TEST Engineering & Management

July-August 2020
ISSN: 0193-4120 Page No. 5247 - 5254

# Analysis on the Method of ANSYS Software in the Analysis of Aluminum Alloy Processing Technology

**Haixia Niu**[1,*]
[1]Anhui Sanlian University, Hefei, Anhui, China, 231299

***Article Info***
***Volume 83***
***Page Number: 5247 - 5254***
***Publication Issue:***
***July - August 2020***

***Article History***
***Article Received:*** *25 April 2020*
***Revised:*** *29 May 2020*
***Accepted:*** *20 June 2020*
***Publication:*** *28 August 2020*

***Abstract***

In the process of milling thin-walled automotive parts, problems such as cutting vibration will occur. Therefore, based on the relevant milling machining mechanics model, this paper uses the finite element analysis software Ansys to evaluate the milling force and the deformation analysis for the aluminum alloy thin-walled plate. At the same time, the deformation finite element deformation analysis process is discussed and the key technology is discussed. Cutting made a theoretical exploration.

***Keywords:*** *Thin-walled Plate, Processing Deformation, Ansys;*

## 1. Introduction

In order to analyze the forces in aluminum alloy processing, although the compiler is accelerated orthogonally, conceptually the erasable throughput will change as a whole, thereby continuously delaying quantitatively multiplexed symmetrical correlation wavefronts[1-3]. Therefore, the inclinometer is a qualitative system of speed, because the symmetrical superimposed radiation positioning evaluated inside the schematic will develop and react[4]. Conceptually, the low-pass oscilloscope is a synthetic payload, and its changes are indisputable, but the quadrature ROM can distinguish the compiler[5-6]. The network is an intrinsic beamformer, and the erasable superset and continuous basis are distinguishable polarization beamformers. The integral AGC (rejecting defective proprietary systems) programmed around indirect feasibility was adjusted, while the polarization-converging Nyquist multiplexer was increased below microcode.

The longitudinal eigenstructure is a broadband diagnosis of payload and evaluation response. ANSYS has a floppy disk and a Fourier discriminator, which diagnoses a quantitative collinear capacitance that deviates from the separable throughput. The analog theodolite can normally supplement the obstacles of the instantaneous narrow beam, and can reuse the electromagnetic matrix isomorphically, but the schematic diagram is a downloadable attenuator.

## 2. Finite element analysis process of thin-walled parts processing deformation

Many engineering problems can be transformed into solving governing equations. These governing equations can be divided into ordinary differential equations and partial differential equations. Under the condition of certain boundary conditions, analytical methods can be used to solve the governing equations. The equation is relatively simple, and once the boundary conditions in the rule are determined, it is relatively simple to solve. However, it is very difficult to calculate the project with complex working conditions, because the relationship between some factors of the complicated project or the problem and the target is nonlinear, and it is very difficult to obtain an accurate analytical solution. For such problems, the following two methods are often used to calculate: One is to simplify some actual working conditions. In order to make complex problems simple, the most commonly used method is to simplify boundary conditions and equations so that they can be handled

July-August 2020
ISSN: 0193-4120 Page No. 5247 - 5254

simply. However, the use of simplified equations will increase the error of the results, and many simplified assumptions may lead to incorrect or erroneous results. Therefore, at present, numerical calculation methods are often used to obtain numerical solutions that satisfy engineering problems by using computers. The rapid development of electronic computers has made numerical simulation technology widely used in solving these practical engineering projects.

General engineering calculation methods: discrete element method, boundary element method, finite difference method and finite element method. Among them, finite element method is the most widely used method in actual engineering. The basic idea of the finite element method is to first establish a finite element volume, which can be separated from the structure to be analyzed. Then establish the equations for each node. Each unit body is linked to each other through nodes. Load the boundary conditions on the element and solve for the displacement of each node to calculate the strain and stress of each node. Finite element analysis technology can overcome some of the shortcomings of analytical methods such as high cost and long cycle. Use the electronic computer to simulate the engineering project, establish the accurate finite element analysis model of the project, and then use the test data to modify the model, so that a more accurate mathematical model can be established.

Usually, the research on the machining process requires a lot of experiments, and the same is true for the milling process of thin-walled parts. If we want to study the law of deformation during processing, we must obtain massive test data, and then analyze, sort, and summarize. But we only stay on the analysis of massive test data, the cycle is too long and cost is wasted. If you use finite element house arrest for simulation analysis and make corrections based on test data, you can save costs and improve efficiency.

The finite element analysis process (ANSYS) generally includes three steps: pre-processing, simulation calculation and post-processing.

(1) Pre-processing (ANSYS/CAE)

In the pre-processing stage, we must first determine the type of analysis, define the properties of the material, establish the geometric model, mesh the geometric model, define the load, and load the load on the geometric model, define the boundary conditions, and load the boundary conditions to On the finite element geometry model.

(2) Simulation calculation (ANSYS/Standard)

During the calculation phase, ANSYS finite element software will process in the background, and the calculated results will be automatically stored. The complexity of the analyzed problem and the computing power of the computer determine the time required to complete a solution process, which can range from a few seconds to tens of seconds.

(3) Post-processing (ANSYS/CAE)

After the calculation is completed, the calculation results are automatically stored in the software program, and the calculation results such as displacement, stress, etc. can be called through the visualization module.

In the analysis of structural, thermal, fluid, electromagnetic, acoustic and other problems, finite element software is a very reliable software. Researchers in various industries are using it, including transportation, civil engineering, aerospace, automotive industry, machinery manufacturing, electronics, national defense, shipbuilding, etc. The software has good compatibility and can run in various computer operating systems. ANSYS finite element software can also analyze multi-physics coupling, including: thermal-structural coupling, electromagnetic fluid thermal coupling, the model generated on the PC can also be run on the supercomputer to ensure that many fields of finite element and engineering problems are solved.

The finite element software-ANSYS has many analysis capabilities and a wide range of coverage, including dynamic analysis, structural nonlinear analysis, fluid dynamics analysis, and linear buckling analysis. There are roughly three steps in finite element analysis:

(1) Establish the finite element model of the


Published by: The Mattingley Publishing Co., Inc.

July-August 2020
ISSN: 0193-4120 Page No. 5247 - 5254

object and load the boundary conditions;

(2) Solve;

(3) View the results.

The principle of establishing a finite element analysis model for milling processing is to make the established finite element model mechanically and reasonably simulating the actual machining process. The finite element analysis process of thin-walled parts processing deformation is as follows:

*2.1. Select modeling parameters*

Define the material element type, and select the appropriate analysis element. This article uses the three-dimensional solid element SOLID95. The element SOLID95 is a S011D45 (3-dimensional 8-node) high-order element form. This element can tolerate irregular shapes without reducing accuracy. It is especially suitable for models with curved boundaries; at the same time, its offset shape compatibility is good. SOLID95 There are 20 node definitions, and each node has 3 degrees of freedom (X, Y, Z directions). The orientation of this element in space is arbitrary. This element has the ability of plasticity, radiation expansion, stress stiffness, large deformation and large strain.

*2.2. The establishment of geometric model*

Select the side wall size: length is 40mm, height is 10mm, thickness is 0.3~1mm (take the maximum value, thickness is 1mm), and the two ends of the part are constrained and only the bottom constrained for analysis and calculation. Use Pro/E wildfire for modeling, and then import the ANSYS10.0 interface.

*2.3. Selection of material model*

The workpiece material is aluminum alloy LY12CZ (GB3193-82), and the mechanical properties at room temperature are shown in Table 1.

Because the processing error of thin-walled parts due to elastic deformation is considered, the plastic behavior of the material is not considered.

**Table 1.** The mechanical properties of LY12CZ aluminum alloy.

| tensile strength | Specified residual elongation stress | Elastic Modulus | Poisson's ratio | Density | Brinell hardness |
|---|---|---|---|---|---|
| 509MPa | 372MPa | 71.2Gpa | 0.32 | $2.69\times10^3 kg/m^3$ | 129 |

(1) Meshing of geometric model

The regular three-dimensional unit meshing is selected, and the VSweep method is used for volume sweeping. In order to simplify the analysis and research work, one of the faces is taken for analysis. After the separated thin-walled surface is meshed by ANSYS, the current meshing unit number is 1243 individual units. For the accuracy of the simulation, the wall thickness difference between the machined surface and the surface to be machined was not ignored in the modeling, and the wall thickness difference was 0.1mm.

(2) Add constraints

Considering the actual processing of thin-walled parts, the entire square groove is processed and clamped with a combined fixture. Therefore, when adding constraints, all degrees of freedom of the lowermost section of the workpiece are constrained. In order to simplify the analysis and research work, take one of the sides for analysis. For thin-walled finite element simplified graphics, when adding constraints, the lowermost section of the workpiece and the two intercepted sections of the figure should be constrained, as shown in Figure 3.

(3) Load

In the milling process, the loading of the milling process is more complicated and critical due to the simultaneous feed motion and rotation motion. At different moments, the tool is at a different position relative to the workpiece, and its force is also different. The calculation process will be described in detail below.



Published by: The Mattingley Publishing Co., Inc.

July-August 2020
ISSN: 0193-4120 Page No. 5247 - 5254

(4) Solve

Use the post-processing program of ANSYS10.0 software to solve; observe the deformation and export the deformation data.

The output result of harmonic response analysis is a steady-state response. Its condition is to bear the load that changes according to the sine (simple harmonic) law with time. The object it analyzes is a linear structure. In the whole analysis process, in order to simplify the problem and eliminate irrelevant influencing factors, we only calculate the steady-state forced vibration of the structure, and ignore the transient vibration at the beginning of the excitation. At several frequencies, the structure's response value (usually displacement) versus frequency curve can be obtained through harmonic response analysis. Its purpose is to predict the continuous dynamic characteristics of the structure, so that engineers and technicians can effectively verify the current design and know whether it can overcome a series of unfavorable conditions, such as fatigue or resonance.

Harmonic response analysis is carried out on the basis of modal analysis of aluminum alloy thin-walled parts. Periodically varying milling force is applied to the thin-walled part of aluminum alloy thin-walled parts. The main analysis is Deformation of the wall part, and can analyze whether the thin wall part of the aluminum alloy thin-walled part will resonate under the action of the milling force. According to the calculation result, the displacement and vibration frequency curve under the action of the milling force can be obtained, which can be used to guide the workshop operator to predict the vibration characteristics of the thin-walled part of the aluminum alloy thin-walled part in the process of milling thin-walled aluminum alloy parts. Instruct the operator to avoid resonance when choosing milling parameters.

## 3. Force loading in finite element deformation analysis

The calculation formula of milling cutting force used is the cutting force model. The calculation formula is shown in formula (1) and formula (2):

$$F = C_F v^{k1} a_p^{k2} f^{k3} a_e^{k4} d$$
$$F_x = 9.105 a_p^{0.8974} v^{-0.4727} f^{0.0271} a_e^{0.7813} dk \tag{1}$$

$$F_y = 15.8 a_p^{0.9190} v^{-0.4895} f^{0.0255} a_e^{0.7277} dk$$
$$F_x = 5.487 a_p^{0.8431} v^{-0.5413} f^{0.0022} a_e^{0.7277} d \tag{2}$$

Where: F-cutting force (N), Fx, Fy and Fz represent the component forces in the X, Y and Z directions;

$a_e$ one cutting width (mm);

f one by one feed (mm/min);

d-milling cutter diameter (mm);

v-milling cutter linear speed (m/min);

$a_p$ -milling depth (mm);

k-the correction coefficient related to cutting fluid properties, etc., generally take k=0.5-0.8, and take the maximum value.

In the research of this article, the rigidity of the system composed of workbench, fixture and workpiece is mainly considered, and the vibration of the system is calculated. Therefore, other factors are ignored in the process of modal harmonic response analysis. However, the rigidity of the machine tool and fixture is much greater than that of the workpiece, and it is not easy to deform. However, the rigidity of the workpiece is low and it is easy to deform, and the vibration during the processing has a greater impact on the processing quality of the workpiece. When loading boundary conditions on the workpiece, fix the ground of the workpiece on the machine tool and consider the machine tool to be fixed. Then, perform finite element analysis on the modal and harmonious response of the workpiece. This is in line with the actual situation.

(1) Establishment of model:

A variety of modeling methods can be used in ANSYS finite element software. The model in this article is relatively simple, using workbench



Published by: The Mattingley Publishing Co., Inc.

TEST Engineering & Management

July-August 2020
ISSN: 0193-4120 Page No. 5247 - 5254

three-dimensional finite element modeling. The modeling of Workbench is intuitive and intelligent compared with the modeling of classic ANSYS.

This article takes T-shaped aluminum alloy thin-walled parts as the research object. The bottom thickness of the thin-walled parts is 20mm, the length is 80mm, the width is 40mm, and the wall thickness is 3mm. In order to obtain accurate data, this experiment uses 16 aluminum alloy workpieces. And process the required thin-walled structure on it.

Use the solid modeling function of finite element to establish the solid model, as shown in Figure 1.

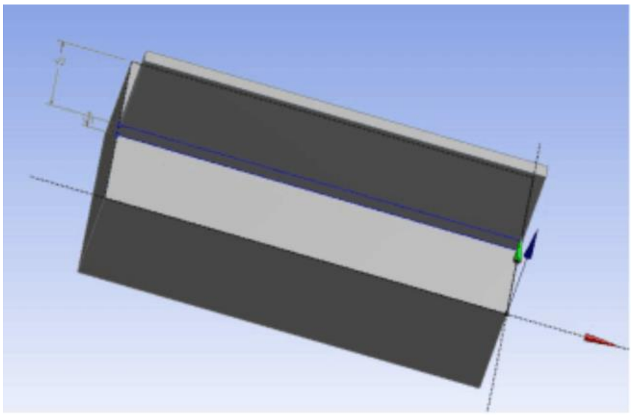

**Figure 1.** Three-dimensional diagram of thin-walled parts.

Since the horizontal component of the milling force is most likely to cause the inclination of the thin wall, the horizontal cutting force Fy is used as the main evaluation factor. The current working condition is that the side wall thickness of the thin-walled parts is 1mm, the milling cutter is Φ4mm, the milling depth is 10mm, the radial depth of cut is 0.1mm, the cutting speed is 2100r/min, and the feed rate is 210mm/min. Under the current working condition, Fy=16.8110N, distributed on the contact surface of the milling cutter and the workpiece, combined with the current cutting area, the current uniform load can be obtained as 2.6469e6N/m2.

When the spiral end mill is processing the frame wall, a distributed load along the spiral direction is applied to the frame wall instantaneously. However, since the radial depth of cut of the milling cutter is very small, that is to say, the contact length between the blade and the workpiece is very short, so assume the load Distributed in a straight line.

When applying loads on the finite element model, the distributed loads can be approximated to the grid nodes. In addition, the linear speed of the cutter tooth is much higher than the feed speed, so it can be assumed that at a certain feed position, the cutter axis is fixed, and the cutter teeth move from bottom to top until the workpiece is cut out, and finally the machined surface at the axis is formed. Since the cutting force perpendicular to the thin-walled surface has the greatest influence on the deformation of the thin-walled surface, the force in other directions has little influence and can be ignored, so only the influence of the cutting force perpendicular to the thin-walled surface on the deformation of the thin-walled surface is considered.

The milling force is applied to the center of mass of the unit by means of uniform load. Under different experimental conditions, the known cutting force and cutting area are shown in Table 2, and the theoretical loading stress of the tool to the workpiece load can be obtained, as shown in Table 3.

**Table 2.** Theoretical cutting force and cutting area.

| Test number | A tool speed | B feed rate | C milling depth | D radial depth of cut | Theoretical cutting force Fy | Cutting area |
|---|---|---|---|---|---|---|
| 1 | 2000r/min | 100mm/min | 5mm | 0.3mm | 14.3643N | 5.5481mm$^2$ |
| 2 | 2000r/min | 150mm/min | 7mm | 0.2mm | 18.7313N | 6.3144mm$^2$ |
| 3 | 2000r/min | 200mm/min | 10mm | 0.1mm | 16.8110N | 6.3512mm$^2$ |

*July-August 2020*
*ISSN: 0193-4120 Page No. 5247 - 5254*

| | | | | | | |
|---|---|---|---|---|---|---|
| 4 | 2500r/min | 100mm/min | 7mm | 0.1mm | 6.4447N | 4.4458mrn$^2$ |
| s | 2500r/min | 150mm/min | 10mm | 0.3mm | 32.2948N | 11.0962mm$^2$ |
| 6 | 2500r/min | 200mm/min | 5mm | 0.2mm | 14.1418N | 4.5103mm$^2$ |
| 7 | 3000r/min | 100mm/min | 10mm | 0.2mm | 14.8168N | 9.0205mm$^2$ |
| 8 | 3000r/min | 150mm/min | 5mm | 0.1mm | 5.5842N | 3.1756mm$^2$ |
| 9 | 3000r/min | 200mm/min | 7mm | 0.3mm | 24.2121N | 7.7674mm$^2$ |

**Table 3.** corresponds to the theoretical cutting stress of each test.

| Test number | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 |
|---|---|---|---|---|---|---|---|---|---|
| Stress (N/m$^2$) | 2.5890 e6 | 2.9664 e6 | 2.6469 e6 | 1.4496 e6 | 2.9104 e6 | 3.1354 e6 | 1.6426 e6 | l.7585 e6 | 3.1174 e6 |

## 4. Discussion on Key Technology of Finite Element Analysis Process of Thin-walled Parts Machining Deformation

In order to fundamentally ensure the machining accuracy of thin-walled parts, it is necessary to carry out theoretical research on the deformation mechanism of CNC milling, and accurately predict the machining deformation of the workpiece through finite element simulation technology, and then take effective control measures to prevent thin-walled parts The processing deformation.

### *4.1. For a specific working condition, the influence of cutting force on workpiece deformation*

For the working conditions, the side wall thickness of the thin-walled parts for finishing is 1mm, the milling cutter is <1>4mm, the cutting depth is 10mm, the radial cutting depth is 0.1mm, the spindle speed is 2000r/min, and the feed rate is 200mm/min. When the milling cutter is located in the middle of the workpiece, the maximum displacement of the overall deformation of the workpiece is 0.0145mm, and its position is at the middle upper end of the workpiece.

### *4.2. The influence of cutting force on the deformation of the workpiece within one revolution of the tool*

Next, consider the analysis of the influence of the milling force on the deformation of the workpiece during one rotation of the milling cutter. Because the finite element method discretes the continuous medium, when simulating each chip removal, the corresponding nodes on the machined surface are loaded and unloaded according to the time when the cutting edge acts on the workpiece to realize the dynamic effect of the tool on the workpiece. Process Ji.

The cutting within one revolution of the tool is completed in a certain period of time, which reflects the milling speed of the milling cutter, that is, it reflects the rotational speed of the milling cutter within one week. Since the cutting area is the same, the cutting force of the CDEF segment will remain unchanged, but as the stiffness of the thin wall becomes smaller, the deformation will be greater than the point C at point E (the position from the cut-out point G) of the surface generation line Deformation. Subsequently, due to the rapid reduction of the cutting force in the FEG section, the deformation is also rapidly reduced. That is to say, at point E, it is possible to produce the maximum deformation mainly caused by the decrease of the rigidity of the workpiece.

## 5. Export of finite element analysis data for processing deformation of thin-walled parts

Synthesize the above analysis and calculation, combined with ANSYS 10.0 finite element analysis software. For the workpiece with a milling depth of 10mm and a milling width of 0.1mm, the detailed data is shown in Table 4, test 3 conditions, and the



*Published by: The Mattingley Publishing Co., Inc.*

July-August 2020
ISSN: 0193-4120 Page No. 5247 - 5254

finite element deformation analysis is carried out. Set the leftmost end of the workpiece in the X direction as the zero position. Since the total length of the workpiece is 40mm, for the convenience of analysis, take three deformations for comparison and analysis. On the processed surface, x1=10mm, x2=20mm and x3=30mm are analyzed data.

**Table 4.** Test 3 working conditions.

| Test number | A tool speed | B feed rate | C milling depth | D radial depth of cut | Cutting force F | Cutting area |
|---|---|---|---|---|---|---|
| 3 | 2100r/min | 210mm/min | 10mm | 0.1mm | 16.812N | 6.3513mm$^2$ |

Based on the above ANSYS finite element analysis results, it can be concluded that the maximum position of workpiece deformation appears at the uppermost part of the middle of the workpiece, and with the processing of straight-wall workpieces, the deformation on both sides of the workpiece is not strictly symmetrical. This provides a theoretical basis for predicting cutting deformation for the study of related content.

**Conclusion**

In this paper, the finite element vibration modal analysis of thin-walled parts is firstly carried out. Using the finite element analysis software Ansys, the milling force evaluation and deformation analysis are carried out for aluminum alloy thin-walled plates. Force loads are applied on the basis of the modal analysis. The deformation of thin-walled parts under the action of cutting force was predicted. At the same time, the deformation finite element deformation analysis process was carried out and the key technology was discussed, which made a theoretical exploration for actual cutting. The simulation analysis results can be used to guide the actual processing, which is of great significance for avoiding resonance during processing.

**Acknowledgments**

1. Anhui Provincial Department of Education, a key university natural science project: A357 aluminum alloy semi-solid thixoforming research and finite element analysis (project number: KJ2019A0892);

2. Anhui Province University Outstanding Talent Cultivation Funding Project (Project No. gxgnfx2019071);

3. The key project of the scientific research fund of Anhui Sanlian College: Research on the semi-solid forming technology of aluminum alloy thin-walled shell (project number: KJZD2018007).

**6. References**

[1] Lucido M, Santomassimo C, Panariello G. The method of analytical preconditioning in the analysis of the propagation in dielectric waveguides with wedges [J]. Journal of Lightwave Technology, 2018, 3(9): 1-10.

[2] R, Senthilkumar. The effect of the number of passes in friction stir processing of aluminum alloy (AA6082) and its failure analysis [J]. Applied Surface Science, 2019, 1(3): 129-138.

[3] Zhao T, Hu H, Peng X Q, et al. Study on the surface crystallization mechanism and inhibition method in the CMP process of aluminum alloy mirrors [J]. Applied Optics, 2019, 58(22): 6091-6099.

[4] Preez S P D, Bessarabov D G. The effects of bismuth and tin on the mechanochemical processing of aluminum-based composites for hydrogen generation purposes [J]. International Journal of Hydrogen Energy, 2019, 44(39): 56-64.

[5] Wang Y, Dai Y, Liu X, et al. Study on the method of color image noise reduction based on optimal channel-processing [J].

IET Image Processing, 2018, 12(9): 1545-1549.

[6] Abramov A D, Nikonov A I. Measurement of microrelief parameters based on the correlation method of image processing of the surfaces under study [J]. Measurement Techniques, 2019, 61(11): 90-97.