Analysis system of power tiller’s general machine components based on VB and ANSYS

Shiqing Lu¹, Hui Jin¹, Miao He¹ and Zhe Xu²*

¹School of Mechanical Engineering, Chongqing University of Technology, Chongqing, 400054, China
²Vehicle Engineering Institute, Chongqing University of Technology, Chongqing, 400054, China
*Corresponding author’s e-mail: xuzhe@cqut.edu.cn

Abstract. Power tiller is one of the most representative agricultural machinery in hilly mountain areas in China. While the design and analysis of the power tiller’s components is still using the traditional methods with low efficiency. This paper developed a parametric finite element analysis system of the general machine components of power tiller based on ANSYS APDL and Visual Basic language. The system achieved the functions of ANSYS launching automatically, finite element parametric modeling, structural analysis and results output. The application of this analysis system is expected to shorten the design cycle and improve analysis efficiency and accuracy of the power tiller’s general machine components.

1. Introduction
Power tiller is a multi-purpose hand-held agricultural machine which is widely used in southwest China. Power tiller plays an irreplaceable role in advancing the process of agricultural mechanization in these areas. There are more than 1000 power tiller producers in China, and the production and consumption markets of power tiller have been kept fairly fast growth in recent years. However, the research investment is less due to the low product profit of the power tiller. The modern design and analysis methods are lacking of application on power tiller. Therefore, some key components of power tiller, like gears, shafts, rotary blades, are with long designing cycle and low reliability. Although some research work has been carried out to simulate the stress and deformation, and to optimize the design of power tiller’s key components, there is no systematic guidance for the analysis of power tiller’s general machine components [1-2]. With the function of secondary development of software, many analysis systems are developed and applied on general machine components of other machinery [3-5]. Considering the problems mentioned, this study carried out a second development of FEM software ANSYS via APDL language, and then established an analysis system of power tiller’s general machine components through the visualization software VB connecting and invoking ANSYS. This system presents a new approach for the optimized design of power tiller’s key components.

2. System Architecture
ANSYS provides the solution for batch command with the APDL language, a powerful tool for parametric finite element analysis. The parametric modeling and finite element analysis could be executed by running the APDL codes automatically. While the understanding and using of APDL is difficult for non-professionals, therefore, a finite element analysis system has been developed in this
study with the VB integrated environment. As shown in figure 1, the working procedure of this system contains: inputting the modeling parameters and boundary conditions via VB interface; generating the APDL codes automatically with the inputting parameters; running ANSYS in the background, executing the APDL codes; generating finite element model and analysing; displaying the results and outputting report.

![Programming flowchart of the analysis system](image)

Figure 1. Programming flowchart of the analysis system

3. Parametric modeling of power tiller’s key components

The parametric modeling methods of shaft, gear and rotary blade are discussed in this study. Shafts and gears are the most important components of power tiller. The rotary blade is the main working part of rotary tiller. These components are widely used on power tiller as standard and general parts.

The most common shaft is stepped shaft on power tiller. The shape of stepped shaft is regular. With the parameterization of the shaft’s diameter and length, the parametric model of stepped shaft could be built easily. In the shaft’s parametric model, the effect of chamfer and rounded corners were omitted, and the keyway was modeled by its length, width and relative positions. Then, the 3D model of stepped shaft could be built with “Extrude”, “Rotate” and “Boolean” operations in ANSYS by APDL language.

The parametric model of gear was based on involute equation. The involute equation on X-Y plane was expressed as below:

\[
x = r_b (\sin \phi - \phi \cos \phi) \\
y = r_b (\cos \phi + \phi \sin \phi)
\]

Where \(r_b\) is the radius of base circle. To realize the parametric modeling in ANSYS, the involute equation was transformed in to the APDL language form as shown in equation (3) and equation (4).

\[
X = \text{fai} \times \sin(\text{anglet} - \text{thet}) \\
Y = \text{fai} \times \cos(\text{anglet} - \text{thet})
\]

Where \(\text{anglet}\) is the pressure angle, and \(\text{thet}\) is involute angle. With the involute equation, the gear tooth's x and y coordinates could be calculated and sequential discrete points were generated as shown in figure 2(a). Figure 2(b) and figure 2(c) show the line model and surface model, respectively. With the extruding code of surface model, the 3-D model of gear ring could be generated as shown in figure 2(d). Boolean operation would produce features like keyway, circular hole, etc. as shown in figure 2(e).

![Parametric modeling of the gear](image)

(a) point model (b) line model (c) surface model (d) gear ring (e) gear with keyway

Figure 2. Parametric modeling of the gear
Rotary blade is another key component of the rotary tiller. The blade cuts soil by rotating at high speed for the purpose of loosening soil and weeding. And in the meantime, the rotate tiller is propelled by the counterforce of the soil during cutting. During working, the blade is subject to shock and friction, generating unbalanced and uneven forces on the rotate tiller, and leading to wear and tear of the entire blade eventually. Rotary blades could be classified as seat type blade, disc type blade and stubble crushing blade according to the different structural style. The shape and parameters of a typical seat type blade is shown in figure 3.

The parametric model of the blade was derived from bottom-up approaches. The construction of the blade model was from the low-level of primitives. The user first defined the key points of the hilt boundary, and then the related lines, planes and volumes in turn as shown in figure 4. The parametric modeling of the irregular curve was generated from the rounding command according to different fillet radii.

4. Data transmission mode between VB and ANSYS

There are two types of data transmitted between VB and ANSYS. One is the parameters for modeling and the other is the finite element analysis results. Parameters for modeling include the structural parameters, material characteristics, gridding parameters, boundary conditions and load parameters. Through the VB interface, as shown in figure 6, the modeling parameters data were written to a txt command file following the patterns of APDL language. Then ANSYS could be invoked to execute the command file through the following codes “Result=shell("C:\..\Ansys Inc\Ansys version 17.0\bin\intel\ansys version-b-p ansys_product1 -i input_file -o output_file")”.

When the finite element analysis completed, the analysis results needed to be displayed in VB interface, as shown in figure 7. The analysis results, such as the stress nephogram, could be transferred
to VB interface with the following codes “PictureX.Picture = LoadPicture ("C: \Users \.. \vb \picture")”.

5. Parametric analysis interface

Based on the study of modeling approaches above, the finite element analysis system interface for power tiller’s key components were designed in VB. As shown in figure 5, the system contains shafts, gears and rotary blades analysis modules.

![Figure 5. System startup interface](image)

Take a stepped shaft as an example, the material properties and structure parameters were set firstly in the interface of figure 6. Then the parameters for constrains and loads could be written in the similar interfaces. After the input of pre-process parameters, the program was generated with APDL format and then executed in the background of ANSYS.

![Figure 6. Parameters setting interface](image)
Once the calculation was completed, the analysis results could be extracted and displayed in the Analysis results viewer, as shown in figure 7. The analysis results could provide a reference for the structural safety condition and its improved design.

Figure 7. Analysis results display interface
6. Conclusion

This paper studied the parametric modeling methods for shaft, gear and rotary blade of power tiller, and provided several key techniques related on the secondary development of ANSYS using VB. Finally a finite element analysis system with geometric modeling, parameter settings, solution and results display was developed for power tiller’s key components. This system could be easily operated by non-professional finite element structural analysts through the simple man-machine interface. The application of this system would shorten the development cycle and improve the design efficiency and accuracy for the designers of power tiller’s key components.

Acknowledgments

This work was financially supported by “Coupling vibration mechanism and reliability analysis of power tiller’s key components project” of science and technology research program of Chongqing Education Commission of China(No KJ1600919) and “Multi-objective cooperative control of electric vehicle’s hybrid brake system”(No KJ1709229), and “Study on multigrain modular design method considering uncertainty of agricultural machinery in hilly and mountain area” (No KJ1709233).

References

[1] Sahay C S, Thomas E V, Satapathy K K. (2009) Performance evaluation of a novel power-tiller-operated oscillatory tillage implement for dry land tillage. Biosystems Engineering, 102(4):385-391.
[2] Kang A S, Cheema G S, Singla S. (2014) Wear Behavior of Hardfacings on Rotary Tiller Blades. Procedia Engineering, 97:1442-1451.
[3] Chen J, Jincheng M A, Wang Q, et al. (2014) Structural optimization of the wind turbine blade combining the ANSYS secondary development. Journal of Harbin Engineering University, 35(7):895-900.
[4] Amaral N, Rencis J J, Rong Y. (2005) Development of a finite element analysis tool for fixture design integrity verification and optimisation. International Journal of Advanced Manufacturing Technology, 25(5-6):409-419.
[5] Shi Z, Zhang R. (2010) Numerical simulation of interfacial crack growth under fatigue load. Fatigue & Fracture of Engineering Materials & Structures, 32(1):26-32.