A method for geometric tolerance annotation and position adjustment of engineering graphics

Youzhi Jiang*, Fuyun Liu, Qiang Wu, Yunzhan Xu

School of Electromechanical Engineering, Guilin University of Electronic Science and Technology, Guilin, Guangxi Zhuang Autonomous Region, China

*bemail: Liufuyun310@aliyun.com, cemail: 282224461@qq.com

demail: 1356697814@qq.com

*Corresponding author: aemail: 19012201009@mails.guet.edu.cn

Abstract. In the field of engineering design, 2d engineering drawing is the core content of machining. Most drawings are datum symbol old mark phenomenon, manually modify the notation and variant design can lead to the emergence of the form tolerance of interference, using CAD software provided by the API interface, combined with the Microsoft visual studio 2012 development platform for secondary development, makes the old national standard in the engineering drawing, form and position tolerances of interference is able to automatically adjust and modify the notation, and meet the international requirements. It can not only shorten the design cycle and improve the drawing efficiency of designers in product production, but also avoid the human error in drawing and ensure the quality of drawing.

1. The introduction

As a tool to describe geometric parts, 2d engineering drawing needs to transfer part information to machining personnel. In the field of engineering design, how to get a production requirements and beautiful drawings is extremely important. Most of the drawings have the phenomenon of the old national standard of the reference symbol, cumbersome manual modification and annotation, and the variation design will lead to the interference of shape and position tolerance, which often requires a lot of modification of 2d engineering drawings to complete the final design work.

With the help of Solidworks 3d modeling software, its own functions can be called by its API functions, which are obtained by accessing the Solidworks program COM interface. This paper uses C++ language to call its related COM interface, and develops a DLL plug-in to realize the self-adaptive adjustment function of engineering drawing. This plug-in can be seamlessly combined with SolidWorks software, which is convenient for designers to use.

2. Datum composition and batch modification

![Old and new datum annotation styles](image)

FIG. 1 Old and new datum annotation styles
The old and new gb marking styles of the datum in 2d engineering drawings are shown in Figure 1. In most of the old drawings, there is a phenomenon of the old GB marking styles of the datum, so a method of batch modifying the standard marking styles is proposed to batch modify the marking forms of the old GB of the datum. Datum in two-dimensional engineering graphics object corresponding to the Interface is DatumTag Interface, each Interface has its corresponding attributes and methods, to determine the datum object style under the datum method is to use an object Interface GetDisplayStyle() method, the method of the return value is the value of a long plastic, and GetDisplayStyle() method to determine the datum in the form of an enumeration of annotation style, as shown below:

```csharp
Enum swDatumDisplayType_e
    SwDatumDisplayType_Roundgb = 2 // Circular
    SwDatumDisplayType_Square = 1 // Square
End Enum
```

When the return of long integer value compared with enumeration values 1 and 2, you can determine whether the annotation Style of datum for the new and old national standard, to determine the datum of annotation Style of old gb, adopting SetDisplayStyle (UseDoc, Style) method to set the datum show Style, the first parameter UseDoc is a Boolean type, two way of setting only True and false, True represent the Style using the file, it refers to the basic reference standard in document properties of the default Settings; False means that you can change the style, that is, you can change the style of the reference annotation. The second parameter sets the annotation style of the datum in the form of an enumeration, consistent with the GetDisplayStyle() method. The reference objects in the view are traversed, the obtained reference objects are sorted, the reference objects in the view are classified by the GetDisplayStyle() method, the serial number of the reference object with the return value of 1 is put into the container OldGBnum, and all the old gb reference styles are modified in batch by SetDisplayStyle(). When need to datum object notation, artificial

![Datum interface](image)

FIG. 2. Datum interface changes extremely inefficient, namely design a man-machine interactive interface is shown in figure 2, according to the GetLabel() method, get the reference object tag Text, and then based on the datum of input symbols to judge whether as the object of this datum, then USES the SetText (WhichText, Text) method, set datum fluctuation before and after the Text.

3. Geometric tolerance mark adjustment
In the geometric tolerance labeling of two-dimensional engineering drawings, the geometric tolerance labeling style is used for marking. The labeling style consists of the tolerance grid, the geometric tolerance symbol, the reference letter and the geometric tolerance value. The labeling style is shown in FIG. 3.
Need to modify multiple shape tolerance annotation, artificial modification of low efficiency, which developed a human-computer interaction Interface as shown in figure 4, first to extract the two-dimensional engineering graphics drawing information, using the SolidWorks secondary development principle to obtain the geometric tolerance under the view object Interface Interface (Gtol Interface), using GetGtolCount () method to obtain geometric tolerance quantity, so as to determine whether the view visible geometric tolerances annotation symbols, using GetGtols traverse geometric tolerance object () method, and geometric tolerance objects in the form of serial number mark up and recorded in a listbox, by using the GetFrameValues (FrameNumber) method under geometric tolerance interface, a view on the geometric tolerance of text, the first parameter FrameNumber represents the frame, frame number refers to geometric tolerance properties can be set in two lines of the geometric tolerance parameters, users usually edit geometric tolerance to the first frame, its method of the return value is an array of strings, respectively is a geometric tolerance value of 1, geometric tolerance value 2, benchmark 1, 2 and 3, After the geometric tolerance information is obtained, 5 ListBox controls are added to the MFC interface. Each geometric tolerance object is obtained, and the acquired geometric tolerance information is stored in the ListBox control. If the parameters are not filled, blank is replaced. GetFrameSymbols3() method is adopted to obtain the geometric tolerance symbol on the view. The return value is an array of 6 lengths, and the first number in the array represents the geometric tolerance symbol. As shown in table 1 return value corresponds to the geometric tolerances of symbols, the value of the second and third respectively in the array represents the first and second tolerance values of the material conditions of symbols, and in the array after three values respectively
represent the benchmark benchmark benchmark 1, 2 and 3 symbols material conditions, will return to the form tolerance of symbols in turn in a listbox button (figure 4) Get the Info will Get every tolerances on the drawing object, and to form a set of tolerance information displayed in the human-computer interaction interface, when the user wants to modify a certain geometric tolerance information, enter the serial number and geometric tolerance information in the interface, Can be shown on the drawing.

Table 1 Tolerance symbols of form and position

| Geometric features   | Return value  | Geometric features   | Return value  |
|----------------------|---------------|----------------------|---------------|
| Straightness         | <IGTOL-STRAIGHT> | Perpendicularity     | <IGTOL-PERP>  |
| Flatness             | <IGTOL-FLAT>   | Angularity           | <IGTOL-ANGULAR> |
| Roundness            | <IGTOL-CIRC>   | Round to beat        | <IGTOL-SRUN>  |
| Cylindricity         | <IGTOL-CYL>    | All the beat         | <IGTOL-TRUN>  |
| Profile of a line    | <IGTOL-LPROF>  | Position             | <IGTOL-POSI>  |
| Profile of a Surface | <IGTOL-SPROF>  | Concentricity        | <IGTOL-CONC>  |
| Parallelism          | <IGTOL-PARA>   | Symmetry             | <IGTOL-SYMMETRY> |

4. Geometric tolerance position adjustment

FIG. 5 Interference of shape and position tolerance

In the field of engineering design, the variant design has become a hotspot, when in variant design of 3D modeling software, the two-dimensional engineering graphics in the geometric tolerance will be did not meet the requirements of drawing, as shown in figure 5, there are two kinds of form tolerance marked form, is a lead, a is more twist wire tagging, adopt GetLeaderStyle () method to distinguish the form of lead. When fuses in turn lead more annotation, adopting GetArrowHeadAtIndex2 () method to obtain the coordinates of point A, when for variant design, determine the coordinates of point A two-dimensional engineering graphics template and variant design have any change, the coordinates of A point if A coordinate change, namely the need to geometric tolerance adjustment, if no change point A coordinates, i.e. no need to adjust, when parts for variant design, geometric tolerance sash in view of the position will not change, so will cause leads appear in figure 5, get too close to the contour line are too far away from contour and tilt. To solve this problem, the method GetPosition() is adopted to obtain
FIG. 6 Is a parallel schematic diagram of geometric tolerances

\[ \begin{align*}
x_B' &= x_B + (x_{A'} - x_A) \\
y_B' &= y_B + (y_{A'} - y_A)
\end{align*} \tag{1} \tag{2} \]

the coordinates of point B, and point A and Point B in the template drawing are taken as the benchmark. GetArrowHeadAtIndex2() method was used to obtain the coordinates of the point A’ after the variant design, and EditDelete() method was used to delete the geometric tolerance object. Based on the point A’ after the variant design, InsertGtol() method was used to re-insert the geometric tolerance symbol. Is simply asking too coordinates because vector \( \overrightarrow{AB} \) and vector \( \overrightarrow{A'B'} \) parallel and in the same direction, namely according to the formula (1) (2) can be calculated point B’ coordinates, using the point A’ and point B’ and the coordinates of the default size of the sash and fuses the coordinates of the turning point, using ConvertToMultiJog () method to set the coordinates of the turning point, using SetPosition() method sets the B’ coordinates, insert the geometric tolerance is completed; When the lead is labeled with a single lead, the coordinates of B’ need only be obtained and fixed with SetPosition() method.

FIG. 7 Drawings of interference after variant design
FIG. 8 Drawing after adjustment

The 2d engineering drawings are adjusted by this method as shown in FIG. 7 and FIG. 8. The geometric tolerance marks can meet the production requirements.

5. conclusion
In two-dimensional engineering graphics, each a piece of paper in the number of view is different, so the introduction of the container, the container is equivalent to an array of unlimited length, when the traverse in the view annotation information can be stored into the container, when at the end of the traversal, use function to know the length of the container, because two-dimensional engineering graphics tagging information uncertainty is introduced into container this concept can solve the problem; View acquisition is a sequential process, so the data in the container needs to be cleared after processing to ensure the normal operation of the next view. When using C++ to call API interface, after completing the required functions, it must release its interface, otherwise it will cause the 3d modeling software to crash.

This method solves the bulk modification of the old national standard of the datum, the modification of the marking text of datum and geometric tolerance and the adaptive adjustment of geometric tolerance. This method is specific to any two-dimensional engineering drawing, not limited to a certain drawing. Due to the diversity of 2d engineering drawing marking information, the program can only be used to fine-adjust the drawing to make it clear and clear, reduce the manual adjustment time of the designer and improve the design efficiency.

Acknowledgements
Guangxi Special Fund Project for Innovation-Driven Development (Guike AA18242037)

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