GEARS PARAMETRIC MODEL IN CATIA

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Abstract: The paper presents a solution for insertion of gears parametric features in CATIA and the steps used for platform integration of the model. From specific geometric data on spur gears, the source solid is created as base for the digital 3D gears family with involute teeth, with parametric coupling, helix and external toothing.

Keywords: design automatization, gears, parametric feature

I. INTRODUCTION (11 PT TIMES NEW ROMAN, STYLE SUBTITLEA)

CATIA V5® is an complex engineering design software used, among many others, in parametric 3D modeling, surface modeling and cinematic and structural simulations. It contains comprehensive libraries for standard parts and has the possibility to create user defined libraries for repetitive content. As gears are not included as accessible parametric parts, this paper offers a simple yet complete solution for integration of gear repetitive content as accessible function, embedded in the program platform.

The geometry of a spur gear is controlled by well-known parameters (Figure 1), so the generic gear will be controlled by those parameters, while its mounting will be considered [1, 2]:
- with a single keyway for key mounting on the shaft
- for involute splined joint on shaft
  - spur spline
  - helical spline

II. THE GENERIC GEAR DESIGN

1. Parameters definition

The complex geometry of the tooth recommends the Generative Shape Design [3] mode. In order to obtain a structured hierarchy, the tooth will be defined in a Geometrical Set – a definition set that relates intelligent geometric features [3]. The first step will be the definition of the parameters: modulus, teeth number, outer, pitch, root radiuses etc.

![Figure 2. Parameters definition](image2)

The geometric parameters will be related with specific laws [2, 3], as dependent, while the principal parameters remain independent, used for user input on the interface:

![Figure 3. Dependent parameters in the law editor](image3)

The involute spline curve will be the...
support of the tooth profile, design based on the defined parameters. The curve will be rotated against the ZX plane. The outer and root circle will be used for trimming, filleting, in order to finish the profile, which will be mirrored to complete a tooth.

Figure 4. Construction of the tooth profile from parametric spline curve.

The tooth profile will be arrayed, joined for a sketch and extruded in Part Design mode as a basic Pad feature will be obtained.

Figure 5. Spur gear PAD.

2. Mounting parameterization

In order to solve the mounting type representation specific design parameters will be used as well as logical parameters for the status of the parametric block [2]. Newly defined parameters will be used for the corresponding pocket definition and for square splined mounting definition [4, 5]. The parameters linked with the key mountin will be deactivated and a new pocked with the corresponding array will be defined to obtain the spline mounting profile [4].

Figure 6. Different mountings for the spur gear. The geometric elements for the spline mounting will be deactivated also in order to define the involute spline mounting profile parameters, in a new Geometrical Set. Further, a new pocket feature will be used to define the whole array, as previous described. The geometric definition of the spur gear is finalized.

3. Teeth type parameterization

In order to derive the helical gear from the generic spur gear pad it is neccessary to deactivate all the extruzion definitions, including the initial Pad definition up to the initial plane sketch. The supporting rail will be defined as a parameterized helix element. The toth will be obtained as a RIB feature along the helix, with the joined plane sketched structure. This will be the base for the future generic helical gear.

Figure 7. Helical gear PAD.

Figure 8. Helical gear mounting variants

III. GEAR AUTOMATIZATION

In order to obtain an automatic access to the various types of the gears, three boolean parameters will be defined: „spur“, „involutespline“, „key“:

Figure 9. Gear formulas - boolean parameters
A new Rule will be used to implement these new status parameters, on the basis of six macros: spur-helical, helical-spur which correspondingly transforms the spur gear in helical gear. At the beginning, the geometric parameters are defined and, consequently, the mounting parameters are deactivated. Depending on the user’s choice, the spur gear PAD or the RIB of the helical gear are deactivated, then brought in the desired state.

Three macros for each type of gear (spur or helical), for which will be declared, in the corresponding order, all the elements linked with the mounting type. In figure 10 the hierarchy of the macro tree derived from the choices is represented.

![Diagram of macro hierarchy]

Figure 10. Logical hierarchy of the macros

After the gear model representation, the bidirectional associativity permits its editing in any of the part, assembly or drawing modes. The gear table can be represented using the Design Table command. Due to the automatic structure of the parametric gear model, the data can be easily exported in any of data base software, or imported from structured libraries with standard data.

III. CONCLUSIONS

This gear model can be used in assembly design and can be fully interfaced with data bases. This study can be extended up to building in geometric data and mechanical elements libraries, using Generative Structural Analysis module or for editing the graphic interface and new functions definition, using Microsoft Visual Basic, for easy geometry insertion in the model space.

This newly defined function can integrate automatic calculus of the assembly features, maximum torque etc. This study can be extended to bevel gears, and more detailed geometry (structural holes or gear base).

LITERATURE:


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