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ORIGINAL ARTICLE

A geometric feature-based design system of full parametric association modeling of standard cam for automotive stamping dies

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Abstract Cam mechanism is the most difficult part in the design and manufacture of stamping dies of automobile panels. The same series of cam products need to meet the needs of different working angles. Each specification needs to design and prepare a three-dimensional model, which brings great challenges to the design, editing, and management of the models. Therefore, a new design system of full parametric association modeling based on geometric features for standard cam was proposed, which is seamlessly integrated in NX software platform. The models of different work angles of the same series of cam are integrated into a fully parametric model. The specification and assembly location of standard parts, activation and suppression of detail features will automatically change with work angle based on parametric feature and association constraint technique. By modifying the work angle, the system can automatically instantiate different specification models of cam. Furthermore, the optimized design of cam structural is easier to achieve by motion simulation and finite element analysis based on the parametric model and geometric features. The system can also output the BOM table, when the model is instantiated. The cam of KMACG 600 demonstrates that the newly developed system shows an excellent performance on the model simplification, data management, and optimization design of cam, which can generate high quality design and reduce cost and designing time significantly.

Keywords Standard cam · Parametric association modeling · Geometric feature · Optimized design · Stamping dies · Siemens NX software

1 Introduction

Cams are the most commonly used mechanism in stamping dies, especially in the automotive panel stamping dies. The shape of the automotive panel is very complicated, which is obtained through many stamping operations in different directions. However, the stamping press blocks move along the vertical direction, which cannot meet the other direction of the work requirements. In order to solve this problem, the cam mechanisms are used to convert the vertical movement into the horizontal or inclined movement in the panel stamping dies of automotive panels, as shown in Fig. 1. The manufacturing of cams is very complex and it requires high accuracy. In order to reduce the difficulty of manufacturing the cam stamping dies, the cams are gradually formed a common standard part, which is manufactured by the standard of specialized enterprises, such as MISUMI, SANKYO, and PUNCH. The cam is generally installed in the die structure as a relatively independent mechanism, which is used to finish cam horizontal or inclination piercing, trimming and flanging, and side shaping by moving together with other parts of the die.

Standard parts of cam consist of the main structure (the base, slider, and driver or the holder, slider, and driver) and a series of standard components (like V-type guides, forced return block, nitrogen cylinder). The working angle of the cam is a multiple of 5°, which is generally from 0 to 75° to the horizontal direction, as shown in Fig. 2. Therefore, many different angles and

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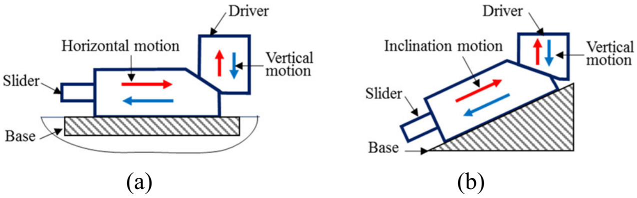


Fig. 1 The composition and movement mode of the cam mechanism. **a** Horizontal cam. **b** Inclination cam

specifications of cam parts will be derived from the same series of cam. The angle serves as the main parameter to drive the cam in the same series. The shape of its main structure varies in different angles, and the key size will not change, neither will the types nor the number of standard parts of the same series. However, the standard specifications will change along with the change of the angle. If every angle cam has a model, it will lead to the data redundancy. Any modification of the model will lead to the change in all the model of angle, which contributes to repeated operations. The correctness and feasibility of the changes are difficult to guarantee, which is not conducive to the management and maintenance of the data. Therefore, obvious problems exist in the management and modeling of cam model, which waste many manpower, material, and financial resources. Parametric modeling enhances the efficiency of management and maintenance of cam data, which can save repetitive work and reduce mistakes, and improve the production efficiency of the standard parts to realize the goals of parametric, intelligent and automatic generation, and the collaboration of design, drawing, manufacturing, and programming. Just need one model to realize the cam with the different angles in the same series, which drive the model structure of cam with different angles, and the automatic generation and update of standard parts through angle parameters.

Parametric and intelligent design of automotive stamping dies and its standard parts are always research focuses on the field of die design, which can improve the designing quality and efficiency effectively. In recent years, many scholars engaged in the research of digitalization die design and proposed many methods which are widely used in the field of designing automation of stamping dies. Tang et al. [1] proposed a new conceptual assembly modeling framework, which employed the zigzag mapping method to implement the function design of the die. The concepts of logic component and logic assembly are presented to satisfy the characteristic of the conceptual die design and represent the abstract information of die assembly. Every logic component can be converted into an instance model in the traditional 3D modeling system. This method can support the conceptual die design and the top-down design process. Lin et al. [2] developed a knowledge-based parametric design system for drawing dies implemented on top of the Pro/E software. But after 1 year, Lin et al. [3] developed another structural design system for 3D drawing dies based on functional features. It achieves automation design by replacing the process data of old die layout with a new one based on the knowledge engineering and fully parameterized template. The results show that these systems can greatly improve the design quality while reducing the development time and cost of drawing dies. Then, Lin et al. [4] continued to study the mold optimization design through fuzzy-based Taguchi method, for the die structure model is parametric design, which can be automatically modified according to the optimization results. Optimization results indicate that the die weight can be greatly decreased. The researches above indicate that the parameterization is crucial for stamping die design. Drawing dies are relatively simple structures in the stamping dies of automotive panels, which are easy to realize automation and parametric design. Azamirad et al. [5] also did the similar research work to optimize the die structure and reduce the volume of die structure. For deep drawing dies of axisymmetric parts, Naranje et al. [6] developed a knowledge-based automated design system on the AutoCAD.

Stamping die structures are generally composed of standard parts and non-standard parts. The above design systems of drawing die mainly focus on the parametric design of non-standard parts. However, the parametric design of 3D standard parts is as important as non-standard parts. Jia et al. [7] developed a design system of punches and dies for progressive die based on functional component, which was integrated with SolidWorks CAD system. The number of punches and dies is very large, especially in blanking die structure, accounting for a large portion in the entire design time. The design time for punches and dies can be shortened to about 1/3 and can realize intelligent design by using this system. Fan et al. [8] developed a die sets standard parts library on the Pro/E software, which is based on parametric modeling and part family. Kim et al. [9] presented a 3D CAD library which is constructed using the standard components and is used for designing a press die.

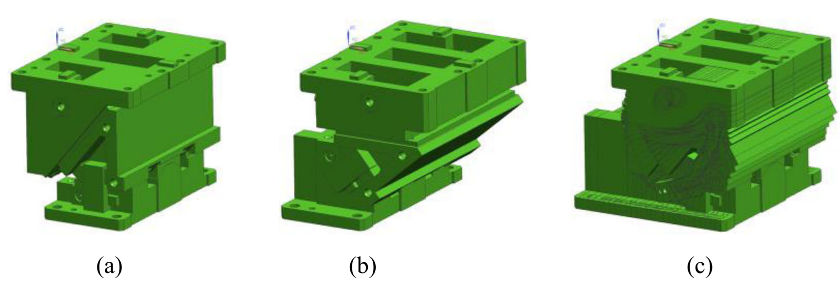


Fig. 2 Different working angle models of the same series of standard cams. **a** 0 degree. **b** 50 degrees. **c** All of the working angles

The key of parametric modeling is feature, which is the smallest unit of 3D model. Various researchers have defined features differently on the application domain. Ranta et al. [10] introduced feature modeling into the early stage of product functional design. A product prototype can be co-designed by multiple designers in different places on the network based on features. Tang et al. [11] proposed an intelligent feature-based design for stamping system. The stampability evaluation of parts has been carried out based on features. Cheok et al. [12] proposed an integrated feature-based modeling and process planning system for bending operations in progressive die design. Ohashi et al. [13] developed computer-aided die design system to design forging sequences and die profiles by eliminating features. Each feature represents a forging process, which is used to characterize the flow of materials during forging. Álvares et al. [14] developed a web-based remote CAD/CAPP/CAM system of design and manufacture based on feature for cylindrical parts. The activities of collaborative design, process planning and manufacturing activities are integrated in a web platform by the use of features. Singh [15] developed a feature library for die design. Pao et al. [16] proposed an algorithm combined with expert formula and geometric feature for extrusion die design. The locality and maximum number of the die orifice in the die head can be optimized by this system. He et al. [17] proposed an intelligent design methodology of product based on feature. The knowledge of functional design can be transferred from the conceptual design to the structural design, even the final detail design.

These researches in literature show the importance of feature and parameter design. Moreover, most of the researches focus on the design of die structure, general product, and die standard parts. There is less research on the parametric modeling of the standard cam. The cam acts as a standard part of stamping die, which need to be tested and optimized repeatedly before being released to the market. The same series of cam is made up of cams with several different work angles. The conventional non-parametric design ideas lead to many drawbacks, such as mass storage of data, cumbersome storage, low efficiency of modification and update. The parametric design and model data management become more and more important. Therefore, the main purpose of this paper is to develop a parametric modeling design system of standard cam for automotive stamping dies based on geometric features. The system is seamlessly integrated in the Siemens NX platform, and the user can operate the assembly cam model instantiated with a specific input angle to realize the functions, such as motion simulation, stress analysis, and BOM (bill of material) output.

2 Geometric feature-based parametric modeling methodology

The standard cam consists of three main parts: holder, slider, and driver (or driver, slider, and base). They are assembled together by other standard parts to form a certain motion rule. The same series of cam are made up of cams with several different work angles. The shapes of three main parts and the types and quantities of standard parts of the same series of cam with different angle specifications are the same. In addition, the details of the features are different. Therefore, in order to ensure consistency and correctness of the same series of cam, all of them have to be integrated into a fully parameterized model by means of activating and suppressing geometric features. The following KMACG600 series of cam is used as an example to illustrate, which is a kind of suspension cam.

2.1 Feature extraction of the three main parts

KMACG 600 series of cam contains eight different specifications of working angle (WA), detailed parameters of which are shown in Table 1. Generally, the WA and the returning angle (RA) of the cam are added to 50°, if the WA is greater than or equal to 50°, then the RA is usually 0°, that is, the wedge return along the horizontal direction. The total height and length of the cams also remain unchanged relative to the center point. The shape and size of the cam slider always remain unchanged in different specifications in the situation of rotating around the center point to obtain different working angles. The widths of the holder and driver are respectively 350 and 190 mm; although they remain unchanged, other dimensions of them will change with the change of WA.

On the basis of the mentioned analysis, the three main structures of the cams are modeled by top-down methodology. The parametric sketch is used to design its outline from top design, as shown in Fig. 3. The middle of the slider is composed of a working face and two guide faces, and the inclinations of the two guide faces are WA and RA, respectively. The holder and slider are moving along the guide of the upper guide face, while the driver and slider are that of the lower guide face. Therefore, the guide faces among the three main parts are parallel to each other, but the distance between them is the thickness of a plane guide plate of standard part. Firstly, the outline of the slider is drawn by sketch, and the relationship between the lines is fixed by constraint. Upper guide face

Table 1 Detail parameters of KMACG 600

Variable name	Specifications							
	1	2	3	4	5	6	7	8
WA	0°	10°	20°	30°	35°	40°	45°	50°
RA	50°	40°	30°	20°	15°	10°	5°	0°
Height	350 mm							
Length	600 mm							
Width of holder	350 mm							
Width of driver	190 mm							

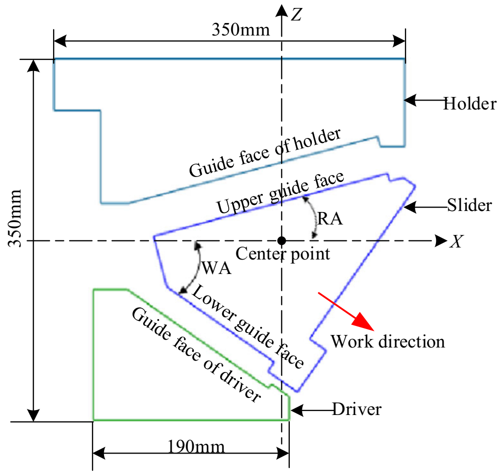


Fig. 3 Sketch outline of the three main parts of cam

and lower guide face are offset by 30 and 101 mm to obtain two reference lines, respectively, and the intersection point of which is constrained to the origin of absolute coordinate system, as the center of the cam. Then, drawing the top line with a width of 350 mm in the position of 175 mm above the horizontal line, another key guide line is obtained by parallel offsetting the upper guide face to complete drawing the other outlines of the holder, according to these two key control lines. Finally, the outline of the driver is drawn by the same method based on the above. The solid structure models are obtained by extruding the sketch outline 300 mm bidirectionally along the Y direction, as shown schematically in Fig. 4.

Obviously, the outlines are designed as geometric sketch features, which are full parameter models. The WA is designed as a master variable. The height of cam, length and width of holder and driver are fixed variables. In addition, all other dimensions are dependent variables, which are relational statements or expressions with WA in NX software. Figure 5 shows three relation expressions of dependent variables with WA. When the WA is 30°, then the RA is 20°, and the DriverLH (the height of driver) is 90 mm and the HolderLH (the height of holder) is 198 mm. As long as the value of WA is changed, other parameters will be changed

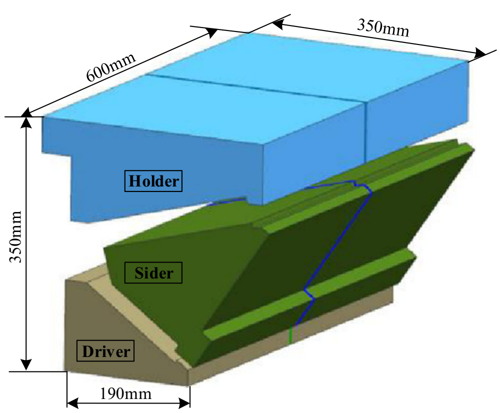


Fig. 4 Top solid model of the three main parts of cam

automatically, and the model will also be updated automatically. Table 2 shows the models of different working angles.

2.2 Association assembly and update of standard parts

Create three main assembly parts under the root node of the NX navigator tree and then copy the sketch outline to each new part. Based on the linked sketch to generate solid body of main parts, the relationships between the main parts and the sketch outline are established. The next step is to realize the motion between them. The motion of the cam includes working and returning, and the three main parts are involved in the motion. In this example, the driver is fixed, the slider and the holder are always locked together, but there is a relative sliding between them. When working, the holder and the slider come down together, once they contact with the driver, the three will move together to complete the side stamping work, and when returning, the holder and slider return back along the driver and then separate from it. The accuracy of the motions described above is guaranteed by the standard parts between them, including between the slider and the driver, as well as between the slider and the holder. The standard type and quantity of the same series of cam are the same, but the size of standard parts will be different with the change of WA. All standard parts are relatively independent, which are assembled on solid bodies of main parts by constraints, and the assembly relationship will be updated automatically with the change of the main part structure, for the constraints themselves are geometric features.

Only the slider and the driver have relative motion. Therefore, they only need the guiding of guide plates. The constraint relationship between the standard parts and the driver is shown in Fig. 6, the V-type guide plate is in the middle, which is used to ensure that the movement does not shift toward both sides, and two plane guide plates distributed on both sides. The reference center planes of guide plates are parallel constrained with *YOZ* datum pane by distance, and the bottom face and one side face of them are bonded together with upper face and a vertical face of the driver respectively by touch align constraints. In this way, the standard parts and the driver are completely assembled together. The standard parts guiding with driver are assembled on the slider which takes the same approach, as shown in Fig. 7. The only difference is that the semi V-type guide plates and forced return hooks on both sides are not constrained with slider, but with the standard parts in driver. However, standard parts in driver cannot be directly involved in the constraints, since they belong to different sub assemblies, and they must be waved to slider part in order to use them as reference objects of constraints. The left forced return hook is fastened to the plane guide plate of driver by three touch align constraints, and the right side is the same. The semi V-type guide plates are constrained with two inclined faces of V-type guide plate respectively by touch align method. In addition,

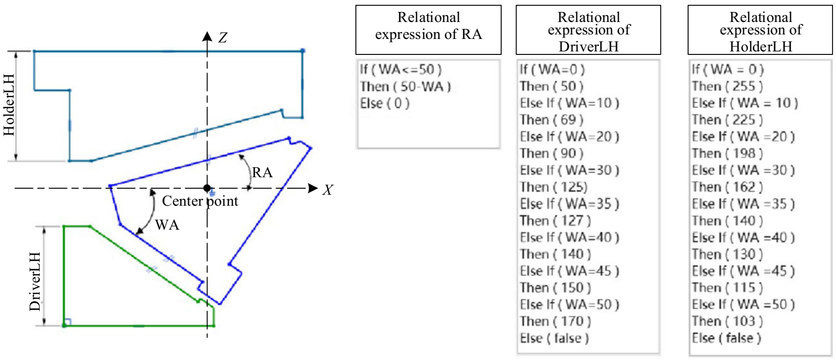


Fig. 5 Example of three relation expressions of dependent variables with WA

their ends are constrained by distance. Consequently, there are no constraints between the forced return hooks and semi V-type guide plates with slider, and there is serious interference between them, also including the driver, as shown in the top view and transparent view of Fig. 7.

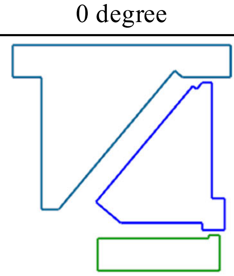
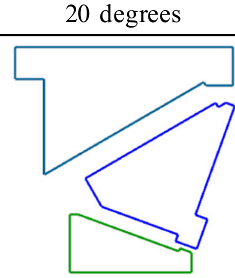
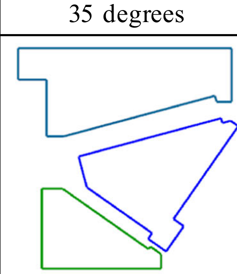
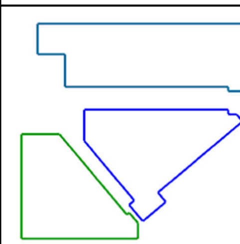
In order to eliminate interference and establish the parametric constraint relation, a concept and technology of reference set in NX software is used in the modeling of standard parts. Figure 8 shows the following two examples of reference set modeling of standard parts. The reference set and solid of standard model belong to the same NX part, but the modeling parameters of former are controlled by the latter. Therefore, the reference set will change with the change of standard solid. The reference set is a subsidiary of standard component in NX, which has the same function as the geometric features of standard model, but when needed, they will not be displayed in general. For this example, the reference sets of forced return hooks are associated duplicated in driver and slider separately by WAVE technology, and that of semi V-type guide plates only need to be copied to slider. Then duplicated solid bodies will be subtracted from the driver and slider, respectively, that is, the Boolean subtraction operation. Accordingly, this interference has been eliminated, as shown in Fig. 9. The associated relational between all the parts of the driver and slider is also established at the same time.

According to the same methodology and technology, standard parts between holder and slider are assembled, and the results are shown in Fig. 10. The number of standard components between the holder and slider is much more than that between the slider and driver, because the slider needs to be mounted on the holder safely by two T-type hanging plates, which also have the same effect with V-type plate to ensure the slider does not shift along both sides. When returning, the slider needs to overcome its own gravity, and the force is provided by nitrogen cylinders installed on holder and T-type hanging plate.

At this point, a detailed description of relationship between assembly and parameterization is needed. Figure 11 shows the mechanism of the fully parametric association and assembly of the cam, and all the parts are strung together by global public parameters and constraints. As shown in Figs. 6 and 7, the distance constraint of 150 mm is a global public parameter, which can be used in any part model under the assembly root node. In the global public parameters, the WA is the master control parameter, which is associated with other parameters. Therefore, the geometric model of the cam is driven by the global public parameters. However, the constraints between components are dependent on the global public parameters and features of geometric models, such as plane faces, datum planes, and center lines. Then, with the change of WA master parameter, all the components and assembly between the models will be automatically updated. Therefore, the design of the fully parametric association modeling of the cam can be implemented robustly, quickly, and accurately.

The specifications of some standard parts will be different due to the change of WA of the wedge. These standards are generally simple and regular, which are fully parametric models. The parametric models are built based on part family

Table 2 Sketch outline of three main structures under different working angles

Working angle	0 degree	20 degrees	35 degrees	50 degrees
Sketch outline				

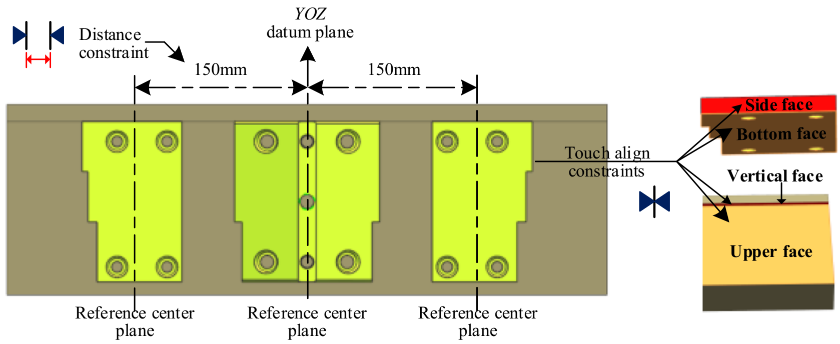


Fig. 6 Constraint relationship between standard parts and driver

modeling technology, which is an important technology for NX. Firstly, the parametric model of standard parts is designed. Then, spreadsheet of the part family is built, in which the members of the part are defined, and each member represents a specification for standard parts, as shown in Table 3. When standard parts are used, any of the members can be selected and added to the component, the parameters of the member in spreadsheet are automatically updated to the model, and moreover, the member can be switched at any time. Therefore, each member of the standard component is also controlled by the WA, and each WA corresponds to a member of the standard component. When the WA is changed, the corresponding members are automatically loaded and updated.

2.3 Parametric modeling of detail geometric features

So far, the key modeling process of cam has been finished, but many details of the features need to be further improved. These features are used to modify the three main parts; moreover, there is a large difference in the models of different working angles, and the modeling of that should be based on global public parameters and standard parts. These details must be able to avoid the installation of standard parts, and they can be updated automatically with the change of WA and specifications of standard parts.

Figure 12 shows the detail feature modeling of driver, and all of these features are Boolean subtraction operations based on the top model of driver. The modeling process follows the following steps: firstly, drawing parametric profile sketches on the plane face of driver, for example, the sketch of detail 1 is on the bottom face, and the sketches of detail 2 and detail 3 are on the top inclined faces. These parametric sketches are constrained by reference to the driver boundary or coordinate system, such as the block in detail 3 with the size 50 × 110 mm, and position is constrained by distance reference *Y*-axis and one edge of driver. However, the sketches of blocks in detail 1 and detail 2 need to refer to the edges of standard parts, which are projected on the sketches by creating interpart link. Therefore, the features will be updated with the change of the standard parts to ensure that the standard parts have sufficient space for installation. Secondly, the block solids are obtained by extruding the profiles of sketches, the normal direction of plane faces that sketches attached are used as the extruding direction, and the distance of extruding is determined by the structure shape of driver. In the blocks of detail 1 and 3, the starting point of extruding is the distance from the reference face 30 mm, and ending point is controlled by the expression of variable DriverLH, which is the global public parameter of driver. In this way, the reasonable sizes of blocks are fully guaranteed. Finally, the blocks are subtracted from driver by Boolean subtraction operation. Figure 12 is the

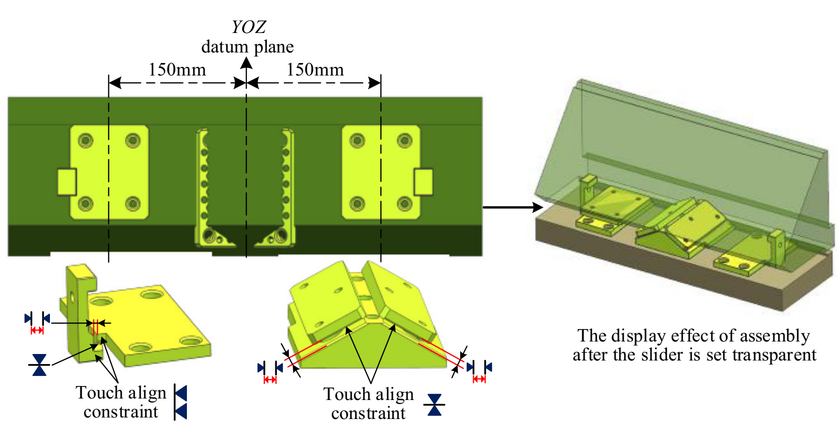


Fig. 7 Constraint relationship between standard parts and slider

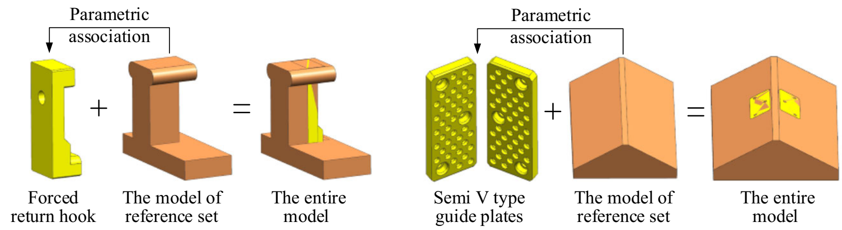


Fig. 8 Examples of reference sets of standard parts

final fully parametric association model of driver. With the same method, the detail features modeling of holder and slider are completed, as shown in Figs. 13 and 14.

However, some detail features only belong to the model of a certain WA specification. These features need to be activated or suppressed with the change of WA. When the feature is suppressed, it is not removed from the model, but is isolated and hidden from the model. Once the suppressed features are activated, they can go back to the state of being suppressed. Figure 15 shows an example of a feature suppression and activation of slider. Moreover, all the features of a part's modeling history in NX can be suppressed and activated by expressions. It provides a favorable technical condition for feature-based parametric modeling.

Table 4 shows the final cam models of different working angles, and only need to change the WA, the structure of three main parts, including all detail features, and assembly position of standard parts and their specifications will be updated automatically. Therefore, a parametric cam model is implemented to integrate all the specifications of the cam, and the continuity and uniformity of the same specification model are guaranteed from beginning to end.

3 System components and development

Based on the above research design, the models of KMACG600 series of cam are successfully created. Meanwhile, the rule of parametric model based on the parametric modeling technology are established in the NX, including the parametric modeling of CAM main structure, detail features, and standard parts of CAM, which can update the CAM model through the working angle drive. Follow this rule in the subsequent work, and the expansion of other series of parametric data of CAM models can be easily implemented. In order to generate instantiation model quickly and to manage all designing models conveniently, an optimization design and analysis system of parametric modeling of CAM is developed in the NX based on the secondary development technology of NX/Open API. All the data of models and standard parts are stored and managed by SQL database technology, which can realize the data reduction and the automatically load of different data model in different platforms, such as designing, 2D drawing, and manufacturing.

3.1 System architecture

Figure 16 illustrates the architecture and components of this system, which consist of three modules: the cam model based on the full parametric design, instantiation of specific cam model, and data manipulation of cam model. The cam models with full parametric design stored in the server with SQL database, which includes design knowledge and history of main parts and standard parts. The instantiation of CAM model and data manipulation integrated in NX enable the operation of cam data through the interface program, which consist of a model operation interface and three application interfaces, including motion simulation, finite element analysis of stress, and output BOM.

Figure 17 shows the flowchart of this system. The front setting and operation are relatively simple, and the key lies in the cam parametric modeling and data management in the background. The core steps of the process are to realize fully parametric modeling of cam and fully parameter-driven assembly-based, which is one of the difficulties. Standard parts used in different series of cam are the same type to achieve the cam data management. Therefore, all the standard parts with the

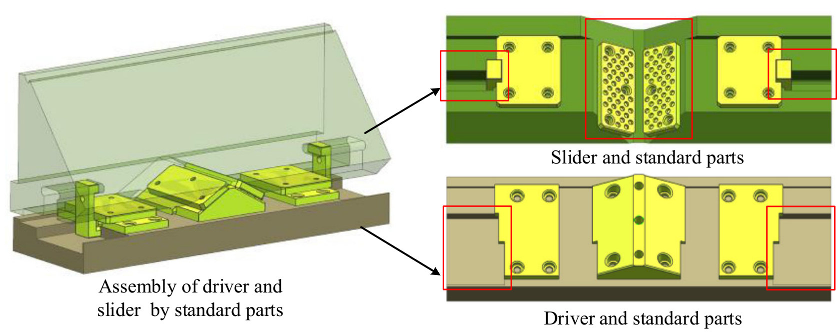


Fig. 9 Final assembly effect of driver and slider

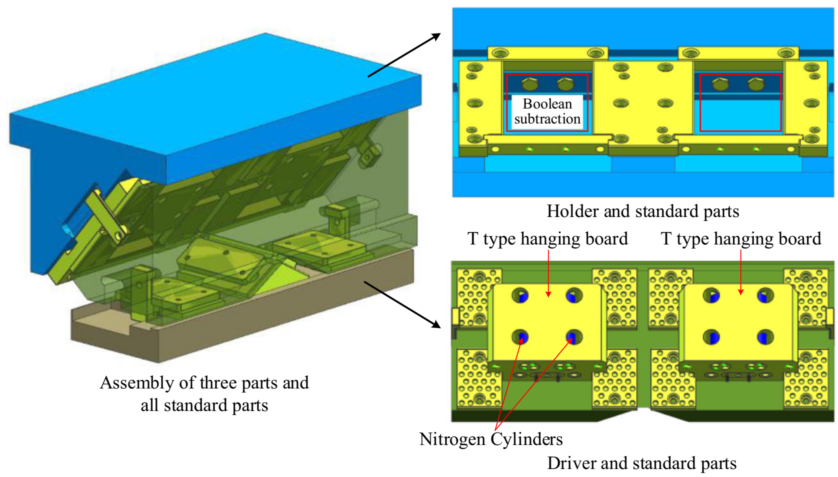


Fig. 10 Final assembly effect of parts

same type in the cam are instantiated by the same standard part, which ensure that all the models used this standard part will automatically update after the standard part updates. The standard part is the public data of all cams. The instantiation of cam needs to drive standard part to be parameterized, which is the second difficulty. This project is combined with the NX parametric modeling and SQL database technology, which is seamlessly integrated into the NX platform to realize the CAM parameter management, maintenance and updating, instantiation of the model, motion simulation, and BOM output. The entire technical scheme is the simple and convenient human-computer interaction and fully meets the development requirement.

A series of control parameters of three pieces corresponds to different working angles, which can output in the form of Excel (Table form), which is similar to Table 3. The values in the Excel can be added, deleted, or modified as needed and can directly correspond to the model, which is convenient and quick. Get the corresponding parameterized standard parts according to the structure of standard parts, control parameters, and model type. Different types of the standard parts are collected in the same parametric model, the idea of which is the same as the creation of three pieces aforementioned. Parameters of the standard parts are outputted to the Excel table, and the different models correspond to different parameters, as shown in Table 3. Cams of different working angles differ not only in the size parameters of three pieces, but also in the types of standard parts. Therefore, all the parameters of the cam model and specifications of standard parts are controlled by a unified table. A more intuitive introduction to the entire process described above is shown in the following flowchart of Fig. 18.

3.2 Cam data storage and management

Figure 19 shows the data hierarchy of the cam. The parametric cam model in the NX modeling is constructed in assembly. For example, the cam in the CAM500 series with the assembly name “CAM500.prt.” All the cam parts are assembled with the form of part in the node. The slider, driver, and holder are the main parts of cam, and the rest are standard parts. We can get them from the library of standard parts in the assembly form. However, the standard part will change with the change of angle as shown in figure. When the angle of CAM500 series is 30°, the standard part 1 is 50 with the angle of 20° and its size is 75, which is instantiated. Therefore, for the maintenance of standard parts, when standard parts in the library updates, the instantiated standard parts in the CAM can also be updated in real time. The main structure and standard parts of cam parts are stored in different folders. We just

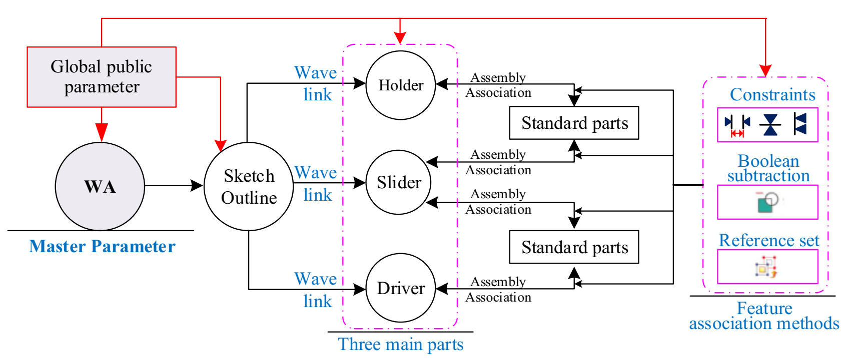


Fig. 11 Mechanism of fully parametric association and assembly

Table 3 Spreadsheet of part family for plane guide plate (mm)

Part number	b	L1	b2	L2	L3	Bool1	Bool2	Bool3	Bool4
MWF-100 × 100	100	100	0	25	75	True	False	False	False
MWF-100 × 125	100	125	0	25	100	True	False	False	False
MWF-100 × 200	100	200	0	25	175	True	False	False	False
MWF-125 × 100	125	100	75	25	75	False	True	False	False
MWF-125 × 125	125	125	75	25	100	False	True	False	False
MWF-125 × 200	125	200	75	25	175	False	True	False	False

store the main structure in the cam and standard parts are stored separately in the library.

Each series of cam has a separate folder stored in the path of "CAM-Model." The independent folder stores the assembly root of this series of cam (recommended to name it after the title of cam series generally) and the three main parts. Moreover, the standard parts are not stored and we only store all the type of standard parts under the "CAM-STD." Each type of standard parts is the parameterized model. All the type can be instantiated by modifying the specification parameter. Each cam series is recorded with the cam type and specifications. The system will automatically find the type of standard parts in the "CAM-STD," and instantiate a specific standard part, then copy this and the main parts of cam to the temporary folder. The system will update the main parts model by updating the angle parameter. Reopen the cam model in the NX, and the cam model with a specific working angle is ultimately required.

4 Structure optimization design based on parametric features

The cam as a standard part needs to be researched and optimization designed repeatedly before being manufactured and released to the market. Motion simulation and stress analysis are the main methods for the optimization design of the cam. The motion interference between parts can be eliminated by motion simulation, and the motion relationship can also be optimized further. While through the stress analysis, the internal structure of the cam can be optimized to improve the comprehensive performance of that. Therefore, the system has carried on the omnidirectional design from the macroscopic and the microscopic to the cam. Based on the parametric features, which are the key link in the optimization design, the closed design and optimization can be realized successfully.

4.1 Motion simulation of cam

The motion law of the cam is more complex, which is triggered by contacting and constrained by the geometric relationship between the contact parts. The key problem to realize the motion simulation is to judge the contact time between the parts and to solve the motion of each part. NX provides two methods of 3D contact and motion function to drive the cam motion. The contact force equation of 3D contact is shown in Eq. (1). The simulation solver will automatically drive the active part to contact the passive part by calculating the active part, without adding to the driver. This method needs to be very accurate in order to achieve a better simulation result, and it needs a large number of steps, which increases the processing time and difficulty.

$$F_c = kx^e \tag{1}$$

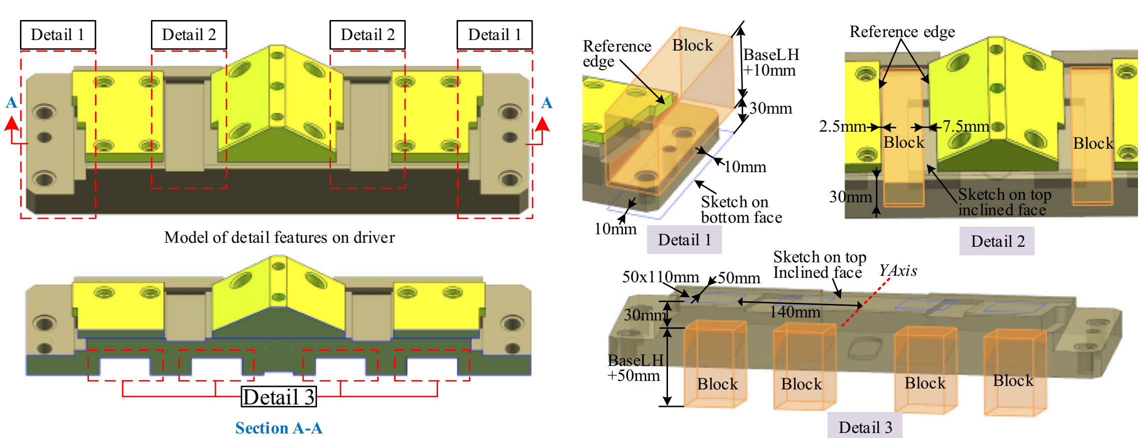


Fig. 12 Detail features modeling of driver

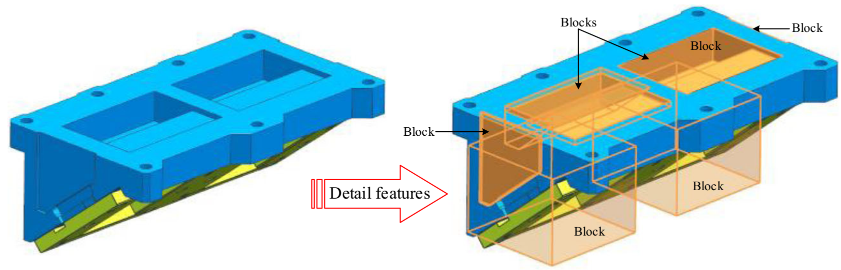


Fig. 13 Detail features modeling of holder

where F_c is the contact force, k is the stiffness matrix, x is the penetration depth of contact ball, and e is the nonlinear stiffness index.

However, the motion function control method is based on the analysis of the motion law and contact relation of each part of the cam to solve the motion equations of each part. Although the real contact between the parts is not considered, through the correct determination of the contact time, the motion law of each part is correctly matched in time. Therefore, it is possible to realize the coordinated motion of all the parts and achieve the effect of the whole motion simulation. The STEP function is used to express the motion law of each part, as shown in Eq. (2).

$$\mathrm{STEP}(\mathrm{time}, t_0, h_0, t_1, h_1) = \begin{cases} h_0 & (\mathrm{time} \leq t_0) \\ h_0 + \left(\frac{\mathrm{time} - t_0}{t - t_0}\right)^2 \cdot (h_1 - h_0) & (t_0 < \mathrm{time} < t_1) \\ h_1 & (\mathrm{time} \geq t_1) \end{cases} \quad (2)$$

where *time* is an independent variable, which is time usually; t_0 is the initial value of t; h_0 is the initial value of step function; t_1 is the termination value of t; and h_1 is termination value of function. The STEP function can be implemented in two forms, nested and incremental, but an integral error may occur when the nesting level is too high, so this system uses the form of incremental to construct the motion driving function.

As shown in Fig. 20, in the stamping process, the holder always moves with the upper die along the stamping direction, and the driver is always fixed with the lower die, but the slider is always lifted on the hold. At the initial moment, the slider is at the starting point of the stroke under the action of elastic force and gravity, which remained relatively static and moved with holder. Once the slider contacts with the driver, it will be forced to move between the guide faces of holder and driver, if the holder continues to move. Until the driver motion is terminated, the slider will move to the bottom dead center. Figure 21 shows the stroke relationship between the parts of the cam, the stroke diagram as shown on the right triangle. Therefore, the following Eqs. (3) and (4) can be obtained.

$$h = s^*(\sin\alpha + \cos\alpha^*\tan\beta) = s'^*(\sin\beta + \cos\beta^*\tan\alpha) = v^*t \quad (3)$$

$$l = \frac{h}{\tan\alpha + \tan\beta} = \frac{v^*t}{\tan\alpha + \tan\beta} \quad (4)$$

where h is the motion stroke of holder along the stamping direction, v is the motion velocity of holder, t is the motion time, s is the working stroke of slider, s' is the stroke of slider along the drive direction, l is the stroke of slider along horizontal direction, and α and β are the working angle and driving angle, respectively.

According to the relationship between the various parts of the motion, the motion function curves of them can be created. A complete motion of the cam includes two sub processes: working and returning, which are just two opposite processes, as shown in Fig. 22. The driver remains stationary during the whole motion process. The holder always keeps uniform motion along the stamping direction, and the motion curve is

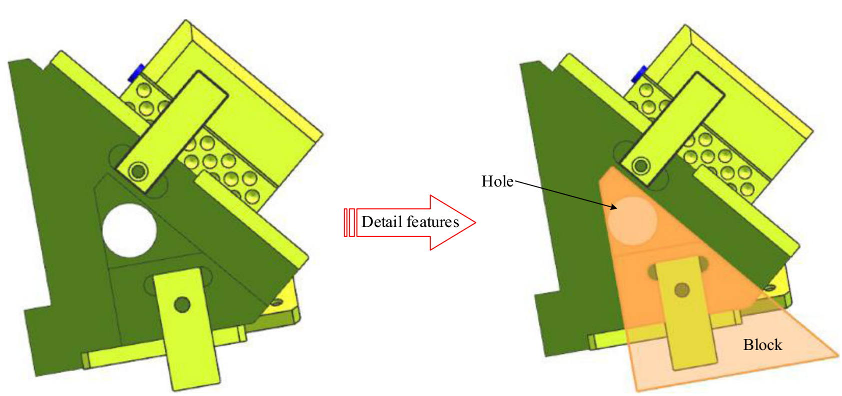


Fig. 14 Detail features modeling of slider

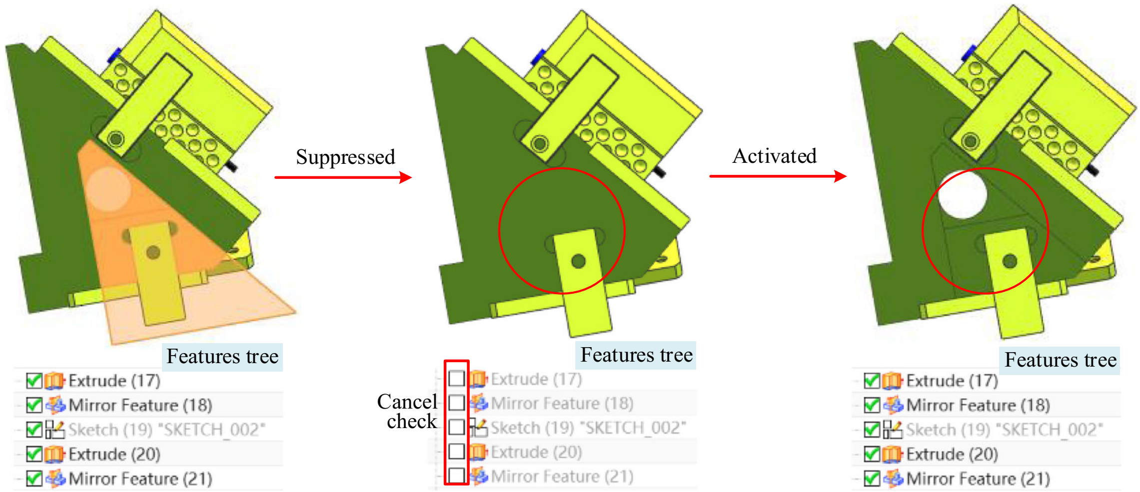


Fig. 15 Suppressed and activated of geometric features

shown in Fig. 22a. However, the slider motion needs to be decomposed into two directions of the horizontal direction and stamping direction, as shown in Fig. 22b. It is illustrated that the all motion curves are linear with time. It is possible to obtain a higher accuracy by using motion curve to simulate the motion of cam.

The motion curves cannot be used directly, which need to be discretized in order to extract the effective motion data. The motion simulation is a process of cyclic motion, so a cyclic process is represented by 360°. Therefore, the time axis as a reference axis is discretized into 360 segments by equal time to obtain 361 time nodes. The relation between displacement and time are as shown in Eqs. (5) and (6).

$$h_i = h\left(i^*\Delta t\right) \quad (i = 0, 1, 2, \cdots, 361) \tag{5}$$

$$l_i = l\left(i^*\Delta t\right) \quad (i = 0, 1, 2, \cdots, 361) \tag{6}$$

$$\Delta t = \frac{t_2\text{-}t_0}{360} \tag{7}$$

where i is the node number of discrete time, Δt is the time increment step, and h_i.and l_i are the coordinates of the trajectories at i node time.

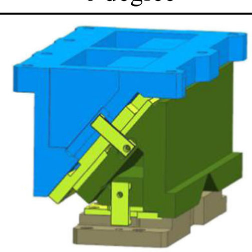
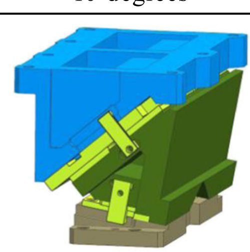
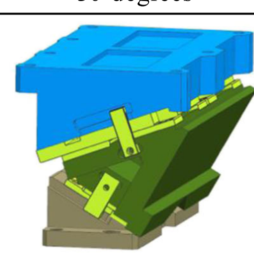
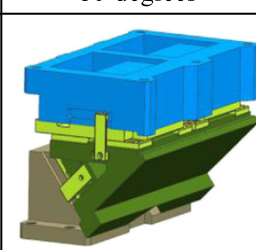
Therefore, the *STEP* functions of the h and l of holder and slider can be described as follows: Eqs. (8) and (9). The motion of holder is controlled by the step function of Eq. (8), but that of slider is controlled by the step function of Eqs. (8) and (9) simultaneously.

$$\begin{aligned}\mathrm{STEP}(\mathrm{time}, 0, h_0, 1, h_{01}) &+ \mathrm{STEP}(\mathrm{time}, 1, 0, 2, \Delta h_1)\\ &+ \ldots + \mathrm{STEP}(\mathrm{time}, i, 0, i+1, \Delta h_i)\\ &+ \ldots + \mathrm{STEP}(\mathrm{time}, 359, 0, 360, \Delta h_{359})\end{aligned} \tag{8}$$

$$\begin{aligned}\mathrm{STEP}\left(\mathrm{time}, 0, l_0, t', l_0\right) &+ \mathrm{STEP}\left(\mathrm{time}, t', 0, m_1, \Delta l_{i_1}\right)\\ &+ \ldots + \mathrm{STEP}(\mathrm{time}, i, 0, i+1, \Delta l_i)\\ &+ \ldots + \mathrm{STEP}\left(\mathrm{time}, m_2, 0, t'', \Delta l_i\right)\\ &+ \mathrm{STEP}\left(\mathrm{time}, t'', 0, 360, 0\right)\end{aligned} \tag{9}$$

where $\Delta h_i = h_{i+1} - h_i$, $\Delta l_i = l_{i+1} - l_i$ ($\boldsymbol{i}$ = 0,1,2,…,359), t' is the moment of slider contacted with driver, m_1 is the integer time node after the contact time, t', t'' is the moment when the slider and driver are out of contact, and m_2 is the integer time node

Table 4 Final cam models of different working angles

Working angle	0 degree	10 degrees	30 degrees	50 degrees
The final cam model				

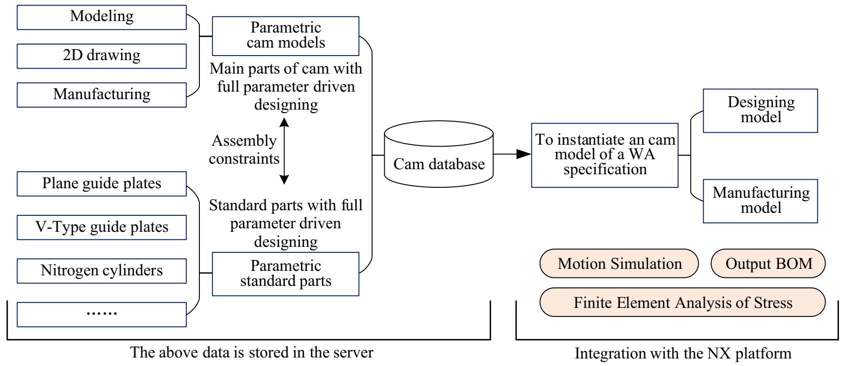


Fig. 16 System architecture

after the contact time t''. In this way, the motion of each part of the cam is closely matched with time.

The motion simulation can truly reflect the interference and reasonableness between the internal parts of the cam. If interference occurs, or the structure is unreasonable, the parameters or constraints of the original parametric model can be modified directly, and then, the simulation model can be updated automatically without having to be defined again.

4.2 Finite element analysis and optimization

Finite element analysis is used to simulate the stress and deformation of cam. According to the simulation results, the structure of the wedge can be further optimized. The finite element analysis is generally divided into three steps: pre-processing, calculation, and post-processing. The pre-processing of FEM (finite element method) is a huge and complex process, including the simplification and abstraction of the model, the establishment of finite element mesh, the definition of material, and boundary conditions. The effective and reasonable pre-processing of CAD model will greatly save the time of analysis and ensure the correct and reasonable results of finite element analysis. On the basis of parametric modeling, this system can realize the automatic pre-processing and calculation of the finite element analysis based on the features.

There is a big gap between the parametric model and the finite element mode. The parameterized model needs to be simplified to remove or suppress some features. What are affected are not the results of finite element analysis but the efficiency. This simplified model is called idealized model in the system, which is the input model of finite element analysis. There are two key steps in the generation of idealized models.

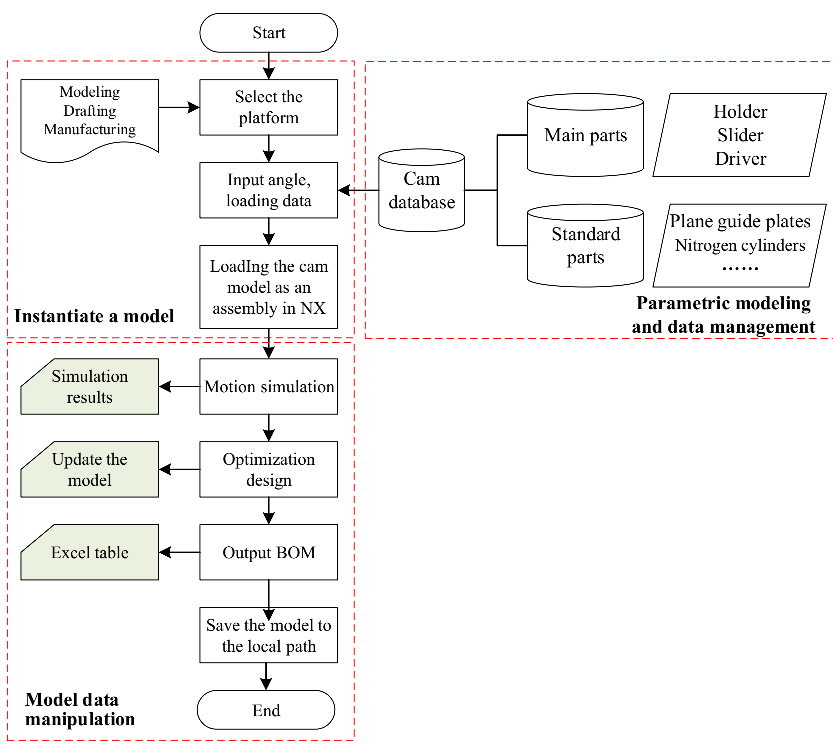
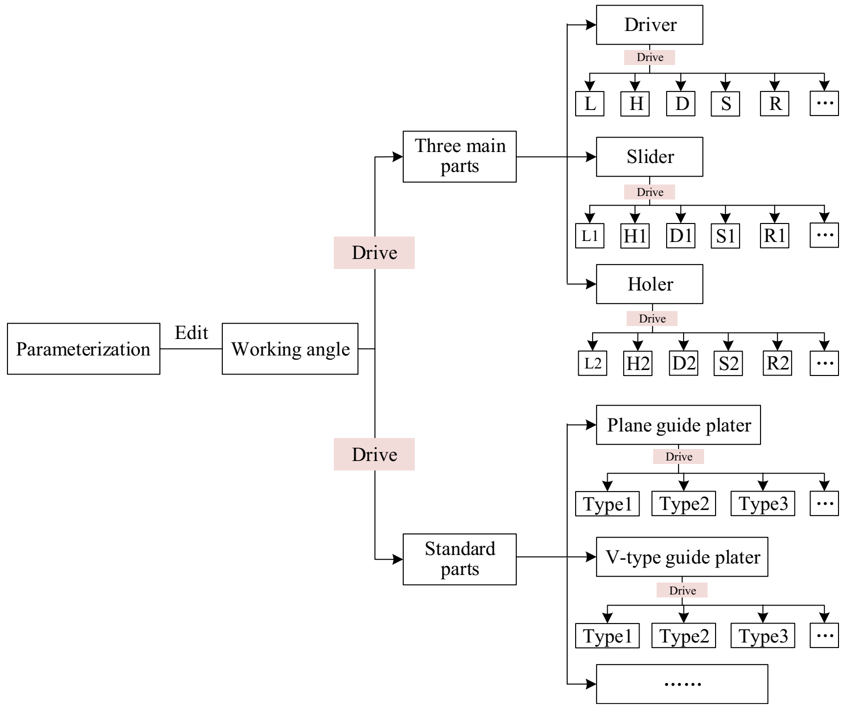


Fig. 17 Flow chart of system technology

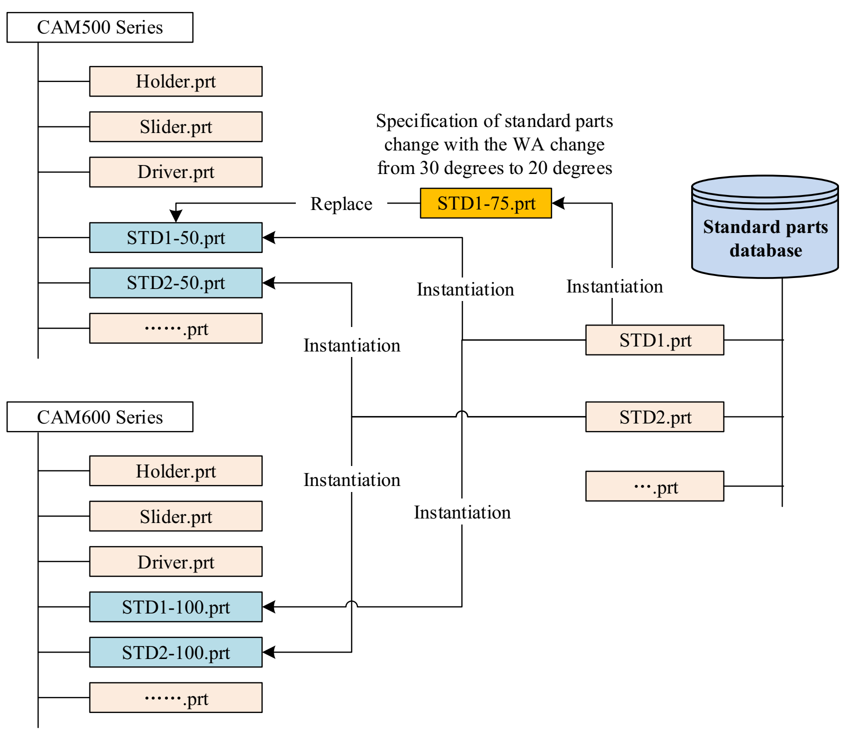
Fig. 18 Parameters driven relationship of cam



1. Suppression of small features. These features include the details of three main parts, structure and standard parts, such as screw and pin holes and some modified chamfers, which have a great influence on the size of element, and little influence on the precision of analysis. Figure 23 shows the features suppression of idealized model of driver. All the corner modifications, screw, and pin holes of the model parts are suppressed based on the parametric model.

2. Model integration based on Boolean addition operation. There are only three moving parts: holder, slider, and driver, and the rest of the standard parts are fixed on the

Fig. 19 Data hierarchy of cam



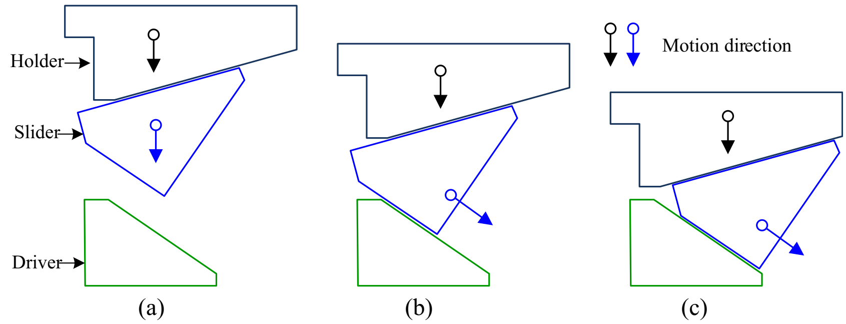


Fig. 20 Motion process of cam. **a** Initial moment. **b** Contact moment. **c** Termination moment

mounting surface by the bolts. In this case, the two parts are generally considered as a whole in finite element analysis, which can greatly simplify the finite element analysis of boundary conditions between parts. The simplified standard parts attached to the three main parts are merged into the main part model by Boolean addition operation, respectively. Therefore, a simplified idealized model for the finite element analysis of the cam is obtained.

After the model is simplified, the finite element analysis model is switched directly in NX. Meshing the three models respectively by using the ten nodes tetrahedral element with dimensions of 20 mm, the finite element mesh model is shown in Fig. 24. Among them, the change from the parametric model to the idealized model is the most time consuming process, if the simplification of the model needs to be handled manually.

The cam is subjected to two external forces, and the force is applied on the holder by Press and is acted on the slider by working. The force on the holder by Press is uniform force, which is acting on the upper surface of the holder, while the working force acted on the slider is generally concentrated force, which is determined by the working conditions of cam. Because the work content is not known, the work force is loading by uniform application. The force of the nitrogen cylinder in the slider is considered internal force, which is determined according to the specification and stroke. This force is considered to be the action and reaction force acted on the contact surfaces of slider and holder with nitrogen cylinders. The loading of external and internal force is shown in Fig. 25a. The driver always keeps fixed during the working process; therefore, the boundary condition of fixed constraint is applied on the bottom nodes of driver. The guide faces between the three parts are applied to surface-to-surface contact constraint, the friction coefficient of which is assumed to be 0.15. Figure 25b shows the boundary conditions and constraints of the FEM model.

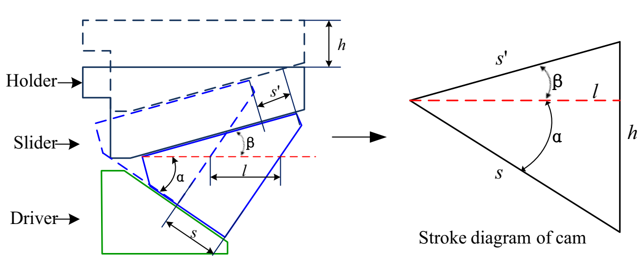


Fig. 21 Stroke relation diagram of cam

After the finite element model is prepared, it needs to be submitted to the solver to solve the various physical fields. The Nastran with third-party solvers is embedded in the NX, which has high simulation accuracy. The results of the deformation of cam under the action of F1 = 42kN and F2 = 45kN are shown in Fig. 26a. The maximum amount of deformation is 0.006 mm in the weakest position of holder. The deformation of this position can be defined as a target, which is 0.1 mm as the limit, and then the maximum allowable force of the cam is optimized, because the cam is used to complete side piercing or flanging. In order to ensure the accuracy of piercing, the cam can not have too large deformation. In general, the highest accuracy of the hole is 0.1 mm. Therefore, the maximum deformation of the cam optimization is set to 0.1 mm. The aim is to optimize the internal structure of the cam so that it can withstand a greater external force to produce deformation less than 0.1 mm. In this way, the accuracy of the hole can be guaranteed. The maximum force F1 = 412kN and F2 = 479kN. Figure 26b shows the deformation of the cam under the maximum force. Figure 27 shows the stress distribution of the cam model under the maximum external force. Therefore, the maximum allowable force of the cam can be optimized by finite element analysis.

The above is only the case where the working angle is 10°, and the rest model of the other working angles also needs to use the same method to carry on the optimization. Taking advantage of parametric model, the idealized models and finite element models are also parameterized. The system will automatically update all the models by changing the working angle. Then the system

Fig. 22 Motion function curves of holder and slider. **a** Motion function curve of holder. **b** Motion function curves of slider

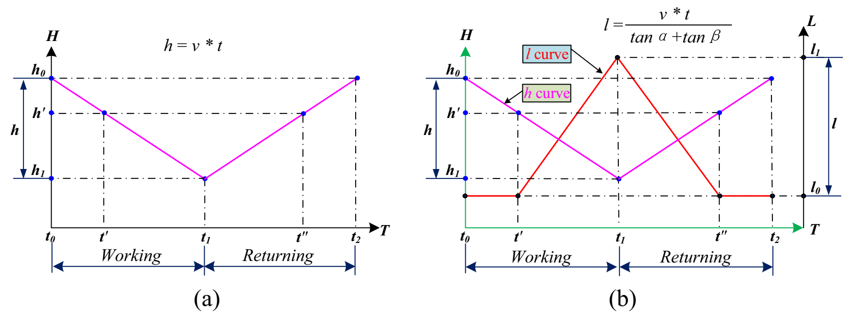
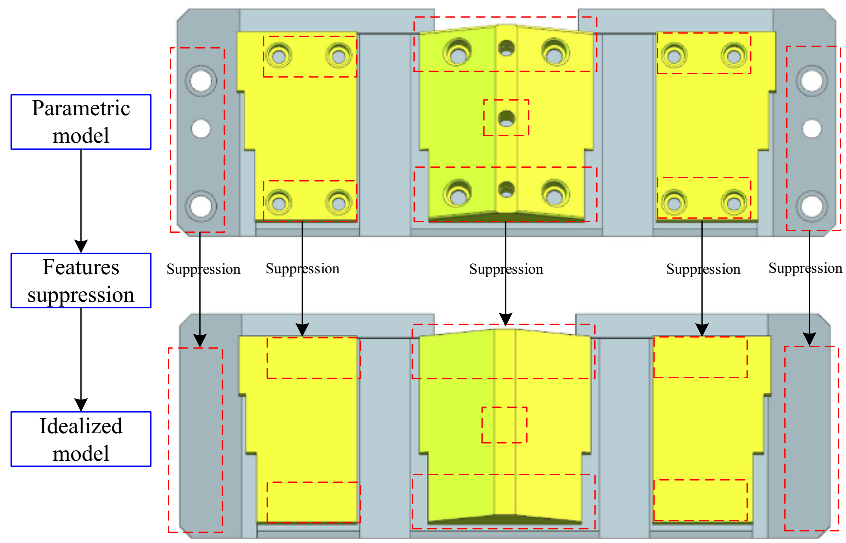


Fig. 23 Features suppression of idealized model of driver



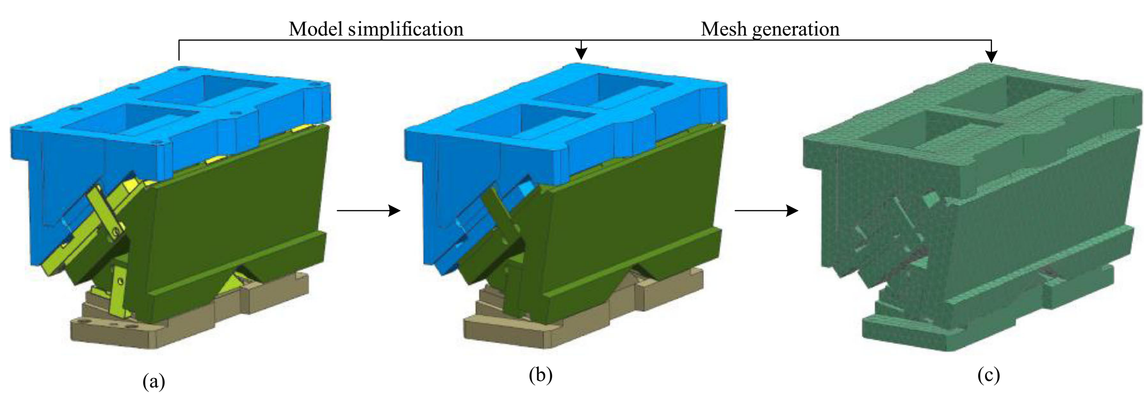


Fig. 24 Generation process of finite element mesh model. *(a)* Parametric model. *(b)* Idealized model. *(c)* Finite element mesh model

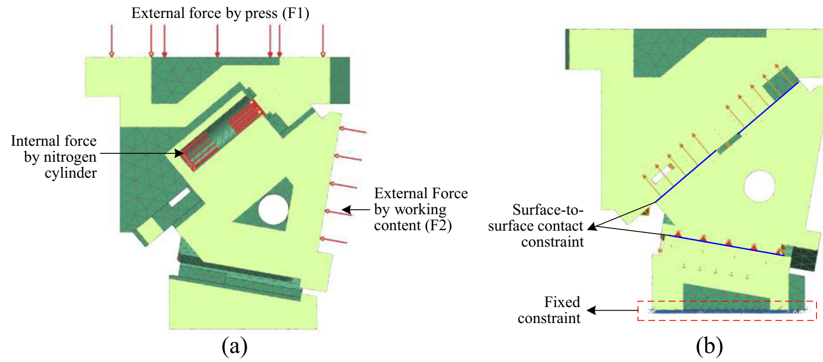


Fig. 25 Force loading and boundary conditions. **a** Force loading. **b** Boundary conditions and constraints

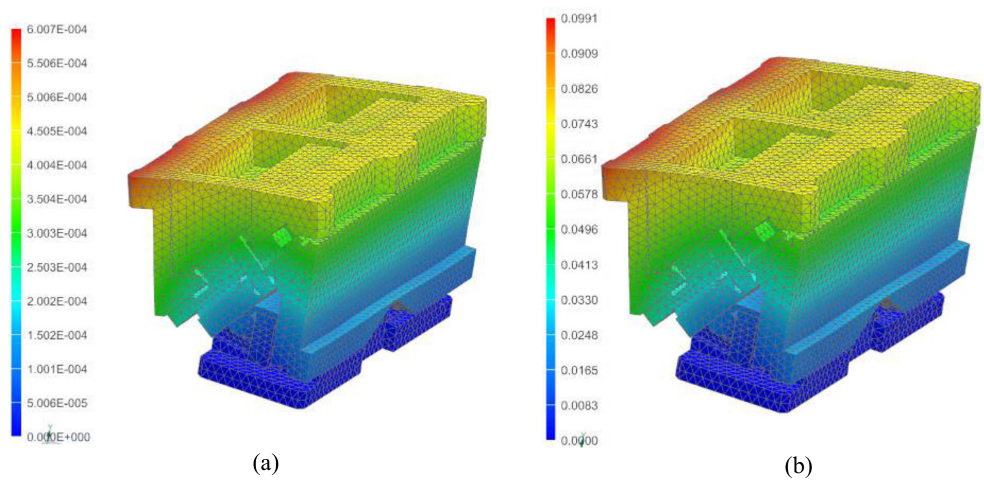


Fig. 26 Deformation of the cam model under the action of F1 and F2. **a** F1=42KN, F2=45KN. **b** F1=412KN, F2=479KN

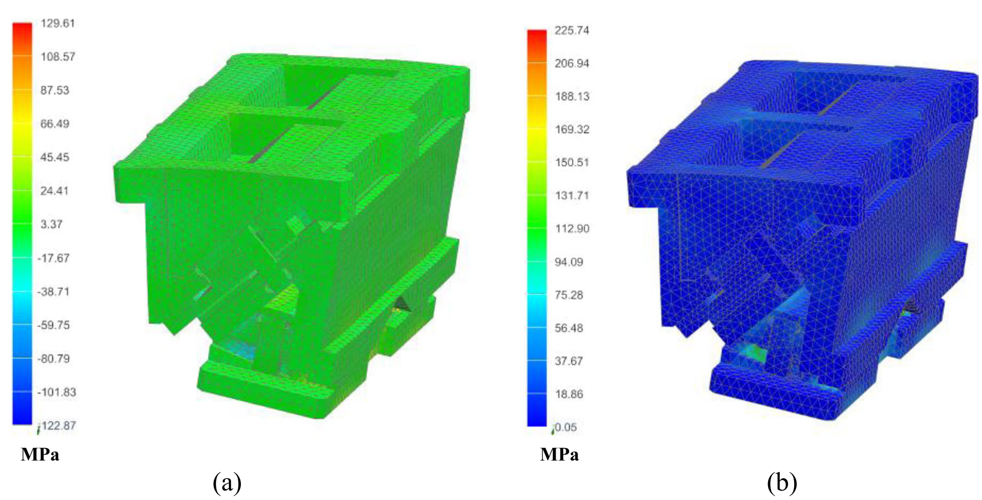


Fig. 27 Stress distribution of the cam model under the maximum external force (F1 = 412kN, F2 = 479kN). **a** Mean stress. **b** Von-Mises stress

automatically calls NX solver for iterative optimized calculation. The maximum allowable force for each specification model of working angles is shown in the following Table 5, which can provide a useful guide to the use of cams. The results of the optimization analysis show that the parametric modeling can effectively improve the design efficiency and quality, which is an important technology of intelligent manufacturing.

5 Output of bill of materials

BOM is used to identify the structure of the product and the link of business communication. The BOM runs throughout the manufacturing process of the cam, such as the procurement of standard parts and assembly parts. Therefore, it is important to ensure the accuracy of the BOM because the cam has a lot of angle specifications, and they are also very similar, which are easily confused. How to quickly and accurately output BOM is also an important research content of this system.

A parametric model of cam contains multiple work angle specifications, which can be instantiated at the time of manufacture, and each angle model corresponds to a BOM table. The BOM tables are generally excel forms and have a fixed template. The number of parts in the BOM will not change for cam, but the specification of some parts will change. Table 6 is a BOM of working angle of 30°, including part number, name, quantity, material, specification, manufacturer, weight, and type. Different companies have different requirements for BOM tables, which need a customized development.

In order to ensure BOM of cam is accurate, all the information of the part is stored in the attributes of the model in NX. Some of the information is static, such as part number, name, quantity, and type, while others are dynamic, which will change with the change of the model, such as size and specification of some standard parts. The size is calculated according to the model in real time, which requires updating. The specifications of standard parts are obtained from the “OS_PART_NAME” in the spreadsheet of part family, and the system obtains its specification attributes by intercepting the characters. Figure 28 shows the BOM attributes of one parts.

The BOM table in the installation folder of the system is the BOM of current angle specification of cam and also the BOM table template. Once the angle has changed, the system first updates the attributes of each part in the model and then writes them to BOM table in the installation folder. Therefore, the consistency and correctness among the current angle specification of the model, the attributes of each part, and the BOM table have always maintained. When the model is instantiated for manufacturing, the system will automatically output the BOM of the current angle specification of cam to the user specified file path. Moreover, in the generation of two-dimensional drawings, the system will automatically read the part number from the property to generate the explosion of assembly of the cam.

Table 5 Maximum allowable external force of the cam

Max. external force	Specifications							
	1	2	3	4	5	6	7	8
WA	0°	10°	20°	30°	35°	40°	45°	50°
F1 (kN)	350	412	471	452	464	462	507	512
F2 (kN)	406	479	480	444	450	469	495	507

6 Conclusions

A new design system of standard cam based on the geometric features and parametric association modeling technology was researched and developed for the stamping dies of automotive panels. The example of KMACG 600 demonstrates that the system has an excellent performance in the design and development of new cams for stamping dies. The system presents the following remarkable characteristics:

1. The models of different work angles of the same series of cam are integrated into a fully parametric model, which can reduce the repeated modeling, simplify the model data and improve the design quality greatly.
2. The closed design and optimization system realizes the whole process of cam development, from the model's parametric design to optimization design of motion simulation and the finite element analysis and finally to output the BOM table of the cam.
3. The model of new series of cam can be derived from an existing parametric model of cam quickly by modifying parameters and editing geometric features, for all the history of parametric modeling process is preserved on model, which is an important technology of intelligent manufacturing. This can greatly shorten the development cycle of new cam and reduce the design difficulty.
4. The database and application interfaces are seamlessly integrated into a unified operating platform,

Table 6 BOM table of working angle of 30°

No.	Name	Number	Material	Specification	Manufacturer	Heat treatment	Size	Remark
1	Holder	1	HT300	KMACG500513		Casting annealing		
2	Slider	1	HT300	KMAC500023		Casting annealing		
3	Driver	1	HT300	KMAC400533		Casting annealing		
4	hanging plate	3	QT450 + graphite	KMACG20HUNG		Casting annealing	87 + 105 + 130	
5	Safety hook	2	45	CAQG-15030095		HRC40–45	15 + 30 + 95	
6_1	Guide plate	1	42CrMo	CDBX-25058240-L-B		Carburization/HRC more than 55	25 + 58 + 240	
6_2	Guide plate	1	42CrMo	CDBX-25058240-R-B		Carburization/HRC more than 55	25 + 58 + 240	
7	Guide plate	2	42CrMo	CDBX-25040120		Carburization/HRC more than 55	25 + 40 + 120	
8	Guide plate	4	Copper alloy + graphite	CDBY-12055100		HB more than 200	12 + 55 + 100	
9	Spring top	3	45	CTDK-30029-B		HRC38–42	D30 + 29	
10	Buffer	3	Polyurethane	CHCQ-20020-G			D20 + 20	
11	Limit block	3	45	CXWK-25025122		HRC28–32, Blackening	25 + 25 + 122	
12	Guide plate	2	Copper alloy + graphite	CDBY-12060150		HB more than 200	20 + 60 + 150	
13	Key	4	45	CKEY-14032050		HRC28–32, blackening	14 + 32 + 50	
14	Key	4	45	CKEY-12032026		HRC28–32, blackening	12 + 32 + 36	
15	V-type guide plate	1	42CrMo	CVDX-42125160		Carburization/HRC more than 55	42 + 125 + 160	
16	Rigid retum	2	45	CFHQ-19035081		HRC35–40, blackening	19 + 35 + 81	
17	Guide plate	2	42CrMo	CDBX-25080240-C		Carburization/HRC more than 55	25 + 80 + 240	
18	Guide plate	4	Copper alloy + graphite	CDBY-12080100		HB200以上	12 + 80 + 100	
19	guide plate	2	Copper alloy + graphite	CDBY-12060120		HB200以上	12 + 60 + 120	
20_1	Guide plate	1	42CrMo	CDBX-12070160-L		Carburization/HRC more than 55	12 + 70 + 160	
20_2	Guide plate	1	42CrMo	CDBX-12070160-R		Carburization/HRC more than 55	12 + 70 + 160	
21	Nitrogen spring damper	3	45	DSDB-20048105		HRC38–42	20 + 48 + 105	
22	Gas spring	3		X350–075	KALLER			
23	Nitrogen spring block	3	45	CSDB-33014		HRC40–45	D33–14	
24	Retarder	4		MC150MH3	ACE			
25	Lock block	2	45	CXWK-19064095		Blackening	19 + 64 + 95	

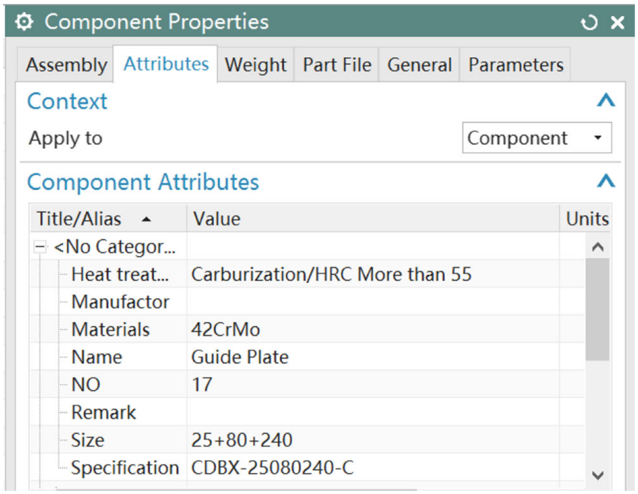


Fig. 28 BOM attributes of part in NX software

NX software, and the system operation is very simple and practical, which can reduce the dependence on experience of parametric modeling and optimized design.

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