Irregular Object Packing Production and Processing

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Abstract: In order to improve the efficiency of the irregular object packing, the project establishes the irregular three-dimensional graphics library using Autodesk Inventor as the development platform. The graphics can be accomplished parametric design and expanded into plane figures. The plane figures are processed on CNC machine by converting into CN code format. The products can be used the packaging of goods directly. The project provides an idea for combining learning with research.

Key words: packing; irregular graphics; parametric; graphics expanded; Autodesk Inventor

1 Introduction

Products and packaging design is a work of comprehensive and modeling design, the design idea is often completed in the two-dimension and the three-dimension between reciprocating constantly adjust changes. Therefore, the cycle, the cost, the modeling, and the appearance of the products and packaging design become the focus of the design. Before long, the long cycle, the high cost and the difficult modification of the manual design has already could not satisfy the modernized needs[1].

The working way of the designer has been changed with the information society featured by the computer and network. The form and the connotation of the design are changing. Different product packaging design has different requirement, so the project goes on the development design combining the CAD software Inventor to establish an irregular three-dimensional graphics libraries .The graphics of the library are parametric design .The graphic designers can build ,modify and design the product by calling the graphics libraries according to their needs. At the same time the three-dimensional graphics can be expanded, mosaic, and packing designed[2]. The expanded graphics on CNC machine by converting into CN code format. The products can be packaging of goods directly.

2 Irregular graphic library building

2.1 Parametric designs of the graphics

Packaging graphics is designed by the soft of Autodesk Inventor. The Inventor was three- dimension mechanical design software developed by Autodesk Company. It can create a unique style of various complicated and delicate parts[3].

Most irregular graphics can be set by the regular ones, so firstly the various regular graphics libraries should be set up, such as square, cone, column, etc. Graphics must realize the parametric design in order to facilitate the other researchers' calling and modification. That is accurate geometry model automatic exported

from the geometric parameter model to achieve the automatic modification of geometric model by modified the local parameter. Such as a rectangle, only given its length, width and height in the coordinate system and the position, the rectangle with the different size and the different location can be got by modifying the several parameters.

The process of achieving the parameterization is as follows in the Inventor:

1) In the state of the sketch editing, creating the basic sketch, marking the size. And that the 'tolerance' instead of 'name' in the size display.

2) Return to the feature modeling environment, creating each features, such as drawing, rotate, etc.

3) Taking 'f_x' parameters. The parameters 'd' is named by name, such as the parameter 'd0' is named the 'wide of the rectangle' to facilitate the users understanding, as shown in figure 1. Users may need to be modified the default value of the equation.

4) Choosing the 'extracted feature' of the 'tool', apparent the 'create ifeature' dialog box. Clicking save and choosing the saving path.

So, graphic library is established. But clearly, graphics libraries exist plenty of graphics which is the modeling rather than the entity.

参数名称	单位	等式	公称值	公差	模型数值
长方形的宽	mm	8.00 mm	8.000000	◯	8.000000
长方形的长	mm	10.000 mm	10.000000	◯	10.000000
距离x轴距离	mm	15.000 mm	15.000000	◯	15.000000
距离y轴距离	mm	12.000 mm	12.000000	◯	12.000000
长方形的高	mm	5.000 mm	5.000000	◯	5.000000

Figure 1. 'f_x' preferences dialog

1.2 The structures of the irregular graphics

All kinds of irregular graphics can be generated according to the certain modeling form, by invoking the mode in the regular graphics libraries after the regular graphics libraries established. Below the example given is a cylindrical and conical structure.

1) The establishment of the cylinder. In the environment of the feature modeling, we select '插入 iFeature' parts under the menu features, finding the cylinder model in the regular graphics libraries. Selecting reference points, generally, the 'center point' as the reference point, and then selecting the reference plane. There we select the XY plane as the bottom plane of cylindrical. After the selection, the square with arrow will become two hooks, as it didn't, please repeat it again.

Clicking 'next button', the dialog will be shown like figure 2. X, Y is the distance from reference to the XY axis, D for the cylinder diameters, H for cylindrical height. Clicking on the D or H, it will appear accordingly illustrate in the text box. Designers can modify these values according to their needs, and then clicking 'complete button'. The cylinder diameters is 10cm, high is 10cm, the distance from the X axis is 8cm, distance from the Y , is generated.

Figure 2. parameter dialog

2) The construction of the cone and the cylinder. The cone transferred is same as the cone transferred. In the selection of the reference point, select the center point as the reference point. Also due to putting the cone into the cylinder, so when select the conical bottom surface as the reference, click the cylindrical top, then the two surfaces in the same plane. Clicking the next to revise parameter values, here noted is the distance from the XY axis is equaled with the distance with the cylinder. The bottom diameter of the circle will be equaled, other with the cone will run, both are not good build together. At last complete, the cone and the cylinder will be the one, and as a whole graphics.

After some common irregular graphics building, are deposited into the irregular graphic library for designers using. Designers only modify the length, width and height distance etc, the graphics they want will be got.

As for some graphics do not existed in the irregular graphic library, we can mosaic the needed graphics as the graphics in the regular graphic library.

2.3 The expansion and the Splice of the graphics.

The Inventor soft supports the open application program interface (API). Inventor API is a component object model (COM) or ActiveX interface, the interface for most popular language support, such as using VB, VBA, VVC＋＋、Delphi、Java etc. In the Inventor, it is embedded in a complete program design patterns VBA, this is a powerful development tools. Therefore, the project uses the VBA language for second development of Inventor to realize the three dimensional figures expanded.

The Concrete ideas for expanding: the solid body uses the boundary reflection (Brep) to define. Therefore, first the program traverses all geometry graph element, then creates a tree structure. The tree structure contains the information of the dot, line, and face parameters. If one side is selected to be expanded, the program will extracted all edge loops in this surface, as well as all the edge of each point of characteristic parameters. And then the program will all points 3d matrix transformation, making arbitrary Angle space plane of all points of data into a view point of data. Using the points of the data creates each side, and then each side will be connected into the coil. All coil are defined of a profile, a profile corresponds to a surface. In order to traverse all options surface expanded, then create the corresponding Profile[4].

Next we use a straight under the edge of matrix transformation example to illustrate the process.

A plane known space in a straight edge of P at both ends of A and B, are solving the transformation of Pxy X to segment A1, B1 transformation matrix.

First step: calculating rotation transformation matrix 'Matrix1'.

The plane P's vector is V_P for hypothesis, the line of X direction's vector is V_X, then rotation transformation matrix Matrix1 can use the following statement:

Matrix1. SetToRotateTo (VP, VX).

Second step: calculating translation transformation matrix 'Matrix2'.

What are the coordinates of A hypothesis endpoint (x, y, z), then according to the top of the rotation transformation matrix can be obtained in the plane of the Pxy point A' rotation transformation of coordinates (x', y', z') :

A'(x',y',z')= A(x,y,z).TransformBy(Matrix1).

Assume the coordinates for endpoint A1 is (x_1,y_1,z_1), so the translation Matrix2 transformation matrix can be used below statement:

Matrix2: Set Translation. (A), p. A1.

Third step: for the transformation matrix.

The line AB to segment A'B' transformation matrix can be obtained by the following statement:

M= Matrix1.MultiplyBy(Matrix2).

This plane P's all endpoints can through Matrix transformation to Pxy plane, such as: B1=B.T transform (Matrix).

In the process of the building, the program records the information of the connection of the surface, and the information of the shared surface. All these information are stored into the sketch Outlines. On joining one side, the program searches the side adjacent to it firstly. and then obtains the shared information. The program finds out the common side and the adjacent surface, another side translation. Through the two side of the geometry information, the program calculates the rotation matrix and translational matrix, and the will of the matrix multiply stitching, thus make two faces together. Joining in all adjacent facets and Mosaic finished.
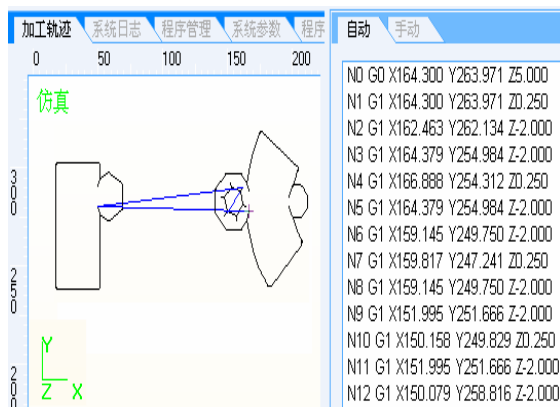

Figure 3. Machining Simulation.

3 Products processing

The expanded graphics are generated in the three-dimension environment. In order to realize those processing, we must be transformed into CN code which can be recognized by machine tool. Conversion process is as follows:

1) In the Inventor, we set up a Standard.idw file. The expanded graphics are converted by certain propor-

tion F to the document to generate a dxf file.

2) In the software of the sculpture, we read the document to dxf file, then transfer the expanded graphics into CN file.

3) In the CN machining software, we load the CN file, namely the CN code, and Set spindle speed, feed processing parameters. Last we can carry on process.

Figure 3 is shown the expanded graphics Process Simulation about a cone and cylinder together. Right is the CN code[5] .

4 Conclusion

The project Based on the platform of the Inventor for secondary development, is realized the building of the irregular graphics libraries and its spread function, and realize the graphic design parameterization, and provides great convenience for users products and packaging design. The project is combining the Research and Production, and directly transforms the theory research into the productivity, and lays the foundation for practical production.

It should be noted that this project is to further explore, such as: *1)* the expansion of the graphics libraries and humanization of operation. *2)* Realized in computer process optimization discharging[6].

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