Development of Parametric Modeling and Finite Element Analysis System for Cylindrical Spiral Tension Spring

Yunping Pan*, Yili Ming, Wenzhe Wan

School of mechanical and electrical engineering, Wuhan University of Technology, Wuhan, China

*panyunping@whut.edu.cn

Abstract. The survey shows that there are few researches on automatic modelling and finite element analysis by changing the relevant parameters of cylindrical spiral tension springs. This paper discusses the method of combining parametric technology with finite element analysis, using Visual Studio 2010 as the system development platform and Visual Basic (VB) as the program development language to establish a VB window operation interface. Automated modelling is carried out through the input feature size, and the VB event-driven feature is used to link automated modelling and automated finite element analysis to establish a parametric design and finite element analysis system for cylindrical spiral tension springs.

1. Introduction

Springs can be divided into helical compression springs, helical extension springs and helical torsion springs according to their loading methods. Among them, cylindrical helical compression springs are the most widely used, and there are many related studies on them, especially the reliability and life research results of valve springs. Into the national standard, and there is very little research on the reliability of extension springs [1]. On the other hand, with the continuous progress of computer technology and the continuous improvement of computer-aided engineering software, these new design methods can speed up the process of spring design to a certain extent.

Relevant studies have shown that the use of parametric modelling methods can significantly increase the speed of three-dimensional (3D) modelling of standardized parts. Li Yilian conducted related studies on the fracture positions and fracture modes of several tension springs actually used, and found that most of the spring failures occurred at the bends of the end rings and shackles [1]; Starting from the contact theory, Zhou Yong studied the fatigue life of the circuit breaker tension spring under impact load based on ANSYS finite element software [2]; Li Zhenhua et al. developed the SolidWorks software based on VB and realized the parametric design of the three-dimensional geometric model of the high-speed EMU spring, which is a spiral type. The 3D solid modelling of parts provides an effective method [3-5]; Shang Yuejin et al. conducted finite element analysis on the static strength, stiffness, fatigue life and modal of the cylindrical compression spring of the vehicle, and obtained relatively reasonable analysis results [6-7]; Miao Yugang established a parametric three-dimensional solid model of the tooling spring, and performed finite element stiffness analysis on the cylindrical spiral compression spring in SolidWorks Simulation, and the analysis results are basically consistent with the theoretical calculation results [8]; Feng Wang applied the secondary development technology to the entire product design process of helical compression spring modelling and finite element analysis, but did not extend it to the field of helical tension springs [9].
2. Methodology
This article introduces a method to integrate tension spring modelling and finite element analysis into an integrated analysis system. Firstly, a 3D model of the cylindrical spiral tension spring is established manually, and the characteristic size of the tension spring is defined as a variable, and using the macro recording function of SolidWorks to obtain the original code. After making relevant modifications, assigning different values to the variable parameters through the man-machine dialogue window in the VB interface to automatically generate the cylindrical spiral tension spring model; then using the relevant code to automatically import the model into ANSYS finite element software for finite element analysis to realize automatic modelling of tension springs and automatic finite element analysis.

2.1 Development ideas
ANSYS is a powerful engineering finite element simulation software, but it is inconvenient in modelling. Therefore, it is necessary to use SolidWorks to establish a spring model first, and then import the model into ANSYS for finite element analysis. At the same time, the system framework is established in Visual Studio, and the two Softwares are organically combined to establish the parametric design of the tension spring and the finite element analysis system. In terms of automated modelling, the system can perform modelling based on the feature size input by the user and output general model files; in terms of automated finite element analysis, the system performs finite element analysis according to the material characteristic parameters and boundary conditions of the tension spring set by the user, and outputs relevant calculation results. The system operation process is shown in Figure 1.

2.1.1 Secondary development of SolidWorks based on VB
SolidWorks software has a complete interface tool-API, and users can use Visual Basic, Visual C++ or other programming languages to develop SolidWorks and build their own application systems. This paper uses Visual Studio 2010 as the development platform and uses VB to perform secondary development on SolidWorks. With the help of the macro recording function of the 3D software, first
drawing the 3D model of the tension spring in SolidWorks, obtaining the original code of the modelling, and then setting its feature size as a variable, and modifying its feature point and feature surface accordingly to ensure the normal operation of the program.

2.1.2 Determine the spring characteristic size
Cylindrical tension springs are slightly different from cylindrical compression springs-the former is subjected to tensile load, while the latter is subjected to compression load; due to the different loads, the end structure of the former is generally hooked and looped. According to GB/T 2088-2009, the geometric parameters of cylindrical spiral tension spring mainly include: material diameter d, spring outer diameter D2, spring inner diameter D1, spring middle diameter D, effective number of turns n, free height H0, working load F1, etc. The characteristic dimensions of the cylindrical spiral tension spring selected in this paper are the spring outer diameter D2, the effective number of turns n, the material diameter d, the pitch p, the free height H0, and the bending radius r. The structure of the tension spring is shown in the figure below.

![Cylindrical spiral tension spring structure chart](image)

In this paper, a template part is established for a tension spring with a circular cross-section and an end structure of LII. Using this as a template, the user sets the relevant parameters when using it to derive the corresponding tension spring. When the program is running, to avoid repeated calling of model dimensions, this article names the model feature dimensions accordingly, as shown in Table 1.

| Variable parameter name     | Variable naming in the program |
|-----------------------------|--------------------------------|
| Spring diameter             | dblHelixDiameter               |
| Effective number of turns    | dblRevolution                  |
| Spring wire diameter        | dblSectionDiameter             |
| Spring pitch                | dblPitch                       |
| Direction of rotation       | blnDirection                   |

2.1.3 Related program codes
- Establish the connection between VS2010 and SolidWorks
  Private Sub Button1_Click()
- Define each variable
  Public dblHelixDiameter As Double
Public dblRevolution As Double
Public dblPitch As Double
Public dblSectionDiameter As Double
Public blnDirection As Boolean
Public boolstatus As Boolean
Public Part As SldWorks.ModelDoc2
Public Swapp As SldWorks.SldWorks

• Create SldWorks object and display SolidWorks program interface
  Swapp = CreateObject("sldworks.application")
  Swapp.Visible = True
  Part = Swapp.ActiveDoc
  Part = Swapp.NewDocument()

• Assign different values to variable parameters
  dblHelixDiameter = Val(TextBox1.Text - TextBox3.Text) / 1000
  dblPitch = Val(TextBox4.Text) / 1000
  dblRevolution = TextBox2.Text

2.2 ANSYS secondary development based on VB
ANSYS is a large-scale finite element analysis and calculation software that integrates structural, thermal, electromagnetic, fluid, and coupled field analysis types. It has complete simulation calculation and post-processing capabilities, and can solve more complex engineering problems such as nonlinearity and multi-physics coupling [10]. At the same time, ANSYS provides users with a complete scripting language-APDL, which provides a solid foundation for the secondary development of ANSYS. The VB programming language has two major characteristics: object-oriented and event-driven. The former can easily call subroutines in the object to meet the corresponding functional requirements, and the latter can trigger specific events, so as to drive the preset program associated with the event. Considering comprehensively, this paper uses VB and APDL for secondary development of ANSYS. The command stream file can be obtained through APDL, and then the code is modified and transformed accordingly to establish a VB-based Windows Form application. Finally, the user can set the material parameters and boundary conditions of the spring to perform the finite element analysis of the tension spring.

2.2.1 Defining material characteristic parameters and boundary conditions
Regarding material properties, the material characteristic parameters selected in this paper mainly include material density, Poisson's ratio, and elastic modulus. Regarding the boundary conditions, the boundary conditions of the spiral tension spring in this paper are that a fixed constraint is applied at one end of the spring, and the four key points at the other end apply an axially outward concentrated force.

2.2.2 Secondary development method and process
• Obtain the command stream file
  There are two main ways to obtain the command stream files——modifying the command stream files generated by ANSYS; in-depth studying of the way of writing ansys, and directly writing the command stream files through the APDL language [11]. In this paper, the command stream acquisition method is adopted as the former. First, performing finite element analysis on the tension spring in a graphical user interface on ANSYS, and exporting its log file, and then obtaining the original command stream file. Then, deleting the irrelevant code in the original file and modifying the file code related to parameterization. After further debugging, the command stream file of the tension spring can be obtained.
  • Use VB to start ANSYS
    VB mainly uses the Shell function in the VB programming language to start ANSYS for background analysis, the specific codes are as follows.
Shell("C:\Program Files\ANSYS Inc\v182\ansys\bin\winx64\MAPDL.exe -p ansys -np 2 -lch -dir C:\Users\33076\Desktop -j spr -s read -l en -us -b -i C:\Users\33076\Desktop\spring.mac -o C:\Users\33076\Desktop\spring.lgw", 1, False, -1)

Among them, ‘C:\Program Files\ANSYS Inc\v182\ansys\bin\winx64\MAPDL.exe’ is the installation location of the finite element analysis execution software on the computer; ‘-p ansys’ is the product module of ANSYS; ‘-dir C:\Users\33076\Desktop’ is the command stream file and the save path of the file generated during the analysis; ‘-j spr’ means that the Job name of the ANSYS project file needs to be set to spr; ‘-i C:\Users\33076\Desktop\spring.mac’ is the command stream file used as the input file for batch processing; ‘-O C:\Users\33076\Desktop\spring.lgw’ is to set the save path and file name of the output document

Automated finite element analysis operation process

After obtaining the command stream file for the finite element analysis of the tension spring, in order to facilitate the parameter assignment, it is also necessary to convert it into VB usable codes, and the conversion format is ‘PrintLine(2, "PrintLine(1, ", str & ",")")’, where ‘str’ is the string variable, which represents the APDL statement in the command stream file. After the assignment is completed, the new APDL command stream file is output, and finally the finite element analysis of the tension spring can be completed by calling the shell function. The specific analysis process is as shown in the figure below.

![Diagram of automated finite element analysis process](image)

Figure 3 Automated finite element analysis process

2.3 System interface design

According to the analysis of system functions, it can be divided into three interfaces——the main interface of the system, the parametric modelling interface, and the finite element analysis interface. The specific interface design is as shown in the figures below.
3. Results
In this paper, a finite element analysis is carried out for a cylindrical spiral tension spring with a circular cross section and an end structure of LII. The characteristic dimensions of the spring are shown in Table 2, and the relevant theoretical values refer to GB/T 2088-2009. The spring material is 65Mn. At room temperature, its elastic modulus is 211Gpa, Poisson's ratio is 0.288, and the material density is 7820kg/m3. A fixed constraint is applied at one end of the spring, and an axial load of 500N is applied to the four key points at the other end.

| Spring outer Diameter (mm) | Effective number of turns (rad) | Spring wire Diameter (mm) | Spring pitch (mm) | Corner radius (mm) |
|----------------------------|--------------------------------|---------------------------|------------------|-------------------|
| 30                         | 10.5                           | 5                         | 5                | 6                 |

3.1 Spring stiffness analysis
The 8-node SOLID185 3D solid element selected in this paper is used to discretize the spring model, and the mesh is divided into 35355 nodes and 165872 elements, its deformation under the action of 500N axial tension is 7mm. According to GB/T 2088-2009, the initial tension F0 of this type of spring is 232N, and the standard spring stiffness is 37.6N/mm; according to formula (1), the spring stiffness obtained by finite element analysis is 38.3N/mm; at the same time for coil springs, If the material used is isotropic, according to its stiffness calculation formula (2), its theoretical spring stiffness is 39N/mm. By comparison, the error of the simulation analysis value is within the acceptable range, indicating that the finite element analysis of the tension spring in this article is reasonable.
\[ F' = \frac{F - F_0}{\Delta L} \tag{1} \]

\[ F' = \frac{G d^4}{8D^3n} \tag{2} \]

### 3.2 Spring stress analysis

Figure 7 is a cloud diagram of the static stress strength analysis of the spring under a load of 500N. It can be seen from the figure that the maximum Von Mises stress of the spring is about 544MPa, which appears on the inner surface of the working ring. After applying a load to the spring, the shear stress distribution of the spring is shown in Figure 8, and the maximum shear stress is 207MPa, which is located inside the connection between the effective loop part of the spring and the shackle. Relevant studies have shown that the bending stress and shear stress of the shackle will be generated after the bending part of the shackle is loaded. Among them, the shear stress \( \tau \) can be approximated by the formula (3) [13]. After approximate calculation, the shear stress of the spring is 194MPa. Comparing the simulation result with the analytical solution, the error between the two is 6.70%, which is within the engineering error range, which further illustrates the rationality of the finite element analysis results.

\[ \tau = \frac{8F_2D r_3}{\pi d^3 r_4} \tag{3} \]

#### Figure 7 Spring equivalent stress

#### Figure 8 Spring shear stress

### 4. Conclusion

In this paper, the parametric technology and finite element analysis method are combined to establish a parametric design and finite element analysis system for cylindrical spiral tension springs. In actual use, the system can perform modelling according to the feature size input by the user, and the modelling accuracy is better than the spring design of Manufacturing Today. After outputing the general model file, the system performs finite element analysis according to the material characteristic parameters and boundary conditions of the tension spring set by the user. This paper only carries out secondary development for the extension springs with circular cross-section and LII end structure, and more spring types can be added in the future to enhance the applicability of this system.

### References

[1] Li Y L, Wang H, Zhang J R, Li K F. Research on the reliability of the tension coil spring[J]. *Physical Testing and Chemical Testing (Physics Section)*, 2004(11): 547-549+572

[2] Zhou Y, Zhou B Y, Li X S. Reliability analysis of tension spring based on contact theory[J]. *Machinery Design and Manufacturing*, 2009(09): 111-113

[3] Li Z H, Shang Y J, Wang H. Parametric design of EMU spring based on the secondary development of SolidWorks[J]. *Computer Aided Engineering*, 2015, 24(5): 38-41
[4] Zhang X Q. Parametric modeling of cylindrical spiral compression spring[J]. Mechanical and Electrical Engineering, 2011, 28 (3): 305-309

[5] Yu Q, Qu Y S. Parameterized design of cylindrical spiral compression spring based on SolidWorks[J]. Journal of Donghua University (Natural Science Edition), 2012, 38(5): 605-608

[6] Shang Y J, Wang H, Meng G P. Three-dimensional finite element analysis and fatigue life estimation of coil springs for new type freight car bogies[J]. China Railway Science, 2008, 29(1): 65-69

[7] Xue H, Li Y C, Liu W X. Research on the test method of coil springs for rolling stock[J]. Journal of Lanzhou Jiaotong University, 2013, 32(4): 139-142

[8] Miao Y G, Zhao F. Parametric modeling and analysis of a tooling spring[J]. Journal of Shaanxi Institute of Technology (Natural Science Edition), 2014, 30(3): 24-28

[9] Feng W, Chen H, Yang Y L, Xu S. The overall parameterized design and finite element analysis of spring[J]. Software Guide, 2020, 19(06): 160-165

[10] Gong S G, Xie G L, Huang Y Q. ANSYS parameterized programming and command manual[M]. Machinery Industry Press, 2009

[11] Yu X M. Development and Application of Drum Brake Auxiliary Calculation and Analysis System[D]. Wuhan University of Technology, 2019

[12] Zhang Y H, Liu H H, Wang D C. Spring Handbook[M]. Beijing: Mechanical Industry Press, 1997.8