普通高等教育“十三五”规划教材

3D Mechanical Design and 3D Annotation Using UG NX

UG NX 三维机械设计及三维标注

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Abstract

The objective of this book is to help the readers to understand the fundamentals of computer-aided design (CAD) concepts and become familiar with the operations in CAD modeling so as to effectively use CAD approaches for mechanical product design. The commercial CAD system, UG NX is employed as the platform for CAD environment. In Chapter 1, the brief introduction to the development process of 3D modeling techniques in CAD and its applications to mechanical design are given. Following this introduction, important topics in product modeling using UG NX, including the basics of UG NX, sketch, part modeling, assembly modeling, drawing and 3D annotation are provided.

This book aims at providing a textbook for graduate students of science and technology universities whose major are Mechanical Design Manufacturing and Automation, Automobile Engineering, Mechanics, Process Equipment and Control Engineering, etc. This book provides the main guidelines for the reader to apply 3D software tools to support practical design application. Moreover, this book provides engineering senior a comprehensive reference to learn advanced technology in support of engineering design using 3D CAD technology. In addition to classroom instruction, this book should support practicing engineers who wish to learn more about the e-Design paradigm at their own pace.

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PREFACE

With the development of science and technology and the globalization of the economy, traditional manual design is gradually being replaced by computer aided design (CAD). The application of computer technology in product design has evolved from the calculation and drawing in the past to the integration of today's 3D modeling, optimization design and simulation, which greatly shortens the design and manufacturing cycle of the product and improves the design quality.

The machinery industry plays a pivotal role in the entire industrial production process. Mechanical CAD is of great significance to promote the development of the machinery industry and the improvement of the level of science and technology. With the continuous improvement of China's manufacturing informatization level, the demand for high-level talents in the machinery industry who are familiar with both professional knowledge and 3D CAD technology is becoming more and more intense. Therefore, in the preparation of this book, we strive to reflect the contemporary mechanical 3D CAD. The basic content and development level, focusing on the basic content of mechanical CAD technology and the application of 3D CAD technology.

This book uses UG NX software as a 3D CAD platform. It mainly introduces the basic content and development history of mechanical CAD technology, UG basic knowledge, sketching, 3D modeling and typical mechanical parts modeling, assembly design, engineering drawings and product manufacturing information (PMI). This book takes the design of the planetary gear reducer as the main line. While introducing the UG command and function, it also introduces the ideas and methods of the typical parts and assembly design of the reducer. In the process of learning, readers can not only master the concept of CAD, the commands and functions of UG, but also know well about the methods and steps of engineering design using 3D CAD.

As a combination of theory and practice, the content is easy to understand. The main functional commands are explained with the operation examples step by step, so that beginners can apply learning commands to specific design. This can more effectively stimulate readers' interest in learning and improve learning effect. The book was written by the College of Mechanical and Vehicle Engineering of Taiyuan University of Technology, and Zhang Ruiliang was the editor. Zhang Ruiliang wrote Chapter 1 and Chapter 4; Ding Hua wrote Chapter 2 and Chapter 5; Yao Aiying wrote Chapter 3; Xin Yupeng wrote Chapter 6; Liu Feng wrote Chapter 7. The entire book was organized by Zhang Ruiliang, and Xin Yupeng reviewed the manuscript.

We are thankful to graduate students, Muhammad Yousaf Iqbal Rao and Anuj Desaihave, for their friendly help in the process of writing this book. At the same time, the book refers to a large amount of literature and materials, and we would like to express our deep gratitude to the original author. Due to the limited editorial level, there might be certain inadequacies or mistakes in the book

and we sincerely hope that readers will criticize and correct.

Example files and slides needed for this book made by authors from College of Mechanical and Vehicle Engineering of Taiyuan University of Technology are available and could be obtained by author's e-mail (rl_zhang@163.com).

Written at Taiyuan University of Technology

2018-11-20

CONTENTS

Chapter 1 Introduction to CAD in Mechanical Engineering

This chapter aims at introducing the development history of CAD technology and its application in mechanical design, the advantages of 3D mechanical design and the data exchange standard of CAD.

The main aspects of this chapter are:

(1) Master development of CAD technology;

(2) Understand the advantages of 3D mechanical design;

(3) Understand CAD data exchange standards.

1.1 Development History of CAD Technology

The first computer graphics system was invented by the United States in the 1950s, then a passive computer-aided design techniques with simple plot output capabilities was developed. At the beginning of 1960s, surface modeling in computer-aided design (CAD) application became a reality, and commercial computer graphics equipment was introduced in the mid-1960s. In 1970s, a complete CAD system began to form. Thereafter, a raster scan display capable of producing realistic graphics appeared and various forms of graphic input devices such as manual cursors and graphic input boards were introduced that promoted the development of CAD technology. It began to grow rapidly as drawing on the computer screen became feasible. People wished to use this technology to eliminate the traditional manual sketching that is cumbersome, time-consuming and low in drawing accuracy. At that time, the starting point of CAD technology is to use the traditional three-view method to express the parts and use 2D engineering drawings produced on paper as media for technical communication. This technology is known as two-dimensional (2D) modeling technology. The meaning of CAD at this time is only a substitute for the drawing board, it means

Computer Aided Drawing (or Drafting) rather than CAD that has been discussed here. At the beginning, 2D system AutoCAD was produced by Autodesk, dominated the CAD drawing market in China. Nowadays CAD users, especially in the initial CAD users, 2D drawings still occupy a substantial proportion in China.

CAD technology has tremendous experienced development in its almost 70-years evolution history, and its technological development process is shown in Figure 1-1.

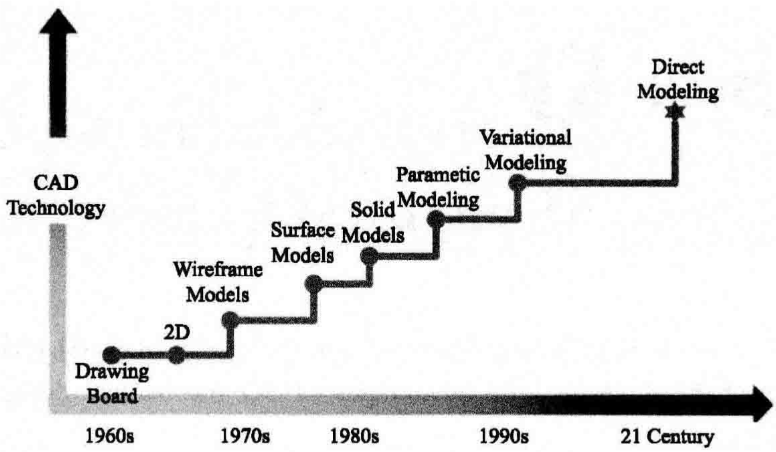

Figure 1-1 Development history of CAD technology

1.1.1 Wireframe models

In the end of 1960s, we began to study the construction of 3D solids with wireframes and polygons. The model using this form is called wireframe model. This modeling method describes objects with a set of edges formed entirely by vertices just like the frame made by the wire. Thus, the wireframe model gets its name. The wireframe model brings accessibilities for the production of engineering drawings such as a simple structure, low requirements on computer performance, can represent three-dimensional data of basic objects, can generate arbitrary views, and can maintain a correct projection relationship between views. In addition, trimetric views and perspective views can be generated, which brings a big improvement in 2D system. It is inconclusive for the reality of the object in a wireframe because all the curves are displayed. It is impossible to represent the nonpolygon surface such as a cylinder or a sphere due to the lack of surface geometric information. In addition, especially the relationship information between edges and faces, faces and faces are missing in the data structure. Because of that, we cannot construct a solid entity, cannot identify the face and body, cannot distinguish the inside or outside of a solid object. A wireframe model is not able to support a finite element mesh for structural analysis of a physical object other than beam or truss structures. The initial wireframe modeling system can only express basic geometric information and cannot effectively express the topological relationship between geometric data. Due to the lack of surface information of the model both CAM and CAE can't be realized (Figure 1-2).

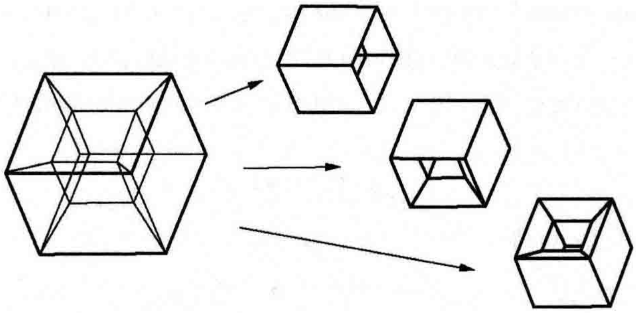

Figure 1-2 Wireframe model

1.1.2 Surface models

In the 1970s, the aircraft and automotive industries are in the growing development. There were a large number of free-form surfaces problems encountered in aircraft and automobile manufacturing. At that time, only multi-section views and characteristic parallel lines could be used to determine the free-form surfaces. Due to the incompleteness of the three-view representation method, it was often seen that the sample produced was very different or even completely different from what the designer imagined after the design was completed. Designers can't guarantee whether the shape of the surface designed can meet the requirements. Therefore, the clay model was often produced proportionally as the basis for design evaluation or scheme comparison. The slow and complicated production process had greatly delayed the development time of the products, and the requirement for updating the design means was getting higher and higher. In the 1970s, the Bézier algorithm was proposed by Pierre Bézier, which provided the designer to deal with surface and curve problems with a computer (Figure 1-3). The surface modeler CATIA was released by Dassault Systemes in 1981. A surface can be thought of as an infinitely thin shell stretched over a wireframe. Based on the data structure of the wireframe model, surface models represent a shape by its surface geometry. It is formed by adding various related data that can form the solid surface. A surface model includes information about the faces and edges of a part and can express the topology between the edge and the face. In surface model we can achieve face-to-face intersection, rendering, surface area calculation, hiding and other functions. The application of the surface model indicates that CAD technology is freed from the three-view mode that simply acts like the engineering drawings. First time the main information of the product parts is fully described by the computer. A surface model is good for visualizing complex surfaces and supports NC tool path generation. The surface modeling system CATIA brought the first CAD technology revolution. The development methods of complex products such as airplanes and automobiles are better than the old ones. The design development cycles has also been greatly accelerated and the automotive industry has begun to use CAD technology in large quantities.

Since the surface model can only represent the surface and boundary of the object and it cannot be cut. The physical properties such as mass, centroid, and moment of inertia of a solid object represented in a surface model is hard to determine, and it is difficult to express complex manufacturing information. Additional information must be added to a surface model in order to specify in/out and top/bottom of the physical object that the surface model represents.

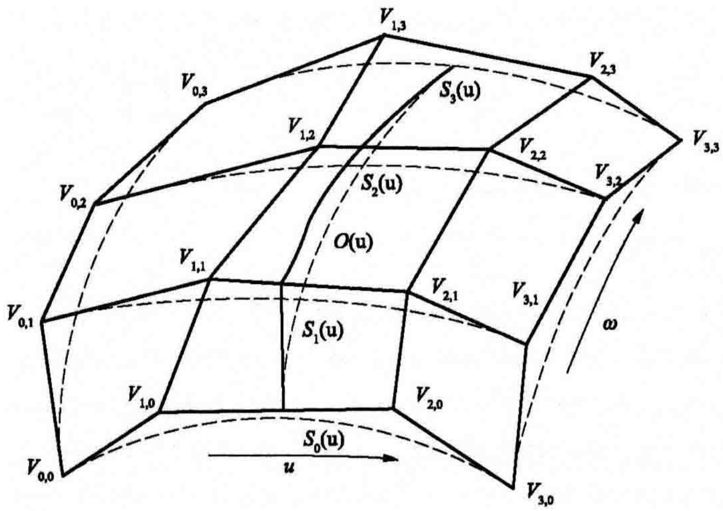

Figure 1-3 Bézier surface

1.1.3 Solid models

The problem of CAM can be basically solved with the surface model. However, since the surface model technology can only express the surface information of the shape, it is difficult to accurately express other characteristics of the part such as mass, center of gravity and moment of inertia etc., which are very unfavorable to CAE. The biggest problem is that the pre-processing of the analysis is particularly difficult. Based on the exploration of the development of CAD/CAE integration technology, Integrated Design and Engineering Analysis Software (I-DEAS) based on solid modeling technology was produced by Structural Dynamics Research Corporation in 1982, and used primarily in the automotive industry, most notably by Ford Motor Company and General Motors. Solid models contain information about the edges, faces, and the interior of the part. The mathematical description contains information that determines whether any location is inside, outside, or on the boundary surface. In terms of mathematically representing a solid object, two major modeling methods, constructive solid geometry (CSG) and boundary representation (B-rep), are widely employed by geometric modeling kernels, which are the core of CAD systems. CSG approach expresses the sequential process of modeling, and B-rep model specifies connectivity information of vertex, edge, face, and volume that must be defined to construct the solid model. The solid modeling technology can accurately express all the attributes of the part, and contain information about 3D entity of the part. It gave guarantee that only the parts that can actually be realized are shaped will not lack edges, faces, and no edge will penetrate into the part entities, thus avoiding errors and unrealizable designs. An advanced overall shape definition method can be provided to support new models from old models through Boolean operations. The solid model theoretically helps to bring together model expressions in CAD, CAE and CAM which brings amazing convenience to the product design.

A solid model is the ultimate way to represent general objects, which are physically solid objects. It represents the future way of CAD technology. Based on this consensus, it became the mainstream of CAD technology development at that time. It could be said that the popular application of solid modeling technology marks the second technological revolution in the history of CAD development (Figure 1-4).

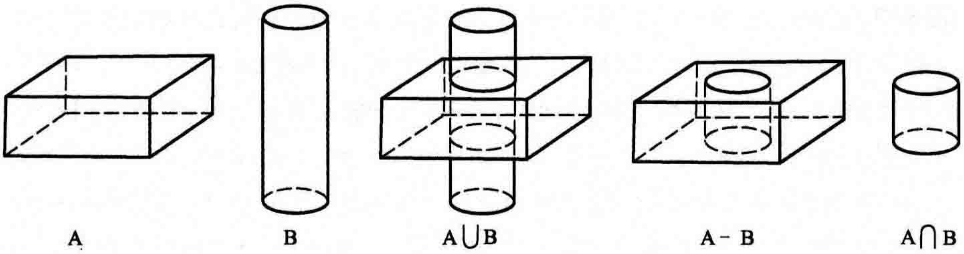

Figure 1-4 Schematic diagram of the union, difference, and intersection of a 3D primitives

Solid modeling technology not only brings the improvement of the algorithm and the hope of future development but also brings the extreme expansion of the amount of data calculation. Under the hardware conditions at that time, the calculation and display speed of solid modeling was very slow and it was rather reluctant to design in practical applications. CAE based on the physical model is originally a higher-level technology that's why the popularity is cramp and the reflection is not strong. In addition, in the face of the contradiction between algorithm and system efficiency, many companies give favor solid modeling technology that do not have the power to go. Develop it, but instead turn to the surface model technology that is relatively easy to implement. The technical orientation of each company has once again divided. The solid modeling technology has not been fully promoted in the entire industry.

1.1.4 Parametric solid model

Parametric is a term used to describe a dimension's ability to change the shape of model geometry as soon as the dimension value is modified. In the mid of 1980s, for the problems of unconstrained free-form modeling techniques, the researchers proposed a better algorithm than the unconstrained free-form modeling, called parametric solid modeling method. In 1989, Parametric Technology Corporation (PTC) developed Pro/ENGINEER software using object-oriented unified database and fully parametric modeling technology that providing an excellent platform for 3D solid modeling. Parametric modeling manipulates parameters to control the geometric shape of a solid objects. Parameters come from dimensions in 2D profiles in sketch, dimensions on 3D solid features and variables in user-defined equations. The core of parametric modeling is to use geometric constraints, engineering equations and relationships to illustrate the shape characteristics of the product model. If defined properly, the entire part geometry can be controlled by a small number of key parameters. Design intents can therefore be captured through the change of the small set of parameters. The dimensions of the geometric entities that make up the product shape can be parametrically related as equations. The parametric modeling technique is also called the dimension-driven geometry technology. It brought the third technological revolution in the history of CAD development.

The guiding thought of the parametric system is only need to operate in the way specified by the system. The system gives guarantee of correctness and efficiency of the design that you generate otherwise it refuses to operate. This kind of operation also has a lot of side effects. First of all, the user must follow the intrinsic use mechanism of the software such as never allowing underconstraint and solving in reverse order etc. Secondly, it is difficult for the designer to put all the dimensions when the cross-sectional shape of the part is complicated. It is difficult to determine which dimensional modification will satisfy the design intents. Finally, the dimension-driven range is also limited, if an unreasonable parameter is given to cause a feature to interfere with other features, a change in topological relationship is caused. Therefore, from the application point of view, the parameterization system is particularly suitable for the parts industry where the technology is quite stable and mature. In such an industry, the shape of the part changes little, and often only the analog design is used that is the shape is basically fixed, and only a few key dimensions need be changed to obtain a new serial design result.

1.1.5 Variational modeling technology

Parametric technology requires that all the dimensions are fully defined, that is the designer must consider the shape and size together in the early design stage and the whole design process. It controls the shape through the size constraint and drives the shape modification through the dimension change. Dimensional parameters are regarded as a starting point. Once the shape of the designed part is too complicated, how to change these sizes to achieve the desired shape is not intuitive. Moreover, while topological relationship of key shapes are changed in the design, geometric features losing certain constraints can also cause system data to be confused. In fact, full constraint is a rigid rule for designers which will interfere with the creativity and imagination of the designer.

In fact, when we are doing mechanical design and process design, we always hope that the parts can be built as we desire and can be dismantled at will, allowing us to construct three-dimensional designs on the computer screen. We hope to keep each intermediate result for use in iterative design and optimal design. In response to this demand, SDRC developers have proposed a more advanced variable technology, variational geometry extended (VGX), based on parametric technology.

The variable technology further distinguishes the size "parameters" defined in the parametric technique into geometry constraints and size-constraints, rather than constraining all geometries by size only as in the parametric technique. The reason for adopting this technology is that in the conceptual design phase of a large number of new product developments, the designer first consider the design ideas and then fixes them in certain geometric shapes. The exact dimensions and the strict dimensional positioning relationships between shapes are difficult to be fully determined at the initial stage of design, so it is naturally desirable to allow the existence of under size-constraints in the initial stages of the design. In addition to considering the geometry constraints the variable design can also directly solve the engineering relationship as a constraint directly with the geometric equation without additional model processing. The advantages of using variable technology are:

(1) The designer can adopt the design method of the size after shape, allowing the use of incomplete size constraints and only giving the necessary design conditions. In this case, the correctness and efficiency of the design can still be guaranteed.

(2) The modeling process is a process similar to an engineer thinking about a design in his mind. The geometry that meets the design requirements is the first and the dimensional details are gradually and subsequently improved.

(3) The design process is relatively free and relaxed. The designer can consider the design scheme more and does not need to care too much about the inherent mechanism of the software and the design rules. Therefore, the application field of the variable system is also vast.

(4) In addition to the general serialized part design, the variable system is especially handy when doing conceptual design, and is more suitable for innovative design such as new product development and old product remodeling design.

Based on the idea of variational modeling theory, SDRC launched I-DEAS Master Series software in 1993, formed a set of unique variational modeling theory and software development methods. The variational technology not only maintains the original advantages of parametric technology but also overcomes many of its disadvantages. Its successful application has provided more space and opportunities for the development of CAD technology and also driven the fourth technological revolution of CAD development. At present, most CAD systems provide support for variable technology, and variabilization is also the mainstream of current 3D CAD systems.

1.1.6 Direct modeling and synchronous modeling techniques

The core of solid modeling is constructive solid geometry (CSG) and boundary representation (B-rep) modeling method. CSG expresses the sequential process of modeling and B-rep is the vertex, edge, face, and body information of the 3D model. The feature-based parametric modeling system adds the concept of feature tree to CSG. This is the kernel principle of popular mainstream feature-based 3D mechanical CAD systems at this time.

The direct modeling core has only B-rep information but doesn't have CSG information, because the order of the modeling is not considered, so designers can quickly define and edit geometry by simply clicking on the model geometry and moving it. In 2008, Siemens PLM Software took the lead in releasing synchronization technology in the PLM industry. The 3D modeling technology was further improved forming a variety of modeling methods including direct modeling, feature modeling, surface modeling and synchronization technology. The synchronization technology is a three-dimensional modeling method and constraint solving technique that combines feature-based modeling and direct modeling to achieve dimension-driven and stretch deformation (Stretch) in a three-dimensional environment. Direct modeling method can retain the solid feature information of the part and implement the design changes. Thus, the history-less modeling and feature-based parameter modeling are perfectly compatible, enabling rapid modification of the 3D model, helping achieve rapid design changes and product family design.

Synchronous technology advances a large step on top of existing history-based parametric modeling technique. Synchronous modeling technique examines the current geometric relation of the product model in real time and combines them with the parameters and geometric constraints added by the designer. Thus, the designer can evaluate and build new geometric models, and edit the model without having to repeat the entire modeling history. Designers no longer have to research and analyze complex constraint relationships to understand how to edit models. They don't have to worry about subsequent model associations for editing. Synchronous modeling technology breaks through the inherent architectural barriers of history-based design systems. The drawbacks of the current design process caused by parametric modeling technology can be avoided. Designers can effectively perform dimension-driven direct modeling and no re-creation or conversion is required. Synchronous modeling technology can modify the product more quickly, which will greatly improve design efficiency and reduce product development costs, therefore shorten the time to market (Figure 1-5).

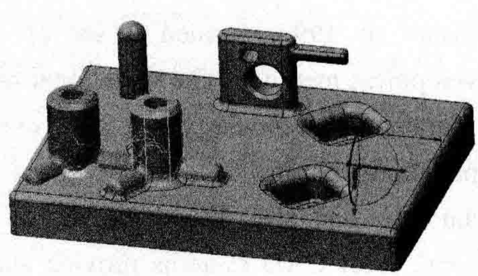

Figure 1-5 Synchronous modeling

According to the design task and the product characteristics of the design, the designer should determine what kind of modeling method will be employed. For example, the art