Method and Implementation for Generating Profile Sets on CATIA V6

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Abstract. CATIA V6 has found extensive applications in the water conservancy and hydropower industry. However, when it comes to generating sectional views, CATIA V6 still faces some challenges. For instance, the generation of sectional views relies on the selection of projection profiles, which are limited to the XY, XZ, and YZ planes by default. This restricts the ability to create custom profile sets based on specific rules. To address the issue of generating profile sets, this paper proposes a method based on CATIA V6's custom features and the CAT3DBagRep interface. This method enables the rapid generation of a series of profiles based on specified rules, filling the gap left by CATIA V6's inability to generate profile sets. This approach facilitates the batch generation of sectional views with irregular projection profiles.

Keywords: CATIA V6; Profile Sets; CAA; custom features; CAT3DBagRep

1 Introduction

In the rapidly evolving digital technology era, the field of engineering design is undergoing profound changes. CATIA V6, as an outstanding product from *Dassault Systemes*, has gained widespread recognition across various industries such as aerospace and automobile manufacturing. However, despite its powerful functionalities, CATIA V6 still faces limitations in specific tasks, particularly in generating sectional views. Specifically, the generation of sectional views relies on selecting projection profiles which are limited to the XY, XZ, and YZ planes by default, without the ability to create custom profile sets based on specific rules. This paper delves into the issue of profile set generation and aims to make significant improvements.

With its excellent performance and design capabilities, CATIA V6 provides invaluable support to engineers and designers.In recent years, scholars both domestically and internationally have conducted a series of studies on CATIA V6. In 2013, *Ye, Z.H. et al.* [1] used CATIA V6 to establish an improved model of gymnastics equipment based on virtual human body data, exploring the best evaluation method for human-machine simulation in gymnastics equipment design. The results showed that CATIA

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V6 significantly shortened the product development cycle, enabling seamless integration from 3D design to manufacturing. Its efficient and precise design simulation capabilities brought revolutionary breakthroughs to the development of gymnastics equipment.

In 2018, *Tang, X.Y. et al.* [2] studied the application of BIM technology in the design of long-span arch bridges using the Guangxi Pingnan Bridge as an example. The results indicated that CATIA V6 not only significantly improved design efficiency but also made the design more scientific and accurate. Additionally, it could simulate the construction process, conduct collision interference and error analysis, providing strong technical support for arch bridge design.

In 2019, *Xu, S.H. et al.* [3] discovered the unique advantages of CATIA V6 in generating finite element meshes for hull structures through its in-depth application. The software could quickly generate high-quality finite element meshes, effectively reducing the work time for mesh generation. By reusing models through associative design, it significantly improved the efficiency of finite element generation, providing strong support for CAD/CAE integration.

In 2022, *Wu, P.P. et al.* [4] adopted a method of defining vector objects using normalized data formats, expanding CATIA V6's vector filling styles, and achieving high-quality and rapid filling of engineering drawings based on CATIA V6. This method significantly improved the quality of engineering drawing filling styles, enhanced drawing efficiency, and further met the filling requirements for engineering design in 2D drawings.

In the same year, *Xue, X.N.* [5] used CATIA 3D design software to conduct research on outfitting equipment engineering template design, assembly design, parametric design, etc. It was found that CATIA software could achieve systematic and visual design of the hull, thereby improving designers' efficiency and ensuring the quality of design drawings.

In summary, CATIA V6 has achieved remarkable results in multiple industry sectors. However, with the continuous progress of technology and changes in market demand, the native functions of CATIA V6 still have certain limitations in some aspects. To address this, *Dassault Systemes* provides the CAA (*Component Application Architecture*) component library, a powerful custom development tool that includes a comprehensive application programming interface such as mathematical libraries, geometric libraries, and underlying frameworks. This provides users with a broad space for customized secondary development. Numerous scholars have conducted countless research studies on this topic.

In 2003, *Li, Z.S. et al.* [6] conducted a detailed analysis of CATIA's interface technology to deepen the secondary development and application of CATIA software, and systematically expounded on its development steps, providing valuable guidance for subsequent developers.

In 2008, *Zhou, X.E. et al.* [7] focused on CAA, the main development platform of CATIA, and deeply explored its composition, architecture, principles, and functions. They proposed a basic method for CATIA secondary development based on CAA, making the secondary development process of CATIA more efficient and flexible, greatly improving design efficiency and software application capabilities.

In 2012, *Liang, D.C. et al.* [8] further introduced CATIA's CAA secondary development and its development environment RADE. They successfully used secondary development technology to traverse the product structure tree and effectively extract the parent-child assembly relationship and attribute information of components. This achievement not only demonstrates the advantages of CAA in improving CATIA's customization and scalability but also significantly enhances the flexibility and efficiency of the design process, providing strong technical support for complex engineering design.

In 2018, *Sui, G.D. et al.* [9] proposed a parametric modeling method based on CATIA secondary development technology to address the complexity and time-consuming issues of 3D modeling of earth-rock dams. This method achieves the association of feature parameters with 3D entities by calling CATIA's API, enabling fast and efficient 3D modeling of earth-rock dams. Compared to traditional methods, this solution significantly improves modeling efficiency and saves a lot of time for subsequent simulation calculations.

In 2022, *Cheng, Z. et al.* [10] proposed a multi-mode structural drawing method based on the CATIA V6 platform. This method combines CAA and EKL secondary development technology to develop an efficient drawing program that utilizes 3D structural models to achieve semi-automated drawing, greatly improving drawing efficiency.

In the same year, *Kang, H. et al.* [11] conducted secondary development based on CATIA/CAA technology to solve the problem of rapid modeling of O-ring seals. They customized menus and toolbars using the CAA interface, completed interface development using Visual Studio, and adopted a hybrid drive method to achieve parameter-driven functionality. This greatly simplifies the tedious steps of changing dimensions in CATIA, making the O-ring seal modeling process more convenient and efficient.

In 2023, *Mo, J. et al.* [12] developed a tool for rapid generation of engineering drawings based on template modularization and the Component Application Architecture (CAA), combined with Visual Studio. This tool can quickly generate frame, title bar, detailed tables, and change marks. Engineering example tests have proven that the tool can quickly generate standard-compliant drawings. Additionally, the use of CAA simplifies the development process, improves the flexibility and customizability of the tool, and can significantly improve the efficiency of drawing generation.

In conclusion, the application of CATIA V6 in the field of engineering design is quite extensive. However, its limitations cannot be ignored, and it is difficult to fully meet the diversified needs of designers. CAA, as the secondary development platform of CATIA V6, undoubtedly opens a new door for designers. It provides rich development tools and interfaces, enabling designers to carry out personalized customized development according to their own work characteristics and needs. In recent years, with the continuous development and innovation of technology, more and more enterprises and individuals have begun to realize the enormous potential of CAA secondary development. They utilize the CAA platform to conduct in-depth customization and optimization for certain functional shortcomings or specific needs of CATIA V6, thereby greatly improving the efficiency and flexibility of CATIA V6.

This article addresses the shortcomings of CATIA V6 in profile generation by utilizing the CAA interface for in-depth secondary development. We propose an innovative solution that customizes profile features through the OSM file and uses the *CAT3DBagRep* interface to draw profiles. Our choice to use OSM files for defining profiles is based on the flexibility of OSM files in data storage and feature representation. Additionally, OSM files enable convenient description and modification of the geometric attributes and characteristics of profiles. Secondly, we opted for the *CAT3DBagRep* interface to render the profiles. This selection was made due to its powerful 3D graphics rendering capabilities and seamless integration with CATIA V6. Through the *CAT3DBagRep* interface, we can efficiently generate profiles that conform to specified rules while ensuring compatibility with the CATIA V6 system.This method can quickly generate a series of profiles that meet specified rules, effectively filling the gap in CATIA V6's ability to generate profile sets. We hope this research can provide strong support for batch generation of irregular projection profiles and further promote technological progress and innovation in the field of engineering design, injecting new vitality into the sustainable development of the industry.

2 Overall Design Plan

In this paper, by customizing profile features in the OSM file and utilizing the CAT3DBagRep interface to draw profiles, a series of profiles following specified rules can be rapidly generated. The specific technical route is illustrated in ***Figure 1***.

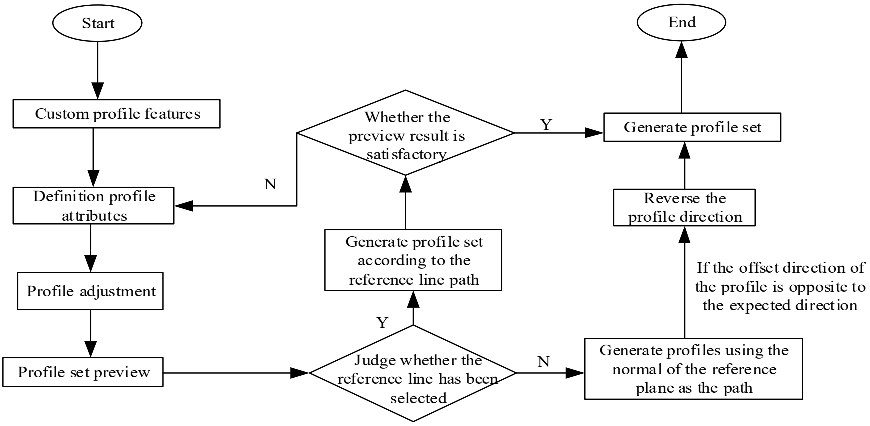
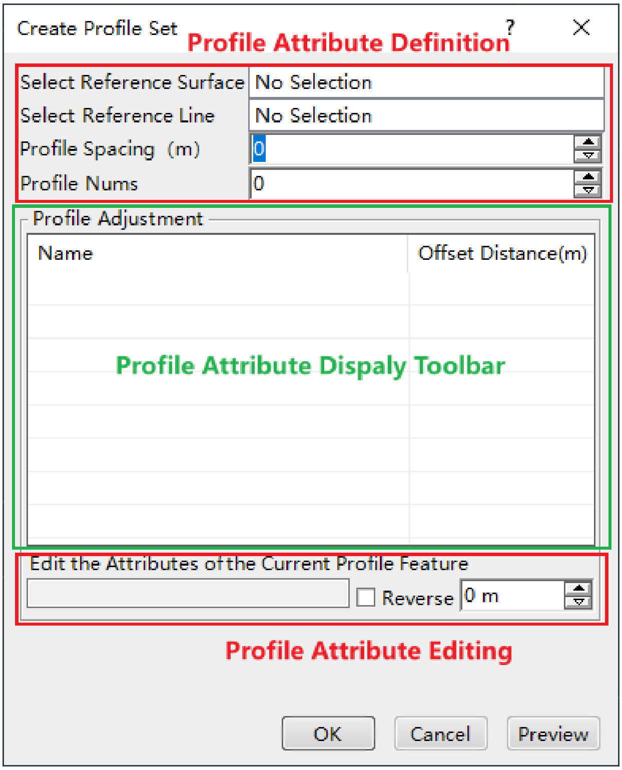


Fig. 1. Batch Generation of Profile Set

Firstly, developers define the profile features in the OSM file; subsequently, users activate the profile set generation function in the 3D view and define the attributes of the profiles to be generated based on actual needs, including selecting a reference plane, reference line, and defining profile spacing and nums. These attribute information will be instantly displayed in the toolbar. The functional interface is shown in ***Figure 2***.



Select Reference Surface: No Selection
Select Reference Line: No Selection
Profile Spacing (m): 0
Profile Nums: 0
☐ Reverse 0 m

Fig. 2. Functional Interface Diagram

If users need to adjust the offset distance of individual profiles, they can modify it directly in the attribute editing box, and then click "*Preview*". The program will execute different operational logics based on whether the user has selected a reference line:

If a reference line is selected, the program will generate a set of profiles along the path of the reference line. If the preview results meet expectations, after the user clicks "*OK*", the program will automatically call the *CAT3DBagRep* interface provided by CAA to draw each profile individually, generating a profile set. If the preview results are unsatisfactory, the user can redefine the profile attributes and re-execute the profile set generation function until satisfaction.

If no reference line is selected when defining the profile attributes, the program will use the normal of the reference plane as the path to generate the profile set. During this process, if the offset direction is found to be opposite to the expected direction, the user can adjust the profile offset direction by checking the "*Reverse*" box in the profile attribute editing bar. After adjustment, clicking "*OK*" will prompt the program to generate the desired profile set.

3 Key Methods

The key technical methods proposed in this paper mainly cover the following aspects:

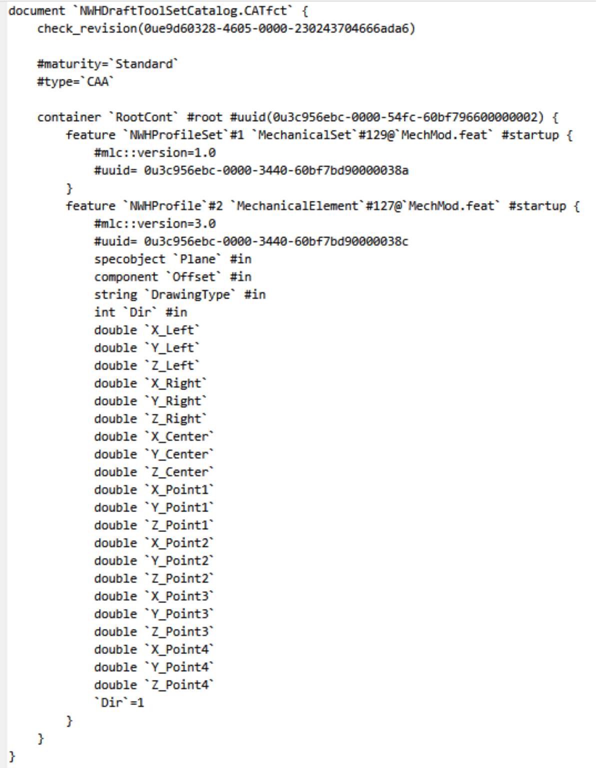
- Defining custom profile features in the OSM file;
- Generating a series of auxiliary points on the reference line based on the defined offset distance;
- Creating a sequence of auxiliary planes along the reference line using the coordinates of the auxiliary points;
- Projecting the eight corner points of the bounding box of the geometric figure in the view onto the auxiliary planes, and selecting the four corner points that form the largest area profile;
- Finally, utilizing the *CAT3DBagRep* interface provided by CAA, drawing each profile based on the coordinates of the four corner points obtained in the previous step to generate a profile set.

The following sections elaborate on the above key techniques in detail.

3.1 Custom Profile Features

Firstly, enter the runtime prompt control window of the CAA development environment and execute the command "*CATfctEditorAssistant -create -new -catalog -name NWHDraftToolSetCatalog.CATfct -with -client -id NWHDraftToolSetCatalogID*" to generate a CATfct file named *NWHDraftToolSetCatalog*. Then, use the command "*CATfctEditorAssistant -describe -as -osm -catalog -name NWHDraftToolSetCatalog.CATfct -with -client -id NWHDraftToolSetCatalogID -as NWHDraftToolSetCatalog.osm*" to generate an osm file named *NWHDraftToolSetCatalog* based on the CATfct file. Subsequently, define the feature attributes of the profile set (*NWHProfileSet*) and profile (*NWHProfile*) in the *NWHDraftToolSetCatalog.osm* file, with specific details shown in ***Figure 3***. Finally, execute the command "*CATfctEditorAssistant -update -catalog -name NWHDraftToolSetCatalog.CATfct -with -client -id NWHDraftToolSetCatalogID -with -osm NWHDraftToolSetCatalog.osm*" to update the content of *NWHDraftToolSetCatalog.CATfct* file through the osm file, thus completing the customization of profile and profile set features.

To facilitate the subsequent creation of profiles and profile sets, it is necessary to define implementation classes *NWHEProfile* and *NWHEProfileSet* based on the attribute information in *NWHDraftToolSetCatalog.osm*, respectively. Define set and get methods for each attribute, referring to the relevant content of custom features in the CAA encyclopedia for the specific method writing format. Later, the attribute information of profiles and profile sets can be stored by instantiating objects of *NWHEProfile* and *NWHEProfileSet* classes. Additionally, it is required to define a tool class *NWHEDraftToolFactory*, in which methods for creating profiles and profile sets are implemented. These methods can be called later to update profiles and profile sets to the CATIA V6 structure tree.



```
document `NWHDraftToolSetCatalog.CATfct` {
    check_revision(0ue9d60328-4605-0000-230243704666ada6)

    #maturity=`Standard`
    #type=`CAA`

    container `RootCont` #root #uuid(0u3c956ebc-0000-54fc-60bf796600000002) {
        feature `NWHProfileSet`#1 `MechanicalSet`#129@`MechMod.feat` #startup {
            #mlc::version=1.0
            #uuid= 0u3c956ebc-0000-3440-60bf7bd90000038a
        }
        feature `NWHProfile`#2 `MechanicalElement`#127@`MechMod.feat` #startup {
            #mlc::version=3.0
            #uuid= 0u3c956ebc-0000-3440-60bf7bd90000038c
            specobject `Plane` #in
            component `Offset` #in
            string `DrawingType` #in
            int `Dir` #in
            double `X_Left`
            double `Y_Left`
            double `Z_Left`
            double `X_Right`
            double `Y_Right`
            double `Z_Right`
            double `X_Center`
            double `Y_Center`
            double `Z_Center`
            double `X_Point1`
            double `Y_Point1`
            double `Z_Point1`
            double `X_Point2`
            double `Y_Point2`
            double `Z_Point2`
            double `X_Point3`
            double `Y_Point3`
            double `Z_Point3`
            double `X_Point4`
            double `Y_Point4`
            double `Z_Point4`
            `Dir`=1
        }
    }
}
```

Fig. 3. Definition of feature attributes in NWHDraftToolSetCatalog.osm file

3.2 Generation of Auxiliary Points

Based on the offset distance defined by the user and the selected reference plane, a list of auxiliary points is created along the reference line. Initially, the *GetPoints* method provided by CAA is utilized to obtain the endpoints of the selected reference line. Subsequently, by analyzing the distances of these two endpoints relative to the reference plane, the offset direction of the profile is determined. Specifically, the endpoint closer to the reference plane is set as the starting point, while the endpoint further away is designated as the end point, thereby establishing the offset direction of the profile.

Next, the *CreatePoint* method is invoked, using the reference line as the path, to generate a series of auxiliary points with varying offset distances along it. Finally, the auxiliary point objects are stored in a list for future use.

3.3 Generation of Auxiliary Planes

Based on the auxiliary point list generated in the previous step and the selected reference line, a series of auxiliary planes required for generating the profile are created. If the number of auxiliary points generated in the previous step is not zero, the *CreatePlaneNormal* method provided by CAA is utilized. The tangent line of the reference line at the auxiliary point locations is used as the normal of the plane, thus generating a series of auxiliary planes corresponding to the auxiliary point list.

If the user does not select a reference line when defining profile properties, the number of auxiliary points generated in the previous step will be zero. In this case, the *CreatePlane* method can be used to offset the reference plane along its normal direction by a specified distance, thereby generating the required auxiliary plane. The offset direction of the reference plane can be adjusted by checking the "*Reverse*" box in the property editing bar to meet different user needs.

3.4 Obtaining Profile Corners

To accurately generate the maximum area profile, it is first necessary to extract the spatial coordinates of the eight points of the geometric bounding box from the three-dimensional view in CATIA V6. These coordinate points are then projected onto the previously created auxiliary planes. During the projection process, duplicate projection point coordinates must be eliminated to ensure data accuracy.

Next, the positional relationships of the projection points on the auxiliary planes are analyzed, while removing any stagnation points that could potentially cause three points to be collinear. Finally, four projection points that are relatively far from the auxiliary points are identified. These four points are then sorted based on the distances between them, ensuring that they form a relatively regular rectangle when connected in sequence.

These four projection points constitute the four corners that enclose the maximum area profile, and they are stored in the attribute information of the profile object generated by the instantiation of the predefined *NWHEProfile* class for profile features.

3.5 Drawing the Profile

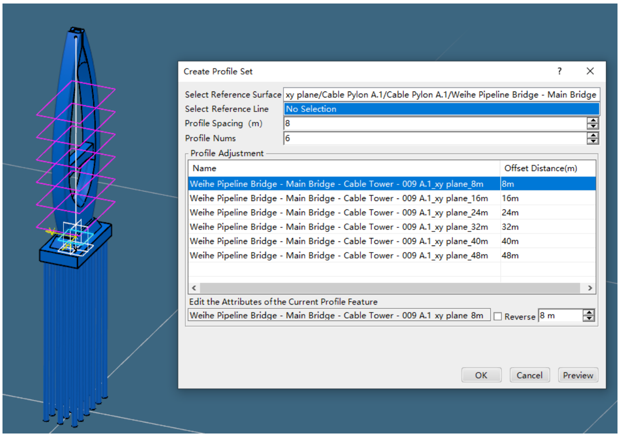
After storing the sorted four corners in the attribute information of the profile object, an event notification is sent to the *CATExtIVisu* interface through the *CATIModelEvents* interface provided by CAA to initiate the drawing of the profile shape. Upon receiving the event notification for drawing, the CAT3DBagRep provided by CAA is first called to initialize the view collection. Then, *CAT3DlineGP* is invoked to draw three-dimensional lines based on the coordinates of the four corners stored in the profile object's attribute information. Meanwhile, the *CATGraphicAttributeSet* interface is utilized to set a vivid color for the drawn graphics, ensuring their visibility in the three-dimensional view.

Subsequently, the *AddGP* method of the *CAT3DCustomRep* interface is called to add the drawn lines and these color to the graphics collection. Finally, the *AddChild* method of the *CAT3DBagRep* interface is used to add the configured graphics collection to the view collection. This completes the profile drawing task and updates its display in the three-dimensional view. The generated profile objects are then stored individually in the profile set object created by instantiating the *NWHEProfileSet* class for profile set features. Once all profiles are drawn and stored in the profile set object, the generation of the profile set is completed and updated in the structure tree for easy viewing and management by users.

Additionally, a preview function is provided for users to visually inspect the effect of the generated profile set in the three-dimensional view. If users are not satisfied with the generated profile set effect, they can redefine the profile properties and re-generate the profile set. Finally, by simply clicking the "*OK*" button, the generation of the profile set is completed.

4 Application and Conclusion

To validate the practicality and efficiency of the profile set generation function developed in this study, we conducted tests on two specific models: a cable pylon model and a test model of a certain irregular curved path. Within the CATIA V6 environment, we applied the methodology presented in this paper to both models and successfully generated profile sets, as illustrated in ***Figure 4*** and ***Figure 5***.



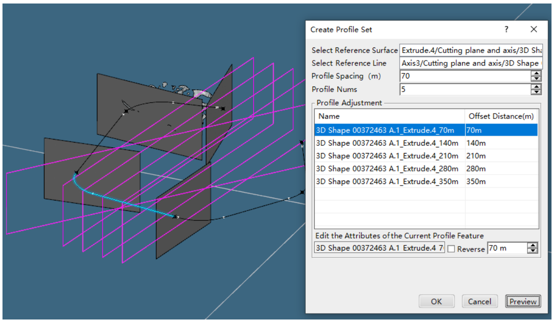
Select Reference Surface: xy plane/Cable Pylon A.1/Cable Pylon A.1/Weihe Pipeline Bridge - Main Bridge
Select Reference Line: No Selection
Profile Spacing (m): 8
Profile Nums: 6

Profile Adjustment

Name	Offset Distance(m)
Weihe Pipeline Bridge - Main Bridge - Cable Tower - 009 A.1_xy plane_8m	8m
Weihe Pipeline Bridge - Main Bridge - Cable Tower - 009 A.1_xy plane_16m	16m
Weihe Pipeline Bridge - Main Bridge - Cable Tower - 009 A.1_xy plane_24m	24m
Weihe Pipeline Bridge - Main Bridge - Cable Tower - 009 A.1_xy plane_32m	32m
Weihe Pipeline Bridge - Main Bridge - Cable Tower - 009 A.1_xy plane_40m	40m
Weihe Pipeline Bridge - Main Bridge - Cable Tower - 009 A.1_xy plane_48m	48m

Edit the Attributes of the Current Profile Feature
Weihe Pipeline Bridge - Main Bridge - Cable Tower - 009 A.1 xy plane 8m ☐ Reverse 8 m

Fig. 4. A certain cable pylon model



Select Reference Surface: Extrude.4/Cutting plane and axis/3D Sha
Select Reference Line: Axis3/Cutting plane and axis/3D Shape
Profile Spacing (m): 70
Profile Nums: 5

Profile Adjustment

Name	Offset Distance(m)
3D Shape 00372463 A.1_Extrude.4_70m	70m
3D Shape 00372463 A.1_Extrude.4_140m	140m
3D Shape 00372463 A.1_Extrude.4_210m	210m
3D Shape 00372463 A.1_Extrude.4_280m	280m
3D Shape 00372463 A.1_Extrude.4_350m	350m

Edit the Attributes of the Current Profile Feature
3D Shape 00372463 A.1 Extrude.4 7 ☐ Reverse 70 m

Fig. 5. A test model of a certain irregular curved path

The results demonstrate that our profile set generation function not only rapidly produces a series of profiles but also accurately generates profile sets along user-specified irregular curved paths.

The native section view functionality in CATIA V6 has limitations, particularly in that it relies on predefined projection profiles (defaulted to XY, XZ, YZ planes) for section view generation, lacking flexibility to produce profile sets based on specific user requirements. However, the implementation of our profile set generation function significantly enhances CATIA V6's capabilities in this regard, overcoming the restriction of its native function, which cannot generate a series of section views according to specific rules. Additionally, feedback from designers indicates that our method enables them to effortlessly create profile sets along designated paths, facilitating the bulk generation of regular section views and significantly improving work efficiency and design quality.

Furthermore, we have investigated potential limitations or issues that may arise during the implementation of our method. For instance, when dealing with particularly complex models or large-scale data, performance bottlenecks or computational delays may occur. To address these challenges, we suggest enhancing performance through algorithm improvements, leveraging more efficient computing resources, or adopting distributed computing techniques.

In conclusion, the profile set generation function developed in this paper exhibits significant advantages in terms of practicality, efficiency, and flexibility, holding promise for substantial progress in the field of engineering design.

References

1. Ye, Z.H., Li, X., Li,Y. (2013) The Virtual Prototyping Design and Evaluation of Ergonomic Gymnastic Based on CATIA. International Journal of Hybrid Information Technology, 6(5):67-78.
2. Tang, X.Y., Li, S.G., Lin,G.T. (2018) The application of CATIA V6 in the design of long span concrete-filled steel tube arch bridge. Journal of Guangxi University of Science and Technology.
3. Xu, S.H., Du, W.L., Peng, Y.K., et al. (2019) Method for Generating Ship Structure Finite Element Mesh Based on CATIA V6. Ship Engineering.
4. Wu, P.P., Xing, J.X., Han, Y.N., et al. (2022) Study on Vector Filling Technology of Engineering Drawing based on CATIA V6. Northwest Hydropower, 06:133-137.
5. Xue, X.N. (2022) 3D Design Optimization of Outfitting Based on CATIA V6. Marine Equipment/Materials & Marketing, 30:18-20.
6. Li, Z.S., Zhu, Y., Xiang, Z.F. (2003) Secondary Development Technology Based on CATIA Software. Journal of Xihua University(Natural Science Edition), 01:16-18.
7. Zhou, X.E., Lu, M.W., Zhao H.X. (2008) Research on Secondary Development and Application of CATIA Software Based on CAA. Science & Technology Information, 36:73-74+317.
8. Liang, D.C., Zhang, W.M., Sui, L.J. (2012) Brief Analysis on Secondary Development of CATIA Based on CAA. Aeronautical Manufacturing Technology, 10:65-68.

9. Sui, G.D., Zhang X.X., Dong, F.P. (2018) 3D parametric modeling method for earth-rock dams based on CATIA secondary development. Design of Hydroelectric Power Station, 34:5-8.
10. Cheng, Z., Qiu, Y., Zhou, T., et al. (2022) Multi-mode structural drawing method and secondary development based on CATIA V6. Ship & Ocean Engineering, 51:20-25.
11. Kang, H., Yuan, R.B., Fan J.S., et al. (2022) Secondary Development of O-ring Based on CATIA/CAA. Mechanical Management and Development, 37:21-23.
12. Mo, J., Zhang, S.L., Wang Y.M., et al. (2023) Development of Customization for Drawing Frame,Title Bar and Parts Lists Based on CATIA/CAA. Journal of Hubei University of Automotive Technology, 37:56-60.