Finite Element Analysis of Cylinder Piston Impact Based on ANSYS/LS-DYNA

Mi Yan
School of Automation
Beijing Institute of Technology
Beijing, China, 15011058244
miyan1988@126.com

Wang Tao
School of Automation
Beijing Institute of Technology
Beijing, China, 010-68944054
wangtaobit@bit.edu.cn

***Abstract*—Nonlinear dynamic finite element analysis system ANSYS/LS-DYNA is used to analyze the characteristics of cylinder piston impacts to meet the requirements of safety and stability for cylinder. Based on the fundamental theories of explicit dynamics, a finite element impact model of cylinder piston was established. The impact force variation, the piston surface stress distribution and the relationship between maximum impact force and different piston initial impact velocity were estimated by numerical simulation. The results indicate that the impact force on the force transducer is about 81% of that on the cylinder piston owing to the collar between the piston and the force transducer, and the stress, close to the surface center of the piston, is larger than that of other areas. The results also show that different initial impact velocity don't affect impact time but only the impact force, and the maximum impact force is proportional to the initial impact velocity.**

Keywords- cylinder piston; explicit dynamics; finite element; impact force; stress

I. INTRODUCTION

Cylinder is widely used as actuator in a variety of pneumatic transmission systems, which can convert the air pressure energy into mechanical energy. [1] The movements of reciprocating, swing and spinning were made through the load of drive mechanism. If the working parts moved by air cylinder are sufficiently big and fast, a great deal of impact and vibration are generated when cylinder suddenly get stopped or reversed. So the security and stability of cylinder will be influenced and the service life will be shorted. [2] To satisfy the functional requirements, it is needed to analyze the impact characteristics of the cylinder piston.

Recently, research on cylinder mainly focus on crawling features, the designs of the buffering device and mounting structures, but few is on the impact properties of the cylinder piston. A rough estimate based on experience rather than theory was given by instruction manual of pneumatic company, they selected insurance coefficient and verified the value falls within a range of safety in production. Therefore, the analysis of the impact properties of cylinder piston is great importance for both theory and practice.

During the analyzing and testing of the position impact properties, How to obtain the accurate and real-time parameters of the impact is the key point. In this paper, a numerical simulation analysis is combined with dynamics and finite element method, to solve the problem of impact dynamics of complex structures.

In recent years, the explicit dynamic finite element method has been widely used in the field of collision impact and has showed a strong advantage in dealing with large-scale contact problems. ANSYS/LS-DYNA is a general explicit nonlinear dynamic finite element system software, it can solve a variety of nonlinear problems, such as the nonlinear contact of high-speed collision, explosion, and metal forming in three-dimensional nonlinear structure, the impact of load nonlinearity and material nonlinear problems [3]. This paper will use the non-linear dynamic finite element analysis system, ANSYS/LS-DYNA, to simulate and analyze the impact properties of the cylinder piston, and use the finite element method to build Cylinder piston collision model, therefore, the impact properties of the piston, the general rules and characteristics of the impact response is quantitatively analyzed.

The explicit dynamic finite element theory of collision analysis is introduced in section II. The building of simulation model and the settings of system parameters are introduced in section III. In impact process the impact force changes of every collision parts are detailed analyzed in section IV, as well as the piston surface stress and the stress distribution, the maximum impact force changes under different cylinder piston initial collision velocity and etc. The whole article is summarized in section V.

II. THE EXPLICIT DYNAMIC FINITE ELEMENT THEORY OF COLLISION ANALYSIS

Collision is a transient complex physical process, which belongs to Non-linear dynamics problems. The explicit dynamics finite element method in LS-DYNA is mainly used in solving the problems of collision [4]. Recent years, the explicit dynamic finite element method has been widely used in the field of collision impact and shows a strong advantage in dealing with large-scale contact problems. Unlike the static implicit one, the explicit dynamic finite element method does not need to establish the stiffness matrix and calculate the inverse operation, but using the explicit central difference method for solving finite element equations can effectively avoid the convergence problems which caused due to nonlinear problems. The speed of solving problems will be elevated, via single-point Gaussian integration and centralized quality, which has advantage of saving the computation time and storage space, as well as easy to deal with. [5]

Specific explicit dynamic finite element analysis theory is as follows:

1) Dynamics equation, as in (1)

$$M\ddot{U} + C\dot{U} + KU = F \tag{1}$$

Where, M is the mass matrix of the structure, C is the damping matrix, K is the stiffness matrix, F is external applied load vector, and U is the structure of the displacement vector.

2) If Displacement, velocity and acceleration of $0, t^1, t^2 \cdots, t^n$ are known, and structural response at the time $t_{n+1}(t+\Delta t)$ is to be determined.

The central difference is used to replace the acceleration, velocity derivative central difference, which represented as:

$$\dot{U}_t = \frac{1}{2\Delta t}\{-U_{t-\Delta t} + U_{t+\Delta t}\} \tag{2}$$

$$\dot{U}_t = \frac{1}{2\Delta t}\{-U_{t-\Delta t} + U_{t+\Delta t}\} \tag{3}$$

3) Replace equation (2) & (3) into equation (1), and we get as follows:

$$\begin{aligned} &\hat{M}U_{t+\Delta t} = \hat{R}_t \\ &\hat{M} = \frac{1}{\Delta t^2}M + \frac{1}{2\Delta t}C \\ &\hat{R}_t = F_t - \left(K - \frac{2}{\Delta t^2}M\right)U_t - \left(\frac{1}{\Delta t^2}M - \frac{1}{2\Delta t}C\right)U_{t-\Delta t} \end{aligned} \tag{4}$$

Where, $\hat{R}_t$ is payload vector, $\hat{M}$ is effective mass matrix, F_t is structural load vector.

4) Solving linear equations (4), may get the displacement vector $U_{t+\Delta t}$ at the time of $t+\Delta t$. Substituding $U_{t+\Delta t}$ into the elastodynamics balance equation (5) & (6), we may get the cell stress and strain at the time of $t+\Delta t$.

Geometric equation:

$$\varepsilon_{ij} = \frac{1}{2}\left(u_{ij} + u_{ji}\right) \tag{5}$$

Physical equation:

$$\begin{aligned} &\delta_{ij} = \lambda\varepsilon_{kk}\delta_{ij} + 2\mu\varepsilon_{ij} \\ &\sigma_{kk} = (3\lambda + 2\mu)\varepsilon_{kk} = 3K\varepsilon_{kk} \end{aligned} \tag{6}$$

5) Solving equations with the central difference method by iterative solution, until the closing conditions to meet the computing.

The central difference method is an explicit algorithm, but its condition is stable, which means that when using this method to solve problems, the size of time step must be less than a threshold determined by the nature of the problem solving equation. Otherwise, the algorithm is unstable, and time step Δt contented with:

$$\Delta t \le \Delta t^{crif} = \frac{2}{\omega_{\max}} \tag{7}$$

Where, ω_{max} is maximum natural angular frequency.

For the transient problem, explicit analysis of the time step size is small. Therefore, the problem of collisions in the simulation of piston impact test is feasible via using the explicit dynamic finite element method.

III. SIMULATION MODELING AND SOLVING

According to the finite element theory, establishing a simulation model is divided into solid modeling and finite element model to establish, among which, the finite element model includes the choice of unit type, the set of material properties and the division of finite element meshing. Finally, contact type and contact algorithm are defined, and the boundary conditions, initial loads and constraints are set to finish the solution to the model. Each section will be followed by details.

A. Solid modeling

Model uses solid modeling, in accordance with the actual size of the impact of various parts of the experimental apparatus, to build the geometric model in solidworks, with ANSYS/LS-DYNA imported, as shown in Figure 1.

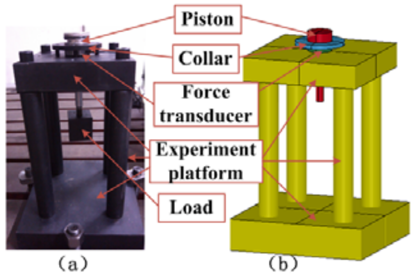

Figure 1. Model diagram of the experimental device and its corresponding.

According to the computing needs, as much as possible to keep the actual structure, details of delivery which do not affect the force between the parts are simplified. The simplifications include: 1) Joints of screw in the actual structure are simplified as direct stick together and the threads are ignored. 2) To simplify the analysis, one fourths of the model was taken in simulation. The units of cm-g-μs are used when the accuracy of the analysis is taken into account.

B. The establishment of finite element model

1) The choice of unit type

The type of unit uses solid elements solid164, which is a hexahedral element with eight nodes, each of which includes the X, Y, Z, such three directions on the state of displacement, velocity and acceleration. This low-level unit of computing has an advantage of fast speed, high accuracy and saving the machine-hour [6]. In this model, because it is a single point of force of the collision, calculation results can be serious hourglass problem. In order to prevent the hourglass problem affecting the accuracy of calculation, the full integration algorithm is adopted.

2) The set of the material properties

Various components of the experimental device materials just as follows. Cylinder piston is made of 7075 aluminum alloy, collar and experiment platform is 45 steel and force transducer is stainless steel. Three kinds of materials are all of No.15 material model *MAT_JOHNSON_COOK (Johnson-Cook constitutive model [6]) in LS-DYNA.

3) The division of finite element meshs

The number of grid directly affects the calculation accuracy and scale. For the convergence problem, the increase in the number of grid will improve the precision of the calculation, but also will increase the size of the calculation. When the grid number achieves to a certain extent, if continue to increase, calculation accuracy improved rarely, but calculation scale is big many. Therefore, in order to analyze the cylinder piston impact characteristics more accurately, the gridding of important structure is minished, especially for piston, collar and force transducer, as shown in Figure 2.

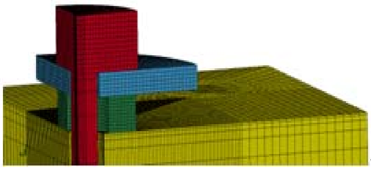

Figure 2. Local finite element model.

C. The definition of contact type and contact algorithm

In the cylinder piston impact process, the contact between different parts is face-to-face contact, so using the Surface to Surface contact type [7] of ANSYS/LS-DYNA. In the possible contact effect between different parts define the master surface and slave surface. Contact between the parts interaction through the contact algorithm to complete, and this paper adopted penalty method [8].

D. The set of the boundary, initial loads and constraints

From mathematical perspective, each kind of boundary conditions and load corresponding to the only solution, so as much as possible to fit the model boundary conditions and load with actual situation and to get the correct solution. Therefore, in the simulation, constraint the bench bottom all freedom degrees and the cylinder piston all freedom degrees except collision direction.

In the experiment, cylinder piston does a free-falling movement in the height of *h*. According to equation, $v = \sqrt{2gh}$, initial impact velocity is applied to the cylinder piston and gravity is added to the whole model.

IV. CYLINDER PISTON IMPACT PROCESS SIMULATION RESULTS AND ANALYSIS

This chapter mainly analyzes the impact properties of cylinder piston, includes: in the impact process the change of impact force, the piston surface stress and the stress distribution, the maximum impact force changes under different cylinder piston initial collision velocity and etc.

In the simulation experiment, the simulation conditions are as follows: suppose the cylinder piston falls from the height of 80mm, i.e. h=80mm, consequently, piston impact collar at the speed of 1.25m/s, i.e. v=1.25m/s. The distance between piston and collar is 0.05mm.

A. The analysis of impact force in the impact process

The simulation results of impact force on the contact interface between piston and collar, and impact force on the contact interface between collar and force transducer are shown in Figure 3. Where, curve A is the impact force changes between piston and collar, curve B is the impact force changes between collar and force transducer.

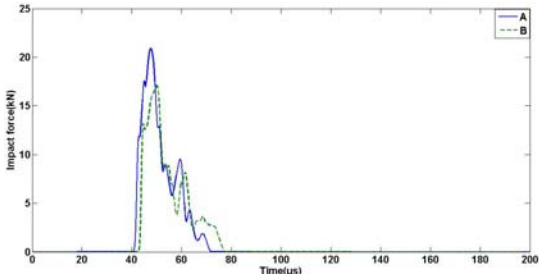

Figure 3. Impact force on the contact interface between piston and collar, and between collar and force transducer

As the pulse shapes, curve A and B seen in Figure 3, (a) Pulse shapes are quite irregular, impact force is rising rapidly and reaching its maximum after collision occurred and then decreasing, mostly due to high frequency response. (b) When t=40μs, piston impacted collar, and when t=72μs they detached, so the impact time is 32μs, and the impact peak is 20.94kN. (c) When impact was delivered to force transducer, due to the damp effect of collar, the time delay is 2μs. While, the impact time is 42μs, and impact peak is 17.08kN. These show that the impact force of force transducer is smaller than the impact force of piston, and the former is about 81 percent of the latter.

B. The analysis of the piston surface stress

The simulation analysis of impact force is the average of piston surface force. In the practical application, there is stress distribution in the piston surface. This chapter will introduce the stress and deformation of the piston through analyzing the stress and stress distribution in simulation, so as to provide reference for the optimization design of the piston.

1) The simulation get unit stress changing when radius were taken 11.5 mm, 12.6 mm, 17.6 mm, 22.6 mm and 31.2 mm, the curves are presented in Figure 4. When r=11.5mm, the maximum stress is 44.5MPa. When r=11.5mm, the maximum stress is 44.5MPa. When r=12.6mm, the maximum stress is 38.6MPa. When r=17.6mm, the maximum stress is 33.3MPa. When r=31.2mm, the maximum stress is 36.8MPa.

As shown in Figure 4, it is known that the stress change time is same to impact happened time in Figure 3, but pulse shapes are less regular than these in Figure 3. This is mainly due to the particularity of individual units, impact force increased rapidly after collision occurred, and then decreased. Stress has a smaller concussion near 0.

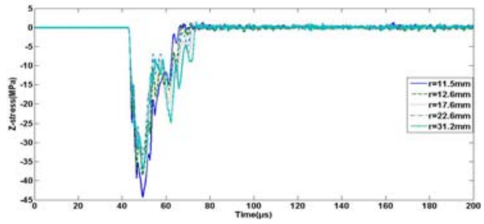

Figure 4. Z-stress comparison of different radius.

2) In the simulation experiments, selecting units of piston surface and the radius from 10 mm to 31.5 mm. Taking these radius units to do stress analysis, maximum stress comparison of different radius is presented in Figure 5.

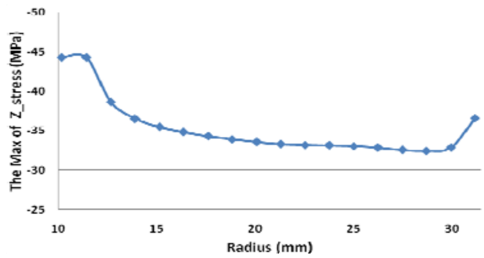

Figure 5. The maximum stress comparison of different radius.

As shown in Figure 5, a) Radius range from 10.5mm to 14mm, maximum stress is much larger. When r=11.5mm, the maximum stress is largest, it is 44.5MPa. b) With the increase of the radius, maximum stress tend to be stable. When r=30mm, the maximum stress is smallest, it is 32.9MPa. c) Radius range from 30mm to 31.5mm, maximum stress gradually increasing, and at the piston edge maximum stress reach 36.6MPa.

3) According to the above analysis results, at t=47μs, the impact force is largest, and the von-mises stress map is presented in Figure 6.

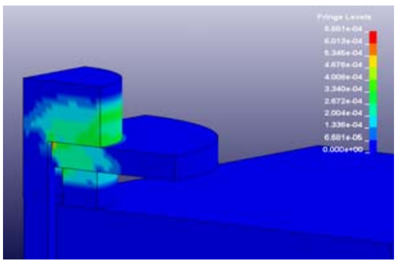

Figure 6. Von-mises stress map,when t=47μs.

As shown in Figure 6, stress wave is diffusion, maximum stress appears in internal position of piston surface, and these arethe same to the analysis of stress distribution.

C. The maximum impact force changes with different cylinder piston initial collision velocity

In addition to the piston initial impact velocity v=1.25m/s, the simulation experiment also compared impact force under different velocities. The comparison results are presented in Figure 7.

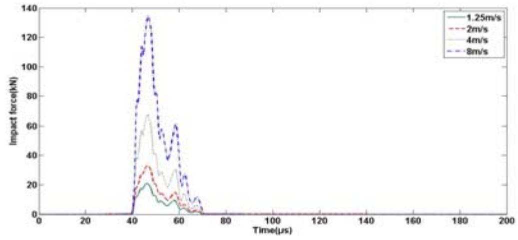

Figure 7. Comparison of impact forces under different velocities.

In Figure 7, the impact force curves under different velocities are similarity, pulse width is 32μs, and the maximum impact force is proportional to initial collision velocity. In other words, different initial impact velocity, would not affect impact time, it can only influence the size of the impact force, and the maximum impact force is proportional to initial piston initial collision velocity in a certain range.

V. CONCLUSION

In this paper, based on the fundamental theories of explicit dynamics and finite element analysis system ANSYS/LS-DYNA, the research and analysis of the cylinder piston impact characteristics is developed.

Theoretical analysis and simulation results show that: a) Due to the buffer function of collar, the impact force on the force transducer is about 81% of which on the cylinder piston. b) Piston surface stress is different, and the inside position of piston surface stress is larger, which can reduce the influence of impact by adding buffer. With the increasing of radius, the stress changes smoothly. While the outside edge stress increases, it can be strengthened by the material. c) Different initial impact velocity, don't affect impact time but only the impact force, and the maximum impact force is proportional to the initial impact velocity in a certain range. These conclusions provide strong foundation for the coming impact experiment and the theoretical basis for the optimization design, maintenance and damage evaluation of cylinder piston.

REFERENCES

[1] SMC（China）Co.Ltd, Modern pratical pneumatic technology, Beijing: Machine Industry Press, 1998.

[2] Liu Hanjun, "Factors in fluencing the dynamic characteristics of the cushion cylinder," Hydraulics and Pneumatics, vol. 1, pp. 33-37, 1987.

[3] Shang Xiaojiang, and Su Jianyu, Ansys/ls-dyna dynamic analysis method and engineering examples, Beijing: China WaterPower Press, 2008.

[4] Bai Jinze, LS - DYNA3D theory foundation and case analysis, Beijing: Science Press, 2005.

[5] Huang Qiao, Li Zhonglong, and Sha Xuejun, "Strain sensor-based wireless measurement system for bridge," vol. 10, pp. 1322-1325, 2007.

[6] J.O. Hallquist, "LS-DYNA Theory Manual," 2006.

[7] Shi Dangyong, "Analysis of explicit dynamic based on ANSYS/LS-DYNA 8.1," Beijing: Tsinghua University Press, 2004.

[8] Zhong Yang, Zhong Zhihua, and Li Guangyao, "Review on Contact Algorithms Calculating the Contact-impact Interface in Mechanical System with Explicit FEM," vol. 47, pp. 44-58, 2011.