Software for CAD Modelling And Design of Monoaxis Shafts

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Abstract: - The proposed software is an original product, realized as a development of the constructive and technological parametrical design process with the supplementary uses of CATIAV5R9 and database software. The modelling of the product consist in describing it, not only from a geometrical point of view but also in relation with a number of characteristics related to its manufacturing. The paper proposes a parameterized modelling for the parts from shafts family, using CATIA modifier and the databases with geometrical characteristics. In order to realize this, a CAD modelling based on technological constructive entities was used. Parameterized modelling is the starting point for automatic design of the execution technology.

Key words: - CAD, CAM, shafts design, shafts technology, revolution surfaces, processing steps.

1 Introduction

The CAD/CAM systems are characterized by the use of complex software that can resolve the difficult problems related with the constructive, and technological design. The modelling of a product using CAD methods is the most usual approach in an integrated system CIM. The CAD module represents fundamental elements in CIM, being the starting point for automatic design of the execution technology [9]. The proposed system is based on the technological and constructive entity concept that allows to define models in a parametric way.

The proposed CAD system follows the classical steps of the automatic constructive design. These are: the design of the system outline, the establishing of the manufactured piece family, the computer aided constructive design of the code-piece and ultimately, the achievement of a specialised software for the particularisation of the code-part in concrete parts, taking the functional necessities of the part component entities into account.

The method is illustrated by examples at the constructive automatic design of the monoaxis shafts and it is put into practice on the concrete parts.

The steps were realized from informational point of view, with the help of parametrical analogical design using CATIA V5, Excel and a software that access database with functional constructive elements; the shafts go into assembling with these elements with a view to an efficient automatic design.

Object orientation is proving to be an enabling technology that makes it easy to construct and maintain complex system from individual components [1]. Object-relational mapping products integrate object programming language capabilities with relational databases managed by Oracle, DB2, Sybase, and other relational database managers. Object-relational mapping products are designed to work well with object programming languages such as C#, C++, and Java[3].

Object-oriented database technology is a marriage of object-oriented programming and database technologies [2]. Perhaps the most significant characteristic of object-oriented database technology is that it combines object-oriented programming with database technology to provide an integrated application development system [5]. There are many advantages to including the definition of operations with the definition of data [4]. First, the defined operations apply ubiquitously and are not dependent on the particular database application running at the moment. Second, the data types can be extended to support complex data such as multi-media by defining new object classes that have operations to support the new kinds of information [6].

2 Code-part design

The Shaft’s parametric modelling based on technological constructive entities as CAD solution has great advantages on productivity growing of the constructive and technological design [10]. We
consider the technological constructive entity as a geometrical shape and an assembly of dimensional, roughness and precision (tolerances) specifications. In order to obtain this entity there is one or more technological possibilities for fabrication [7]. The graphic model of the shaft is obtained using these entities in a convenient way and the real object results by assembling the selected entities [8].

The first step is the design of the concrete object from the representative family (shafts) and the defining of all dimensional, roughness and precision parameters that composed the generalized shaft. The principle that was used at the defining of the technological entities was: a maxim number of shafts to be design with a minim number of entities.

The 3D model was realized with CATIA V5R9 software and in order to obtain a great range of values, database table were created. The reference system was common for all the entities, situated on the shift’s axis and on the frontal left side.

The main entities used at shift’s family defining are:
- cylindrical exterior surface (we call it section);
- chamfers and connect radius related with the sections;
- circular trenches obtained by grooving;
- wedge trenches: parallel wedge, oblique wedge, disk wedge;
- exterior thread surfaces;
- milled surfaces: planed surfaces parallel with the axis, planned surfaces radial to the axis, pocket milling;
- bore holes: axial, parallel to the axis, radial, different position in space;
- thread bore holes;
- grooved cylindrical surfaces;
- exterior cogwheel, etc.

3 Software for Monoaxis Shafts
Program’s main screen window is an active window similar to any active Windows environment. Here the user can perform any Windows specific actions such as redefining the size of the window by dragging the sides or the corner.

From the functional point of view, the program running involves a sequence of specific windows that will be described in the following part of the paper. In the first active window (fig.1) by pressing on one button the user can choose an active workbench (fig.2). In this case, the user clicked on the button named “Cylindrical exterior surface”.

The design of the shaft principal configuration is realised using the interface presented in figure 2. Initially, one must introduce the number of shaft steps/sections (i.e. a cylindrical exterior surface with the same diameter and a given length) and the minimum possible diameter according to the strength calculation.

After that, one introduces the functional role for each shaft steps. Depending on that, the databases, according to the machine elements that are assembled on the respective step, are opened.

In the example shown in figure 3, the functional role of the shaft step is “shafts for rolling-bearing” and the rolling-bearing is a radial ball bearing. After the “Radial ball-bearing” database is opened, one can choose the desired rolling-bearing and taking its dimensions and the imposed precision conditions into account, it is possible to introduce...
the dimensions, the tolerances and the roughness of the shaft steps, in keeping with the interface from the figure 2.

From this point the program will run for each section, so that the next section will be approached only after the previous one was completed.

After the introducing of the revolution principal surfaces, it is possible to introduce the auxiliary surfaces that have an important role from the functional point of view. The buttons according to these surfaces can be access from the principal menu (fig.1).

From the point of view of the data introducing algorithm, the succession of stages is similarly to that from the previous example: introducing of the entity functional role; opening the database according to the machine elements that are assembled on the entity; choosing the desired part and the extraction of the necessary dimensions and precision elements. One must complete the fields: dimensions, tolerances, roughness and position relative to the absolute coordinate system of the shaft.

Figure 4 shows the interface for the design of the entity “Circular channel for safety ring”.

All the mentioned entities are completely defined from the following point of view: shape, dimension, precision and surface quality (roughness). The entities are defined in a parametrical way in a coordinate system that is commonly for all the entities.

When one clicks on the “Export to CATIA” button, our software transforms the database table in an Excel worksheet (fig.5).
The program allows the achievement of an Excel table with dimensional and precision elements for different shaft configuration, each line of the table corresponding at a designed shaft.

The code source that defines the object for transformation is:

```c
#include msword.h
#include excel9.h
#define CR chr(13)
close all

oExcel = createobject( 'excel.application')
if type( "oExcel") == "O"
oExcel.visible = .f.
oExcel.windowstate = wdWindowStateMinimize
lcelimDBC = "Monoaxisshafts.dbc"
if ! file("Monoaxisshafts.dbc")
lcbazaDBC = getfile( 'DBC', lcelimDBC )
endif
if ! empty( lcelimDBC )
open database Monoaxisshafts.dbc
 oExcel.Workbooks.add()
 oWorkbook = oExcel.Worksheets(1)
 USE mgeom
 oworkbook.cells(1,1)='Nr curent'
 oworkbook.cells(1,2)='r1'
 oworkbook.cells(1,3)='l1'
 oworkbook.cells(1,4)='Ra1'
 oworkbook.cells(1,5)='r2'
 oworkbook.cells(1,6)='l2'
 oworkbook.cells(1,7)='Ra2'
…………………………………………………..
with oWorkbook
 i = 1
 scan
 i = i + 1
 .Cells( i, 1) = Nrcurent
 .Cells( i, 2) = r1
 .Cells( i, 3) = l1
 .Cells( i, 4) = Ra1
 .Cells( i, 5) = r2
 .Cells( i, 6) = l2
 .Cells( i, 7) = Ra2
…………………………….
endscan
endwith
```

After this, the CATIA software is open. When the information from the database are read, the configuration of the shafts changes accordingly and some new shapes appear as it is presented in figure 6.

Using the system presented before, the object from figure 7 was obtained.

3 CONCLUSIONS

The main contributions of the authors are the following: structural analysis and synthesis of the information relating to constructive design of the monoaxis shafts; carry out the analysis stage and design of the main logical steps in the procedure, that are very important steps for the development of the software; achievement of a software using relational objectual database Visual FoxPro - as the programming language.

We consider this software as an important step toward an integrated Windows - basic software for computer aided design. The advantages of the CAD method are surely: higher productivity of the
constructive design, facility and elegance of the design process, smaller manufacturing costs by using analogical design and group technologies.

This paper presents a solution for CAD databases. The drawings are stored in RODBMS or ODBMS threw their technological dates. When designer engineer wants to view a drawing, he finds the technological dates of the drawing in database and generates interface file for CAD software. After this, the drawing is automatic generated.

And finally, we appreciate that the software has an important didactical role because it can be used as a design tutorial for the constructive design of the monoaxis shafts.

References: