Parametric Design and Numerical Control Simulation of Cylindrical cam Based on Ug8.5

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Abstract. This paper introduces the method and process of parametric modeling and numerical control programming simulation processing of cylindrical cam by using ug8.5 software, improves the design, manufacturing efficiency and precision of cylindrical cam, and realizes the serial design of cylindrical cam. It has been proved by practice that the performance of cylindrical cam is stable and efficient, which greatly shortens the machining cycle of cam and improves the efficiency.

1. Introduction
The key of cam design is to design the contour of cam working part. The traditional cam design and processing method adopts the drawing method to design the cam contour and the marking method to process the cam. The design efficiency is low and the design and processing accuracy is not high, which restricts the quality of mechanical transmission. With the development of science and technology, mechanical equipment is developing towards high-speed precision and automation, which puts forward higher requirements for the accuracy of cam mechanism [1]. Using ug8.5 software to carry on the parametric design and the numerical control processing programming to the cylinder cam, using the numerical control processing center to carry on the processing, has improved the design, the manufacture efficiency and the precision. With the development of computer technology, CAD / CAM is widely used in cam design and machining, which is of great significance to small and medium-sized machining enterprises in China [2, 3].

2. Parametric Design of Cylindrical cam

2.1. Design Thinking
According to the motion equation of the cam and the parameters in the parameter group, use the expression function of ug8.5 to draw the contour of the cam, substitute the set variable names in the expression, use the function of regular curve to draw the contour of the cam, then use the function of surface stretching and thickening to generate the cam entity, use the function of difference and cylinder difference to generate the cam workpiece, add the parameter variables, real Now it is parameterized.

2.2. Design Method and Steps Based on UG

2.2.1. Parameter analysis of cylindrical cam
Now the cylindrical cam is designed, the radius of the cylinder is \( r_o = 40 \) mm, the height of the cylinder is \( H = 300 \) mm, the stroke of the follower is \( h = 20 \) mm, the width of the curve groove is \( b = \)
12 mm, the depth of the groove is 6 mm, the angle of motion of the thrust is 120°, the angle of far repose is 60°, the angle of motion of the return is 120°, the angle of near repose is 60°, and the thrust and return of the follower are in simple harmonic motion. (1) When the cam turns 120°, the follower will push h according to the cosine acceleration movement law; (2) when the cam turns 60°, the follower will stop moving; (3) when the cam turns 120°, the follower will return h according to the cosine acceleration movement law; (4) when the cam turns 60°, the follower will stop moving again. According to the above conditions, the actual contour curve of the cam is designed and the three-dimensional entity of the cam is generated.

2.2.2. Parameter equation of cylindrical cam
According to the knowledge of mechanical principle, the parameter equation of theoretical contour curve of cylindrical cam [4] is
\[
\begin{align*}
x &= r_0 \cos j \\
y &= r_0 \sin j \\
z &= s
\end{align*}
\] (0° ≤ j ≤ 360°) (1)

Where x, y and z represent the coordinates of any point on the curve, s represents the motion law of the follower, \( r_0 \) represents the radius of the base circle, and j represents the angle of the cam.

2.2.3. UG expression of cam
According to the above conditions, the working contour curve of the cam is designed. According to formula (1), the UG expression of each segment can be obtained as shown in Table 1. It should be noted that \( t \) is the UG system variable with a value of 0 ≤ t ≤ 1.

| Motion process | Name | Formula | Notes |
|----------------|------|---------|-------|
| push stroke    | \( X_{t1} \) | \( r_0 \cos(j_1) \) | push X coordinate value variable |
|                | \( Y_{t1} \) | \( r_0 \sin(j_1) \) | Push Y coordinate value variable |
|                | \( Z_{t1} \) | \( (h/2)(1-\cos(180\*t)) \) | Push Z coordinate value variable |
|                | \( X_{t2} \) | \( r_0 \cos(j_2) \) | Far rest X coordinate value variable |
|                | \( Y_{t2} \) | \( r_0 \sin(j_2) \) | Far rest Y coordinate value variable |
|                | \( Z_{t2} \) | \( h \) | Far rest Z coordinate value variable |
| Far from rest  | \( X_{t3} \) | \( r_0 \cos(j_3) \) | Return X coordinate value variable |
|                | \( Y_{t3} \) | \( r_0 \sin(j_3) \) | Return Y coordinate value variable |
|                | \( Z_{t3} \) | \( (h/2)(1+\cos(180\*t)) \) | Return Z coordinate value variable |
| return stroke  | \( X_{t4} \) | \( r_0 \cos(j_4) \) | Near rest X-coordinate value variable |
|                | \( Y_{t4} \) | \( r_0 \sin(j_4) \) | Near rest Y coordinate value variable |
|                | \( Z_{t4} \) | 0 | Near rest Z coordinate value variable |

2.2.4. Parametric modeling
Substituting \( r_0 = 40 \) mm, \( h = 20 \) mm, \( a_1 = 120° \), \( a_2 = 60° \), \( a_3 = 120° \), \( a_4 = 60° \), \( j_1 = a_1 \* t \), \( j_2 = a_1 + a_2 \* t \), \( j_3 = a_1 + a_2 + a_3 \* t \), \( j_4 = a_1 + a_2 + a_3 + a_4 \* t \), into the above formula (where \( a_1 \) is the thrust angle, \( a_2 \) is the far angle of repose, \( a_3 \) is the return angle, \( a_4 \) is the near angle of repose), using the regular curve, each curve corresponds to each group of object parameters \((X_i, Y_i, Z_i)\) in the table. Then use the derived curve function of ug8.5 to connect the four curves as a whole, as shown in Figure 1. Select ug8.5 surface lifting command, select the curve just connected to the section curve with a distance of 12 mm, as shown in Figure 2, select the thickening command of the offset / zoom function of ug8.5 with a direction inward and a distance of 6 mm, as shown in Figure 3, select the difference command of the combined function of ug8.5, select the cylinder as the target body, and select the body as shown in Figure 3 as the tool selection body, with the result as shown in Figure 4, and then Select the whole column feature command, and the result is shown in Figure 5.
3. Numerical Control Programming of Cylindrical Cam

3.1. Rough Machining
Using the derivative curve function of ug8.5, the on surface offset command offsets the contour curve of the bottom of the groove on the cylindrical surface of the bottom of the groove, selects the curve whose offset distance from the contour on the left side of the cam groove wheel to the middle of the groove is 6 mm as the driving curve, enters the machining environment to create the Φ 6 tool, creates the work order, selects the cylinder at the bottom of the groove as the part geometry, the driving method is curve driving, and the projection vector is The cutter axis and cutter axis shall be far away from the straight line, and the straight line shall be selected as the cam axis. In order to prevent over cutting at the turning point of the driving curve, the advance / retreat cutter mode shall be set as interpolation in the non cutting moving parameters, and multiple cutter paths shall be set in the cutting parameters, with the component offset of 6 mm, each cutter feeding of 1 mm, the component margin of 0.2 mm, and the tolerance of 0.03. The generated cutter path is shown in Figure 6, and the simulation machining is shown in Figure 7.

3.2. Semi Finish Machining
Using the derivative curve function of ug8.5, the on surface offset command offsets the contour curve of the cam at the bottom of the groove on the cylindrical surface of the cam groove. Select the curve whose offset distance from the left and right contour of the cam groove wheel to the middle of the groove is 2.05 mm as the driving curve, enter the processing environment to create the Φ 4 cutter, create the process, select the cylinder at the bottom of the groove as the part geometry, and the driving method is curve driving. The projection vector is the cutter axis, the cutter axis is far away from the straight line, and the straight line is the cam axis. In order to prevent over cutting at the turning point of the driving curve, the advance / retreat cutter mode shall be set as interpolation [5] in the non cutting moving parameters, and multiple cutter paths shall be set in the cutting parameters, with the component offset of 6 mm, each cutter feeding of 1 mm, the component allowance of 0.05 mm, and the tolerance of 0.01. The generated cutter path is shown in Figure 8, and the simulation processing is shown in Figure 9.
3.3. Finish Machining

The processing of the bottom of the cam groove only needs to modify the corresponding parameters on the basis of the tool path of machining and semi finishing. For the processing of cam groove side, the driving method needs to be changed. The side of cam groove is selected as the driving surface and components, and the projection vector is selected to face the driving body [6]. The generated tool path is shown in Figure 10, and the simulation processing is shown in Figure 11.

4. Conclusion

The traditional design method, because of the complexity of the design process, the low precision, and the calculation results can not be directly used in NC machining (the NC machining of the cam needs CAD 3D modeling), has become more and more unsuitable for the current requirements of rapid and accurate cam design and NC machining. UG software ensures the accuracy of part modeling, and provides a variety of simulation processing strategies, which has good operability, and generates NC program code quickly and accurately. Using CAD technology to replace the traditional cam design method can greatly shorten the design cycle, improve the design quality, and meet the objective requirements of NC machining.

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