

CATIA V5 Robust Design Method to Prevent Feature Failure

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Abstract. Cross-modify are the main point of product designing. CATIA V5 software is widely used in automobile, aircraft and other fields. During using CATIA V5, it is a fatal problem for every designer to how to improve the efficiency of modification in the product designing and avoid feature failure. After long-term practice, robust design method is introduced in this paper to solve this problem. It is easy to modify the feature and remain association. It shortens the period of subsequent improvements effectively and provides the assurance for product updating.

Introduction

CATIA V5 has strong capability of feature association and modification[1,2]. Compared with traditional designing, time saving for developing Boeing 777 with CATIA was not significant. But the time for developing subsequent types saved at least more than 50% [3].

Modification is an important point during the designing[4]. Cross modification makes the designing more sound and optimized. Updating is the life of production[5,6]. Continuously modifications improve the capability of product and decrease the cost which makes the product more competitive[7]. At present, much modification runs through the designing process for most of the products. It will take more time to summarize the opinion from different people and view, which causes that the time of modification is much more than that of designing. The initial unbending designing leads to the impossibility to subsequent modify [8,9]. Constraint from irrespective factor may hide some idle association which leads the feature failure during modification. Looking for and Sum up an easy product modeling design method of subsequent modifications, and flexibly used in the design process, it is the effective guarantee of prevent feature failure. In this paper, the method which has steady mode and is easy to modify the subsequent product is called “robust designing method” [10,11,12].

From long-term practice, it draws the conclusion that unclear order is the main reason of feature failure. In this paper, “robust designing method” is introduced to decrease the association and make the designing steady and modification easy.

The paper is organized as follows. In the next section, and some measure criterions are given to prevent feature failure. In Section 3, how to define the coordinate system is presented. Section 4 presents the design factors to ensure the quality of sketch. In Section 5, how to form Geometric feature is given. Section 6 proposes two methods to assembly geometry of product. In Section 7, the steps of modification and adjustment during position assembly among external parts or internal parts are present. Finally, we conclude our paper in section 8.

Measure to prevent feature failure

The feature failure is that the factor loses the basic point of association or the rebuilt failure caused by association going beyond the definition. When rebuilding the model, the 3-D system identifies, analyzes and calculates the features according to the creating order and then displays the model. As soon as error is found, the system will display the relative error and stop rebuilding. Model cannot be displayed and used until correct the error.

Regarding to the analysis mentioned above and combining the experience, measure to prevent

feature failure is as the following:

- (1) Reduce the association with outer factor as best as possible and only associate with the essential and issued outer factor.
- (2) Part geometry is consisted of some geometry instead of geometric feature.
- (3) Constrain feature on the reference system, point, line and plane built by ourselves, not the system reference. As the basement of constraint can be pad and rotate, it is possible to modify the feature.
- (4) Limit the geometrical feature by “up to plane” or “up to surface” rather than “dimension”.
- (5) Deleting the solids which form the part is not permitted but deleting the geometry is Ok.
- (6) It isn't permitted to create the feature which hasn't association with existing feature and close the association otherwise modification can't be carried out.
- (7) Issue the available feature. The feature issued can only be modified but not be deleted otherwise the feature associating with it will be failed.
- (8) It is necessary to have a specific, easy discriminating name or number for every part, draft, frame and reference.

Coordinate system

The coordinate system (Axis system) is consisted of three planes, three axis and one based point which is a perfect datum. System provides this kind of datum to us. But as it is not easy to be modified, it is the best way to set new coordinate.

Under the product and part layer, new a file named as “datum file”. Set coordinate system in this file only to provide point, line and plane datum to the parts. All draft, line frame, geometry and parts associate with the coordinate in order to be modified easily later.

Set datum file of assembly under product and that of part under assembly. Set constraint between assembly and system coordinate, between part and assembly coordinate in order to determine the position and angle of the coordinate system in the space. Issue the axis system and assure the geometrical feature only associate with the part reference, which constrains the position relation among the parts. That is to say that part only constrains with self axis system and axis system only constrains with the upper. The result is that geometry can pan and rotate freely without any spare constraint and the whole assembly will pan and rotate along with coordinate system (See Fig.1).

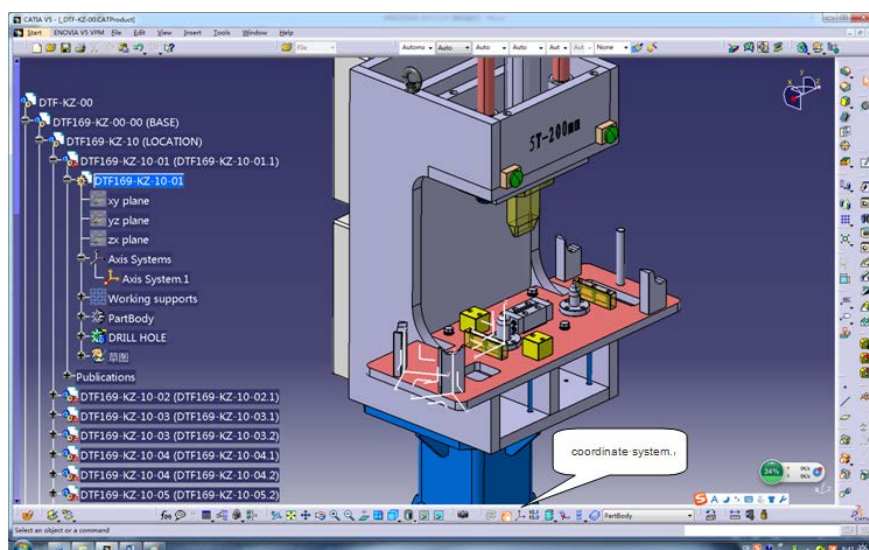


Fig.1: Constraint between part and assembly axis system (Dash means axis system)

Sketch

During the model technology based on feature[4], sketch feature is one of the important parts.

Sketch is the foundation and beginning of model designing. The quality of sketch will affect the quality of 3-D model directly. According to long-term experience, the issue as the following is needed to be pay attention to:

- (1) Sketch is only consisted of some basic geometry without any corner, chamfer and decoration. It is easier to renew a geometry of many simple sketch than that of single complicated sketch.
- (2) Constrain the draft completely rather than less constraint and over constraint.
- (3) Constrain sketch only with self coordinate rather than other factor, otherwise any modification may cause feature failure.
- (4) Sketch under the self geometric atlas in order to check, issue and modify easily.
- (5) The name of sketch should be easy to be check and modify.
- (6) Select the plane newly-built or constrained with the coordinate as the sketch plane.

Form geometric feature

Geometric feature is the basic cell of solid. Based on sketch, solid is formed through adding or cutting material. In order to modify easily and keep association, limit the scope by “plane” or “surface” rather than “dimension”. During modification, geometric feature is renewed automatically along with the plane pan or surface modification (See Fig.2).

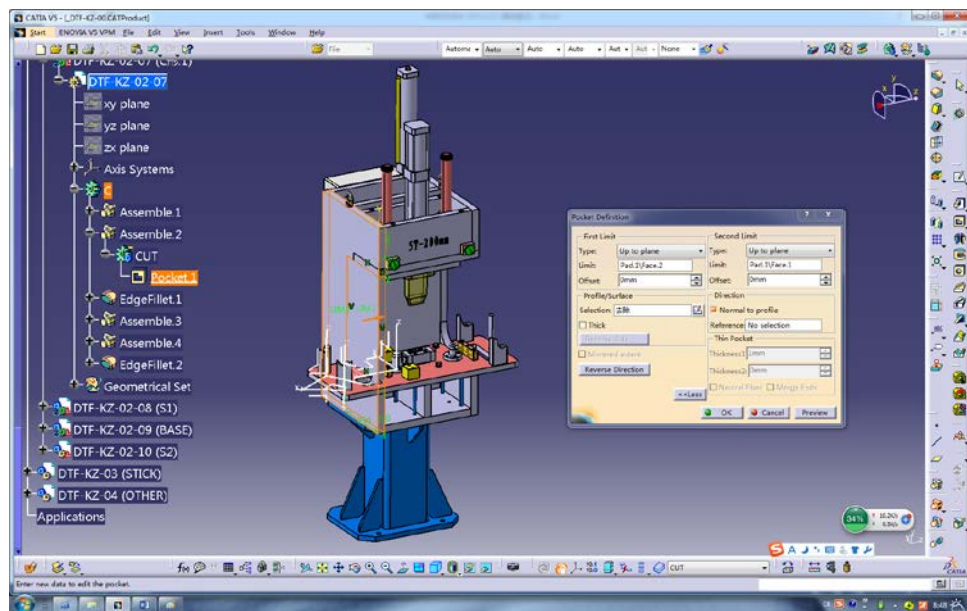


Fig.2: Define constraint

Modeling

If the geometry is consisted of some associated geometric feature, once delete the upper feature, the lower feature will lose association and fail. The only method is to delete all feature or give up deletion and modification. So, geometry consisted of some associated feature is the main potential reason to cause feature failure. (See Fig.3)

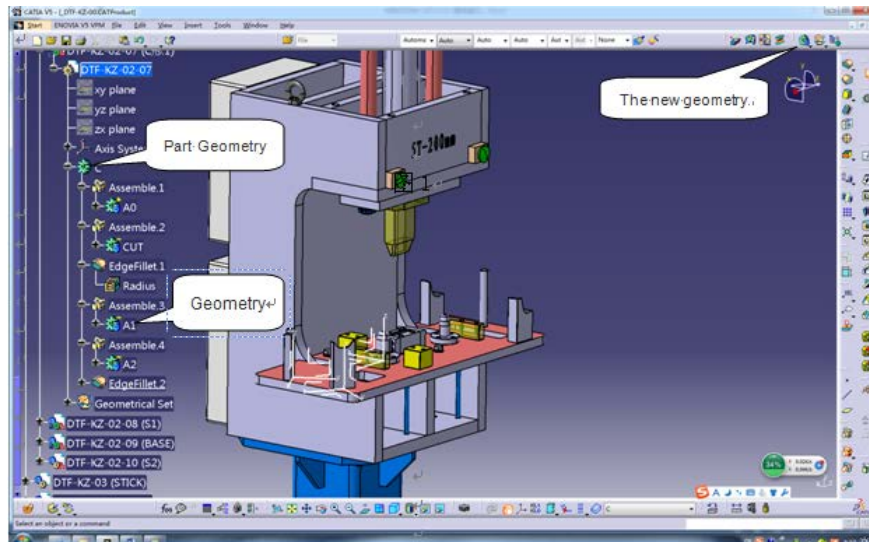


Fig.3: Association among geometries

Part geometry is the default geometric container of software. It is used to define all operation and assembly result. It cannot be involved in Boolean calculation. It's exact structure is geometry of part→geometry→geometric feature.

Part geometry can't be consisted of geometric feature, but of geometry. Assembly some geometry which is called pads or pockets to form solid by Boolean operation. The difference between part geometry and geometry is that assembly the latter to form the former.

There are two methods to assembly geometry. One is to define part geometry, clicking insert, Boolean operate and assembly. The other is to select geometry, click right key, assembly and enter.

So the geometry has respective sketch and feature and no association with others. Deleting and modifying any geometry can't lead to failure of feature renewal. (See Fig.4)

During developing product, part geometry can't be deleted. Because of the association among part geometry, 2-D drawing and CNC program, deleting part geometry will lead to renew feature and reedit CNC program. The only method to resolve this problem is that hides, marks, backups and modifies it in the copy. The 2-D and CNC program will renew automatically along with it.

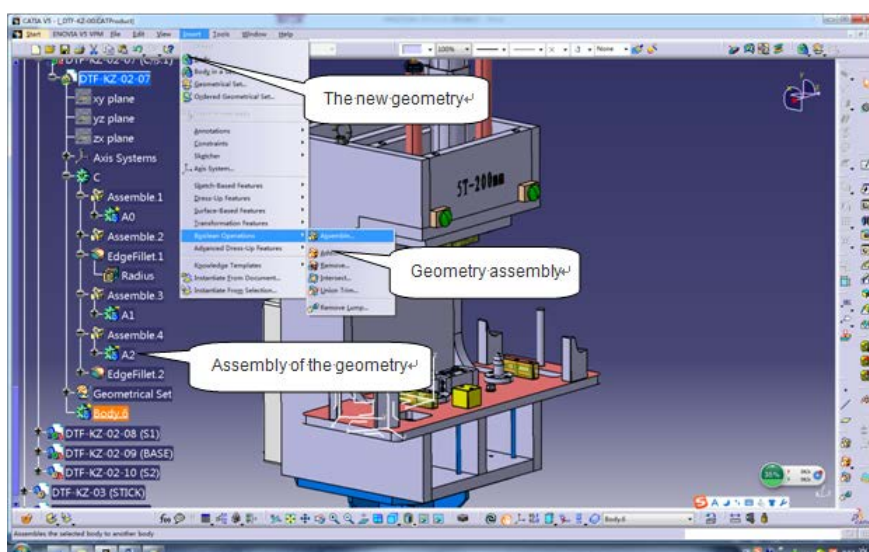


Fig.4: Assembly of geometry

Assemblies

Top down design of new part is done in the absolute coordinate system[13,14,15]. So at the beginning of design, the position relation between new part and other parts is determined and

modification is permitted at any time.

During position assembly among external parts or internal parts, new coordinate system is needed to be set. Combine new part with coordinate system and adjust the position and degree of coordinate system until it is suitable. Modification and adjustment is only to pad the coordinate system rather than constrain the feature of part with other feature.

Conclusions

The design method which form draft by basic geometry graphics, setting single feature and Boolean operation can accelerate calculation and updating, make the order clear, simple the factor of feature and lay a solid foundation for designing and changing of large-type product.

The method summarized above is a kind of designing method to modify the associated designing [16], Modify methods of feature-based design(FBD)[17]. It is easy to be grasped by designer after training. It reached the expectant aim after team popularizing and application and validation in large-scale ally designing. It sped up the change of likewise products, made modification comprehensive and solved the feature failure during modification entirely.

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