PARAMETRIATED ASSISTED DESIGN USING CATIA

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ABSTRACT: *The design activity using the CATIA program often encounters situations in which parts families must be created. They usually have the same constructive shape, but differ in size. Also, in the design of the assemblies it is useful to create dimensional links between two or more components, so that the modification of one of them leads automatically to the modification of the other. This mode of design involves advanced knowledge of modeling, programming and management of parameters and relationships, being called parameterized assisted design.*

KEY WORDS: assembly modeling, simulation of manufacturing, gear tooth, parameter, formula, computing.

1. INTRODUCTION

CATIA (Computer Aided Three dimensional Interactive Applications) is a product of the company Dassault Systemes representing one of the most advanced integrated platform type: CAD/CAM/CAE based assembly modeling on the latest technologies in the field of software industry [1].

CATIA V5 is available as from the year 1999. At the current time CATIAV5 contains more than 140 robust applications covering the following areas of electronic engineering:
- explicit hybrid parametric modeling;
- surface modeling, sheet metal;
- assembly modeling, design optimization;
- generating drawing drawings;
- design of molds and shapes;
- reverse engineering, rapid prototyping;
- analysis using finite element method;
- kinematic analysis using the virtual prototype;
- simulation of manufacturing processes;
- design of electrical parts, pipelines, heating, ventilation and air conditioning;
- CNC programming for CNC machines with 2/5 axes;
- translators for converting entities into / from other design environments.

Using the finite element method can solve some types of problems such as:
-Problems independent of time [1]:
    -analysis of the tensions and registering strains;
    - static analysis of structures;
    - inter-surface contact analysis;
    - temperature stress analysis;
- Problems of spreading or transition:
    -problems of fracture mechanics and fissures under dynamic loads, fatigue behavior, J-integral, cracks, crack growth;
    - the response of structures to aperiodic tasks;
- Own value problems:
    - natural frequencies and own modes of structures;

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2. PARAMETRIATED MODELING OF PARTS

In the design activity, there are frequent cases in which families of parts must be created. They usually have the same geometric shape, but differ in size. Also in the production of assembly drawings (assemblies) it is necessary to create a dimensional connection between two or more components, so that the modification of one of them leads implicitly to the modification of the other.

Thus, the CATIA Knowledge Advisor module allows the efficient integration and use of all the data available for construction, facilitating the technical decision making through rules, formulas, parameters, elements that are considered and used together in a certain context.

The parameters are defined by a name, a type and a value, but instead of this value a relation can be used. In this case, the parameter is constrained by this value and the user cannot change its value directly [2].

The CATIA Knowledge Advisor module manages four types of relationships:

-formulas, which define how a parameter should be calculated based on other parameters. The formulas also use mathematical operators to define the parameters;

- parameterization tables, which contain data in ordered form, used to control the parameters of a document;

- rules, which represent a set of instructions that conditionally executes a group of reports in a context depending on the values of these parameters;

-checks, which are a set of instructions whereby the user is warned whether certain relationships between rules are met or not.

Parameters, operators or functions are used to write a formula. Each formula has a parameter associated with it, called "Activity". Its value is of type "boolean", being defined by "True / False" (fig. 1) [2].
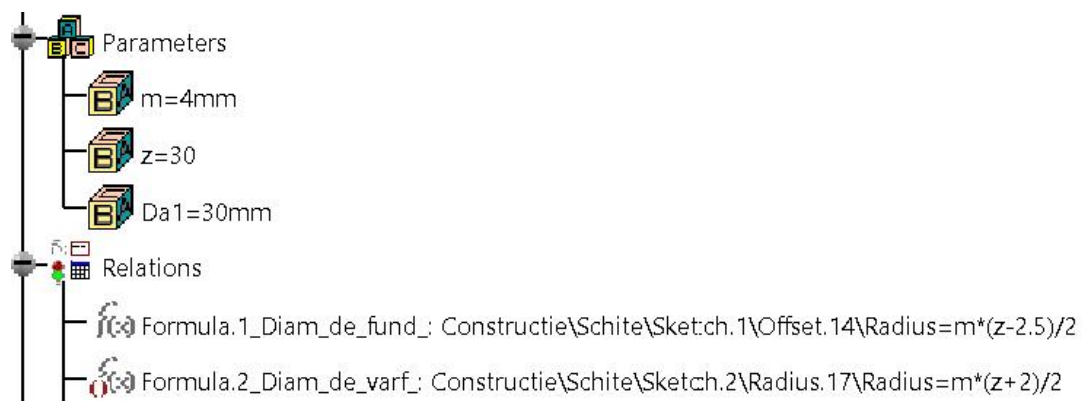

Fig.1. Property of a "true / false" formula

It can be observed that the second formula is marked by a pair of red parentheses, meaning that it is inactive. The formula can be activated or deactivated by right-clicking on the formula and following the windows open until "enabled / disabled" (fig.2)
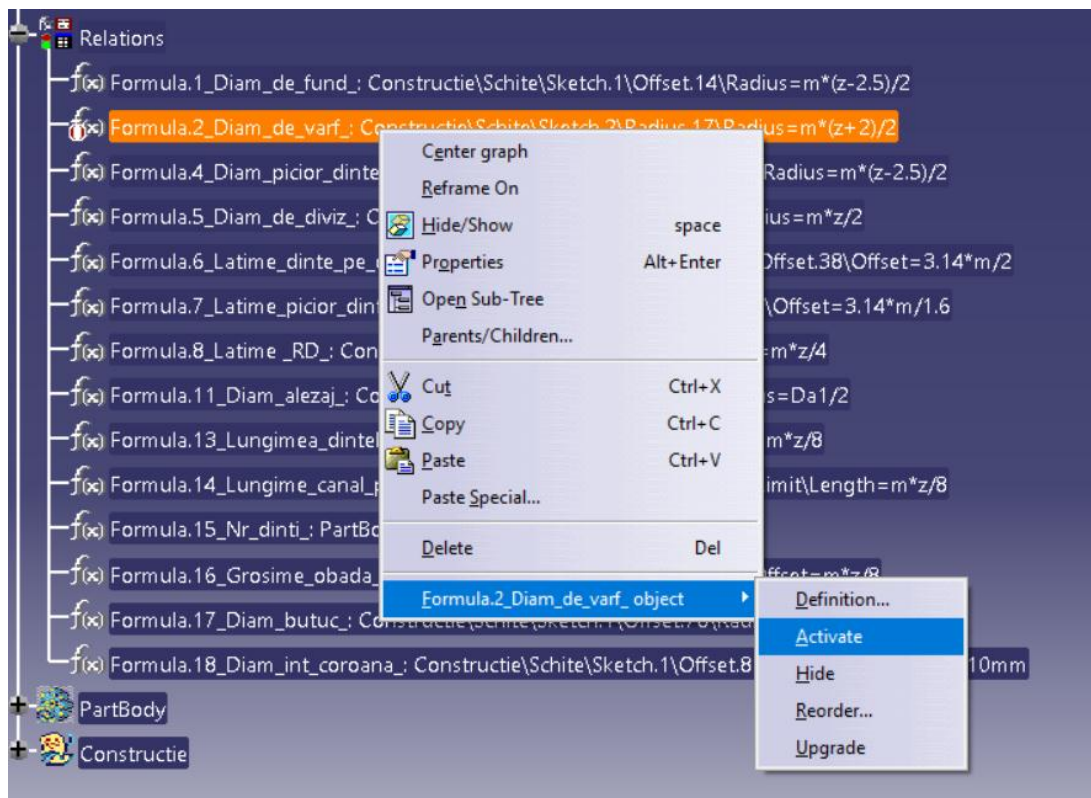
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Fig.2. Activating / deactivating a formula

3. CASE STUDY

For example, a cylindrical gear wheel with straight teeth will be designed by parameterization. First we introduce a series of parameters that are representative of the gear. These are the gear module m and the number of teeth z (fig. 3).
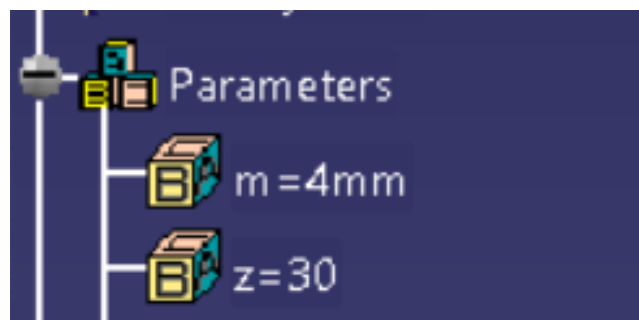

Fig.3. Initial parameters

Then the gear wheel is drawn. During the drawing, in the working drawings, the dimensional constraints will be applied using calculation formulas, these having as working elements only the parameters described previously. Thus the dimensional constraints according to Table 1 will be used.

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Table 1. Formulas applied for dimensional constraints

Formula number	Constraint	Name	Formula
1	Bottom diameter	Constructie\Schite\Sketch.1\Offset.14\Radius	m*(z-2.5)/2
2	Peak diameter	Constructie\Schite\Sketch.2\Radius.17\Radius	m*(z+2)/2
3	The diameter of the foot of the tooth gear	Constructie\Schite\Sketch.2\Radius.34\Radius	m*(z-2.5)/2
4	Diameter of division	Constructie\Schite\Sketch.2\Radius.35\Radius	m*z/2
5	Tooth gear thickness on the diameter of the splitting	Constructie\Schite\Sketch.2\Offset.38\Offset	3.14*m/2
6	Tooth gear thickness	Constructie\Schite\Sketch.2\Offset.39\Offset	3.14*m/16
7	Gearwheel width	Constructie\Schite\Sketch.1\Offset.16\Offset	m*z/4
8	Bore diameter	Constructie\Schite\Sketch.1\Offset.45\Radius	Da1/2
9	The length of the tooth gear	Constructie\Schite\Sketch.1\Offset.45\Radius	m*z/8
10	The length of the keyway	PartBody\Add.1\Dinte\Pad.2\FirstLimit\Length	m*z/8
11	Angular step of the tooth gear	PartBody\CircPattern.2\AngularNumber	z
12	Crown thickness	Constructie\Schite\Sketch.1\Offset.83\Offset	m*z/8
13	Hub diameter	Constructie\Schite\Sketch.1\Offset.78\Radius	Da1/2+10mm
14	Inner diameter of the crown	Constructie\Schite\Sketch.1\Offset.81\Radius	m*(z-2.5)/2 – -10mm

Next to the value of the dimensional constraint, the symbol f(x) appears on the working drawings, which means that the constraint is related to the parameters through a calculation formula (fig. 4).

The whole set of formulas are in the tree, in the order in which they were introduced (fig. 5) [3].

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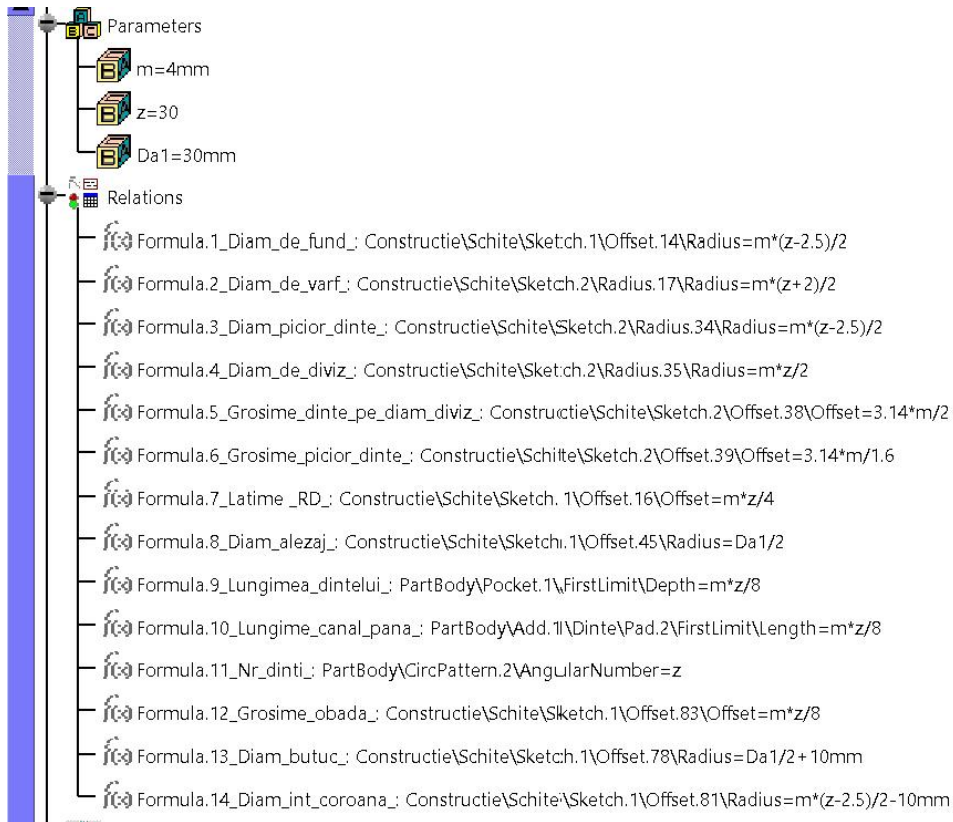Fig.4. Dimensional constraints applied by formulas

Fig.5. Representation of computation formulas in the tree

At this stage of design, the gear wheel is completely defined. For example, we will give values of the parameters **m** and **z** [4, 5].

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Example1: m = 4 mm; z = 30 tooth gear (fig.6).
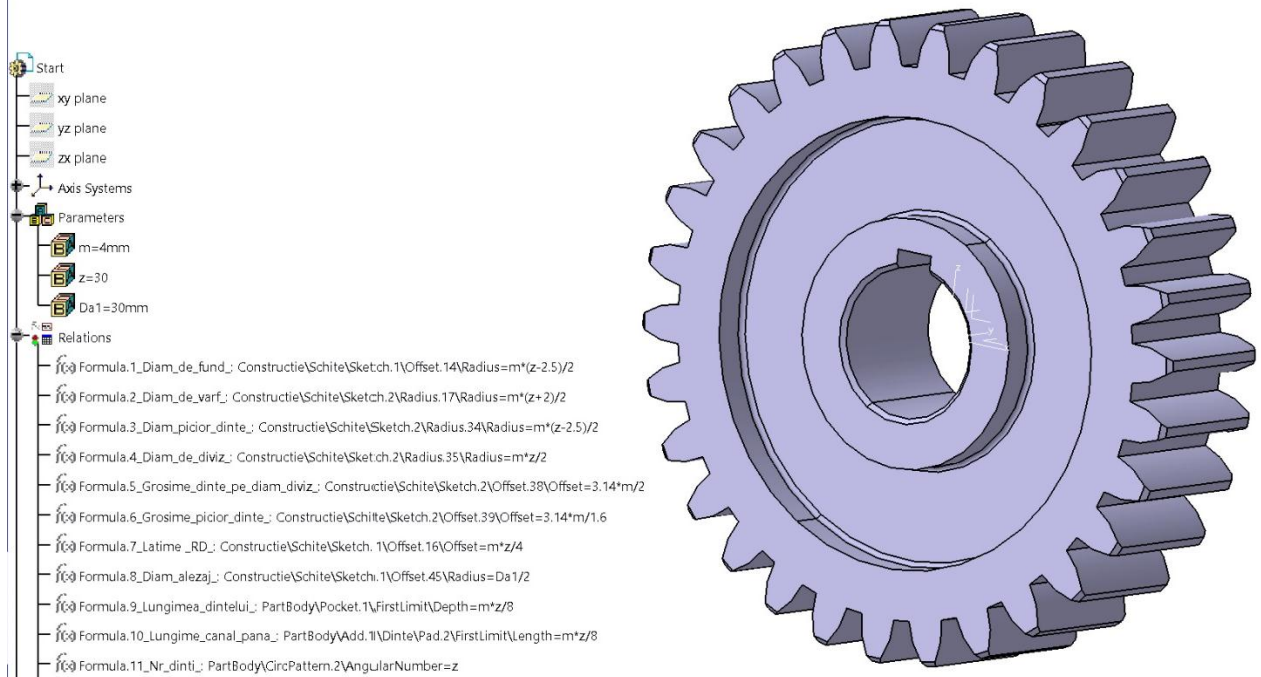
Fig.6. Gear tooth m=4, z=30.
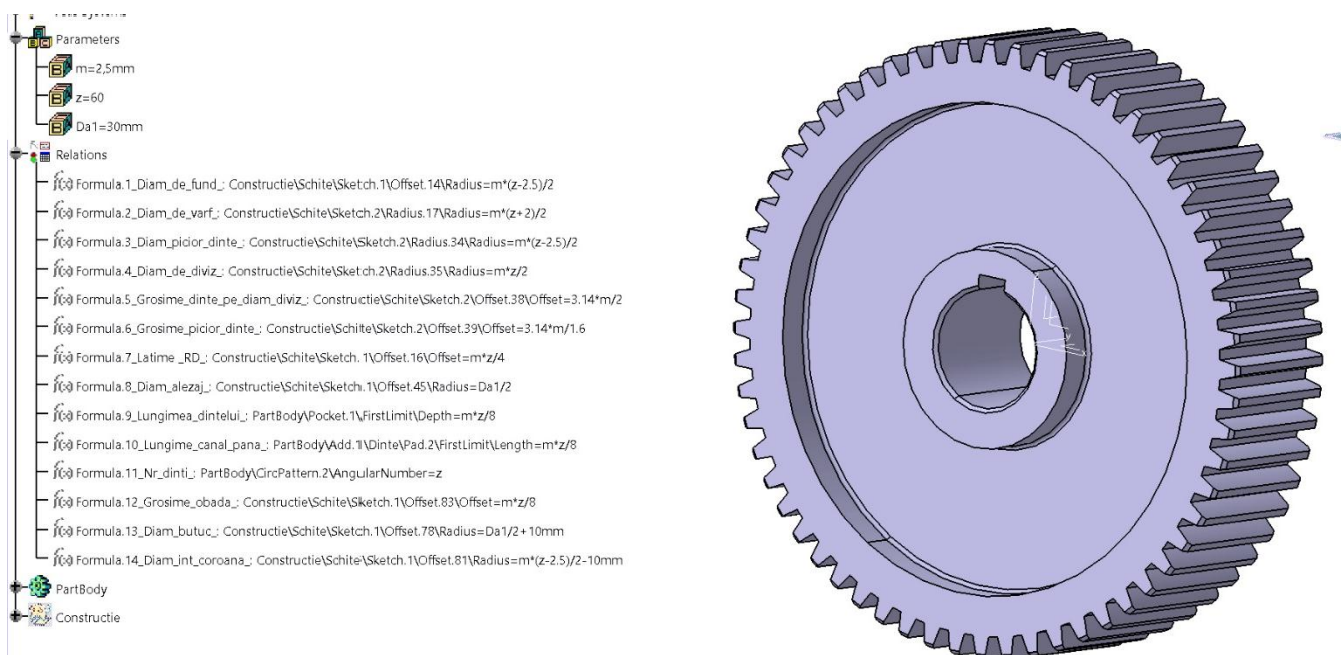
Example 2: m = 2,5 mm; z = 60 tooth gear (fig.7).

Fig.6. Gear tooth m=2,5; z=60

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Example 3: m = 1 mm; z = 100 tooth gear (fig.8).

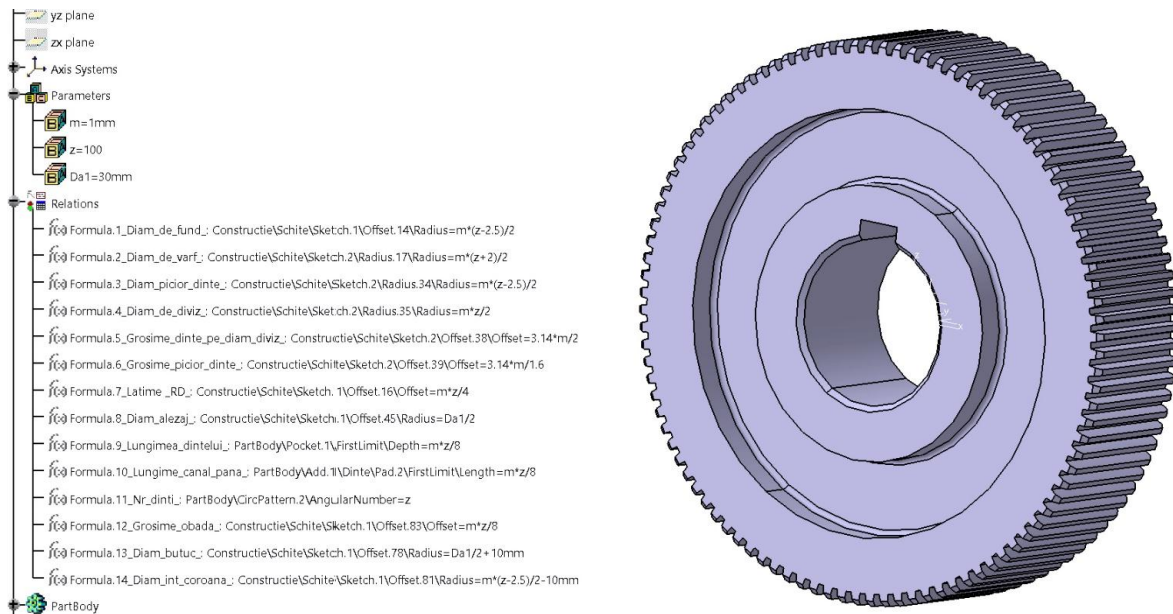
Fig.7. Gear tooth m=1; z=100

4. CONCLUSION

The design activity using the CATIA program often encounters situations in which parts families must be created. They usually have the same constructive shape, but differ in size. Also, in the design of the assemblies it is useful to create dimensional links between two or more components, so that the modification of one of them leads automatically to the modification of the other. This mode of design involves advanced knowledge of modeling, programming and management of parameters and relationships, being called parameterized assisted design.

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Editura "Academica Brâncuşi" , Târgu Jiu, ISSN 1844 – 640X