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An automation of design and modelling tasks in NX Siemens environment with original software – generator module

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Abstract. Nowadays the design constructional process is almost exclusively aided with CAD/CAE/CAM systems. It is evaluated that nearly 80% of design activities have a routine nature. These design routine tasks are highly susceptible to automation. Design automation is usually made with API tools which allow building original software responsible for adding different engineering activities. In this paper the original software worked out in order to automate engineering tasks at the stage of a product geometrical shape design is presented. The elaborated software works exclusively in NX Siemens CAD/CAM/CAE environment and was prepared in Microsoft Visual Studio with application of the .NET technology and NX SNAP library. The software functionality allows designing and modelling of spur and helicoidal involute gears. Moreover, it is possible to estimate relative manufacturing costs. With the Generator module it is possible to design and model both standard and non-standard gear wheels. The main advantage of the model generated in such a way is its better representation of an involute curve in comparison to those which are drawn in specialized standard CAD systems tools. It comes from fact that usually in CAD systems an involute curve is drawn by 3 points that respond to points located on the addendum circle, the reference diameter of a gear and the base circle respectively. In the Generator module the involute curve is drawn by 11 involute points which are located on and upper the base and the addendum circles therefore 3D gear wheels models are highly accurate. Application of the Generator module makes the modelling process very rapid so that the gear wheel modelling time is reduced to several seconds. During the conducted research the analysis of differences between standard 3 points and 11 points involutes was made. The results and conclusions drawn upon analysis are shown in details.

1. Introduction

Formerly a designed product came into being first of all and mainly in a designer imagination. In spite of a continuous technological progress this stage of the design-constructional process has not changed up today. Technological progress and continuous development of computer systems have led to creation of the full range of software which makes the process of design modelling in 3D space easy and efficient. In CAD software a 3D product model is represented by mathematical notation which unequivocally describes its design features. Base on this description a designer is able to prepare product views, cross-sections, make kinematic, dynamic calculation etc. Some of these activities have a routine character. In a daily practice of design and technological bureaus activity there could be
observed problems with large time consumption of routine tasks performing, which in principle do not
add an added value to a product [2,3,4].

On the other hand CAD/CAM/CAE systems users very seldom try to work out and next use their
own software solutions which allow automating routine design tasks. These tasks, that taking into
account their character, are especially prone to automation. In industry it is possible and highly
recommended to apply software automation for this kind of design department activity which could be
characterized by unchangeable, generally known algorithm for exactly defined input data and
precisely defined result. For instance calculation that supports design and geometric modelling
processes of gear wheels, shafts, sleeves.

Software adding and supporting of the design-constructional process, especially geometric
modelling functions, in contemporary integrated CAD/CAM/CAE systems, is possible thanks to
making available by software producers of OPEN/API libraries. In practice, it means the ability to
create original software which preforms various tasks, in given system environment, specified by
program author. These software applications could be made with high level programming languages
such as C/C++, C#, Java, Object Pascal or Visual Basic in different programing environments for
example Microsoft Visual Studio, Embaracadero RAD Studio. This functionality is offered inter alia
by the following exemplary systems SIEMENS PLM NX, CATIA, ProEngineer, Inventor or Solid
Works.

The original software developing process could be realized with the following methods:
- manual, in which a program structure is made by a programmer on their own without application
  of a system internal journaling mechanism. The journaling mechanism allows on automatic recording
  of actions performed by a system user during an element design geometry modelling. The manual
  method requires form a programmer to be very good at knowledge of a system capability to
  automation of considered routine task as well as excellent knowledge of a chosen programming
  language and programming environment;
- mixed, a part of a program structure is made with the journaling technique – a journal script is
  being recorded. The program script contains the subsequent geometric modelling steps; a reaming
  program code is made directly by a programmer.

The benefits of working out and using of original software, made by their own, in daily practice of
design and technological departments could be as follows [1,4]:
- improvement and acceleration of the design-constructional process,
- enable of fast making of part families,
- reduce the risk of a design mistake making at the stage of a product design modelling stage by
  introducing of procedures of an automatic dimension selection for standard parts from relevant
  standards,
- constant supervision of a product being modelled according to, for instance the criterion of fatigue,
  dynamic properties etc.,
- provide possibility of a design optimization,
- automation of process planning,
- provide possibility of fast making of alternative machining routs,
- automatic generation of CNC programs,
- design knowledge storing.

Irrespective of the chosen CAD/CAM/CEA system environment the stage of original software
development should be always preceded by an analysis of programming tools available in the system.
This analysis and subsequent decision play a key role in the whole process of software development
taking into consideration fact that particular programming tools have different abilities so programmer
should find compromise between tools abilities and software development cost [3].

2. Problem formulation
Basis on a critical review and analysis conducted in the context of possibility of routine design and
modelling tasks automation it was decided to work out original software supporting processes of gear
wheel design and geometrical modelling. According to recommendations included in the introductory paragraph the software development stage was preceded by CAD/CAM/CAE system selection and a review of programming tools available in it. As a software development environment Siemens NX was selected. Siemens NX offers the following programming tools:

- NX Open it is a collection of APIs (application programming interfaces) that allows creating software applications for NX through an open architecture using high-level programming languages such as C/C++, Visual Basic, C#, and Java. It could be used for automation complex and routine-repetitive tasks;

- Knowledge fusion which is an interpreter, object-oriented language that allows adding engineering knowledge to a part element by both design and geometric creating rules. The knowledge fusion language is declarative, rather than procedural one, which means that, in general, the rules are only evaluated when referenced or demanded. In KF it is possible to access external knowledge bases, such as databases or spreadsheets;

- SNAP (Simple NX Application Programming) is an easy-to-learn programming tool intended for mechanical designers and other typical NX users — not necessarily programmers. It can speed up work by automating simple routine tasks. SNAP is based on the Visual Basic (VB.Net) language and can be used with the NX Journal Editor or with IDEs, such as Visual Studio.

After a detailed analysis as custom-original software development tools SNAP programming tool was chosen as a compromise between language abilities and development cost.

3. Initial program assumptions and the program structure

The program is designed as a tool that supports the processes of gear wheels parameters calculation, geometrical modelling and relative manufacturing cost evaluation. The range of designed gear wheels is limited to spur and helical gears. The original program consists of the two modules it is the generator module and cost module. The generator module is a main subject of this paper. In the figure 1 the general structure of the worked out program is presented.

![Figure 1. The original program structure.](image)

A program user during the design session is obliged to introduce via program interface the following geometric input data: a teeth kind, wheel design shape (A or B), tooth module (m), a teeth number (z), a wheel rim width (B), a wheel internal diameter/mating shaft diameter (d₀), a tooth depth coefficient (y), a pressure angle (α), a helix angle (β), helix hand (right or left). Basis on the introduced
input parameters the generator module calculates the rest of wheel parameters with gear wheels design parameters calculation algorithms. The calculated parameters are as follows (see the figure 2): a pitch diameter \(D\), a base diameter \(D_b\), an addendum/outside diameter \(D_a\), a root diameter \(D_f\), a tooth depth \(H\), addendum \(H_a\), dedendum \(H_f\). The slot parameters, rim geometrical shape and corresponding dimension set are selected according to appropriate standard.

**Figure 2.** The generator model user interface.

**Figure 3.** The gear wheel example a) the real part model, b) model generated in the generator module: for the following design parameters \(m = 2, z = 15, y = 1\).

**4. Verification of the generator module correctness**

The correctness of the generator module was carried out in the two directions. The first test was aimed at comparison of wheel design parameters calculated in the module with technical data given by...
industrial producers. Basis on this comparison it could be stated that these parameters are identical what proved the correctness of the generator calculation algorithms.

The aim of the second test was to compare geometry of the two involutes, it is the involute drawn by generator module with the involute drawn in a standard way. By the standard approach to involute drawing is understood this way in which the curve is drawn by the three points. These points are usually located at addendum, pitch and dedendum circles. This solution is commonly used in most CAD systems equipped with built-in gear wheel generator. In the presented research it was decided to support the involute curve being drawn on eleven points. This should assure better quality of the involute. Having in mind the test aim the default NX modelling preferences for vector graphics were changed. The distance tolerance value was changed from 0.0254 mm to 0.001 mm and angle tolerance form 0.5 mm to 0.01 mm. Involute curves were drawn in NX sketch environment for the following input parameters: modulus $m = 1$ mm, teeth number $z = 20$ and tooth depth coefficient $y = 1$. The curve drawn in a standard way it is by 3 points was denoted with the red colour whilst this drawn by 11 point with the green one (see: the figure 4).

![Figure 4](image)

**Figure 4.** The involutes being compared.

In the figure 4 the distinct difference between involutes profiles could be seen. In their lower part the involutes deviation is smaller than in the upper part in which the maximum deviations equals to 0.1 mm. It is expected that for bigger tooth modules this deviation would achieve greater values. In the figure the 4 the sides views of teeth for both involute profiles are show. The figure 5 analysis indicated that 11 pts. based tooth width measured at its base is wider than 3 pts. based.

In order to make a more precise analysis of involute profile shape influence on bending stress the FEM analysis in Siemens NX Design Simulation module was carried out. In the figure 6 results of the displacement analysis were shown. In case of the displacement analysis there is not difference between profile $a$ and $b$.

The maximum displacement value equals to 0.001 mm for normal load 500 N applied to the tooth flank. The figure 7 presents results achieved for bending stress analysis. In case of the bending stress analysis difference between maximum (3 pts. based profile) and minimum value (11 pts. based profile) of bending stress equals to 2 MPa which seems to be not so high. In considered case lower value of bending stress for 11 pts. based involute profile could be explained by greater width of tooth measured at its base. Moreover on the basis of the stress distribution analysis it was found that in case of 3 pts.
Based involute the area with increased stress level is higher than 11 pts. based curve. Presumably, it would have influence on fatigue strength especially for gear wheels of large dimensions.

Figure 6. Displacement analysis results a) 11 pts. based, b) 3 pts. Based.

Figure 7. Bending stress analysis results a) 11 pts. based, b) 3 pts. Based.

5. Conclusions
Basis on the conducted research and work it could be found that original software facilitates the design-constructional process. In some cases this kind of software would give better results the CAD/CAM/CAE integrated systems built-in tools. In the considered work the gear wheel modelling time was reduce to particular minutes in opposite to not assisted gear wheel modelling process, realized in sketch module environment, in which modelling time is counted rather in hours. Moreover the software user has greater influence on final product model shape. This allows manufacturing product of better quality which very well meet a client requirements.

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