Research on Computer Aided Industrial Design Based on UG NX 7.0

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Abstract — In order to make the application of computer-aided software in industrial design and manufacturing more popular, and further to improve the production efficiency and product quality in this area, this paper firstly introduced the secondary development of UG software and the application status of UG NX 7.0 in industrial design, then adopted this kind of software, combining with related theory of industrial design, to establish two models of product design in industry. Through the description of actual application of the software and the exhibit of actual effect of auxiliary design in the models, it's proved out that the computer-aided software has many advantages in modern industrial design, so we should apply computer-aided software better in industrial design area and do more related research.

Keywords - UG NX 7.0; The secondary development; Computer-aided; Industrial design

I. INTRODUCTION

In the middle of the twentieth century, the manufacturing industry has changed a lot, the iconic event is that the CAD/CAM software appears and has a wide range of applications in manufacturing, computer-aided technology is widely used in manufacturing industry. At present, the domestic and foreign enterprise product design has widely used CAD software. CAD software can be divided into two-dimensional and three-dimensional CAD according to the function. Now, the application of two-dimensional CAD in the country has been very mature, on its platform, the development of the two-dimensional standard parts library and the national standard environment has been quite perfect. At the same time, it has been fully market-oriented; the application of three-dimensional CAD in China is still not mature. The development of the three-dimensional standard parts library in China has not been fully realized, and national standard environment is not fully established. The mainstream 3D CAD software are basically foreign software, including Pro/E of American PTC Company, CATIA and Solid Works of France Dassault/IBM company, Uni Graphics NX, I-DEAS and Solid Edge of American UGS/SIEMENS company. Uni Graphics, UG for short, is the world's advanced, closely integrated, and the manufacturing oriented three-dimensional CAD/CAM/CAE high-end software. As an integrated engineering solution, UG is widely used in many fields, such as concept design, industrial design, mechanical design, engineering simulation and digital manufacturing, it's used to create and acquire 3D-definition products. UG is a leader in the field of automation technology driven by knowledge, it realized the combination of design optimization technology and knowledge engineering based on product and process, and significantly improved the production of aerospace, aviation, automotive, machinery, consumer products, medical equipment and tools, etc[1]. UG has developed rapidly since it entered the Chinese market in 1990, and has become the preferred software of Chinese aerospace, automotive, large-scale machinery manufacturing, computer and peripherals, home appliances and other fields. Now the latest version of UG is the NX series. The UG's existing modeling function is fully capable of modeling complex parts, and rapid modeling is aimed at improving the modeling speed to shorten the product development cycle. Besides, it is usually aimed at a complex part. This kind of parts is characterized by many design parameters, and the parameters meet certain function relationship. The transform from design parameters to structural parameters needs complex mathematical operations, and the structure parameters are firstly calculated according to the design parameters, which is the usual design method, and then it's the turn to use UG software to establish models. The disadvantage of this method is that the model has a slow generating speed and it's difficult to change its design, because once a design parameter is adjusted, all the structural parameters of the model are likely to be changed, and modeling must be restart from scratch. But if we use UG OPEN API or UG OPEN GRIP to create a user program, and integrate the design calculation and UG modeling together to achieve the design and modeling of automation, the modeling speed will be greatly improved, and the design will also be very convenient, so they are very suitable for modeling design of series parts. What's more, some models are generated based on the measured data, the data can be collected, processed and finally established by the user program[2]. Fig.(1) (a) - (d) shows several aspects of the application of UG NX software.
II. THE SPECIAL DESCRIPTION OF UG NX 7.0

In October 2009, the latest version of the Siemens PLM Software launched its flagship digital product development solutions for NX software. NX 7.0 introduces the function of HD3D, that is an open and intuitive visual environment, which contributes to the global product development team to fully explore the value of PLM information, and significantly improves their ability to develop effective product decisions. In addition, NX 7.0 also added an enhancement function of synchronization modeling technology, which was introduced by Siemens PLM Software last year to improve the efficiency of computer aided design, manufacturing and simulation analysis, and it was widely acclaimed after the launch. The new enhanced capabilities will further improve the development of various types of products, and extended NX's effective coordination ability incomparable with third party's application data of CAD [3]. The NX 7.0 newly increased enhancement in the function of synchronous modeling technology, and introduced innovative HD3D environment that can accept a variety of sources of data, which set up a new standard for the visual analysis of product development. Dr. Ken Versprille, director of PLM research at the industry analysis firm, CPDA, said that "The technology is one of the most important breakthroughs in the field of 3D solid modeling in recent 20 years ". It improved efficiency, increased the reuse rate of the original data and improved the interoperability among the third-party CAD systems by further improving the synchronization modeling technology. NX 7.0 has a strong lead in the modeling of flexibility and productivity. It creates and edits the tools for geometric graphics more quickly, and further accelerates the speed of design, creation and modification of various tasks. The design intent is to be preserved when the change is made, and reliable editing also avoids the failure in the update process, without the replay after a long time. With the most powerful new synchronization modeling capability, users do not need to understand the original creation method in the use the original CAD model, and the expanded the capacity that can achieve shear, copy, paste and mirror function, thus this further improves the productivity. So, it is easier to adapt the old model to the new design, and greatly saves time and cost by improving the design data reuse. The new "model cleaning" tool makes it easier and faster to edit the model from the third-party CAD system. Automatic and semi-automatic mixing and slope identification structured and maintained the relationship between the two most common manufacturing characteristics, and the alternative automatic and manual repair function together with the combine and segment function in geometric model eliminate the excess gap, besides, it can correct geometric model of mismatch[4]. Fig. (2) is the exploded view of UG NX 7.0 software.
III. THE SECONDARY DEVELOPMENT OF UG

A. Brief Introduction of the Secondary Development of UG

The secondary development refers to the development of application program and tools for industry and design process on the software platform, combined with specific application requirements, and summed up the design knowledge and experience of the industry. NX Open is an open architecture that enables different applications and NX to implement a flexible integrated development kit, which aims to enable the integration of applications including NX itself provided and third-party developed applications to achieve data sharing in different software and hardware platforms and on different networks or computers. Through the application and tools provided by NX Open, users can achieve the following functions: Access to the NX object model; Creat and edit objects in the NX object model; Choose the familiar programming language; Creat and manipulating user defined objects; Manage the association between user defined objects and other NX objects; Configure the NX service to create a remote application; User-defined NX interface which makes the NX environment to meet their specific needs of work process; Allow the third party to create custom menu, etc.

B. The Secondary Development Methods of UG

The secondary development of UG is mainly based on the two development tools. We use Fig. (3) to show the relationship between the secondary development tools of UG and the functions.

![Figure 3. The relationship between the secondary development tools of UG and the functions](image)

C. Application Range of UG's Secondary Development

According to the design characteristics of the enterprise's products, the establishment of a unified secondary development environment of NX, so that all of the secondary development staff can be simultaneously on a platform for product development, to avoid repeatedly setting different environmental parameters, and save working time, so as to improve the design quality and design efficiency of the product. Customization of custom tool set mainly include: Configure the default boot file for NX based on business requirements and standards; Provide user specifications, user standard parts library and user oriented interface, etc. Intelligent design system applies the basic principle of the use of knowledge engineering, to combine case-based reasoning and system modeling technology. It's a design system for the development of specific products. It can meet customers' demand for series of products and general products personalized variant design. The development of intelligent design system can not only intelligently design product structure, but also can guarantee the quality of product design. Intelligent design system can greatly help us to shorten the design cycle of the product, to reduce the design cost of product design process to facilitate communication between developers and customers, and to understand the needs of customers timely, so as to improve the market competitiveness of enterprises. In the production process, due to various factors, the initial set of product parameters, functions, and other requirements may change over time, which requires companies to make quick response to this situation. Flexible manufacturing is proposed to meet such needs, it is an important index to test the resilience of an enterprise [6].

IV. APPLICATION OF UG NX IN INDUSTRIAL MANUFACTURING

A. Parametric Design

Parametric design includes two aspects of content: one is the parametric graphic element, and the other is a parametric modification engine. Primitives are in the form of component, we can understand the constraints among the components, all the information of the model parameters in computer aided design is preserved. Parameter changing technology of parametric modified engine can automatically reflect any changes made by the user to a building design or document in other related arts. Intelligent building component, view and comment symbol are adopted to connect each component through a change in the engine. Parameter changes caused by the movement of the component, the deletion and the size change will cause the change of the related parameters, any changes that occur in any view can be parameterized, spread to all views, in order to ensure the consistency of all drawings, no one is required to modify all views one by one[7]. When we use computer-aided software for product development, the efficiency of the product is determined by the speed of the software, there is a direct relationship between them. At early stage of design, the cognitive of the shape and size of the parts has certain defects, so we need to verify that the exact results are obtained in the assembly. This makes using computer to change the size and shape of parts to become an important research topic, and digital design puts out this aspect of problem. The theory of parametric design is to quantify the value of the original quantity. On computer the requirements of the research and development can be realized, and arbitrary adjustment can be made to the size and shape of the parts, therefore, parametric design in the current wave of digital technology is the need to be adhered. In the process of using the CAD software to carry out parametric modeling, modeling is very decisive factor. Whether the model is appropriate or not is directly related to the right product or not. The parametric model represents the geometric constraints and engineering constraints of the part graphics.
Geometric constraints include structural constraints and dimensional constraints. Structure constraint is the topological constraint relationship between geometric elements, such as parallel relationship, vertical relationship, tangent relationship, etc. The size constraint is the dimension constraint representation, such as distance, angle, etc. Engineering constraint is the dimension constraint relationship, which is expressed by defining the dimensions of the variables and the number of their values and logical relations [8].

B. Application of UG NX 7.0 in the Design of Mobile Phone Parts

The following is a case study of 3D modeling of a mobile phone's upper frame. This paragraph introduces the application of UG NX 7.0 in the design of mobile phone parts. Using the sketch function to draw the shape of the cross section of the mobile phone and the guide line of the sweeping. Fig. (4) shows the cross section of screen frame [9].

![Figure 4. The cross section of screen frame](image)

Using sweep function to make the frame section of the mobile phone's display screen by sweeping along the drawing of the guide wire, and to complete entity. And then using the sketch function to draw the cross section shape and guide line of the key parts. Next using the same method as the display frame of the solid shape to complete the solid modeling of the part of the key parts of the mobile phone. After that we need to clip the upper part of the key using the method of slice body to trim the desired entity, and use curve bridging function to combine the frame and key border of the display curve according to the tangency mode, then use the curve grid function to generate the required individual pieces. Finally, stitching each sheet body generated at the forward step in a whole using of the function of the film body, so as to complete the design of the mobile phone's upper frame, as shown in Fig. (5) [10].

C. Application of UG NX in the Process of Gear Design

Gear is one of the important parts in mechanical equipment, the design of the gear part mainly includes two parts, which are the drawing and the rigidity of the gear. The design of the gears in the UG NX can not only reduce the workload of the design staff, but also improve the design efficiency and quality of the parts. Specific methods are as follows:

Under the UG Modeling module environment, using the expression function to put in gear parameters and expressions as shown in Table 1.[11].

<table>
<thead>
<tr>
<th>Parameter name</th>
<th>Parameter symbol</th>
<th>Initial value or expression</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modulus</td>
<td>m</td>
<td>3</td>
</tr>
<tr>
<td>The number of teeth</td>
<td>z</td>
<td>18</td>
</tr>
<tr>
<td>Profile angle</td>
<td>α</td>
<td>20</td>
</tr>
<tr>
<td>Base-circle diameter</td>
<td>d_0</td>
<td>m * z * cos α</td>
</tr>
<tr>
<td>Pitch-circle diameter</td>
<td>d</td>
<td>m * z</td>
</tr>
<tr>
<td>Tooth-top circle diameter</td>
<td>d_α</td>
<td>d + 2 * h_a</td>
</tr>
<tr>
<td>Tooth-top height</td>
<td>h_t</td>
<td>m</td>
</tr>
<tr>
<td>Tooth-root height</td>
<td>h_r</td>
<td>1.25 * m</td>
</tr>
<tr>
<td>Other parameters</td>
<td>omit</td>
<td>omit</td>
</tr>
</tbody>
</table>

From the table above, we can get a formula, which is a condition to calculate other needed parameters, as follows:

$$d_0 = m * z * \cos \alpha$$  (1)

In the formula, $d_0$ is the base-circle diameter; $m$ is the modulus; $z$ is the number of teeth; $\alpha$ is profile angle. On the basis of this formula, we can get the involute equations of gear design, which are the x coordinate equation of involute and y coordinate equation of involute, as follows:

$$x = \frac{d_0}{2} \cos(s) + \frac{d_0}{2} \cdot rad(s) \cdot \sin(s)$$  (2)
In formula (2) and formula (3), $d_i$ is the base-circle diameter in formula (1); $s$ is the curvature of unfolding angle. Then we should calculate the bending fatigue strength of the tooth root. When the gear is in load, bending moment of the tooth root is the maximum, so the bending fatigue strength at the tooth root is the weakest. When the teeth are in top gear meshing, bending moment at the tooth top is the maximum[12]. Therefore, the bending strength of tooth root should be calculated according to the load on the highest point of meshing area of single tooth, and the strength condition is derived for:

$$\sigma_f = \frac{2KT}{\phi m z_i^2}Y_F Y_m Y_\varepsilon \leq [\sigma_f]$$

(4)

In the formula, $Y_F$ is the tooth shape factor; $Y_m$ is the stress correction factor; $K_A$ is the use of coefficient; $K_V$ is dynamic load coefficient; $K_R$ is distribution coefficient of tooth load; $K_\beta$ is a tooth to load distribution coefficient, and can be found in manual; $Y_\varepsilon$ as is a coincidence coefficient, which can be ignored; $\phi$ is width coefficient, which is used to calculate fatigue strength of tooth surface. In the intended service life, the strength condition of the tooth surface that don't produce fatigue pitting is expressed in the follow formula:

$$\sigma_H = \frac{2KT}{\phi m z_i^2}Y_F Y_m Y_\varepsilon \leq [\sigma_H]$$

(5)

In the formula, $Z_{hi}$ is the node area factor; $Z_\varepsilon$ is elastic coefficient; $Z_i$ is a coincidence coefficient, and can be checked out in the manual; The "+" is used for external engagement, and "-" for internal engagement. It's necessary to select gear type, precision grade, material and tooth number as follows: Transmission of cylindrical gear with straight teeth,7 level of accuracy, the number of small gear teeth $z_1$ equals to 28,large gear tooth number $z_2$ equals to 90, and modulus m equals to 3.Finally, we can generate gear by the given parameters of the gear by the following steps: Firstly, we need to go into the UG page, then get into the gear master dialog by clicking on the "generated gear" button to get the gear we designed[13].

V. CONCLUSIONS

The software of UG NX has become an indispensable part of the modern industry, what’s more, NX 7.0 also added a new synchronization modeling technology enhancements, so as to shorten the product design cycle, and improve the production efficiency. The secondary development of the UG software can be applied to the enterprise's one-system of tool environment and product development with intelligent design system, so the function of UG software in industrial design is further enhanced. At the same time, the software can be used in many fields of industrial design, and greatly improves the reliability and effectiveness of industrial product design. So it is very important for the application and research of the software.