Research on Parametric Design Technology of cycloidal pin wheel Reducer for Robot based on SolidWorks

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ABSTRACT

In this paper, SolidWorks software and VB.net programming language are used to accurately design the cycloidal wheel of the cycloidal pin wheel reducer used in robot with 3D parameterization and the 3D model of cycloidal wheel is automatically drawn. It is beneficial to save and modify the drawings and reduce the workload of the designers. In batch design, the design model can be automatically updated by modifying the parameters of cycloidal wheel 3D model or drawing parameters.

Keywords: SolidWorks;Parameterization;Cycloidal wheel;Two-dimensional drawing

1. INTRODUCTION

Robot cycloidal pin wheel (Japan called RV) reducer is developed by the Japanese Emperor company in 1986 [1], and the first application in industrial robot joints on a new type of high precision reduction transmission mechanism. Robot cycloid reducer adopts the combination form of involute gear planetary transmission and cycloid pin gear planetary transmission in structure, which has the advantages of compact structure, small volume, light weight, high transmission accuracy and efficiency, and large transmission ratio range [2]. As an important part of the industrial robot, the end positioning accuracy of the industrial robot depends to a large extent on the transmission accuracy of the cycloid pinwheel reducer for robots [3]. As the core part of the cycloid pinwheel reducer for robots, the design of the cycloid wheel is even more important. As the key parameters of the cycloid wheel are highly coupled with the target of the designed product[4], the result makes the calculation in the design process huge, and the relevant parameters need to be constantly adjusted to achieve the rated transmission target [5], so the parametric design becomes an important subject [6]. In this paper, VB.net language and SolidWorks secondary development technology are used to complete the parametric design of cycloidal wheel and realize the two-dimensional automatic drawing of cycloidal wheel.

2. PARAMETRIC DESIGN OF CYCLOIDAL WHEEL

2.1 Parametric design operation interface of cycloidal wheel

The core component of the cycloid pinwheel reducer for robots is the cycloid wheel. At present, most of the research objects are also cycloid wheels [7]. The parametric design interface of cycloidal wheel is divided into four parts: tooth shape parameter, structure parameter, sample picture and storage information. The tooth profile parameter is to set the teeth of the cycloidal wheel, the structural parameter is to set the overall structure and internal hole structure of the cycloidal wheel [8], and the stored information can be used to name the cycloidal wheel and set the saving path. After the correct input of the parameters[9], the three-dimensional model of cycloidal wheel can be automatically created through SolidWorks by clicking the modeling button, and the program can be finished by clicking the exit program button, as shown in Figure 1.

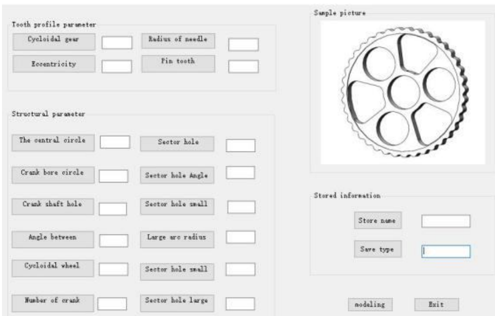

Fig 1 Parametric design and operation interface of cycloidal wheel

2.2 Cycloidal wheel parametric design program

Cycloidal gear is the core part of cycloidal reducer used by robot. The shape of ectoid directly determines the performance of cycloidal reducer used by robot. When cycloidal gear engages with standard needle teeth, the tooth shape is standard tooth shape, and Its equation is shown as formula 2-1 [10]. Where: rp is the radius of the center circle of the needle tooth, mm; rr p is the radius of needle teeth, mm; a is eccentricity, mm; iH is the transmission ratio between cycloidal pin wheel and pin teeth; K1 is the short amplitude coefficient; φ is the meshing phase Angle, °

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When modeling manually, you can directly draw cycloidal Outlines by entering formulas in SolidWorks. But with the program control, it will be too many parameters to achieve. To solve this problem, we can draw 20 coordinate points in SolidWorks by the equation, and then simulate the single tooth shape of cycloidal wheel through the coordinate points, as shown in Figure 2.

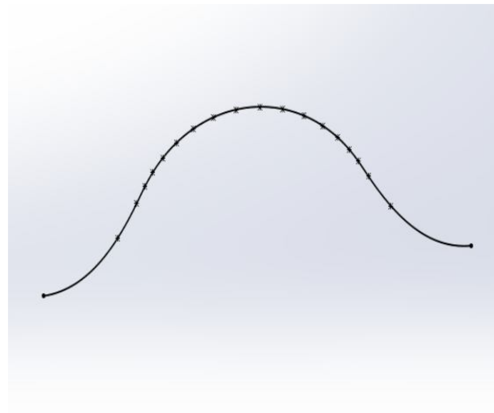

Fig 2 Single gear profile fitting of cycloid wheel

Relevant core code is as follows:

```
Dim Swapp As SldWorks.SldWorks
Dim Part As SldWorks.ModelDoc2
Swapp=CreateObject("Sldworks.applicati
on")
Part=Swapp.ActiveDoc
```

This is a reference for creating Sldworks objects. Since objects and methods modeled later have a parent-child relationship with Sldworks objects, Sldworks objects can be referenced only after they are referenced.

$$\begin{cases} x_c = \left[r_p - r_{rp}\phi^{-1}(K_1,\varphi) \right]\cos(1-i^H)\varphi + \left[a - K_1 r_{rp}\phi^{-1}(K_1,\varphi) \right]\cos i^H\varphi \\ y_c = \left[r_p - r_{rp}\phi^{-1}(K_1,\varphi) \right]\sin(1-i^H)\varphi - \left[a - K_1 r_{rp}\phi^{-1}(K_1,\varphi) \right]\cos i^H\varphi \end{cases} \tag{2-1}$$

```
Dim u、t、i As Double
Dim X(20) As Double
Dim Y(20) As Double
u = 2 * pi / 19
t = 0
For i = 0 To 19
If t <= 2 * pi Then
```

```
X(i)=(Rz-rz*(1+K1^2-2*K1*Cos(t))^(-1/2))*Sin((1-(Zz/Zc))*t)+(A-K1*rz*(1+K1^2-2 * K1* Cos(t))^(-0.5))*Sin(Zz/Zc*t)
Y(i)=(Rz-rz*(1+K1^2-2*K1*Cos(t))^(-1/2))*Cos((1-(Zz/Zc))*t)-(A-K1*rz*(1+K1^2-2*K1*Cos(t))^(-0.5))*Cos(Zz/Zc*t)
 t=t+u
 End If
Next i
Dim skSegment As SldWorks.SketchSegment
Part.SketchSpline(19,0.001*X(0),0.001*Y(0), 0)
.....
Part.SketchSpline(0,0.001*X(19), 0.001*Y(19), 0)
```

Through the linkage of code and solidworks, a single tooth shape spline curve of cycloidal wheel is synthesized, and then the overall drawing of three-dimensional modeling of cycloidal wheel is completed by creating gear teeth, array gear teeth, stretching and cutting, as shown in Figure 3.

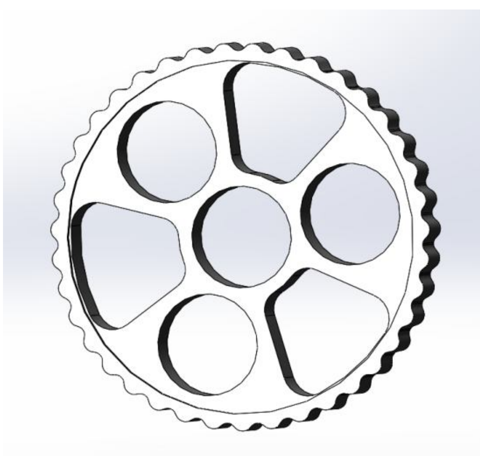

Fig 3 SolidWorks automatically draws cycloidal wheel models

The relevant code is as follows:

```
Part.SketchManager.CreateArc(0, 0, 0, 0, 0.001 * Y(0), 0, 0.001 * X(19), 0.001 * Y(19), 0, -1)
Part.Extension.SelectByID2("Sketch1", "SKETCHREGION", 0,0, 0, True, 4,Nothing, 0)
Part.FeatureManager.FeatureExtrusion2(True,False,False,0,0,T/1000,T/1000,False,False,False,False,0,0,False,False,False,False,True,True,True,0,0,False)
Part.Extension.SelectByID2("Boss-Extrude1","BODYFEATURE",0,0,0,False,4,Nothing, 0)
Part.FeatureManager.FeatureCircularPattern4(Zb, 4*Pi/2, False, "NULL", False,True, False)
Part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, False, 0, Nothing,0)'
Part.SketchManager.CreateCircle(0, 0, 0, d1 / 2 / 1000, 0, 0)
Part.SketchManager.CreateCircleByRadius(d3/2/1000*Sin(a1),-d3/2/1000*Cos(a1),0,d2/2/1000)
Part.Extension.SelectByID2("Sketch2", "SKETCHSEGMENT", 0,0,0,False,0,Nothing, 0)
Part.FeatureManager.FeatureCut3(True, False, True, 1, 0, 0 0, False, False,
False, False, 0, 0, False, False, False,False,False,True,True,True,True,False,0,0,False)
......
```

3. SECONDARY DEVELOPMENT AND DESIGN OF CYCLOIDAL WIRE WHEEL TWO-DIMENSIONAL DIAGRAM

Two-dimensional drawings of mechanical parts generally go through the following five steps [11]: (1) three-dimensional modeling of parts;(2) Create and generate drawing templates;(3) Place all kinds of views of parts;(4) Mark the size of each view;(5) Insert the list, fill in the title bar, etc.At present, various SolidWorks secondary development technologies are widely used in various fields of machinery manufacturing, such as automotive, aviation and other fields [12]. The two-dimensional drawing process of

SolidWorks secondary development is shown in Figure 4:

```
Else
Part=Swapp.NewDocument("C:\Program Data\SolidWorks\SOLIDWORKS 2016\templates\a4-7.drwdot", 0, 0, 0)
  End If
```

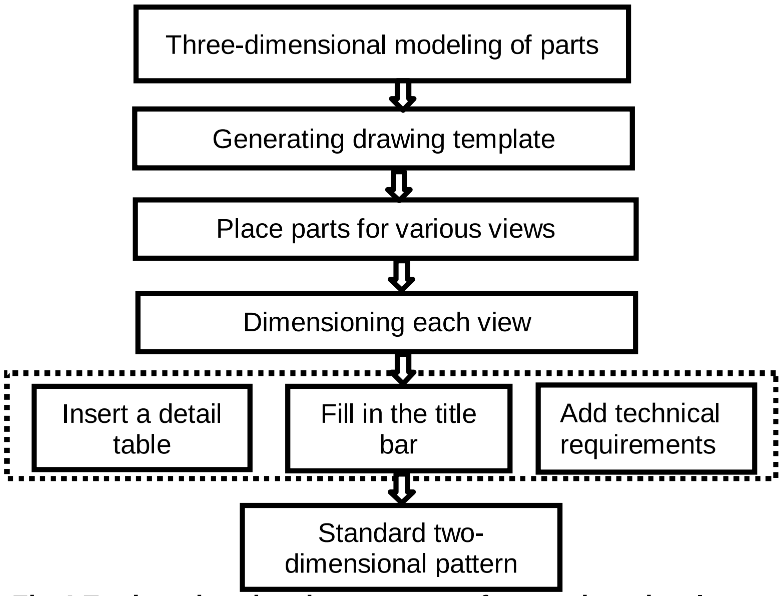

Fig 4 Engineering drawing process of secondary development

3.1 Cycloidal wheel parametric design program

Before writing 2D graphics programs through VB.NET, add references to CommandsTypeLibrary,ConstantTypeLibrary, and TypeLibrary to link SolidWorks to VisualStudio. Then write a program to create a two-dimensional graph document.Before creating a cycloidal wheel two-dimensional diagram, you need to first invoke the created two-dimensional diagram template. The result is shown in Figure 5, and its program code is as follows:

```
Dim myfilename As String
myfilename="C:\ProgramData\SolidWorks\SOLIDWORKS2016\templates\a4-7.drwdot"
If Dir(myfilename) = "" Then
MsgBox("File does not exist")
```

Fig 5 two-dimensional drawing template

After the 2D drawing template is created, the main view and section view of parts need to be created. When creating the section view, a section line needs to be created first, and then the corresponding section view is generated. The relevant program code is as follows:

```
Dim myView As Object
```

```
myView=Part.CreateDrawViewFromModelView3("C:\Users\Administrator\Desktop\parts 8.SLDPRT", "*Forward vision", 0.088, 0.129, 0)
Part = Swapp.ActiveDoc
Part.ActivateView("Two-dimensional graph view 1")
Dim skSegment As Object
skSegment=Part.SketchManager.CreateLine(0#, 0.088, 0#, 0#, -0.091, 0#)
Part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
Dim excludedComponents As Object
excludedComponents = Nothing
myView=Part.CreateSectionViewAt5(0.194,0.129,0,"A", 0,(excludedComponents), 0)
Part.ClearSelection2(True)
```

Dimension annotation, taking the cycloid wheel thickness dimension of 13.6mm as an example, SelectByID2 command first to select two sides, and then AddHorizontalDimension2 command to make annotations.Finally, dimension tolerances are added by the EditDimensionProperties2 command. The associated program code is shown below:

```
Part.ActivateView("Two-dimensional diagram view 2")
Part.Extension.SelectByID2("", "EDGE",0.194-L1/2/2/1000,0.129,-499.96002351577, False, 0, Nothing, 0)
Part.Extension.SelectByID2("","EDGE",0.194+L1/2/2/1000,0.129,-499.96002351577, True, 0, Nothing, 0)
Part.AddHorizontalDimension2(0.205202026460628, 0.0842934923041332, 0)
Part.EditDimensionProperties2(2, 0, -0.00002, "", "", True, 9, 2, True, 12, 12, "", "", True, "", "", False)
Part.ClearSelection2（True）
```

Add a comment to insert text through the InsertNote command, set the text Angle by the myNote Angle property, and set the text style and size by the SetBalloon property. To set the lead of the text using the SetLeader3 property of myAnnotation, the core code is as follows:

```
myAnnotation=mySFSymbol.GetAnnotation()
Dim myNote As Object
Dim myTextFormat As Object
myTextFormat = Nothing
myNote=Part.InsertNote("Uniform distribution")
If Not myNote Is Nothing Then
myNote.LockPosition = False
myNote.Angle = 0
myNote.SetBalloon(0, 0)
myAnnotation = myNote.GetAnnotation()
If Not myAnnotation Is Nothing Then
myAnnotation.SetLeader3(SwConst.swLeaderStyle_e.swNO_LEADER, 0, True, False, False, False)
myAnnotation.SetPosition(0.121954231993969, 0.0963539281900556,
0)
End If
End If
Part.ClearSelection2（True）
```

To insert the specification table, you can use the InsertTableAnnotation2 command to set the coordinate position of the specification table insertion, set the number of rows and columns of the specification table, and obtain the template path.The BorderLineWeight command is used to set the boundary width, and the GridLineWeight command is used to set the grid line width. The code is as follows:

```
Dim swTableAnnotation As Object
swTableAnnotation=Part.InsertTableAnnotation2(False, 0.229347788134974, 0.136902169659606, 1, "C:\Users\Administrator\Desktop\Specification template.sldtbt", 20, 3)
If Not swTableAnnotation Is Nothing Then
swTableAnnotation.BorderLineWeight = 0
swTableAnnotation.GridLineWeight = 0
End If
```

By writing the two-dimensional automatic drawing program of cycloidal wheel, the two-dimensional automatic drawing results are shown in Figure 6.

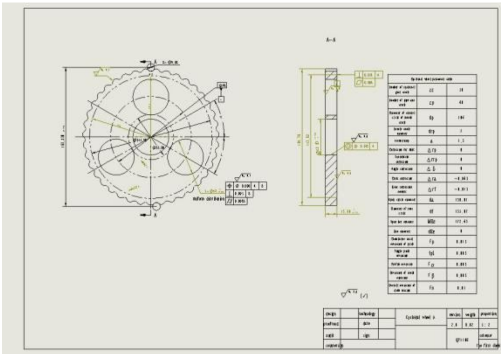

Fig 6 Cycloidal wheel 2 D automatic drawing results

4. CONCLUSION

Based on SolidWorks, the parametric design and two-dimensional automatic drawing of cycloidal gear reducer for robot is conducive to rapid mass design of cycloidal gear, which can save a lot of time and energy of designers. The parametric design method given in this paper is also applicable to the parametric design of other parts. The cycloidal wheel model established in this paper can be used for subsequent automatic assembly, motion simulation and expected to be useful in finite element analysis.

COMPETING INTERESTS

Authors have declared that no competing interests exist.

AUTHORS' CONTRIBUTIONS

'Author Xiaoming Hu' designed the study, performed the statistical analysis, wrote the protocol, and wrote the first draft of the manuscript. 'Author Linshan Han',Yingjia Wang' and Hongwei Wang' managed the analyses of the study and the literature searches. All authors read and approved the final manuscript.

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