Application and Second-development of Thermal-Structural Coupling Analysis of Brake Disk on ABAQUS

SHI YuMin[1, a], ZHANG ZhuLin[1,b] and WANG ZhiPing[1,c]

[1]School of Vehicle Engineering, Shandong Jiaotong University, 250023, Jinan, Shandong, China

[a]zzl790511@163.com, [b]qcxzhang@126.com, [c]wzp1953@126.com

Keywords: second-development, ABAQUS, brake disk, mechanical and thermal coupling.

Abstract. This paper introduced the development method of ABAQUS pre-processing modules using the Python script language as well as the function and the editing process of the ABAQUS script interface. Using the developed parametric GUI user platform, we can achieve different structural parameters of the brake disc model quickly and accurately, and complete the mechanical and thermal coupling calculation and analysis under different running condition. The platform has broad application prospect in the engineering design, and the method has a very good efficiency.

Introduction

ABAQUS is one of the international most advanced general nonlinear finite element analysis software, having the strong computing functions and extensive simulation performance. It was widely used in the nuclear industry, materials science, railway, petroleum chemical industry, aerospace, electronics and civil engineering general industry and scientific research. The CAE of ABAQUS is the preprocessing and post processing modules for finite element analysis, also is the man-machine interactive platform of modeling, analysis and post-processing, thus in the engineering field ABAQUS is widely used[1].

The parametric technology based on finite element method is used in ABAQUS. By the method of combining the graphical user interface (GUI) provided by the ABAQUS/CAE and object-oriented programming language Python, we can realize parameterized modeling. Especially in the complex model, tedious mesh work can automatic process. In the design process, if some characteristic parameters variable given certain engineering meaning or significance process, by changing these parameters, we could achieve the model parameter driving, realize a new round in reconstruction model calculation and greatly improve the work efficiency [2].

This paper developed a brake disc transient temperature field analysis platform based on ABAQUS GUI. Using the developed parametric GUI user platform, we can achieve different structural parameters of the brake disc model quickly and accurately, and complete the mechanical and thermal coupling calculation and analysis under different running condition. The method and development platform have broad application prospect in the engineering design.

The Introduction of ABAQUS Second-development Interface

ABAQUS second-development has the following several ways:

(1) Through the user subroutines we can develop a new model, control the process of ABAQUS and analysis the results;

(2) Through rewrite the environment initialization file, we can change the default setting of ABAQUS.

(3) Through the kernel script we can realize preprocessing modeling and analysis the post processing calculation results.

(4) Through the GUI script we can create the new graphical user interface and the user interaction.

The third method was used in this paper, through writing the Python script to control ABAQUS kernel and realize the automatic process. ABAQUS script interface is a library based on object, embedded scripting language Python, providing a set of application programming interface (API) to operate ABAQUS/CAE and realize modeling/post-processing etc. function. Interface uses the Python programming grammar scripting, but expands the Python script language, providing an additional about 500 object models.

ABAQUS Second-development Principle and Method

In order to realize ABAQUS secondary development, we can first use ABAQUS/CAE to complete numerical simulation: according to the types of simulation problems to build model, such as building a geometry model, material attribute, applying load distribution and boundary condition, setting the mesh and analysis; and then form the input file. ABAQUS/CAE in rpy files using Python script way records all operation orders, so we can use notepad to modify the rpy open files, and then form the second development program code. This paper uses ABAQUS GUI Toolkit to develop the GUI interface, following the Python language format. The designed parameters interface of the brake disc as shown in Fig. 1. When the parameters are entered and click the generate button, a model of brake disk would be generated and all setting would be completed automatically.

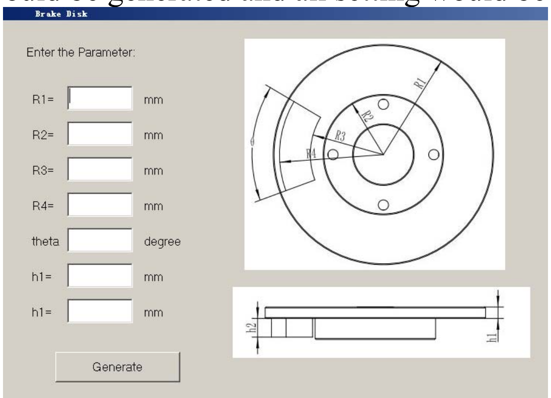

Fig. 1 the interface of the brake disc

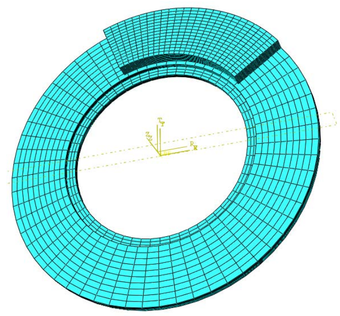

Fig. 2 1/2 brake disc finite element model

After the calculation, a result file could be formed. We could use Python script to read and write the results database, and according to the actual problems and the data storage path change the variables of data; For the convenient of analysis and check, the results were showed in the form of cloud or curve using the ABAQUS/View model.

Calculation Example

Establish the Finite Element Model. Considering the symmetry characteristics of brake disc structure and load, a half of disk finite element model is built. Brake simplified to a 3d contact model between brake disc and friction, the calculation of unit uses 8 nodes cut integral hexahedron units, grid partition was showed as Fig. 2[3-4].

Load and Constraint. Before the braking, both brake disc and wheels are in environmental temperature conditions, so the nodes of the finite element model have the uniform initial temperature. According to a car actual parameters, the pressure on the friction slices is 4 MPa. The boundary conditions which force on the friction plate should be ensure that the friction plate could only be move to the direction pressure of brake disc, the constraint of the brake disc could only rotate around by its axis in a plane, and prevent happening movement of rigid body. In addition to the friction surface, all the heat convection force on the model surface, and the friction surface heat convection forces on the surface effect unit. Brake disc and wheel section are believed to be insulated[5].

The Size of the Brake Disc Structure and Material Parameter. A analysis of a kind of car's solid disc brake, the material of brake disc is HT250, density is 7200 kg/m^3 , and Poisson's ratio is 0.3; The friction plate is composite material, density is 1500 kg/m^3 , and Poisson's ratio is 0.25. The thermal performance parameters of brake disc are showed as Table 1, the thermal performance parameters of friction plate are showed as Table 2.

Table 1 The Thermal Performance Parameters of Brake Disc

Temperature $T_d / ℃$	20	100	200	300
Heat Conduction Coefficient $k_d / w \cdot m^{-1} \cdot K^{-1}$	42.38	43.06	44.23	43.55
Specific Heat $c_d / J \cdot Kg^{-1} \cdot K^{-1}$	503	530	563	611
Coefficient of Thermal Expansion $\alpha_d / 10^{-6} K^{-1}$	4.39	11.65	12.84	13.58
Modulus of Elasticity E_d / GP_a	105	95	90	90

Table 2 the Thermal Performance Parameters of Friction Plate

Temperature $T_p / ℃$	20	100	200	300
Heat Conduction Coefficient $k_p / w \cdot m^{-1} \cdot K^{-1}$	0.9	1.1	1.2	1.15
Specific Heat $c_p / J \cdot Kg^{-1} \cdot K^{-1}$	1200	1250	1295	1320
Coefficient of Thermal Expansion $\alpha_p / 10^{-6} K^{-1}$	10	18	30	32
Modulus of Elasticity E_p / GP_a	2.2	1.3	0.53	0.32

The Determination of Braking Condition. We assume that the car's initial velocity is $v_0 = 100Km/h$, the tire rolling radius R is 0.4 m, then convent the wheel 's brake initial speed is $\omega_0 = v_0 / R = 69.44 rad/s$, the braking duration is 4s[6].

The Simulation Results. When all parameters are entered, using the developed platform to solve thermo-mechanical coupling's results. As shown by from figure 3 to figure 6, respectively for 1s', 2s', 3s' and 4s' brake disc's surface temperature. From the figures, we can see that: the radial temperature distribution on the brake disc is not uniform, inside and outside edges' temperature are obviously higher. At the time of is 1s, the highest instantaneous temperature of brake disc surface reaches 437℃, 2s is 481℃, 3s is 537℃, and 4s is 615℃. Figure 7 is the curve of brake disc surface radial node temperature changing with time, we can see that with the braking time increasing, the node temperature changing as wave, and gradually increasing.

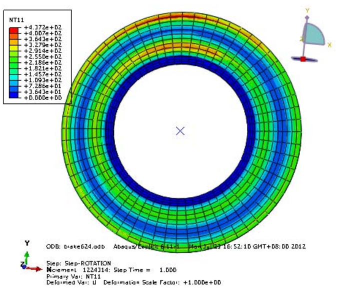

Fig. 3 1s' Brake Disc Surface Temperature

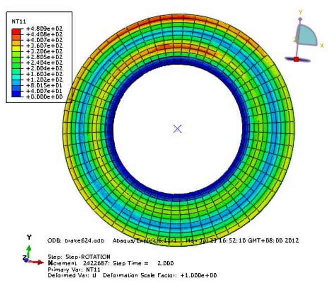

Fig. 4 2s' Brake Disc Surface Temperature

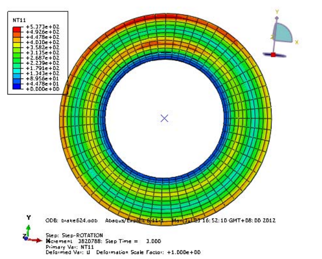

Fig. 5 3s' Brake Disc Surface Temperature

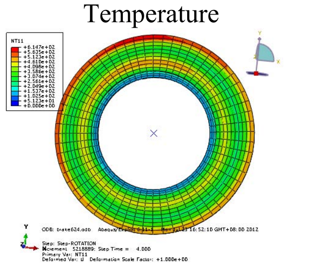

Fig. 6 4s' Brake Disc Surface Temperature

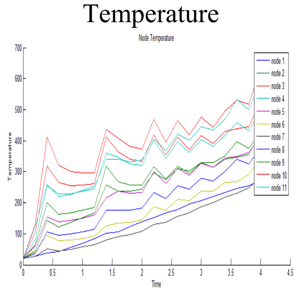

Fig. 7 the Curve of Brake Disc Surface Node Temperature Changes

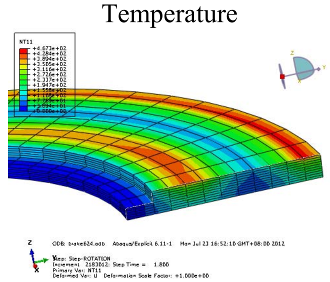

Fig. 8 the Temperature Distribution of Radial Brake Disc Axial

Fig. 8 is brake disc axial temperature distribution. We can also see that along the axial temperature presents the gradient distribution, the surface temperature is the highest, and with the depth increasing, the temperature is reducing gradually.

Conclusion

(1)On the base of ABAQUS software we used python language and ABAQUS GUI Toolkit to develop the platform, making full use of the second development processing. Through the developed dialog box could make application more easily for users, reducing work strength, improving the efficiency in the use of finite element software;

(2)During the braking process, the temperature field of brake disc's radial, axial and circumference, all of which have comparable large temperature gradient.

(3)The brake disc friction areas get friction heat flux impact and heat convection's alternately fiction, then cause the areas' temperature changes intermittently, produce periodic variation thermal stress, and cause the thermal fatigue damage of the brake disc material .

(4)The calculation of the brake disc transient temperature field could found the foundation to analysis the thermal stress and the strength of thermal fatigue.

Acknowledgements

This work was financially supported by Jinan City University Independent Innovation Project (201202078) and Scientific Research Funds of Shandong University(Z201103).

References

[1] Z. Z. Hua, H. J. Hua, Z. Ting: Application of Second-developed on ABAQUS Pre-process and Post-process. Modular Machine Tool & Automatic Manufacturing Technique. Vol. 1(1)(2009), p. 30-34

[2] Z. Qiang, M. Yong, L. S. Chao: Method and Application of Second-developed ABAQUS Based on Python. Ship Electronic Engineering. Vol. 31(2)(2011), p. 131-134

[3] Y. Yang, J.M. Zhou: Numerical simulation study of 3-D thermal stress field with complex boundary. Journal of Engineering Thermophysics. Vol. 27 (3) (2007), p. 487–489.

[4] L. Li, J. Song, Z.Y. Guo: Study on fast finite element simulation model of thermal analysis of vehicle brake. Journal of System Simulation. Vol. 17 (12) (2005), p. 2869–2872 2877

[5] C.H. Gao, X.Z. Lin: Transient temperature field analysis of a brake in a non-axisymmetric three-dimensional model. Journal of Materials Processing Technology. Vol. 129 (1-3) (2002), p. 513–517

[6] J.Y. Li, J.R. Barber: Solution of transient thermoelastic contact problems by the fast speed expansion method. Wear. Vol. 265 (3-4) (2008), p. 402–410