Theory and Practice of Science and Technology
2024, VOL. 5, NO. 6, 11-17
DOI: 10.47297/taposatWSP2633-456902.20240506

A Comprehensive Comparative Study of Creo and Rhino Modeling Methods

Hansheng Yan, Huimin Yi, Zhengtao Xiao, Shiying Qiu, Yao Long

Guangdong Polytechnic of Industry and Commerce, Guangzhou 510510

ABSTRACT
This study focuses on the comparative analysis of Creo and Rhnio software, among which Creo performs outstandingly in the field of parametric design. Its integrated free-form surface design FDX module can seamlessly integrate free-form surface processing and solid modeling functions. In contrast, Rhnio combines its precise 3D modeling capabilities with Keyshot's real-time realistic rendering technology to create an efficient 3D design solution in the current product design field. This study conducted an in-depth comparative analysis of these two software from three dimensions: the flexibility of appearance modeling, the convenience of model modification, and the smoothness of the design process, and further analyzed their respective advantages and focuses in product design practice. Intended to provide practical guidance for college students when choosing design tools, as well as useful references and inspirations for research and practice in the field of product structure design and modeling analysis.

KEYWORDS
Creo; Rhnio; Product appearance modeling; Model modification; Structural Design

With the increasing development and popularity of computer software, large-scale 3D modeling software such as Creo and Rhino have been widely applied step by step. The continuous development of technology has improved design efficiency and led to the diversification of industrial design methods[1].This article starts from the perspective of industrial design, compares and studies the modeling methods and processes of Creo and Rhino, summarizes their respective advantages and disadvantages, and their applications in industrial design. Finally, suggestions are made for the shortcomings of the software, and future development directions are discussed. Through this study, Creo and Rhnio can be combined with the same case to compare their strengths and weaknesses. Students can use this case to analyze the results and achieve the effect of learning from each other's strengths and weaknesses.

1 Introduction to Creo and Rhnio

Creo, as one of the main tools for product industrial design, uses concept sketches as references in the digital design process, provides parametric modeling capabilities, and has flexibility in free-form surface processing. Helping businesses save time and money required to create complex structures. Creo's strength lies in modeling, including solids, surfaces, sheet metal, and piping[2].

Rhino software has demonstrated extensive application value in various fields such as 3D animation production, industrial manufacturing, scientific research, and mechanical design. It can

quickly create 3D NURBS models that require high precision, flexibility, and complexity. Rhino has demonstrated outstanding conversion and enhancement capabilities. I am particularly skilled in rendering, conceptual product ideation, showcasing unreleased products, and market promotion, playing a pivotal role in the industry.

2 Analysis and Comparison of Creo and Rhnio in Product Modeling

Rhino software has the comprehensive ability to create, edit, analyze, and convert NURBS curves, surfaces, and solids without any constraints on complexity, angle, or size during this process. On the other hand, Creo (formerly Pro/E), as a pioneer of parametric design concepts, introduced single database technology to effectively solve the problem of feature correlation[3]. Rhino software has the comprehensive ability to create, edit, analyze, and convert NURBS curves, surfaces, and solids without any constraints on complexity, angle, or size during this process. On the other hand, Creo (formerly Pro/E), as a pioneer of parametric design concepts, introduced single database technology to effectively solve the problem of feature correlation.

2.1 Analysis and comparison of the two in two-dimensional modeling

The three elements of 2D modeling are points, lines, and surfaces. In Rhino, the process of building entities is flexible and varied, usually achieved by constructing surfaces and subsequently merging these surfaces. In contrast, Creo integrates the specific construction and editing of faces as well as the construction and editing of 3D solids in the same toolset, meaning that the tools for creating faces can also be used to create solids.

In terms of methods for constructing points, lines, and surfaces, Rhino provides a richer and more flexible range of options. It not only supports independent building by clicking directly in the view, but also allows users to overlay these elements on existing objects for building. In contrast, Creo's point tool, line tool, rectangle tool, and spline tool have relatively single functions, which is slightly less flexible compared to Rhino[4].

Rhino has added a right-click shortcut function to the design of its creation tools. For example, the control point curve tool activated by the right-click can easily generate curves that pass through multiple specified points, further improving the convenience of operation. However, Creo is unable to achieve this functionality. In addition, in Creo, precise grasp of dimensions and arbitrary modification of numerical values are one of the biggest features of Creo. Unlike it, Rhino's editing function does not rely on parameterization methods. Therefore, the points, lines, and surfaces constructed by it are considered conceptual objects, lacking parameterized properties, and their size, length, and proportion depend more on the modeler's intuitive perception and judgment.

2.2 Analysis and comparison of the two models in 3 D modeling

The 3 D modeling methods of Creo and Rhino can be classified into three categories: stretching, rotation, and multilateral surface.

2.2.1 Stretch, and rotate

Stretching and rotation are two common methods of creating models, achieved by stretching and rotating tools, respectively. Rhino shows a greater diversity when modeling with these two tools. Take the stretching tool (also known as the extrusion tool), it can not only stretch along a line or curve, but also stretch to specific points or form cone structures, providing a more flexible

operation option. In contrast, Creo's stretching tool supports stretching in a straight direction. Rhino also performs well in the use of rotation tools. It can not only rotate around the specified axis of rotation (i. e. Revolve operation), but also rotate along the preset path (i. e. RailRevolve operation), further broadening the possibility of design creation.

2.2.2 Multilateral surface

Creo multilateral structural surfaces are divided into three categories:

(1) scanning categories. Simple scan, variable cross-section scan, spiral scan;

(2) Free class. Simple mixing, scanning and mixing;

(3) Grid class. Boundary mix, styling tool.

The multilateral surfaces in Rhino are divided into four categories:

(1) Scanning class. Monorail scan, dual-track scan;

(2) Free class.Loft；

(3) Grid class. Lattice line structure surface, embedded surface;

(4) Repair category. Mixed, mixing, splicing, etc.

Rhino and Creo are different in solid building: Rhino creates and merges closed surfaces to restore entities to original faces, and Creo creates them directly without surface merging, and it is unbreakable once created. In terms of the flexibility of 3 D modeling, Rhino allows users to build a variety of common 3 D shapes directly in the view, while Creo follows the parametric modeling rules, requiring two-dimensional sketching and then generating 3 D models, which is a relatively complex process.

2.2.3 Editing of the 3 D model

Creo to the entity of editing ways, including adjusting the creation of two-dimensional model parameters or constraints, directly edit the entity line and surface, increase such as empty, shell, bars, drawing Angle, corners and slope characteristics, the material between entity cutting, and by dragging the point on the entity or using specific tools to implement bending, show equal operation.

Rhino, by contrast, can also chamfer the lines of 3-dimensional entities, edit the solid surface, perform Boolean operations between entities, or decompose the entities into surfaces for more detailed editing. However, in addition to the chamfer operation can easily input values for accurate control, other editing operations mostly rely on the object lock point, manual adjustment, and combined with visual judgment to complete, relatively low degree of automation.

3 Analysis and comparison of Creo and Rhnio on model modification

3.1 Model modification method of Creo

As a full parametric software, Creo needs to be very rigorous, suitable for parts design. The modification of models also has great convenience, and can be directly interface with structural designers. In Creo, users can modify or redefine the model completed or being built, and flexibly use the editable function of the software, which can effectively improve the flexibility of product modeling and the efficiency of design. The following is a brief description of the modified parts model.

(1) Creo allows users to modify the features of the parts, such as making the feature read-only, modifying the feature name and modifying the feature section, etc. For example, modify the name of a feature: double-click the feature in the model tree to enter a new name.

(2) Creo allows the user to redefine the existing feature to change the creation process of the feature. Choose different characteristics, and the content of their redefinition is also different. For example, by using a feature where a section is stretched or rotated, the user can redefine the section or redefine the reference for the feature. Creo provides redefinition feature operations for the model dialog and model tree for different features.

(3) Creo allows the user to insert new features between the original features, but note that if the redefined feature removes the original reference, it needs to be revisited and reborn again.

(4) Creo allows the user to rearrange the generation order of each of the established features, thus increasing the flexibility of the design. In the feature ranking, attention should be paid to the parent-child relationship between features. The parent feature cannot be moved to after the child feature, and again, the child feature cannot be moved to before the parent feature.

(5) Creo allows the user to implicitly or delete the resulting features. Implied features can be recovered by the reply command, while deleted features are not recoverable. In addition, if the implied or deleted feature has subfeatures, the corresponding subfeatures are also implied or deleted after the implied or deleted feature. If you do not want to imply or delete subfeatures, you can use them by.

3.2 Rhino's method for model modification

In contrast, Rhino does not have the parameter recording function, and once it wants to be modified, it needs to be rebuilt. Although it can save one backup for each period built, and it can be extracted in the backup file when it needs to be returned, the process is too complicated. Rhino is not completely without modification function, and individual commands have modification function themselves. We will focus on two common modification commands here:

(1) Record the build history command (Record History)

As the name says, the history of construction. Record the link between the input object and the result object, and when the input object changes, the result object also changes. There is a logic of what the original base looks like and what it looks like later. To be even more common, they are a father-son relationship, where the father determines the attributes of the son, and the son has no way to affect the attributes of the father. Commands with this property in Rhino are arrays, replication, mirroring, curve flow, surface flow, projection, etc.

The command operates as follows: first, on the status column, click "Record build history". Then execute instructions that support construction history (for example, Loft) and select some curves to build surfaces. Finally, you edit the input curve, and the surface updates with it. The record construction history panel on the status column shows whether the state of the construction history is currently recorded. When the text on the panel is in bold, the record construction history is enabled and the fine type is deactivated. The following is an example of a vase shape modification.

Step 1: Create a contour curve, and a rotation axis.

Step 2: When you open the record and update the construction history, select the desired rotation curve, and click the record history option at the bottom of the rhino work window. As shown in Figure 1.

Figure 1 The initial curve and the model of the vase

Step 3: Use Revolve to complete the desired surface, and then adjust the parent curve of the generated surface, and the finished subsurface will also change accordingly. Figure 2 is the two updated models obtained by modifying the contour curve on the basis of Figure 1.

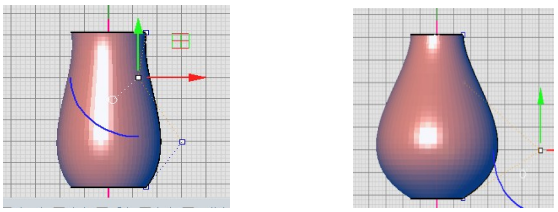

Figure 2 Update model with a modified initial curve

（2）Cancel the construction order

Rhino provides special commands for individual modeling functions to cancel the modeling function, including the following four commands, to make a brief introduction.

First, untrim the surface, whose command actions and typical objects are shown in Figure 3.

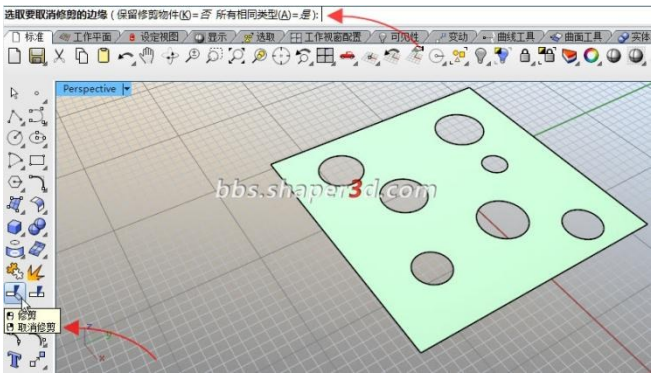

Figure 3 Cancel trim command execution interface

Second, untrim the holes of the entity, whose command operations and typical objects are similar to Figure 3.

Third, untrim all edges, outer edges, and hole features.

Fourth, other untrim tools, such as removing surface edges.

4 Division of labor between Creo and Rhnio in product development

4.1 Rhino: a styling tool in the industrial design field

In the industrial design process, once the designer has completed the idea of two-dimensional sketch, he can use Rhino modeling software to carry out computer-aided design of three-dimensional model. For designers with aesthetic literacy, Rhino provides a free stage for creativity, enabling them to construct points, lines, surfaces and volume elements to fully demonstrate their artistic talent. With Rhino's efficiency, designers can quickly produce multiple creative models in a short period of time.

In addition, with the help of high-quality rendering plug-ins such as Vray and Keyshot, the three-dimensional images generated by Rhino have almost reached the realistic effect of high-end renderers. This means that in the creative design stage, designers can not only quickly model, but also visualize the design scheme immediately, thus greatly broadening the boundaries of design

thinking.

4.2 Creo: an engineering solution for structural design

Creo is an engineering software focused on production and manufacturing. With the parametric design, model tree management and a single database technology, it effectively solves the problem of feature correlation, and makes the creation, editing and modification process of the design model become extremely convenient. However, the high standardization and rigor of Creo in the modeling process also reduces the speed of model creation to some extent.

Given these characteristics, Creo is more often applied to the refinement design stage after the 3 D creative model is determined. Specifically, when the designer uses Rhino to visualize the model, he can turn to Creo to create a post-finalized model with detailed size data. At this stage, the designer will fully consider the processing and production of the product, and conduct in-depth design refinement of the model to ensure that it can meet the requirements of the later structural design.

4.3 Division of labor and development of Creo and Rhino

Rhino is widely believed to be suitable for the creative expansion stage of 3 D conceptual design, while Creo specializes in 3 D model formulation and deepening design. More than 70% of the design agencies choose Rhino to quickly build complex surface models to show creativity. Relatively speaking, Creo, as an engineering software, is more suitable for the structural optimization of product design to ensure that products can be mass produced, including the rationality of mold and process.

With the improvement of ease of Creo ease of use, industrial designers start to use the software to directly express design, can communicate more accurately with structural designers, and speed up the realization of design concept and design development process.

5 Summary

For individual industrial product modeling, Rhino is better than Creo, such as mobile phones, tableware, home appliances, etc. Many enterprises or design companies often use Rhino when designing the appearance of products, and make full use of its rich auxiliary tools, truthfully rendering, layer control, display status of objects, etc.

For complex models of multi-part combinations, such as cars, engines, robots, etc., Creo has strong advantages and can simulate the assembly of solid parts, which is very suitable for the model of mechanical products. Creo adopts modular design, covering multiple independent functions such as sketch drawing, component creation, assembly design, sheet metal design and processing, and realizing the comprehensive coverage of the development process.

By comparing the advantages and disadvantages of Creo and Rhnio, we can learn from each other in the process of modeling, and more modeling skills and difficult points can be solved easily.

Funding

the second batch of national Vocational Education Teacher Teaching Innovation Team Project of the Ministry of Education (ZH2021020101); Research and Practice Project of Teaching Reform of Higher Vocational Education in Guangdong Province (2023 JG 477); Special Key Research Project of Guangdong Vocational College of Industry and Trade (2022-jd-05)

About the Author

Hansheng Yan (1978- -), male, Han Nationality, professional leader / associate professor, Address: Room 605, No. 1, Huaqiao Middle Road, Xinshi Street, Baiyun District, Guangzhou City, Guangdong Province.

References

[1] Han-sheng Yan, Juan Zhao, Huimin Yi, et al. Research on Unity3D Virtual Laboratory for Product Structure Cognition [C].2023.

[2] Wang Wangchun. Structural optimization design of nut drying cyclone filter based on CFD [D]. Shaanxi Province: Chang'an University, 2022.

[3] Zhang Tiecheng. Comparison of Creo free-form surface with T-Splines function [J]. Science and Technology Information, 2013(22): 122-123.

[4] Zhu Rong. Thinking on the teaching of 3 D software courses of Animation major[J]. Science Fiction Illustrated, 2023 (6): 214-215.

[5] Liu Weizhi, Fang Jiayi. Research on the teaching reform of 3 D software courses of animation major in Higher vocational colleges [J]. Toy World, 2024(6): 188-190.