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ROMANIAN GENERAL
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CEurSIS 2014

MINISTERUL EDUCAŢIEI ŞI CERCETĂRII
UNIVERSITATEA TEHNICĂ
DIN CLUJ-NAPOCA

CENTRUL UNIVERSITAR NORD
DIN BAIA MARE
FACULTATEA DE INGINERIE

CEurSIS 2014

THE INTERNATIONAL CONFERENCE OF THE CARPATHIAN EURO-REGION'S SPECIALISTS IN INDUSTRIAL SYSTEMS

-10th EDITION –

Programme & Abstracts

BAIA MARE
September, 11st – 13rd , 2014

SIMTech
INDUSTRIAL SYSTEMS & TECHNOLOGY MANAGEMENT
departament

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DEPARTAMENTUL DE INGINERIE ELECTRICĂ,
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OBJECTIVE

Our objective is to ensure the conditions for high quality research and professional interactions in the benefit of science, technology, and collaboration.

CONFERENCE SECTIONS

1. Manufacturing Technologies;
2. Machine Elements, Tribology, Maintenance;
3. Product Design;
4. Management of Technology;
5. Electrical and Computer Engineering

GENERAL INFORMATION

The first edition of The International Meeting of the Carpathian Region Specialists, focused on Gears, was organized in 1996 under the coordination of Eugen PAY. The following editions had a continuously grown multidisciplinary approach of the research and technological development activities, so the 2014 meeting comprises a wide spectrum of engineering.
Until now more than 300 specialists from Romania, Poland, Hungary, Germany, Czech Republic, Austria, Serbia, Ukraine and Slovakia, with more than 500 papers, attended the CEurSIS.
The scientific activities it possible to be complete with proposed social activities (visit to History and Art Museum, Vilage Museum and Mineralogy Museum etc.).
THE OFFICIAL LANGUAGE: English for all printed material, presentations and discussion.

EVENT SCHEDULE

THURSDAY, September 11st, 2014

Str. Victoriei 76
19,30 Welcome dinner - Universitary Restaurant

FRIDAY September 12th, 2014

Str. Dr. Victor Babes 62A

09,00 – 10,00 - Registration of participants -

10,00 10,15 - OPENING OF CONFERENCE

10,15 – 11,30 - PLENARY SESSION PRESENTATION

11,30 – 11,45 - Coffee break

11,45 – 13,30 - THEMATIC SESSIONS

14,00 – 15,00 – Lunch at Universitary Restaurant

15,30 – 16,45 - THEMATIC SESSIONS

16.45 17,00 Coffee break

17,00-18,30 - THEMATIC SESSIONS

19,30 - Festive dinner

different types of bullets. The plates that have been shoot had different hardness and the bullets had various speed. Specimens were made from the proper segments of the damaged armor plates. The examinations of metallographic specimens help us to define the changes of the crystals, especially the cracks as well as the size of the effect zone.

Vasile KICSI, Simion HARAGÂŞ, Petru BERCE - **Experimental Research on Lubrication of Aluminum Injection Moulds**

Abstract: *This paper presents a case study carried out in order to improve the efficiency in the use of a classical release agent used to lubricate the injection moulds for aluminum alloys. The study had a commercial goal, namely to replace a release agent produced by a competitor company with a product of Chem Trend.*

Ana MORARESCU, Constantin GEORGESCU - **Worm Design of the Spiral Propeller**

Abstract: *Jet Propulsion systems use axial pumps, with rotor shaped like a propeller, similar to those used in aeronautics. Analysing the functioning of such a system, one may notice the possibility of achieving the jet propulsion by the help of screw pumps. This paper presents a designing algorithm for the mating profiles of the worms used for a spiral propulsion system*

Eleonora POP, Marius ALEXANDRESCU - **Study of Electrical Resistance of Hydrodynamic Journal Bearing**

Abstract: *It is paper investigates a mathematical model developed to determine electrical resistance under different parameters of operation so as to predict bearing performance and safe load carrying capacity. It is evident that bearings working in electric fields undergo very specific processes that affect crater formation, lubricant deterioration and life. In a hydrodynamic journal bearing, the zone of minimum film thickness, load-carrying oil film varies along the circumference of a bearing through its length. This has been found to form a capacitor of varying capacitance between the journal and the bearings dependent on permittivity of the lubricant used, the bearing length, the eccentricity ratio and the clearance ratio. Besides this, load-carrying on oil film offers resistance that depends on operating parameters and resistivity of the lubricant.*

Nicolae UNGUREANU, Radu COTEŢIU - **Lubrication of Industrial Equipment – Part of Maintenance Operations**

Abstract: *This paper presents some theoretical aspects of research related to the operation of lubrication of industrial equipment as part of maintenance operations. Are presented the general aspects of operation, the logical flow chart of work, a general program of work, the specific Ishikawa diagram, and a series of measures proposed for smooth running of lubrication operation.*

Section 3: Product Design

Section Chair: Prof. Vasile Năsui / Prof. Adrian Ghionea

Room: L8/1

Adrian BUDALĂ - **Theoretical Modeling and Experimental Testing of a Coupling with Friction Shoes and Centrifugal Driving (Part I)**

Abstract: *The paper presents some elements of theoretical modeling used in dynamic analysis and numerical simulation of the functional performances of a coupling with friction shoes and adjustable centrifugal driving.*

Adrian BUDALĂ - **Theoretical Modeling and Experimental Testing of a Coupling with Friction Shoes and Centrifugal Driving (Part II)**

Abstract: *The paper presents some elements of experimental testing made with a prototype of a coupling with friction shoes and adjustable centrifugal driving.*

Gabriel DIMIRACHE, Adriana ZAMORA - **The Analysis and Some Results of Experimental Research on the Main Parameters of the Feeding Mechanisms of Type R-C Used for Shifting Coal Cutter –Loaders**

Abstract: *The mechanism studied in the present paper belongs to category of the most recent technical solutions used for the self-shifting of coal cutter loaders during operation and the maneuvers for their positionig on the coal face line. The paper presents the results of a study on the main kinematic and kinetostatic parameters of the feeding mechanism R-C, in which the toothed wheel (R) has teeth profile composed of arcs connected by a segment of the right while the rack (C) has pins with circular profile.*

Liliana DRAGAN - **Research Concerning the Actuation of a Revolute Joint Using Braided Pneumatic Muscles**

Abstract: *The paper is focused on the study of the manner in which a revolute joint can come into operation helped by pneumatic actuators, following the model of the biological muscles. A geometrical, analytical and thermodynamical modeling of the artificial muscle has been done and an experimental stand has been built to allow the validation of the theoretical working characteristics.*

Alexandru Vasile ERDEI Adriana COTEŢIU - **Analysis and Study Case on Sizing a Photovoltaic System (PV System)**

Abstract: *The present paper presents some theoretical aspects regarding the main components, structure and operation of photovoltaic panel. It also shows theoretical and experimental study of the sizing method on a photovoltaic system. The experimental study is based on calculating for the photovoltaic cells placed on the UTCN-CUNBM university building using simulation program, called Polysun 6.1.*

Ionuţ Gabriel GHIONEA - **Considerations about the Design Process of Mechanical Products in Parametric Conditions**

Abstract: *The main intent of the design process of products using parametric modeling is to apply a CAD system flexible and complex enough to determine the engineer to consider a variety of possible variants easily and at low costs.*

Modern CAD systems with Product Life Management (PLM) support provide a virtual working environment in which the product's model is designed and manipulated. Using the geometric modeling capabilities, the engineer conceives, models, adds, deforms and edits the product's parts. The 3D virtual CAD model is accompanied by mathematical description, parameters, constraints and these can be modified and influence the future product design at any modeling phase, can be tested in various ways etc., which is a major advantage over the classic modeling systems. This paper presents some considerations about the design process of mechanical products in a modern parametric manner and includes two case studies showing the many possibilities in creating parametric products using the CATIA v5 software.

Marian PIŢIŞ, Vasile NASUI - **Considerations on Snubber Testing**

Abstract: *Snubbers are restraining devices used to control the movement of pipe and equipment during abnormal dynamic conditions such as earthquakes, shock waves caused by turbine trips, valve discharges, rapid valve closure or accidental rupture of piping. There are*

CONTENT

INVITED PAPERS

Section 1: Manufacturing Technologies

Section 2: Machine Elements, Tribology, Maintenance

Section 3: Product Design

Section 4: Management of Technology

Section 5: Electrical and Computer Engineering

Considerations about the Design Process of Mechanical Products in Parametric Conditions

Ionuţ Gabriel GHIONEA[1]

Abstract: *The main intent of the design process of products using parametric modeling is to apply a CAD system flexible and complex enough to determine the engineer to consider a variety of possible variants easily and at low costs. Modern CAD systems with Product Life Management (PLM) support provide a virtual working environment in which the product's model is designed and manipulated. Using the geometric modeling capabilities, the engineer conceives, models, adds, deforms and edits the product's parts. The 3D virtual CAD model is accompanied by mathematical description, parameters, constraints and these can be modified and influence the future product design at any modeling phase, can be tested in various ways etc., which is a major advantage over the classic modeling systems. This paper presents some considerations about the design process of mechanical products in a modern parametric manner and includes two case studies showing the many possibilities in creating parametric products using the CATIA v5 software.*

Keywords: *parametric modeling of products, design process, CATIA v5*

1 INTRODUCTION

In a modern CAD system, a list of curves and surfaces equations, various points coordinates are live updated as the virtual model is conceived, visualized and manipulated on screen by the engineer [6].

The model includes informations about surface connectivity (on how the surfaces from a more complex shape are joined, which surfaces are adjacent to each other by which curves, how each surface was obtained etc.) which is very important because all these informations are useful in many applications. As an example, a NC program to generate and to validate the tool paths on a milling machine will certainly use all the data available to check the gaps or twists in a specific surface where the machining operation is impossible).

In the design process, the engineer includes in the model description a list of equations, different attributes and parameters, constraints and relations between the product's parts in assembly.

David Weisberg, editor of the Engineering Automation Report, wrote about the parametric modeling [8]: "The problem with a pure parametric design technique that is based upon regenerating the model from its history tree is that, as geometry is added, it is dependent upon geometry created earlier. This methodology has been described as a parent/child relationship, except that it can be many levels deep. If a parent level element is deleted or changed in certain ways, it can have unexpected effects on child-level elements. In some cases the user was forced to totally recreate the model. Some people described parametric designing to be more similar to programming than to conventional engineering design."

2 PARAMETRIC DESIGN CONCEPTS

Conceiving parts and assembling products in a modern CAD system is a creative activity that is based on features: modeling and constraining tools, assembly and simulation possibilities, kinematic animations etc. They are very flexible and powerful, interlinked with each other, but there are no universally applicable methods for checking the validity of features. In many cases it is up to the engineer which defines or uses a feature to specify what is valid or not for it's applied result. A feature may be suitable for a part or a group of parts in a subassembly, and not for the whole assembly because of disrespecting important constraints at a upper/parent level. Typical checks that need to be done are: compatibility of parent/dependent features, limits on dimensions and inadvertent interference with others.

In the past, using classic modeling systems, in creating a parametric part of a product, the engineers start from scratch, model features sequentially, one after another, spending time working on the most recent feature without needing to go back to edit previous features. Today, having modern CAD systems, in most cases they don't build models from scratch, but use knowledge data from previous projects, periodically go back to adjust earlier features from time to time. In that process, they'll identify and process most of the dependencies, transforming them in parameters, relations, constraints etc., saved therefore as knowledge for future projects [3].

The common benefit in parametric modeling is to use advanced construction techniques in modeling and feature recognition in editing phases. It's taken almost 35 years of industry research to get the options where a user is clicking on a face of a model and the system recognize that he is pointing to a feature with parametric values to edit the geometry.

The parametric modeling paradigm enables engineers to create new concepts based on previous designs. Users can start with a 2D conceptual model and easily use it as the basis of a 3D model. Parametric modeling tools also enable concept designers to use all types of available data, including ideas, 2D drawings, sketches, surfaces, single parts, or entire assemblies and products. Because these modeling tools offer interoperability in a parametric manner, almost no model re-creation is necessary, giving designers more time to explore a wider range of design alternatives [1].

Special and common products, such as automotive/aerospace parts, jewelry, toys, sporting goods, electronics, various devices etc. include parametric surface and solid modeling. Within these product categories there are many items that incorporate complex/free-form shapes presenting some serious modeling challenges [4].

3 DESIGN SOURCES, CONCEPTS AND PROCESS

There are two general types of design: empirical design (better known as conceptual design) and scientific design (the engineer uses principles of physics, mathematics, mechanics and/or other sciences in the new or revised design of products in such a way to function under imposed conditions). Almost all technical design is a successful combination of conceptual and scientific design. Therefore, an engineer must have both adequate engineering and scientific knowledge.

New ideas or design concepts are born initially in the mind of the engineer. In order to capture, preserve and develop these ideas, the engineer use modern CAD systems, with solid and surface modeling capabilities.

The concept of a successful product evolves from version to version, with more features and functions, redesigned shapes etc. To identify and model all these, the engineer finds in his advantage to share his ideas with others and begin working in a team effort. A design team include members familiar with knowledge of materials, production, management, mechanics etc.

In industry, a project for a product is always a team effort long before the product is produced and launched on the market. Every team member, having his own vision, may impose parameters and constraints in every phase of the project, so the CAD software must easily handle parametric design. Each time a parameter is modified, many others are disposed to be changed according to the relations established, keeping the single part or the whole product in the right shape. So, these relations must be first mathematically correct and then must respect the product's design specifications [3].

The design process is the ability to combine ideas, scientific principles, available resources, existing standards, patents and products into a solution to a problem. This ability to solve problems in design is the result of an organized approach to the problem known as the design process. The design process leading to analysis of needs, conception, modeling, virtual testing, manufacturing, assembly, real testing, maintenance, service and to many other activities necessary for a successful product is composed of several phases.

Although some industrial groups identify them in their own specific way, using various procedures, five stages can be followed: identification of problem, emergence of concepts and ideas, choice of the correct solutions, creation of models and prototypes then simulations of their behaviour in virtual environments to validate the solutions, production. If a particular stage proves to be unsatisfactory, it is often necessary to return the whole project to a previous stage and repeat the procedure. That's why it is so important that the 3D model of a part or assembly have to be created in a modern parametric manner. Every function of a product generates many parameters and constraints between its components. The initial sketches of the design are followed by a study of suitable materials, mechanical correctitude, assembly and kinematic possibilities. Some of these problems are solved graphically with the help of parametric CAD systems [7].

Through them, the design engineer has the tool to model, visualize, assembly, simulate, analyze the continuity of surfaces, interference between the parts, determine paths of motions, areas of maximum stress, possibilities of manufacture certain shapes of parts etc. To reach these simulations, all the parts are carefully designed using parameters. Costs are constantly kept in mind, because no matter how well the product looks and performs in laboratory tests, it must be sell for a profit on a competitive market.

During the design process, a great attention is placed on what was done before in other projects. Previous experience provides a sense of proportion using the correct set of parameters, adding new parameters, relations and constraints, permitting a new design and/or a significant change in product's functions and technical specifications, based on new added features.

A modern approach is to create standard parts, organized in parametric families of parts, that are used wherever possible for reasons of cost reduction, ease of assembly, maintenance and serviceability. The modern idea in a product design approach involves the improvement, development and evolution of an existing product or it's redesign from a previous version in which many details are similar. Some of the product's parts require changes of design, so the parametric modeling is a must, especially when a 3D printed model is obtained for evaluation.

Later, a full-size working model made to the latest specifications is known as a prototype. It is tested, modified where necessary and the results noted in the model revisions, specifying which parameters need to be edited. If the prototype proves to have some improvements to be made or mistakes to be corrected, it is necessary to return to the CAD model and repeat the procedures, any parameter modification leads to an update of the model [2].

4 CASE STUDIES IN PARAMETRIC DESIGN

An important argument to CATIA v5's success is its parametric feature-based surface and solid modeling approach in obtaining 3D models.

Using this software and having the right skills, the engineer is able to conceive and build a virtual CAD model of a complex product in a parametric manner, that leads to dispose of many various parameters to change influencing the product's size, dimensions, functionality, aspect etc. Thus, the engineer gain a great leap forward to rapidly create and explore design alternatives for parts and assemblies, especially in the context of complex products with many variants [4], [8].

In these two case studies there are presented some CATIA v5 parametric applications from simple to complex in order to unveil the working methodology in each case in a very intuitive manner.

The first case study uses design tables, *Microsoft Excel* files linked to *CATPart* files. The parameters used in CAD modeling are stored in sets of values, each set being an unique version of a part or assembly.

Figure 1 shows a 3D model of a simple part, a bridle, and its parameters. The primary values in creating the model are, in mm: L=45, B=22, h=16, N=18, d=5, K=9, m=6, P=15, f=7, b=8, s=5, c=12, d1=6.6, d2=11, t=6.8, r=1. These values are, in fact, one set of dimensions from a family of parts.

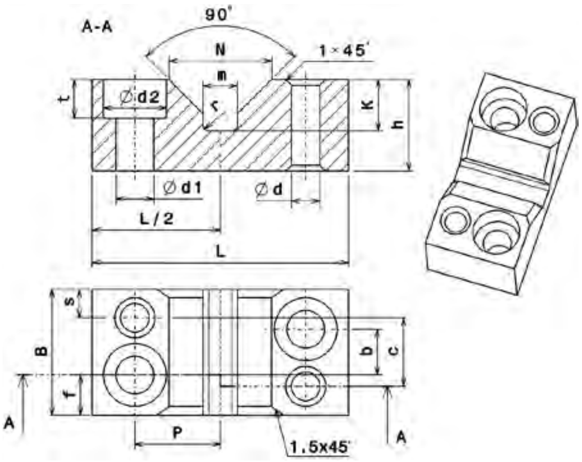

Fig. 1. Bridle model with parameters

The part creation starts with a sketch and insertion of parameters of type length, offset, angle. These parameters are intrinsic with the part because the user needs them to create several sketches and solid features (*Pad*, *Pocket*, *Hole*, *Chamfer*). The initial names of parameters are too long to be easily remembered so their renaming is recommended. This method is a good practice for the whole part and all the parameters.

An *Excel xls* file is then created as the first set of values. The user edits the *xls* file adding more sets of dimensions and specifying the measurement units for each one.

	A	B	C	D	E	F	G	H	I	J
1	L (mm)	B (mm)	h (mm)	N (mm)	d/2 (mm)	K (mm)	m (mm)	P (mm)	f (mm)	b (mm)
2	45	22	16	18	2.5	9	6	15	7	8
3	55	22	20	24	2.5	11	8	20	7	8
4	70	28	25	30	3	14	12	25	10	8
5	85	28	32	37	3	18	16	32.5	10	8

Fig. 2. Values of parameters organised in sets

Figure 2 presents a fragment of a design table with four sets of values for the part parameters. The user associates each column of the table with the respective parameter from the part (Fig. 3).

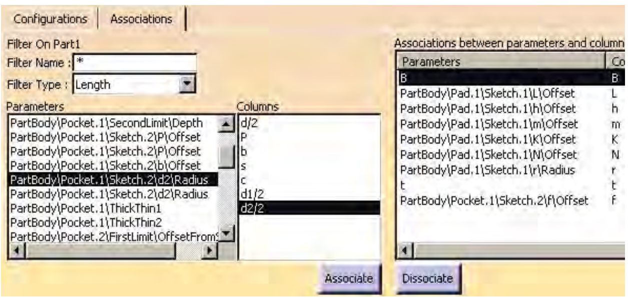

Fig. 3. Association of parameters with columns in sets

Besides the design table, the user may add different formulas, especially for those parameters resulted from the part symmetry. Thus, it's easier to make equal two symmetrical parameters than add one more column in the design table. The user is ensured that the part will be correctly modified at the change of just one parameter instead of two. Editing one parameter and forgetting the other one with the old value will rend the part to become inconsistent.

Figure 4 contains the bridle's versions for each set of parameters. This is a family of parts.

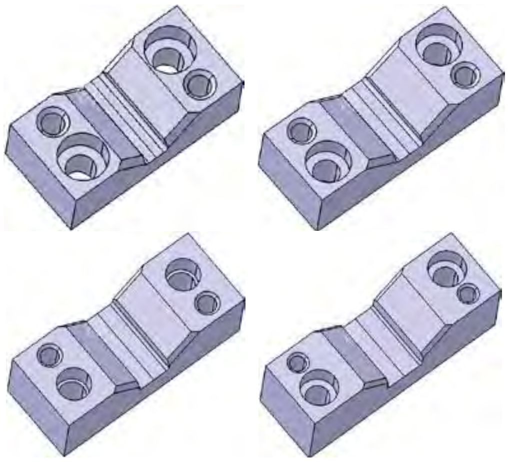

Fig. 4. Parametric versions of the bridle

From the figure 4 it can be observed that the parts in family have similar aspect but different configurations (for lengths, holes, chamfers etc.).

In the second case study, it was considered a simple assembly of few parts from a more complex fixture device (Fig. 5), used in machining by milling of rectangular parts.

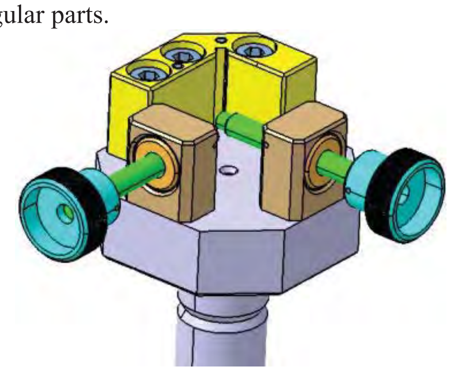

Fig. 5. Parametric fixture device assembly

The subassembly in figure 6 is extracted from the whole assembly and it is composed of four parts: support, bush, fixing screw and a pin.

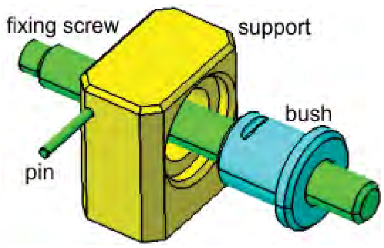

Fig. 6. Subassembly for parameterization

The parameterization of this subassembly consists in the creation of a *Visual Basic* code, a reaction, activated when the user changes the thread dimension of the bush's central hole. The reaction will verify and allow only the choice of the M8 and M10×1 threads, any attempt by the user to overcome these two values being unaccepted. A warning message will be displayed followed by an automatic adjustment of the hole thread in M8, as the default value.

In the *Reaction* window (Fig. 7) the user selects a parameter *PartBody\Hole.1\ThreadDescription* to be the one that activates the reaction when it is changed.

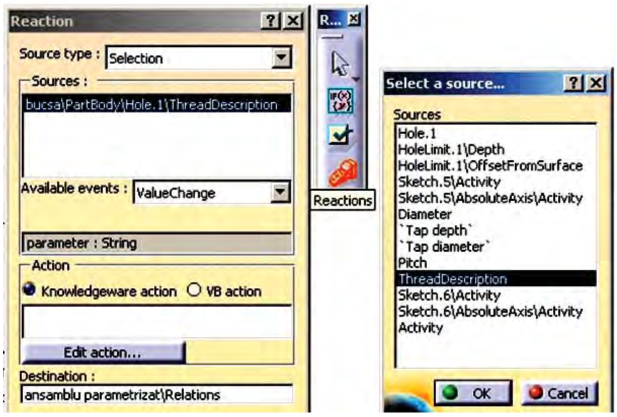

Fig. 7. Choice of the activation parameter of the reaction

Pressing the *Edit action button*, the user may insert the VB code, as shown in figure 8. Considering the complexity of the four parts, however, the number of parameters is relative large, and choosing the right one is simplified by using the three columns acting as filters.

Between the parameters are established many relations modifying some of the parts' sizes for the two cases of the user's selection: thread M8 or thread M10×1. These parameters are of type *string*, but the activation of the reaction leads to changing many other parameters of type *Integer* and *Length*.

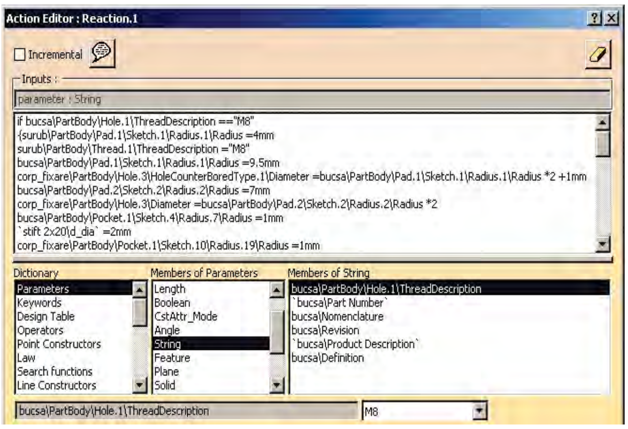

Fig. 8. Inserting the Visual Basic code of the reaction

The VB code allows the user to write some sequences for displaying messages boxes informing about his selection and how the parts are modified.

To simplify all this process of writing relations with parameters, the user may rename them in an explicit way for a better understanding later by other users.

5 CONCLUSIONS

As resulting from the two study cases presented above, the parameterization of parts and of assemblies is a complex activity in the design process of modern products that involves relations and constraints between parameters of different types: string, real, integer, angle, length, Boolean etc. The user must have a good CAD and *Visual Basic* syntax knowledge to handle them in a logical manner in order to obtain various constructive changes based on different criteria linked to functions, dimensions, technology, cost reduction, profit etc.

Acknowledgement
The work has been funded by the Sectoral Operational Programme Human Resources Development 2007-2013 of the Ministry of European Funds through the Financial Agreement POSDRU/159/1.5/S/.138963.

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