НАУЧНАЯ СЕКЦИЯ «АРХИТЕКТУРА. СТРОИТЕЛЬСТВО. ДИЗАЙН»

УДК 72

NUMERICAL SIMULATION AND ANALYSIS OF RESIDUAL STRESS FIELD AT THE LOWER END OF SHOT PEENING STEEL RING

梁潇帝（Xiaodi Liang）[1] 王现鹏（Xianpeng Wang）[2]

1 –大连海事大学（Dalian Maritime University）

2 –白俄罗斯国立技术大学（Belarusian National Technical University）

e-mail: liaodiliang1995@163.com, 937958574@qq.com

Summary. *In this paper, an X-ray diffractometer is used to determine the residual stress of the lower end of the steel ring at the joint surface of the rim lock ring groove. Analyze the distribution of residual stress on the lower end surface of the steel ring before and after shot peening, and perform a finite element model simulation analysis on the lower end of the steel ring based on abaqus/fem-dem coupling, and realize the finite element simulation of a large number of projectiles impacting the target. The simulation results are verified. It provides a new method for the coupling finite element analysis of a large number of projectiles and structures for shot peening.*

Introduce

This paper uses X-ray diffractometer to measure the residual stress of the lower end of the steel ring at the joint surface of the rim shrinking groove, and analyzes the distribution of the residual stress on the lower end of the steel ring before and after shot peening. The shot peening process at the lower end of the steel ring is simulated and analyzed, and the PD3D unit in the abaqus software is used to realize the simulation model of a large number of projectiles impacting the surface of the part during the shot peening process. It provides a new method for the coupled finite element analysis of a large number of shots and structure of shot peening.

1. Residual stress test on steel ring surface before and after shot peening

1.1 Test objects and methods; The clamping surface of the lock ring before and after shot peening at the lower end of the steel ring. Use X350AX ray diffractometer to measure the residual stress on the clamping surface of the lower end of the steel ring before and after shot peening. The position of the 12 test points is divided into 12 points in the circumferential direction.

1.2. Test Results;

The front end of shot peening is in a state of tensile stress, and the back end of shot peening is in a state of compressive stress. The maximum compressive stress is -360MPa and the minimum compressive stress is -113MPa. The shot peening effect is relatively evenly.

2. Based on abaqus/fem-dem steel ring lower end shot peening finite element model simulation

2.1. Meshing and pre-processing

This paper uses abaqus software to realize the coupling of discrete element and finite element to simulate the impact of a large number of projectiles on the surface of the part. Figure 5 shows the DEM-FEM coupling model of shot peening at the lower end of the steel ring. 19109 C3D8R units are used, and the grid of the main contact area between shot peening and the workpiece is encrypted. The grid size of the area is 0.2mm, which is similar to the size of the nozzle. The nozzle diameter is 5mm and 66 SFM3D4R units are used[1-2]. Change the unit type to PD3D unit when inp file. Material properties are shown in Table 1.

Table 1 − Q235 steel material properties

Material	Density (tonne/mm3)	Elastic Modulus	Poisson's ratio
Q235	7.8*e-9	2.0*e5	0.3
Cast steel shot	2.7*e-9	2.5*e3	0.3

3. Verification

Select in turn the maximum residue of the simulation results at t=3*10-4, 4*10-4, 5*10-4, 5.5*10-4, 9*10-4, 2.1*10-3 in the shot peening process compared with the residual compressive stress on the lower end of the steel ring, the results show that the error of each measurement point is controlled within 5 %, see Table 2

Table 2 − Simulation and measurement

Simulation MPA	-208	-218	-170	-205	-250	-266
Measurement MPA	-201	-228.3	-176.2	-201.9	-255	-273
Relative error, %	3	4	3	1.5	2	2.5

4. Conclusion

Based on the abaqus/fem-dem coupling, the finite element model simulation analysis of the shot peening process at the lower end of the steel ring is implemented to realize the simulation model of a large number of projectiles impacting the surface of the part during the shot peening process. A new method. But in ABAQUS-CAE, we cannot directly model the particle parts. In this case, we need to manually modify the inp file or use other methods such as python secondary development or other finite element software.

References

1. H Hertz, Ueber die Berührung fester elastischer Körper, Journal für die Reine und Angewandte Mathematik (Crelle's Journal) 92 (1882) 156–171.

2. Johnson, K L, Contact Mechanics, Cambridge University Press, New York, 1985.

УДК 691.162

ОПРЕДЕЛЕНИЕ ПРЕДЕЛА ПРОЧНОСТИ АСФАЛЬТОБЕТОНА НА РАСТЯЖЕНИЕ СОВМЕЩЕННЫМИ МЕТОДАМИ ИССЛЕДОВАНИЯ

Белоусов А. В., Боровокова Е. С.

Полоцкий государственный университет

e-mail: *a.v.belousov@students.psu.by*

Summary. *Experimental studies in assessing the elastic properties of asphalt concrete samples according to the "Brazilian test" scheme. The relationship between the strength and acoustic emission (AE) characteristics of samples under uniaxial loading is determined, models created in ANSYS.*

Асфальтобетон является одним из наиболее используемых в дорожном покрытии. С каждым годом количесвто автомобилей на дорогах увеличивается, соответсвенно и интенсивность движения. Чтобы сохранить дорожное полтно в целостности, необходимо не только регулярно следить за дорожным покрытием, но и заранее определить, будет ли пригодна асфальтобетонная смесь для данного участка дороги.

Цель данной работы заключалась в определении опытным путем как работает на практике метод акустико-эмиссионной диагностики и выявления закономерностей изменения свойств асфальтобетона при нагружении методом «Бразильского теста». «Бразильский тест» подходит для определения прочности на растяжении.