l_1 l_4 + u_q l_2 l_4 + u_r l_3 l_4$$

$$v = v_i(2l_1-1)l_1 + v_j(2l_2-1)l_2 + v_k(2l_3-1)l_3 + v_l(2l_4-1)l_4 + 4v_m l_1 l_2 + v_n l_2 l_3 + v_o l_1 l_3 + v_p l_1 l_4 + v_q l_2 l_4 + v_r l_3 l_4$$

$$W = w_i(2l_1 - 1)l_1 + w_j(2l_2 - 1)l_2 + w_k(2l_3 - 1)l_3 + w_l(2l_4 - 1)l_4 + 4w_m l_1 l_2 + w_n l_2 l_3 + w_o l_1 t_3 + w_p l_1 l_4 + w_q l_2 l_4 + w_r l_3 l_4$$

$$\underset{3 \times 1}{u} = \begin{bmatrix} u \\ v \\ w \end{bmatrix}$$

Element strain field:

$$\underset{6 \times 1}{\varepsilon} = \underset{6 \times 3}{[\partial]} \underset{3 \times 1}{u} = \underset{6 \times 30}{B} \underset{30 \times 1}{q^e}$$

Element stiffness matrix

According to the physical equationsof plane problems in elastic mechanics, element stress expression, can be calculated and then the element potential energy, and finally is the element stiffness matrix:

$$\underset{30 \times 30}{K^e} = \int_{\Omega^e} \underset{30 \times 6}{B^T} \underset{6 \times 6}{D} \underset{6 \times 30}{B} d\Omega$$

Element stiffness equation.

Give an order extremum to the node displacement q^e of element potential energy, the element stiffness equation can be expressed as below:

$$\underset{30 \times 30}{K^e} \underset{30 \times 1}{q^e} = \underset{30 \times 1}{P^e}$$
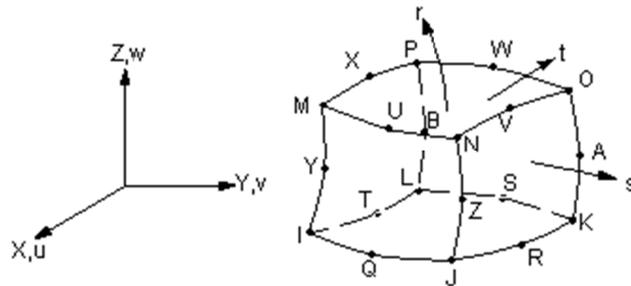
Figure 6. 20 node hexahedron element

In the same way, the node displacement array q^e and nodal force array P^e can be obtained, as well as the element strain field and stiffness equation.

3.2 Result Analysis and Processing

Using finite element software to calculatethe model and realize the constraint face coupling calculation by Lagrange multiplier methodcan effectively improve the calculation precision of the finite element model. According to the measurement test, for plastic materials, the checking result ofthe fourth strength theory is much more accurate than that of the other three strength theories.

According to the rules of the fourth strength theory, the model structure, and the material characteristic, distortion energy density is the main factor causingyield, that is, no matter under what stress state, the material

will yield as long as distortion energy density v_d reachesan limit value which is related to material properties.

When the material is uniaxial stretched, the distortion energy density $\frac{1+\mu}{6E}(2\sigma_s^2)$, correspondingwith yield

stress σ_s, will be calculated, and it is the limit value of the distortion energy density which causes yielding.

Under arbitrary stress state, as long as distortion energy density v_d can achieve the above limit value, the

material will yield and deform. So, there is the yield criterion.

$$v_d = \frac{1+\mu}{6E}(2\sigma_s^2)$$ (3-1)

Distortion energy density under arbitrary stress state

$$v_d = \frac{1+\mu}{6E}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\right]$$ (3-2)

After calculation, here is the yield criterion

$$\sigma_s = \sqrt{\frac{1}{2}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\right]}$$ (3-3)

Replacing extreme stress σ_s with allowable stress, here is the strength condition in fourth strength theory e

$$[\sigma] \geq \sqrt{\frac{1}{2}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\right]}$$ (3-4)

After calculating, the cloud atlases of the stress of cutterhead bracket under extreme condition, the stress variation curve of node in key parts, and the deformation and safety factor are separately shown in Figure 7, Figure 8 and Figure 9.
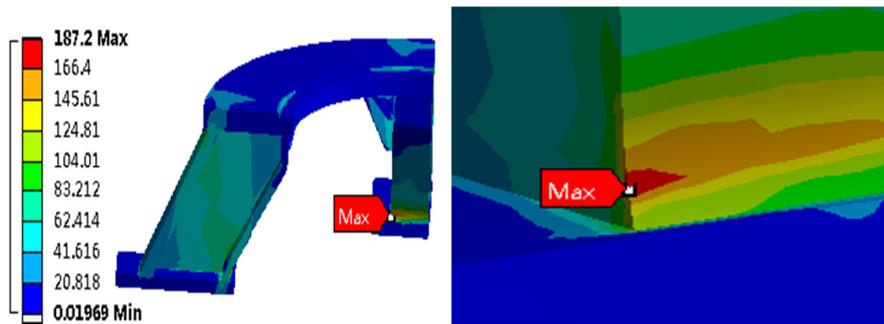
Figure 7. The cloud atlas for the equivalent stress of bracket
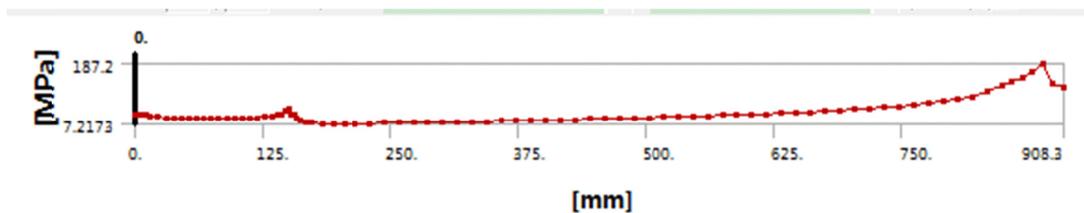
Figure 8. The stress variation curve of node in bracket's key parts
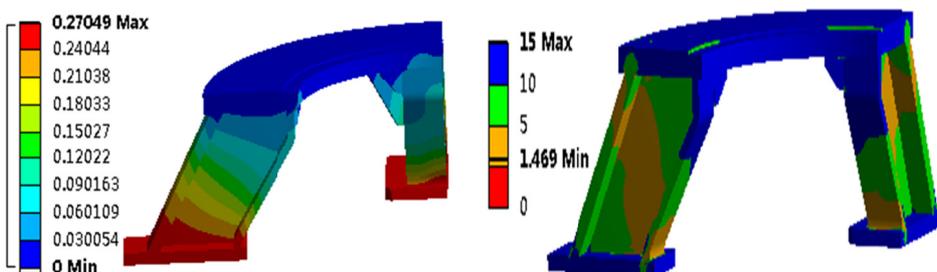
Figure 9. The cloud atlas for safety factor ofthe bracket deformation

According to the calculation results, under the extreme condition, the bracket equivalent stress mostly distributes between 0.0197 MPa and 83.212 MPa , and the stress distributions are rather even. The maximum equivalent stress is 187.2 MPa , which is less than the material allowable stress 275 MPa ; the minimum safety coefficient is 1.4; the biggest Von-Mises stress occurs in the bracket root, while through the measurement in field experiment, the maximum equivalent stress is 166 MPa , so they have a difference of 12%, which can meet the requirements. Although shield construction machine operates continuously, and bracket shall not be replaced in a construction process except in special circumstances, the alternating stress changeslightly, this kind of design schema can meet the construction requirements. Much more attention should be paid on the bracket which can be strengthened properly without impacting the material cost and operating condition, so as to improve the overall performance.

The whole design is safe, and from the view of stress analysis, there is great potential about the ability of the material's resistance to damage, and stress concentration may exist in the maximum equivalent stress of node, for the cutterhead, as the stress surface, is approximately simplified to arigid body structure in order to study the bracket intensity. But the stress value is far lower than the ultimate strength of material, and the stress concentration will not affect bracket stiffness. In actual work process, the power transferred from the cutterhead special structure will not cause stress concentration.

4. Conclusion

Using Solidworks and ANSYS to analyze the stress characteristics of bracket in shield construction machine under extreme conditions, the bracket's stress, deformation and safety coefficient distribution rules are obtained. Through analyzing the designed bracket with actual situation, physical experiment will be more targeted for the test of shield construction machine bracket. Thus shield construction machine bracket which can meet the practical construction requirements shall be designed, as well as reduce the design and experiment cost. Besides, the study results can provide design reference for the similar bracket structure design and engineering construction maintenance.

References

Cheng, B. X., & Wang, Z. Q. (2002). Finite element analys is for cutterhead of shield machine based on CATIA. *Machinery Design & Manufacture, 29*(2), 42-47.

Cong, M., Fang, B., & Zhou, Z. L. (2008). Finite Element Analysis and Optimization Design of the Carriage of Turn Broach NC Machine Tool. *China Mechanical Engineering, 19*(2), 208-212.

Han, H. B., Li, J. Zh., & Liu, H. B. (2010). FEA ANALYSIS OF THE SHIELD MACHINE MAIN BEARING UNDER RADIAL FORCE. *International Conference on Advanced Technology of Design and Manufacture,* 236-239.

Hu, G. L., & He, X. J. (2010). The Parametric Modeling and Optimization for Cutter Head of Shield Tunneling Machine. *International Conference on Computing, Control and Industrial Engineering,* 316-319. http://dx.doi.org/10.1109/CCIE.2010.87

Kimura, F. (1993). Product and Process modeling as a kernel for virtual manufacturing environment. *CIRP Annals, 42*(1), 147-150. http://dx.doi.org/10.1016/S0007-8506(07)62413-5

Srinivas, K. N., & Arumugam, R. (2004). Static and Dynamic Vibration Analyses of Switched Reluctance Motors Including Bearings, Housing, Rotor Dynamics, and Applied Loads. *IEEE TRANSACTIONS ON MAGNETICS, 40*(4), 1911-1919. http://dx.doi.org/10.1109/TMAG.2004.828034

Wang, M. C. (2003). *Finite Element Method.* Tsinghua University Press.

Wu, Z. B., & Xiao, T. Y. (2000). Applications of finite element analysis in virtual Manufacturing. *Journal of Tsinghua University, 40*(7), 66-69.

Xia, Y. M., Zhou, X. W., & Liu, Y. J. (2009). Surface Roughness Forecasting of Spiral Bevel Gear Based on Artificial Neural Network. *Journal of Zhengzhou University (Engineering Science), 30*(3), 71-74.