Impacting load control of floating supported friction plate and its experimental verification

Keyan Ning[1,3], Yu Wang[2], Dingchuan Huang[2], Lei Yin[2]

[1] Science and Technology on Vehicle Transmission Laboratory, China North Vehicle Research Institute. P.R. CHINA
[2] The State Key Laboratory of Mechanical, Chongqing University, P.R. CHINA
[3] Corresponding author, email address: kyning@noveri.com.cn

Abstract. Friction plates are key components in automobile transmission system. Unfortunately, due to the tough working condition i.e. high impact, high temperature, fracture and plastic deformation are easily observed in friction plates. In order to reduce the impact load and increase the impact resistance and life span of the friction plate. This paper presents a variable damping design method and structure, by punching holes in the key position of the friction plate and filling it with damping materials, the impact load of the floating support friction plate can be controlled. Simulation is applied to study the effect of the position and number of damping holes on tooth root stress. Furthermore, physic test was designed and conducted to validate the correctness and effectiveness of the proposed method. Test result shows that the impact load of the new structure is reduced by 40% and its fatigue life is 4.7 times larger. The new structure provides a new way for floating supported friction plates design.
Keywords: friction plate; impact load; fatigue life; damping holes

1. Introduction

Researchers all over the world have done a lot of researches on the friction plates, most of them focus on the research of thermal stress and wear. A lot of study about wear and contact damage were conducted: Yue [1] used finite element analysis to study fretting wear under variable friction coefficient and different contact regimes, the result turns out that when considering partial slip or running-in stage of gross sliding conditions, predictions of FE models with variable friction coefficient are closer to experimental results than the models with constant friction coefficient. Further, Pereira [2] conducted multiscale analysis of the influence of roughness on fretting wear. Kumar D [3] studied the fretting fatigue stress of heterogeneous material in solid mechanics using direct numerical simulations. Bhatti [4] analysised fretting fatigue under out of phase loading conditions using finite element method, it shows that the damage initiation location is greatly influenced by the phase difference. Yue [5] studied the effect of different debris layer factors on fretting wear. Resende [6] studied the convergence of stresses in fretting fatigue analysis. Ferjaoui [7] proposed a method to predict fretting fatigue crack initiation in double lap bolted joint. Yue [8] use finite element model to study stress singularity in fretting wear. Aleksendric´ [9] predicted the wear of friction plate by neural network method considering temperature and pressure, the prediction results of neural network method were validated by bench test. Marklund [10] obtained the variation of friction coefficient, relative velocity and temperature under the condition of oil film lubrication and boundary lubrication based on the friction characteristics test of pin and plate. Zagrodzki [11] analyzes the influence of different pressure distribution on thermal stress, generation of hot spot on friction plate during clutch bonding process and influence of friction material and deformation on hot spots based on thermo elastic theory. Li [12] established a theoretical calculation model for the wear amount calculation of clutch friction plate, and a program was set up to conduct the calculation. Zhang [13] proposed a friction plate life estimation method as the life of friction plate is the ratio between the wear volume allowed to loss and the wear volume lost for one joint.

However, the research on the impact damage of friction plate during the working process is hardly involved, with the study of the wet multi friction plate brake and the wet multi friction plate clutch; it is found that the damage caused by the impact cannot be ignored. The greater the tooth stress is, the more serious the impact damage is. Therefore, the chief work of this paper is to study the impact stress control and prolong the life of friction plate by reducing the impact stress.

The tooth stress of friction plate is affected by the parameters of the friction plate, the pitch error, the installation method and so on. As the parameter of friction plate, the pitch error and the installation way are decided by the design condition and cannot be changed easily. Therefore, in this paper, variable damping is installed to reduce the tooth stress and prolong the life of friction plate.

2. Influence of damping hole on the stress of friction plate

Damping hole is arranged on friction plate. The position and the shape of the damping hole are shown in Figure. 1. Two parameters are needed to decide the position and size of damping hole, named c, the distance from the tooth tip to the centre of the damping hole, and d, the radius of the damping hole.

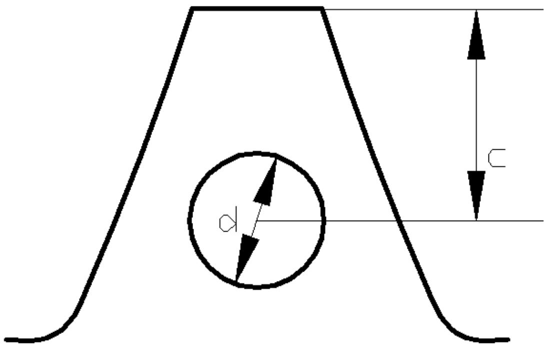


Figure. 1 Schematic diagram of position of damping hole

To study the influence of number and position of damping holes on the maximum root stress. A model of friction plate - inner hub is set up; the parameter of the model is shown in Table 1, here eccentric error exists between the friction plate and the inner hub, the amount of eccentric error is 0.025mm.

Table 1. Parameter of friction plate - inner hub model

Modulus （mm）	Tooth number	Thickness of the friction plate(mm)	Thickness of the inner hub(mm)	Eccentric error(mm)
3	122	3.5	6	0.025

ANSYS element SOLID185 was used to establish friction plate and the inner hub model, as shown in Figure. 2. Each node has 3 degrees of translational freedom. The friction plate and the inner hub are connected through point - surface contact element TARGE170 and CONTA175. The contact algorithm is Augmented Lagrange algorithm, the convergence criterion is set to 0.001. The node on both of the two side surface of the friction plate is fully constrained; torque is applied on the inner surface of the inner hub. The picture of finite element model is shown in Figure. 2, there are 4 layers of elements for the friction plate and 6 layers of elements for the inner hub in the thickness direction.

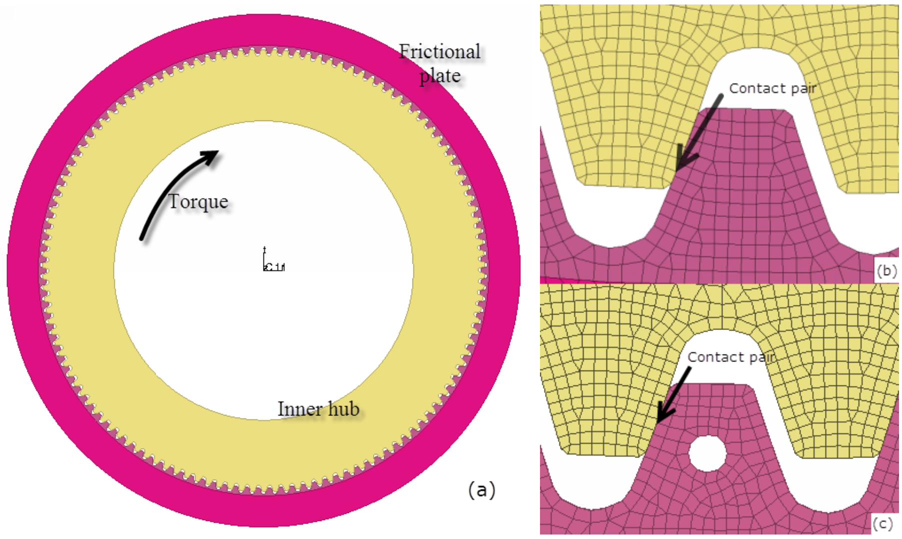


Figure. 2 Finite element model ((a) is the entire model, (b) is the contact area of the model without holes (c) is the contact area of the model without holes)

The stress distribution obtained from simulation analysis of friction plate-inner hub model without holes is shown in Figure. 3. As eccentric error exists, the stress is not uniformly distributed among the teeth of friction plate. The position where the maximum tooth stress exists is set to 0 degree position, 120 degree, 240 degree position are set in a clockwise direction as shown in Figure. 3. The damping hole is opened at a 0 degree position or both at 120 degree and 240 degrees position.

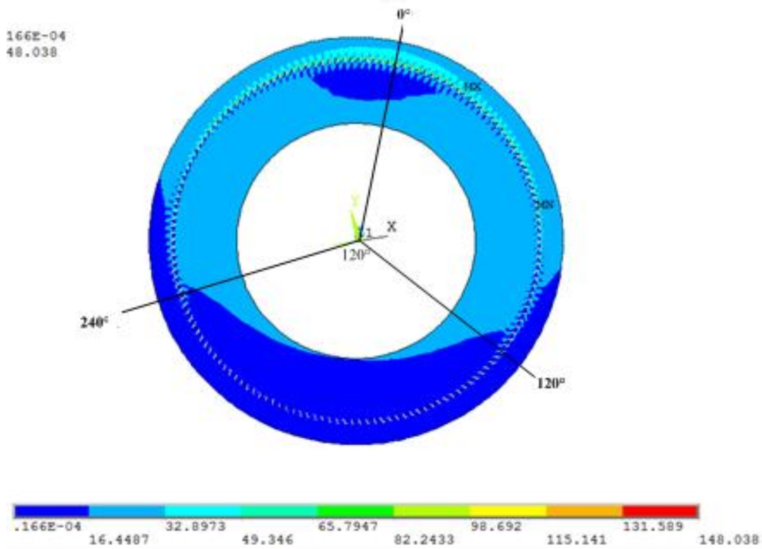


Figure. 3 Schematic diagram of the distribution of damping holes.

2.1.Stress simulation of friction plate without holes

Simulation results of complete friction plate are shown in Figure. 4 . The maximum Von Mises stress of tooth root in different position are shown in Table 2.

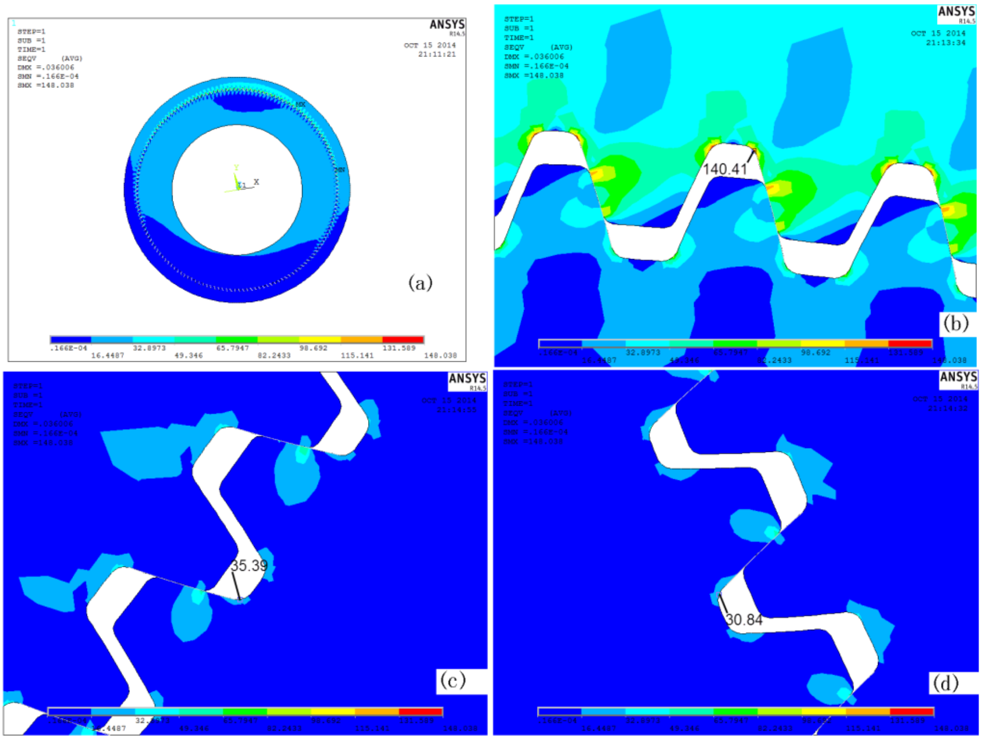


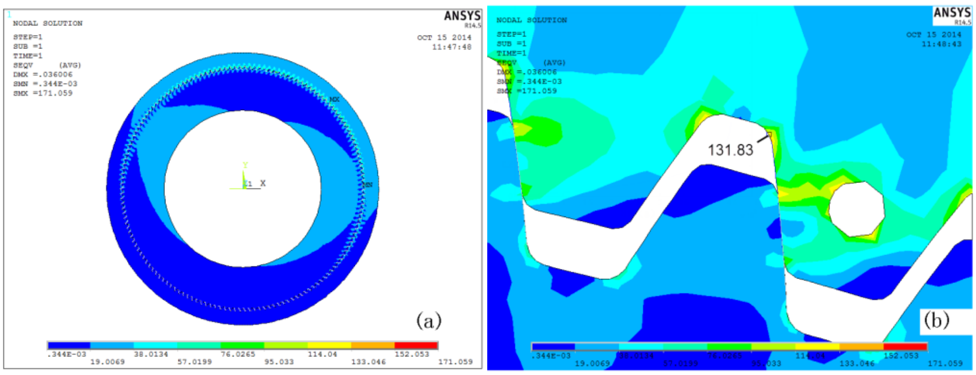
Figure. 4 Stress distribution figure of friction plate without holes ((a) is the overall stressdistribution, (b) is stress distribution of 0 degree position, (c) is stress distribution of 120 degree position, (d) is stress distribution of 240 degree position)

Table 2. Stress distribution of friction plate without holes

Position	The maximum Von Mises stress of tooth root（Mpa）
0°	140.41
120°	35.39
240°	30.84

2.2. Stress simulation of friction plate with damping holes

Stress simulation of the case where 1 damping hole is opened at 0 degree position is shown in Figure. 5. The parameters of damping hole are: d=1.6mm， c=2.87 mm. The maximum Von Mises stress of tooth root in different position are shown in Table 3.



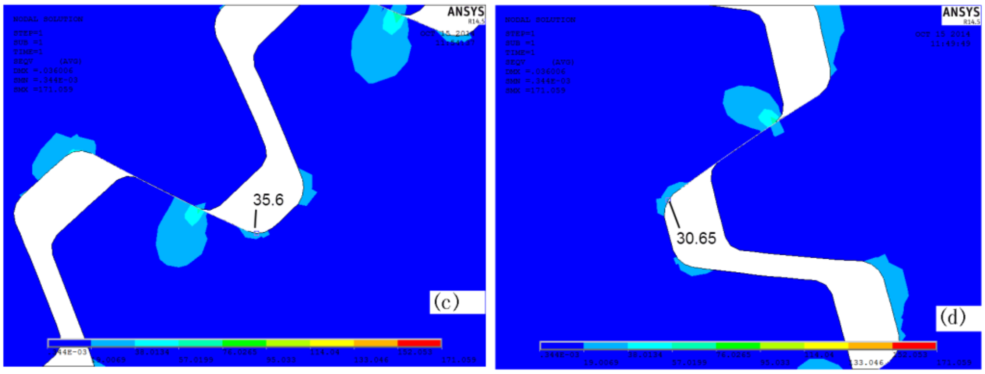


Figure. 5 Stress distribution of the case where 1 damping hole is opened at 0 degree position ((a) is the overall stress distribution, (b) is stress distribution of 0 degree position,(c) is stress distribution of 120 degree position, (d) is stress distribution of 240 degree position)

Table 3. Stress distribution of the case where 1 damping hole is opened on 0 degree position

Position	The maximum Von Mises stress of tooth root（Mpa）
0°	131.83
120°	35.6
240°	30.65

Similarly, Simulation of cases where 3, 9 damping holes opened at the 0 degree position, 1, 3, 9 damping holes opened at the 120 and 240 degrees of the position respectively are conducted. The fitting curves of simulation results of the maximum Von Mises stress of tooth root and the number of holes are shown in Figure. 6. When the damping holes are opened at the 0 degree position, the maximum Von Mises stress of tooth root decreased firstly and then increased as the number of damping holes increases. When the damping holes are opened at the 120 and 240 degrees, the maximum Von Mises stress of tooth root increases as the number of damping holes increases.

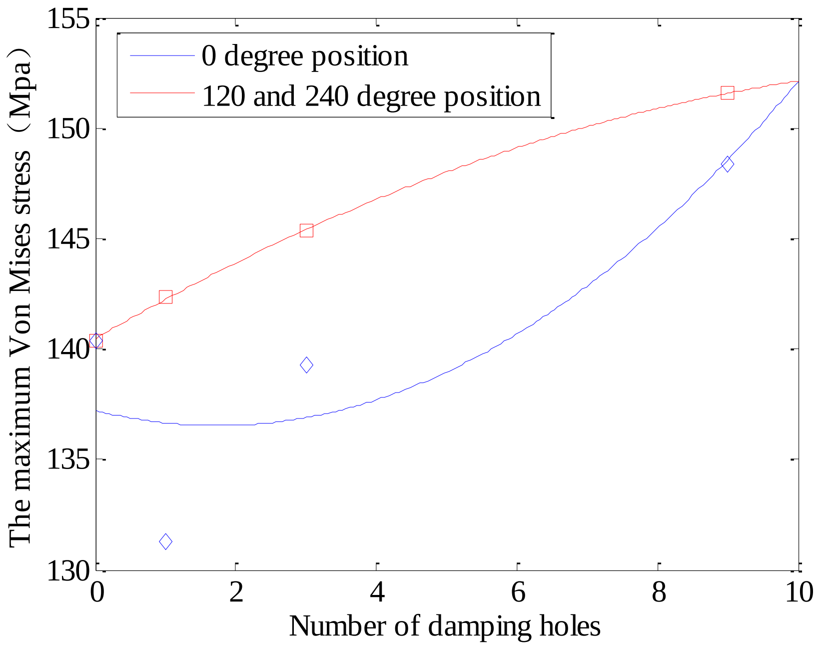


Figure. 6 The relationship between the maximum Von Mises stress of tooth root and the number, position of damping holes

3. Experimental verification

3.1. *Physical experiment*

In order to verify the simulation results, physical experiment was carried out. The experiment was carried out on the impact test rig. The friction plate is floating supported and the inner hub, driven by motor, will hit the friction plate. The parameters of test piece used in the experiment is the same with parameter shown in Table 1, but test piece have only 3 friction plate teeth and 2 inner hub teeth as shown in Figure 7. Experiments of friction plates with variable damping and without variable damping were carried out. Strain gauge is pasted on the tooth root of the middle tooth of the friction plate to measure the strain of friction plate, The damping hole is circular, filled with NBR material and its parameters are d=1.6mm， c=2.87mm.

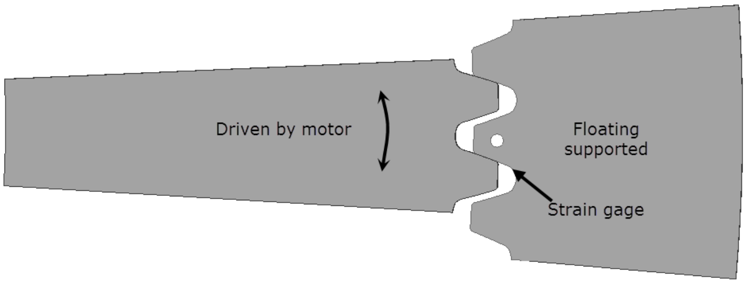


Figure. 7 Position of measuring point and damping hole

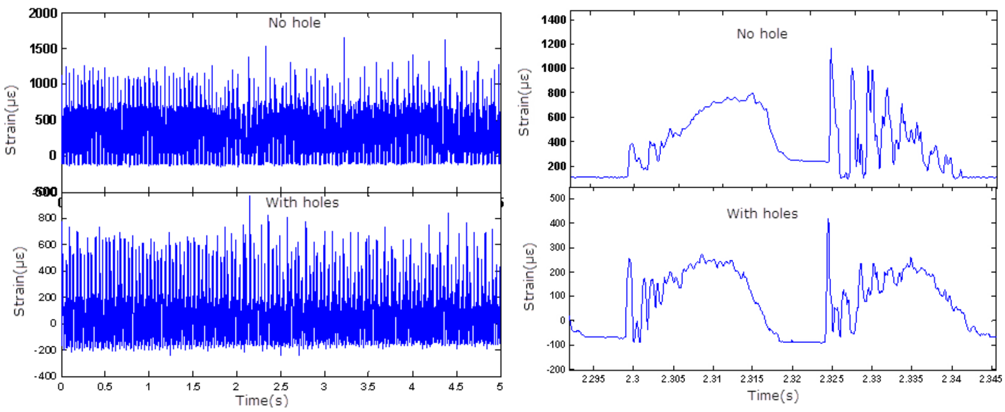


Figure. 8 Tested strain curve and its partial enlargement with and without variable damping

The tested tooth impact strain and its partial enlargement are shown in Figure. 8, the upper part shows the measured strain under the condition without variable damping and the lower part shows the measured strain under the condition with variable damping. Without variable damping, the maximum stress measured at C2 point is: 347.38Mpa, with variable damping, the maximum stress measured at C2 point is: 207.27Mpa. The maximum stress was reduced by 40% with variable damping.

3.2. *Life prediction of friction plate with and without variable damping*

According to the S-N curve of friction plate, life prediction under the condition with and without variable damping as in section 3.1 were conducted, When there is no variable damping, the life predicted is $1.46\text{x}10^6$ seconds, when there is variable damping, the life predicted is $8.31\text{x}10^6$ seconds, fatigue life increased 4.7 times with variable damping.

4. Conclusion

1) In order to reduce the impact load of floating supported friction plate, the method of damping adding was proposed. To study the influence of the position and number of damping holes to the impact load, finite element analysis was conducted; the result shows that damping holes opened at the 0 degree position can reduce the impact load. Later, experiment was carried out; the experimental results show that the proposed method is effective.

2) According to the S-N curve of friction plate, as variable damping reduces the impact load, the life of friction plate become longer.
3) The proposed method prolongs the life of friction plate by changing the body of friction plate, thus the proposed method provides a new way for the design of floating support friction plate.

References

[1] Yue T, Abdel Wahab M. Finite element analysis of fretting wear under variable coefficient of friction and different contact regimes. Tribology International, 2017,107:274-282.
[2] Pereira K, Yue T, Wahab MA. Multiscale analysis of the effect of roughness on fretting wear. Tribology International, in press(http://dx.doi.org/10.1016/j.triboint.2017.02.024).
[3] Kumar D, Biswas R, Poh LH, Abdel Wahab M. Fretting fatigue stress analysis in heterogeneous material using direct numerical simulations in solid mechanics, Tribology International, 2017,109:124-132.
[4] Bhatti NA, Wahab MA. Finite element analysis of fretting fatigue under out of phase loading conditions, Tribology International, 2017, 109: 552-562.
[5] Yue T, Abdel Wahab M. A numerical study on the effect of debris layer on frettingwear. Materials, 2016,9:597.
[6] Resende Pereira KdF, Bordas S, Tomar S, Trobec R, Depolli M, Kosec G, Abdel Wahab M. On the Convergence of Stresses in Fretting Fatigue. Materials, 2016, 9:639.
[7] Ferjaoui A, Yue T, Abdel Wahab M, Hojjati-Talemi R. Prediction of fretting fatigue crack initiation in double lap bolted joint. International Journal of Fatigue, 2015,73: 66-76 .
[8] Yue T, Abdel Wahab M. Finite element analysis of stress singularity in partial slip and gross sliding regimes in fretting wear. Wear, 2014,321: 53-63.
[9] Dragan Aleksendric´, David C Barton. Neural network prediction of disc brake performance. Tribology International, 2009, 42:1074–1080.
[10] Par Marklund, Roland Larsson. Wet clutch friction characteristics obtained from simplified pinon disc test. Tribology International, 2008, 41:824 – 830.
[11] Patankar S V, Rzemyslaw Zagrodzki. Analysis of Temperatures and Stresses in Wet Friction Disks Involving Thermally Induced Changes of Contact Pressure. SAE Technical Paper, doi:10.4271/98203.
[12] Jun Li. Theoretical estimation of wear amount of clutch friction plate of sxz1032 car. Journal of East China Jiaotong University, 1999,4:48-51.
[13] Guanxuan Xie, Siben Zhang. Llife estimation of the friction block (plate) of clutch and brake block. Forging machinery, 2000, 6:15-17.