Modern Applied Science; Vol. 8, No. 4; 2014
ISSN 1913-1844 E-ISSN 1913-1852
Published by Canadian Center of Science and Education

Analysis for EPB Shield Bracket Based on Ansys

Baojiang Du[1], Yajun Yu[1], Pengpeng Ding[1], Weibin Lv[1] & Fei Zhao[2]

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

[2] School of Engineering, Shanghai Ocean University, China

Correspondence: Yajun Yu, School of Mechanical Engineering, University of Shanghai for Science and Technology, China. Tel: 86-188-1826-2081. E-mail: 317750328@qq.com

Received: April 1, 2014 Accepted: May 6, 2014 Online Published: June 25, 2014

doi:10.5539/mas.v8n4p29 URL: http://dx.doi.org/10.5539/mas.v8n4p29

The research is supported by research project of Science and Technology Commission of Shanghai Municipality (project No: 11dz1121300)

Abstract

This paper studies the working condition of EPB, calculates the bearing thrust and torque. It also analyzes the stress characteristics of EPB bracket in extreme working conditions by using the finite element method and gets the stress and deformation distribution of EPB bracket. Through grid refinement, the results of different mesh size are analyzed, the results show that: the maximum strain is convergence, but the position of the maximum stress produces stress singularity. The stress singularity results are amended by theoretical formula method in this paper, meanwhile presents an interpolation method called the critical method of interpolation, the value obtained by this method is more close to the true value. By using this method to interpolation calculate the existing analysis data and get the desired result. By comparing the interpolation results, calculation results and analysis results, it proves the feasibility of this interpolation method, provides a kind of effective method to solve the singular stress results, provides an effectual reference for structural improvements and design manufacturing of shield bracket.

Keywords: finite element analysis, EPB bracket, stress singularity, critical interpolation

1. Introduction

Shield tunneling machine is special tunneling construction machinery, and the EPB bracket is one of the principal constituent parts. The shield machine will encounter different geological condition in underground mining, from silt, clay, sand and hard rock to soft rock and other conditions. In the work process, the mainly parts of shield tunneling machine that is in contact with geological environment is cutter head and bracket. In the excavation process, the EPB bracket suffers complex force, bad working environment, and once the construction begins, the EPB will be placed in the ground and assembling only needs once forming, it is difficult to replace parts in the construction process. So if there is a problem on the equipment, it will give bring huge loss to the whole tunnel. Bracket's structure and quality is directly related to the tunneling engineering of shield tunneling machine. Therefore, the structure design of the bracket is particularly important.

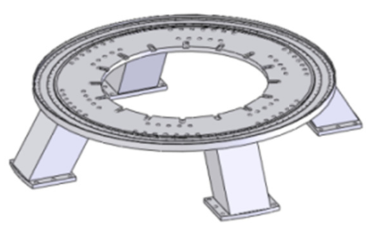


Figure 1. Bracket model

This paper uses the advantages of computer analysis, combines with the finite element theory and mechanics knowledge to analyze the shield machine's bracket structure, to contribute to the achievement of aided design and improve research and production capacity. Meanwhile, gives a more effective approach to solve the problem of stress singularity. The results of this study on the structural design of the bracket construction have certain reference value.

2. Bulid Calculation Model for Bracket

The whole bracket uses the thickness of 50 mm-70 mm sheet, builds their models in SolidWorks. Since EPB bracket structure is complicated, so it's also need to simplify the model to comply with the requirements of finite element analysis, it is also convenient for designers to modify the model after simplification.

According to the model simplification method of EPB bracket, the simplified calculation model of bracket is showed in Figure 2. Because the bracket is mirror symmetric structure, and its force and constraints are symmetrical too. In the FEA (finite element analysis) computational process, in order to improve the utilization of computer resources in the calculation and to reduce the impact on the accuracy of the calculation results due to the hardware limit. According to the theory of symmetry calculation of finite element calculation, you can choose its 1/4 to study after cutting, it can ensure the accuracy of the finite element calculation, and improve the computational efficiency. To properly handle the design model, you can get the corbel finite element model, apply the symmetry constraint on the symmetry plane after cutting, then select 1/4 of the model to calculate, as shown in Figure 3.

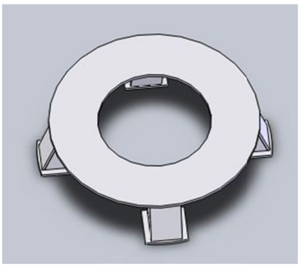


Figure 2. Bracket model's finite element calculation model

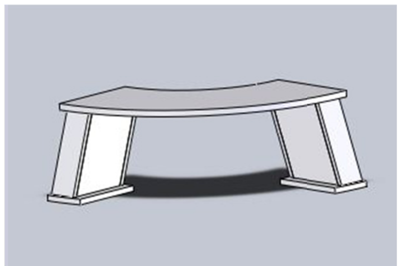


Figure 3. 1/4 model of bracket

3. Analysis of the Forces

The force bearing by EPB bracket is mainly from the cutter, so the study of the forces bracket should start from the forces of the cutter. When shield cutter is working, it will have intense friction with the surrounding soil to produce friction resistance, effects of different soil environments are also distinctive. Due to the difference of the environment, the formula for calculating the friction resistance that you need to use is different. In clay-based geological environment conditions, the frictional resistance due to the soil and the cutter is much smaller than the cohesive strength generated by the clay itself produce, thus the frictional resistance due to the soil and the cutter is often not considered when calculating the frictional resistance, and only consider the frictional resistance generated by the soil cohesive strength. In sand-based geological environment condition, the friction resistance is greater than the cutter soil cohesive strength; therefore, you should regard it as the frictional resistance to calculate. The calculation on the friction between shield shell and formation can according to different geological conditions and the different situation:

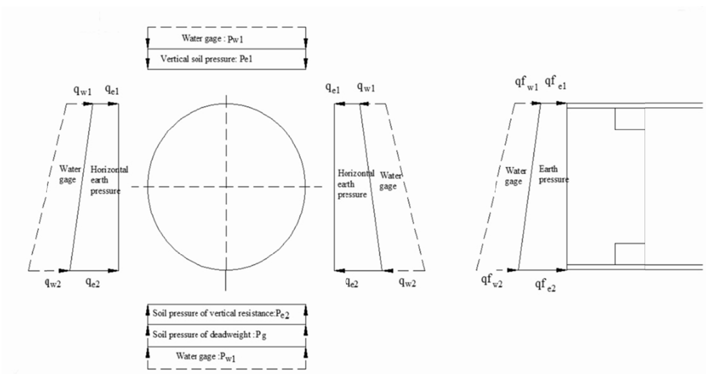


Figure 4. Calculation diagram of cutter's external load

As shown in Figure 4, the shield cutter's frictional resistance can be calculated according to the following formula:

(1) in the case of sand (soil cohesive strength: c = 0):

$$F_1 = \mu(\pi D_0 L p_w + w) = \mu(\pi D_0 L \frac{p_{e1} + q_{e1} + p_{e2} + q_{e2}}{4} + w) \quad (1)$$

(2) in the case of clay (soil friction angle: $\varphi = 0$):

$$F_1 = \pi D_0 B c \quad (2)$$

Type in : B is the thickness of the shield machine cutter (m)

D_0 is the shield machine's outside diameter (m)

L is shield length (m)

c is cutting section soil cohesive strength (KPa)

W is weight of the shield cutter (KN)

μ is the friction coefficient between the strata and the shield shell

P_e is the vertical earth pressure strength acting on the top of shield machine (KPa)

The total propulsion must be greater than the sum of the resistance of various advances, otherwise the shield can't move forward. The resistance includes peripheral and soil shield friction resistance, shield forward resistance, the extrusion force of rock broken by the hob, rear trolley traction resistance. In extreme working conditions shield cutter can't rotate, thus the entire shield bearing capacity is the maximum carrying capacity of the equipment machine, the positive pressure and torque is much larger than any other forces, so you should take the positive pressure and torque suffered by the cutter as the external forces for the research. Bracket is the part to connect the cutter and the shield machine body, the positive pressure and torque that cutter suffers also act on the bracket.

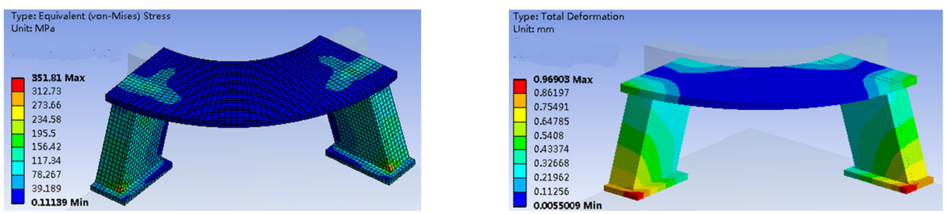
Though calculation according to the corresponding formula: loading of thrust is 35280KN, resistance torque in the geological environment is 5669 $KN \bullet M$.

4. Strength Analysis of Bracket Cutter Head

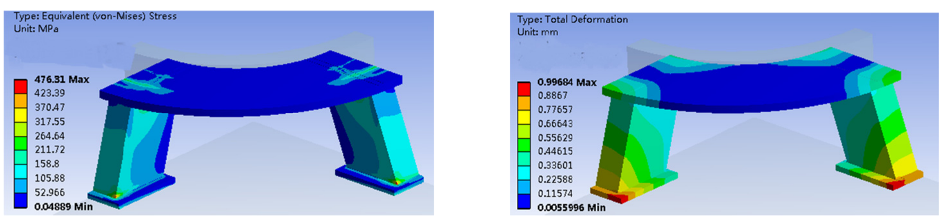
4.1 Analysis of Results

The material of bracket model is Q345B; material's density is 7850kg/m^3; poisson ratio is 0.28; elastic modulus is 2.06×10^{5}MPa; yield strength of the material is 275MPa.

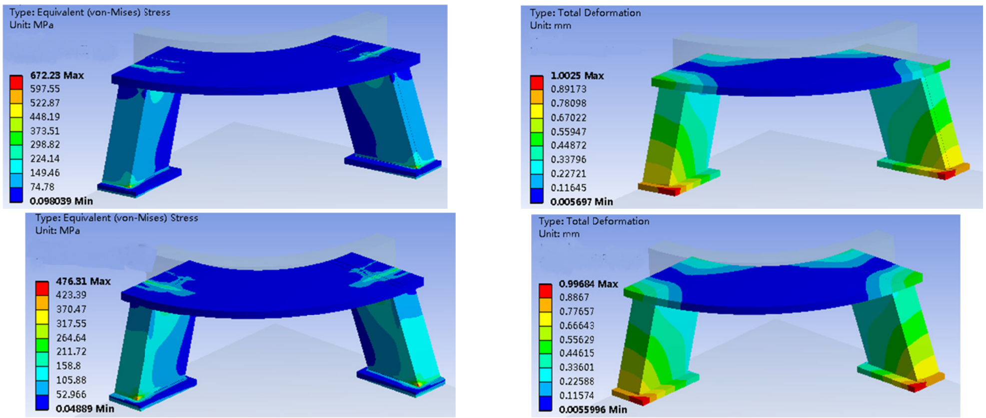
According to the design size of bracket, you can use 40mm, 20mm and 10mm grid size to carry on the grid division of the bracket and calculate. The maximum equivalent stress, the maximum deformation is shown in Figure 5, then analyze the results.



a. 40mm mesh



b. 20mm mesh



c. 10mm mesh

Figure 5. Stress distribution, safety factor and the maximum displacement distribution

After comparison on the maximum deformation and maximum stress that are calculated by each mesh, you can get the result that is shown in the Table 1. The maximum deformation is convergent, but with the mesh refinement, the maximum stress is more and more big. The maximum stress is present at the ends of the support beam bracket, the position of maximum stress cross-section is shown in Figure 6. You can find that only a few nodes' stress is not normal, so the stress distortion happens on this position. The stress obtained by the method of Finite element analysis is not the exact stress value of the bracket.

Table 1. Comparison of results

Mesh size	Maximum deformation (mm)	Change rate	Maximum stress (MPa)	Change rate
40mm	0.969	0	351.8	0
20mm	0.997	2.9%	476.3	35.4%
10mm	1.003	0.6%	672.2	41.1%
conclusion	Convergence		Misconvergence	

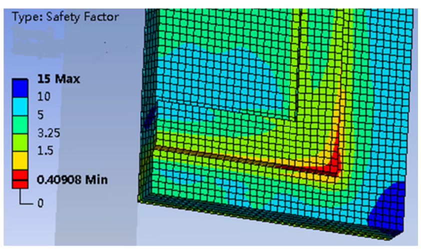


Figure 6. The highest stress profile (10 mm mesh)

In the course of the finite element calculation, the stress singularity makes the convergence rate of the finite element solution is very slow, especially for the uniform grid division, the stress singularity can make the result tend to be infinite, thus affect the accuracy of the results, even lose the reference value for stress of structural calculation. In the finite element analysis of the shield bracket, the geometric model structure should be simplified due to the calculate need. However, the stress singularity phenomenon happens in the calculation, it is necessary to carry out further analysis on it, which can get more accurate stress analysis results.

4.2 Singular Point Calibration Equations

By the analysis of Figure 7 you can see bracket is under the joint action of the tangential force Fb caused by axial thrust Fc of shield cutter head and the torque. You can look each bracket as a single beam, the component perpendicular to the beam axis of Fc and Fb will cause bracket's torsion deformation and integrated view, the deformation belongs to the beam's bending and torsion deformation.

1) The bracket is simplified as a beam and the bending moment diagram is show in Figure 7.

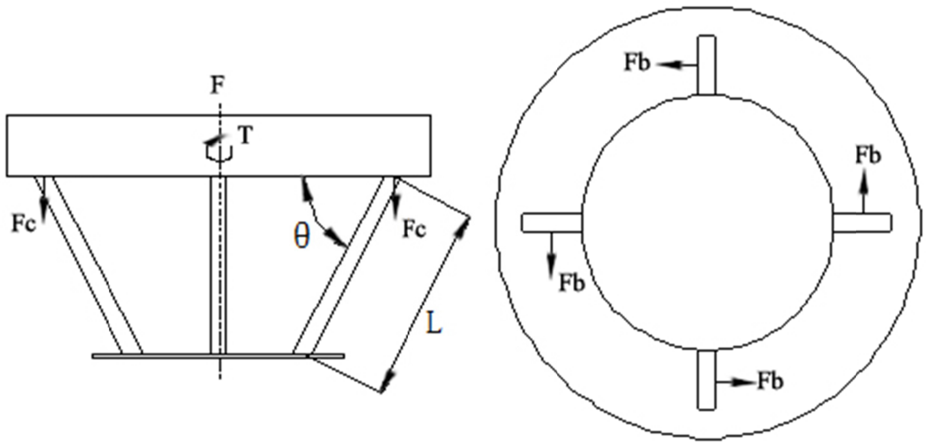


Figure 7. Shield bracket force simplified diagram

In the XZ plane, decompose the force Fc , $Fc\cos\theta$ is the component that is perpendicular to the axis direction, $Fc\sin\theta$ is parallel to the axis direction, The corresponding bending moment perpendicular to the axial direction is

$$M_1 = Fc\cos\theta\times l \tag{3}$$

In the XY plane, the bending moment generated as follows:

$$M_2 = Fb\times l \tag{4}$$

2) Normal stress and shear stress produced by the bending moments

Rectangular cross-section of the beam is shown in Figure 8, synthesis the bending moment M_1 and M_2 in two directions to get the synthetic bending moment M.

$$M = \sqrt{M_1^{\ 2} + M_2^{\ 2}} = \sqrt{(F_b l)^2 + (F_c \cos\theta \bullet l)^2} \tag{5}$$

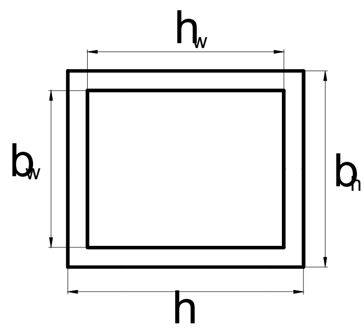


Figure 8. Beam figure

$$\sigma = \frac{M}{W} = \frac{6\sqrt{(F_b l)^2 + (F_c \cos\theta l)^2}}{b_h^{\ 2}(3h - b_h) - b_w^{\ 2}(3h_w - b_w)} \tag{6}$$

$$\tau = \frac{F_s}{A} = \frac{\sqrt{(F_b l)^2 + (F_c \cos\theta l)^2}}{hb_h - h_w b_w} \tag{7}$$

As shield bracket belongs to the plastic material, and it is in a complex stress state, therefore you should follow the fourth strength theory to calculate it:

$$\sigma_1=\sigma \qquad \sigma_2=\tau \qquad \sigma_3=-\tau \lim_{x\to\infty}$$

$$\sigma_{r4}=\sqrt{\frac{1}{2}\left[\left(\sigma_1-\sigma_2\right)^2+\left(\sigma_2-\sigma_3\right)^2+\left(\sigma_3-\sigma_1\right)^2\right]}=\sqrt{\sigma^2+3\tau^2} \tag{8}$$

Experimental results show that: for the plastic material, the fourth strength theory is in better agreement with the experimental results than the third strength theory, it has been widely used in engineering. Because the material of bracket is Q345B, so when performing the strength checking, you should use the fourth strength theory, so the results you can will be more proximate to the actual analysis results. After the above formula you can calculate: $\sigma_{r4}=293\mathrm{MPa}$.Because the actual model and calculation model is slightly different, so the results are not very accurate, but the results can be used as reference.

4.3 Singular Point Interpolation Computation

Sometimes, some details of the model can be simplified significantly, sometimes details seems is not very important at the beginning, but the results show that the details are crucial. At this time, in the singular region, you can use stress linear interpolation and stress fitting combined with Matlab to calculate the stress value fast and accurately.

The previously mentioned formula checking method is a kind of effective method for engineering whose calculation precision request is not high and analysis part has simple structure. But if the engineering requires high accuracy and has a complex structure, because it will suffer many conditions limit such as computer hardware, model of external conditions, then it is difficult to obtain accurate results to follow the above engineering checking method. This paper proposes an analysis and calibration method called critical interpolation for the FEA results. In this method, you should conduct effective multiple FEA calculation on engineering models, and obtain the distribution of Equivalent (*von-Mises*) values, generate the stress values as curve and fits them, and then study the values on the critical point from the overlapping sections and the non-overlapping sections. You should get stress value toward the coincidence of the direction subject to a set of numerical with high quality meshes, take the first adjacent stress numerical difference of 10% stress points, and the critical point linear combination gives a linear transformation equations, then solve the results, get the singular point stress value.

The critical values obtained by the FEA and the fitting curve is (x_a , y_a), the numerical difference between the 10% stress points is (x_{a+1} , y_{a+1}),

$$\text{Slope:}\quad k=\frac{y_{a+1}-y_a}{x_{a+1}-x_a} \tag{9}$$

critical interpolation equation

$$y-y_a=k(x-x_a) \tag{10}$$

obtain the interpolation equations by (1), (2)

$$y=\frac{y_{a+1}-y_a}{x_{a+1}-x_a}(x-x_a)+y_a \tag{11}$$

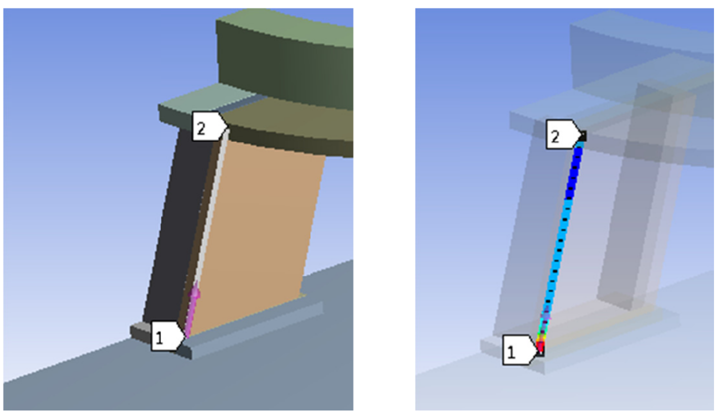


Figure 9. Stress picking route

Make the edge that the maximum stress belongs to as the path, as shown in Figure 9, you can select the node stress values based on three mesh size and compare them., from Figure 10 you can see that the values in the range of BD are consistent.

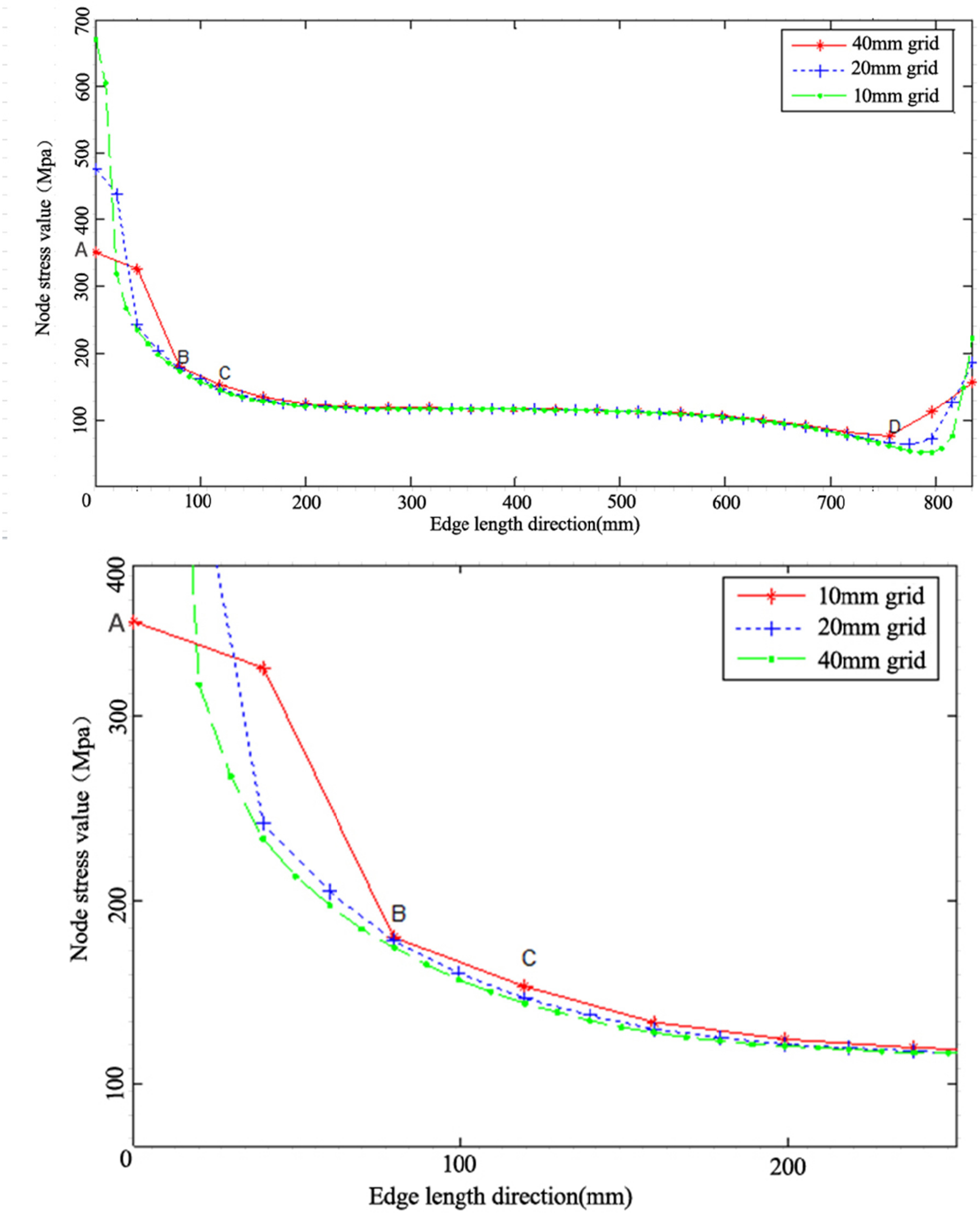


Figure 10. Comparison of node stress value on the edge of three kinds of grid size

When it's close to the stress singularity A, the calculation results divisive. Use the calculating data of 10mm grid, you can find the numerical difference between the 10% stress point near point B. According to the method above mentioned, you can use linear interpolation to estimate the stress value at the point A. Take a stress intensity factor for 1, the point stress P_A is 272(Mpa). Compare the results of three different sizes of mesh and the interpolation results, it's obvious that the interpolation results is closer to the mechanics analysis results

(293Mpa). Therefore, the results obtained by critical interpolation method have a better accuracy than any other results.

5. Conclusion

This article proposes a measure to solve the stress singularity phenomenon that appears in the structure stress analysis, and applies the measure in the finite element analysis of shield bracket. Through the simulation calculation of EPB brackets, we obtain the stress, deformation and other related parameters. Then, using interpolation method combined with MATLAB to further analyze the stress results, provide validation basis to shield bracket structure improvement.

References

Choi, S. K., Chtm, M. S., Van Tyne, C. J., & Moon, Y. H. (2006). Optimization of open die forging of round shapes using FEM analysis. *Journal of Materials Processing Technology, 172*(1), 88-95. http://dx.doi.org/10.1016/j.matdes.2009.06.046

Guan, H. S, & Gao, B. (2008). Composite earth pressure balance shield machine tool chip research of torque. *Modern tunnel technology, 45*(2), 73-78.

Laurent, DucMnea, & Anne Marie Habraken. (2005). Analysis of the sensitivity of FEM predictions to numerical parameters in deep drawing simulations. *European Journal of Mechanics A/Solids, 24*(4), 6l4-629.

Mondal, D. P., Amakrishnan, N. R., & Das, S. (2006). FEM modeling ofthe interface and its effect on the elastio-plastic behavior of metal matrix composites. *Materials Science and Engineering, 433*(1-2), 286-290. http://dx.doi.org/10.1016/j.msea.2006.06.094

Sun, A. B. (2013). Mechanical Analysis and Improvement of Research in Shield cutter and bracket. Shanghai: *University of Shanghai for Science and Technology,* 45-47.

Sun, A. B., Zhang, Y. L, Min, R, Ji, C. Q., & Zhuo, P. (2012). Design and Analysis for EPB Shield Bracket Base on Ansys. *Modern Applied Science, 6*(8), 26-31. http://dx.doi.org/10.5539/mas.v6n8p26

Sun, G. Q., & Shang, D. G. (2010). Prediction of fatigue lifetime under multiaxial cyclic loading using finite element analysis. *Materials & Design, 31*(1), 126-133. http://dx.doi.org/10.1016/j.matdes.2009.06.046