

A Systematic Teaching Method of NX sketch

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Abstract. The sketch is a fundamental expression means of product design, in the current mainstream 3D design software; it is a common feature which can implement the translation of the two-dimensional contour to three-dimensional solid. For the current lack of sketch knowledge points in teaching, in the paper, taking the teaching of NX software for example, propose a systematic teaching method about the sketch of NX. And elaborate some knowledge point such as the sketch type, the internal sketch, and the external sketch etc. At the same time, it also introduces the knowledge including sketch curves, geometric constraints and dimensional constraints etc. In the end, it illustrates the application of teaching methods and techniques with two examples.

Introduction

The sketch is a fundamental expression means of product design, and it occupies a very important position in the product design process [1-2]. Concept design is the early stage of product design. In this stage, the design is vague, incomplete, and original; there is only simple structural relationship and foundation design dimensions [3]. The sketch will record the design ideas and creativity of designers, and it is the important step converting design ideas to a visual product [4]. The sketch is the base of 3D (Three-dimensional) modeling and the first step to create the product [5].

In the current mainstream 3D design software, such as NX, Pro/E, CATIA, the sketch is a common feature which can implement the translation of the two-dimensional contour to three-dimensional solid. According the shape of the product and the idea of the designer, draw the sketch, and then create solids or sheets through the subsequent commands like Extrude, Rotate, and Sweep etc. The sketch can draw complex contours and achieve much more complex shapes than the rectangular, cylinder, sphere, etc. Due to the modifiability of the drawing curve by using the sketch feature, the modified curve can be updated automatically, and the subsequent generated solids by using Extrude and Rotate etc. will also be updated. There is no need the designer to re-run the operations like Extrude and Rotate etc. And it can simplify the design process.

After years of product design and teaching process, for the teaching of the sketch, the authors concluded the existing general problems about the majority of sketch teaching. As follows: ignore the importance of teaching the sketch, and have fewer hours of teaching; the knowledge point about the teaching content of Sketch is disorder and unfocused; Sketch exercises arranged is inappropriate, and failure to effectively target the relevant knowledge points; Difficulty of the exercises is inappropriate.

For the current lack of sketch knowledge points in teaching, in the paper, taking the teaching of NX software for example, propose a systematic teaching method about the sketch of NX. The teaching method can extended to other design software.

The Systematic Teaching Method Model

Fig. 1 shows the teaching method model based on NX Sketch system produced by the paper. The teaching model includes six parts including basic concept, important knowledge, geometry creation,

constrains, sketch preference and comprehensive exercise. The model includes all the knowledge points on the sketch, and the knowledge points have been arranged for a reasonable order.

‘Geometry creation’ and ‘constrains’ are the key contents of NX sketch, and they are also the parts spent the most time. And the two parts are arranged appropriate exercise for consolidating the knowledge learned. ‘Basic concept’ and ‘important knowledge’ are arranged before the Core knowledge of sketch, and the purpose is to enable the participants to have a global, general understanding for the sketch problems about the basic concept of sketch, how to create a sketch, and how to apply sketch before studying the sketch. ‘Comprehensive exercise’ arranged at last is the integrated application for ‘important knowledge’, ‘geometry creation’, ‘constrains’ and ‘sketch preference’. And it is the test and feedback on the above knowledge. It is the key of sketch training to choose ‘Comprehensive exercise’ appropriately.

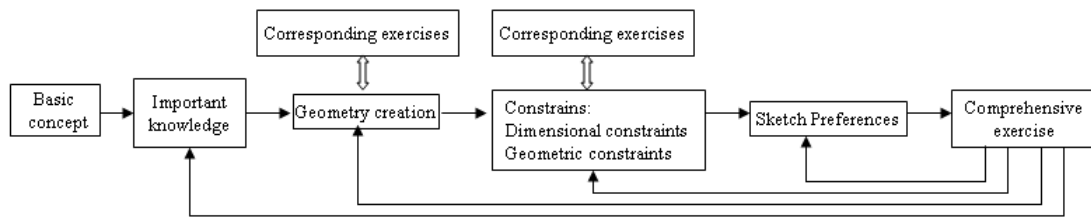


Fig. 1 The systematic teaching method model

The Key Content

Basic Concepts. In a sketch environment with friendly interface, it can be created a sketch feature. Sketch feature is a collection of two-dimensional curves and points on a plane. This feature is a reusable object, and it can be used to build new feature solid or sheet body. Associated with the subsequent creation of features, the sketch is the base to implement parametric feature modeling of NX software. As the sketch changes, the new features will also be updated accordingly.

The sketch is widely used in the modeling process. It is often used in the following conditions: it needs to control the curve parametrically, create the model with more complex shape, as a guide path for the sweeping feature, and the generated generatrix for free-form features, if the shape itself is adapted to extrude or rotate, it can be used as the base feature of a model. Fig. 2 shows some instances of the sketch application: Fig. 2a shows the sketch curves of Extrude, Fig. 2b shows the sketch curves of Rotate, Fig. 2c shows the sketch curves of Sweep as guide line and section line, Fig. 2d shows the sketch to generate sheet as section line.

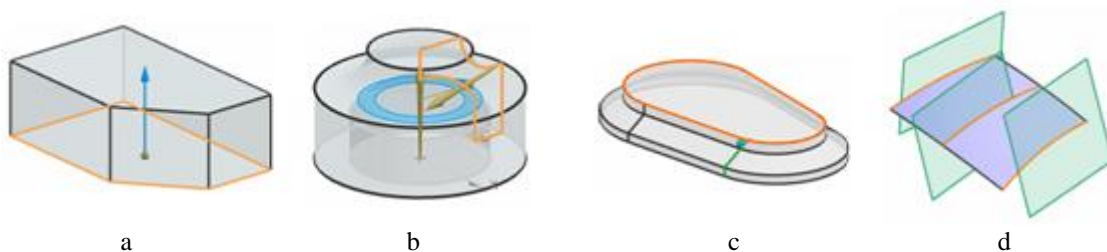


Fig. 2 The example of sketch application

Important Knowledge

The Type of the Sketch. Sketch module contains two types of sketches: ‘Sketch on plane’ and ‘Sketch on path’. ‘Sketch on path’ is a kind of Sketch with special constraint type, it can be used to create an outline for ‘Variational Sweep’. In addition, Extrude, Rotate features can also use this type of sketch. This type of sketch path is generally the free-form curves; the sketch plane is defined according to the selected position of the curve path.

‘Sketch on plane’ is the most common type of a sketch, that is, creates a sketch on the existing planar surface or on a datum plane. When using this type of sketch to create model, with particular attention to the following two aspects [6]: firstly, when creating a new file, there is no any solids in model space, it needs to select the saved reference plane or coordinate system plane (XC-YC, XC-ZC, YC-ZC) as the sketch plane, as shown in Fig. 3. Secondly, if there is solid in model space already, according correlation principle, it needs to select the planar surface of the existing solid or the datum plane associated with the solid.

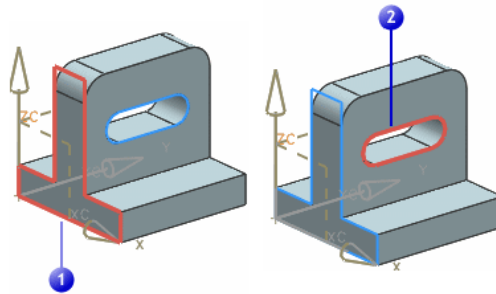


Fig. 3 Sketch on path

Direct Sketch and Sketch in Task Environment. You can create a sketch by using direct sketches and sketches in the task environment. There is no need to enter the sketch environment to make the operation simpler and is easy to be modified. But when you need to use the commands including ‘project curve’, ‘intersection point’, ‘intersection curve’, you must enter the task environment sketch. It is recommended for beginners to use the sketch command in tasking environments, in order to more intuitively recognize the overall knowledge of sketch command.

Internal Sketch and External Sketch. With the command ‘extrude’, ‘revolve’, the generated sketch is internal sketch. And the external sketch is directly shown on the part navigator, and may be used for the subsequent features. Internal Sketch and External Sketch can be finished mutual switching through carrying out ‘make sketch external’ and ‘make sketch internal’.

Geometry Creation and Constrains. Sketch curve reflects the designer's thinking. How to achieve the creating quickly and easily and to be modified easily is particularly critical [7-10]. As shown in Fig. 4, in sketch environments, ‘sketch tools’ provides a rich command to complete the creation of the sketch curve. Use the command ‘profile, line, arc, circle, fillet etc.’ to create a simple sketch curve. Use the command ‘quick trim, quick extend, make corner’ to modify the sketch curve. And for more complex and regular curve, it can use the command ‘pattern curve’ to achieve linear array shown in Fig. 5a, and circular array shown in Fig. 5b. Use the command ‘mirror curve’ to achieve curve mirror shown in Fig. 5c.

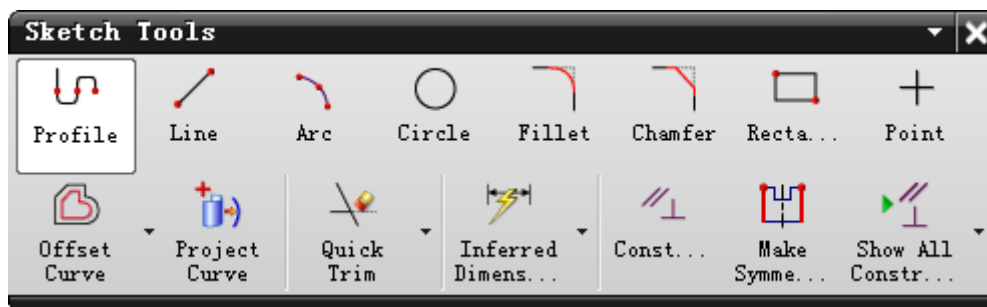


Fig. 4 Sketch tools

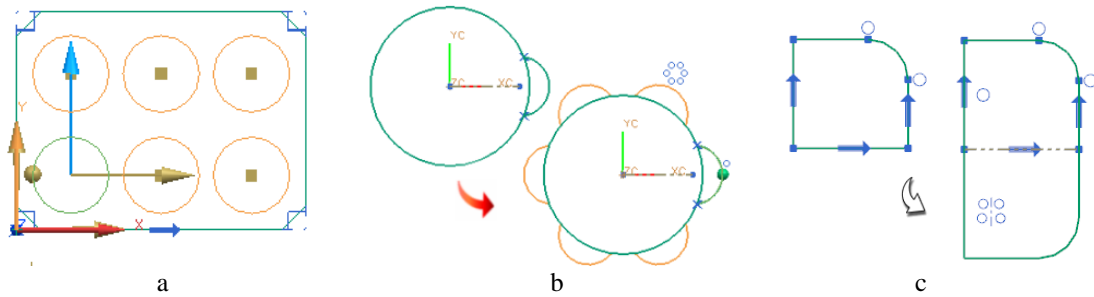


Fig. 5 complex curve

The constraint relationship among the sketch objects also is an important way to reflect a designer's design ideas. 'Constraints' of NX contains geometric constraints and dimensional constraints. There exists Degree of Freedom (DOF) in the place which the sketch curve is not constrained, and DOF arrow will be removed when the system add a reasonable restraint to a sketch. The final sketch might be Under Constraint, Full Constraint, Over Constraint and other states.

For geometric constraints, it can achieve geometric characteristics of an object of the sketch (such as the position of the point), and the relationship between objects sketch of two or more (It requires two lines orthogonal or parallel, tangency between the two arcs etc.) Common geometric constraint types include: Fixed constraint class (fixed, constant length), Point object class (coincident, point on curve, midpoint), Linear object class (horizontal, vertical, perpendicular, collinear, equal length), Arc object class (concentric, tangent, equal length). In addition, there are some advanced types of constraints, such as 'mirror curve, make symmetric, offset curve etc.'

'Dimensional constraint' is to establish the size of the sketch curves object, such as the length of the line, radius of the arc etc. It can also define the relations of objects, such as the distance between two points. A dimensional constraint is equivalent to the size of the drawing, Changing the sketch dimensions can drive the curve object controlled by the sketch, thereby can change the solid model controlled by the sketch curves.

Comprehensive Exercise. In the process of drawing the outline and adding constraints, the application of appropriate skills can make the process of creating the sketch easily. Through the following examples of sketches, it will illustrate the relevant skills when creating sketch.

Example 1, it needs to create the sketch curve as shown in Fig. 6a. Firstly, select the suitable geometric constraints to fix a sketch point S, and use the dimensional constraints to constraint the maximum outline of the sketch, as shown in Fig. 6b, the dimensional constraints is 'p0 and p1'. And then complete the rest of the curve.

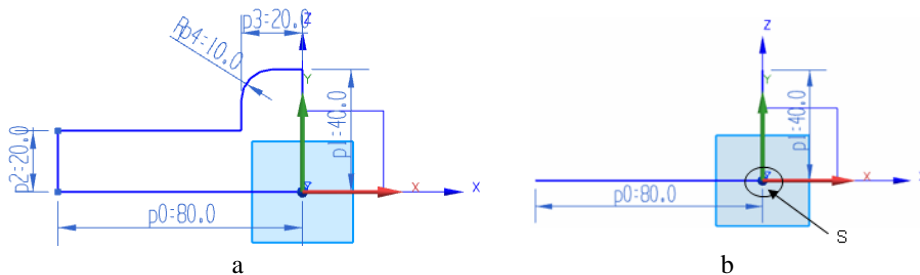


Fig. 6 exercise I

Example 2, it needs to create the sketch curve as shown in Fig. 7a. Firstly, select the suitable geometric constraints to fix a sketch point S1, and then constraint some special sketch point S2, as shown in Fig. 7b, in addition, try to add constraints for it while drawing contour lines, and each time add a sketch curve, it will be restricted as completely as possible.

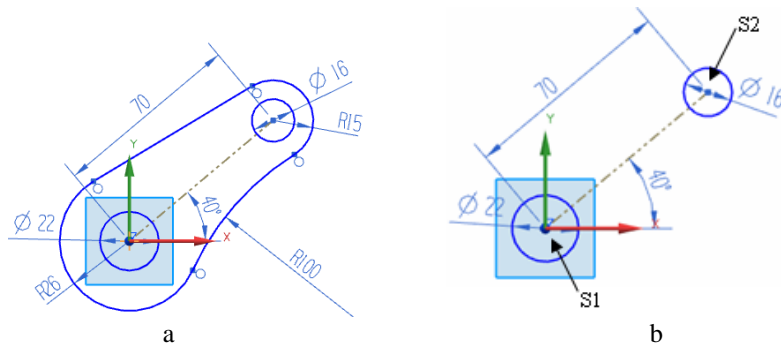


Fig. 7 exercise II

Summary

The systemic teaching method of NX sketch proposed by the paper has been verified in the actual teaching, and it is complete and effective for the teaching. When a sketch is created, how to meet three aspects of contents including sketch curves, geometric constraints and dimensional constraints is the key of mastering the sketch knowledge.

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