IMPROVING THE QUALITY OF MACHINE DESIGN, BY USING THE DATABASE OF STANDARDIZED ELEMENTS IN THE NX SYSTEM

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ABSTRACT

The following study is an attempt to create an additional application, which would be possible to use in practice. The methodology of creation and range, as well as the mechanism of running of the standard elements library for NX system [4], created by Siemens, was herein explained. Having in mind the time needed for appropriate modelling of a part being a subject of a particular norm, one decided to take into consideration only selected elements. Over one hundred and sixty the most commonly used and simultaneously the most desired, by constructors, norms were considered during creation of the following library.

KEY WORDS

quality, design

INTRODUCTION

Application of CAD/CAM systems in the design and production process enables to decrease costs due to the quick reaction on constantly changing market demands [1, 2, 3]. One of the methods to accelerate the computer-aided design process is for example application of libraries of standard elements. Repetitive modelling of the standardized parts, even those of simple structure, is inefficient. Continuous repetition of the same actions is laborious and time consuming, which favours the designer's distraction, and consequently may result in presence of construction errors. It is worth to mention, that the errors resulting from the design stage are the most expensive and difficult to remove, therefore every effort should be made not to let them appear. Engineering software producers understand the expectations of the constructors, and consequently, they continuously develop those programs, paying attention to advance and accelerate the design process. This acceleration is caused mainly by improvement of information swapping between specialist of different fields, well advanced personalisation as well as facilitating decisions making by the constructor.

The paper presents the additional application of base standardized elements for the NX system. What is more, a description of the employment procedure of the standard elements library was presented, as well as the possibilities of its expansion and applications.

PARAMETRICAL MODELING OF THE LIBRARY ELEMENTS

The standard elements library was created with a view to the NX system users, who from different reasons apply the standard parts in their projects [5, 6, 7]. The standard elements, which the following library consist of, were modelled accordingly to the guidelines included in the appropriate Polish norms and were parameterised in order to create their series of types. It was decided, that only a couple of element groups the most commonly applied in engineering mechanical constructions would be taken into consideration. Due to this solution, the NX software was enhanced by a ready elements library, such as: screws, nuts, washers, edging elements, inlets and bearings.

The NX system enables to create part families, due to which the design automation in wide range would be possible. As a constructional template, any element of construction may be designed, and its automation is to adjust the dimensions into the exact parameters, appearing in the designed construction.
When commencing the work under a new element of the library in the NX software, a file should be created, units of measurements and its name should be chosen. In the example publicised, the course of actions for the element group under the common name "Socket Cap Screw" in accordance with PN-EN ISO 4762 was presented.

The next step is to move into sketch settings – "File\Properties", and in the "Display Part Properties" window (pic. 1), in "Attributes" tab enter the element name and the norm, accordingly to which, it would be created. In "Wright" tab, an option of mass measurement while saving the element should be selected. This data may be used in future while creating the constructional documentation and automatically generated in table including parts applied in a group.
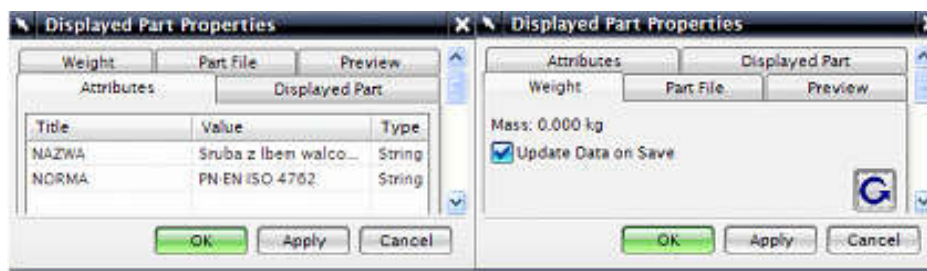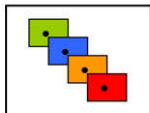
Pic. 1. File properties setting window

The following stage consists of defining changeable parameters for a created nut with the use of Expressions sheet. The equations herein defined are closely related with geometrical parameters used for modelling of the basic element. By applying the basic commands of the CAD module of the NX software, a fully parametrical model of the element was created, which constitutes a basis for the whole element group in the library.

The next action to take is to define the basis of controlling parameters with the use of the command "Tools\Part Families". This option displays a dialog box, in which the parameters controlling the element should be chosen. With the application of "Create" option, the system switches the user into the Excel spreadsheet window with the parameters visible in the appropriate columns. In this spreadsheet, the rows should be filled with another series of types of the screw, visible in the pic. 2.
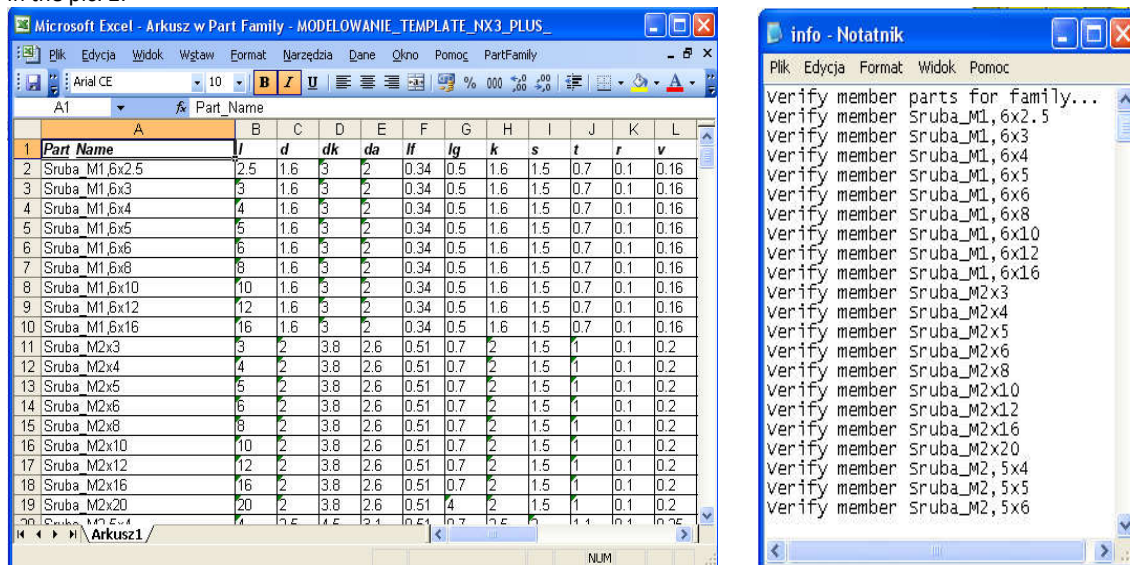

Pic. 2. Database building stages: a) Part Family window fragment, b) series of types verification window

This database is saved in the model's file and does not appear as an additional folder, which is a great advantage of this software, as it eliminates errors originating from copying data tables for appropriate elements. A useful functionality of the system is "PartFamily\Verify Part" command, which enables to verify and eliminate errors of each new element in the part family, which decreases the risk of committing mistakes in the database almost to zero. The above ensures, that any errors would appear in a new element of the database. In case a fault appears, the system is going to notify precisely, which parameter is erroneous and restrain new element from being generated.
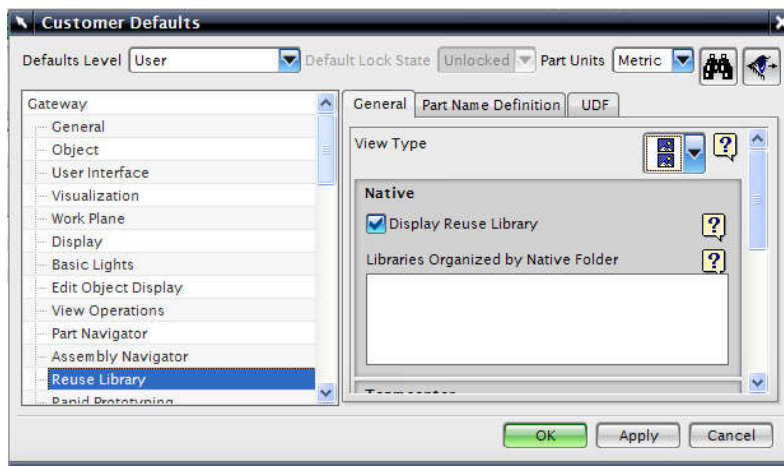
After executing the abovementioned actions, the complete element was obtained, with the database embedded in the "Socket Cap Screw in accordance with PN-EN ISO 4762.prt" file, ready to generate the part family elements. It was saved under the name, which would be unambiguously identifiable and would facilitate to complement the file with an explanatory figure of the element later on.

LIBRARY INTERFACE

The standard elements library was based on the ready-to-use "Reuse Library" module, provided by the NX software producer along with the program. In order to display the library interface, one has to move to software settings "File\Utilities\Customer Defaults", depicted in the pic. 3. In the main tree "Getaway", the "Reuse Library" subtree was chosen and the "Display Reuse Library" option was selected. It results in displaying a graphical window of the standard
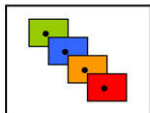
elements library in the toolbar "Resource Bar", visible in the pic. 6. The software producer did not equip the system with the standardised elements. The NX software user is capable to modify freely the library content, adjusting it to his/her own preferences.


Pic. 3. NX system main settings window

After executing the abovementioned actions, one should create a catalogue, in which newly generated elements would be saved. In order to do this, one should move to the "Part Name Definition" tab and choose the target folder in "Part Family Directory" field. When this task is completed, confirmation by pressing "Reuse Library" module enables to add new resources. This is performed by pressing right mouse button on "Name" field and choosing the location of standard elements catalogue. An important advantage is the possibility of adding an element located in any place; it is not necessary to copy the database into a site imposed in advance. It ensures a stable work and prevents from the problems arising in library resources management, OK button.
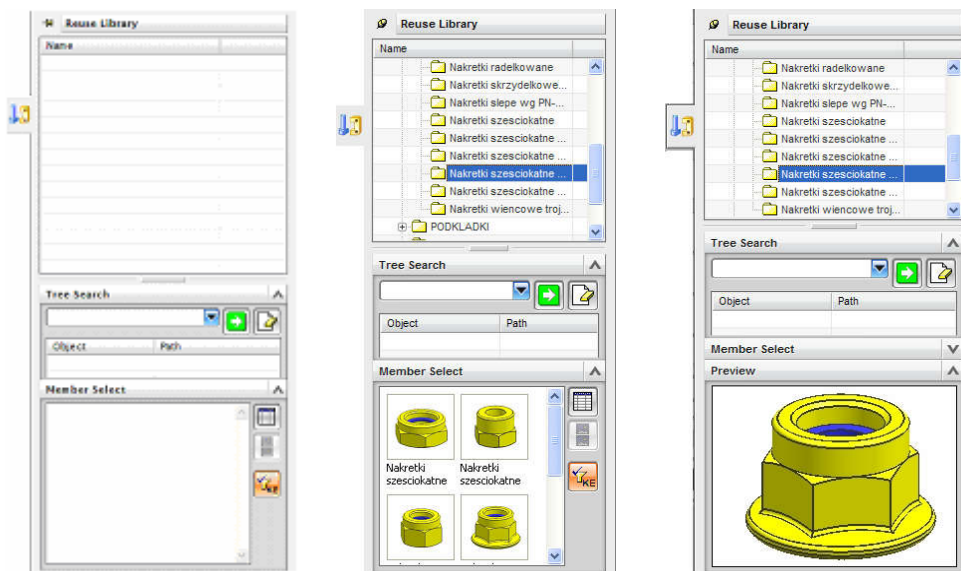

Pic. 4. Possible styles of displaying standard elements database

The application enables to search the elements by the "Tree Search" option. There is a possibility of elements preview at two different modes, by one element view or a couple of elements view, which permits for freedom in library management. What is more, there is also possibility of part preview with a flat explanatory figure, previously created.

RESOURCE CONTROL FILE
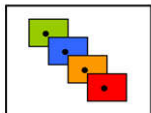
A control file is an indispensable component of the standard element library for the NX software. This file name is created automatically from the "Reuse Library" level. All the model control parameters (variables) are saved in this folder, due to which it is possible to create, on the foundation of a base file, derivative folders of the particular norm.
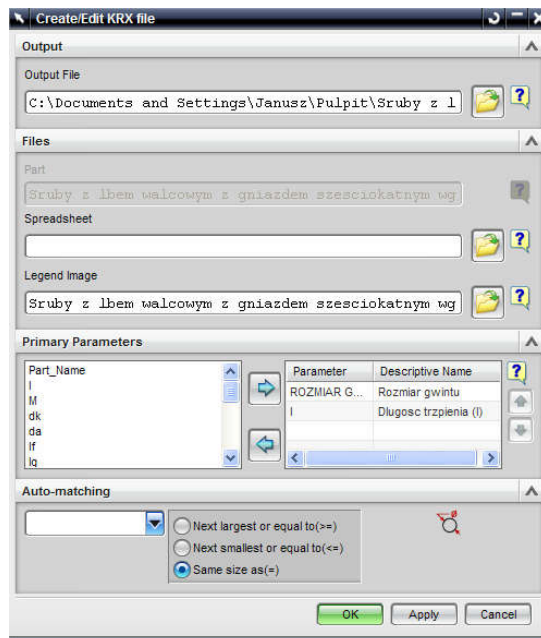
Pic. 5. Creator wizard window of resource control file

In the following section, the creation rule of this file was described, as well as its connection with the model files and the explanatory figure.

The creation rule of the resource control file was presented on the basis of previously generated model and the explanatory figure. In the "Member Select" command window an appropriate element should be chosen and right mouse button should be pressed. A dialogue box would be displayed, in which the "Create KRX file" option should be chosen. The system automatically launches a control file creator, presented in the pic. 5. In the "Legend Image" window, an access path to the explanatory figure was indicated. Subsequently, the element controlling parameters were selected. In case of screws, it would be a size of thread and a screw shank length, so "thread size" and "l". Double-click in the "Descriptive Name" field permits for entering a name, which would be displayed when the element would be inserted. In the "Spreadsheet" window, one may indicate the access path for the additional file, created in the Excel software, supplementing the database with an extended series or with additional information and variables. "Auto-matching" command enables to generate a stack of elements. A controlling parameter should be chosen and it has to be indicated if the created parts should be higher or lower in comparison to the selected parameter. Only a single element with the parameter equal to the selected one was generated. After executing those movements, the choice should be confirmed by pressing the "OK" button and the system automatically creates a file of .krx extension.

INSTERTING THE ELEMENTS FOR ASSEMBLY

In order to add each of the standard elements for an exemplary assembly, the "Reuse Library" application was employed. It permits for two-way inserting the part for an assembly: the first one is to press right mouse button and to select the "Add to Assembly" command, or by applying the "Drag-and-drop" technology – to select the element and to drag it into the model area.

As the exemplary one, the mode of inserting a ball bearing for assembly was described. In the "Reuse Library" ball bearings were searched, the appropriate ones were selected and dragged into the NX software workspace.

After this action, the system displays the dialog box visible in the pic. 6, in which the main bearing sizes may be chosen. The crankshaft diameter, onto which the bearing is going to be mounted, was selected from the drop down box, afterwards, the same happen to the bearing external diameter. If there are different bearing width for the previously defined parameters, an additional variable "Bearing width (B)" would be activated. The attention should be drawn to the fact, that the "Details" window includes all the bearing dimensions, accordingly with the explanatory figure, as well as its full name, size and the norm, from which the dimensions were taken. Those additional attributes may serve to generate automatically a table with the parts directory, as well as to cooperate with the Teamcenter application, helpful in live
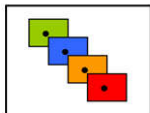
product management, which enables fluent control over the changes in the production and permits for considerably quick modification in the appropriate parts ordering and assuring the smooth production process.

The "Placement" tab serves for selecting the element insertion method. The most commonly used is the "Mate" command, which refers to the command box with the assembly bindings. The "Inferred Only" option inserts the part accordingly to the coordinate system, which is assigned to this element. In case the "Absolute Origin" command would be applied, the component would be placed by the global coordinate system. This method is particularly useful in large compositions, where the constructor by modifying the element, inserts it accordingly to the previously defined global coordinate system. "Reposition" command switches the user to the dialog box permitting for autonomous components placement, however it should be noticed that this assignment does not consist of any assembly bindings, and may lead to errors during the unit assembly. In the "Settings" tab, in the field "Name", there is located the name of the inserted element, which may be modified, however it should not consist of Polish diacritical signs.

In the "Reference Set" field, the component inserting mode for the assembly was selected as the solid model without auxiliary planes and axis. The "Lightweight" command is applicable in enclosure of models, which would not be analysed. In such a referential setting, measuring or balancing the part is impossible. The most commonly applied command is the "Entire Part", which inserts the element along with already existing constructional planes and axis, which permits for quick and precise component building. The "Empty" option inserts the part in a hidden form, it is not visible in the workspace. The choice of the referential setting depends on the kind of the element included or on the constructional needs. The "Layer Option" tab inserts the component in the appropriate layer. It is extremely useful in large assemblies as well as in one-assembly work executed by several constructors. The "Preview" command displays the element before its insertion to the assembly. Additionally, the software enables to rotate freely the component, with the aim of its inspection from each side.
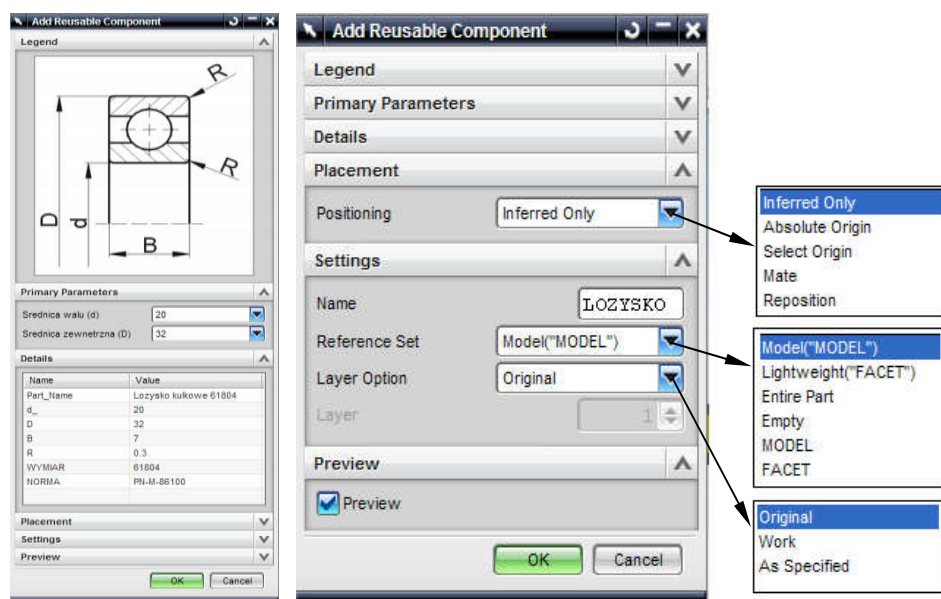

Pic. 6. Bearing insertion window

Insertion of several identical components may be executed by the "Add Component" command, which permits for enclosure of an already existing part. It may be done numerous times, however in an assembly consisting of abundance of the same elements it is inefficient and engage large amount of the system internal memory. During insertion of, for example a screw along with a washer, more efficient is the "Cloning" method. It requires only a single definition of the element group, and subsequently their insertion as an entirety. In the assembly resource management window "Assembly Navigator", a subassembly would be created, which would be inserted in various places, without additional engaging of the computer internal memory.

DATABASE EXPANSION

The created database may be expanded not only with additional series of types, but also with supplementary information, enabling for creation of intelligent and self-repairing models. It was presented on the basis of an inlet and an additional parameter "t1", being the groove depth for the inlet designation.. In order to achieve it, the database in the spreadsheet was complemented by the requested parameter and its location was indicated in the "Spreadsheet" window
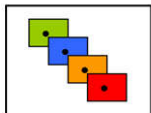
visible on the picture 7. The attention should be drawn to the fact, that the system simultaneously operates on two data files: the first one is embedded in the model file and the second, external, one with the additional parameter.
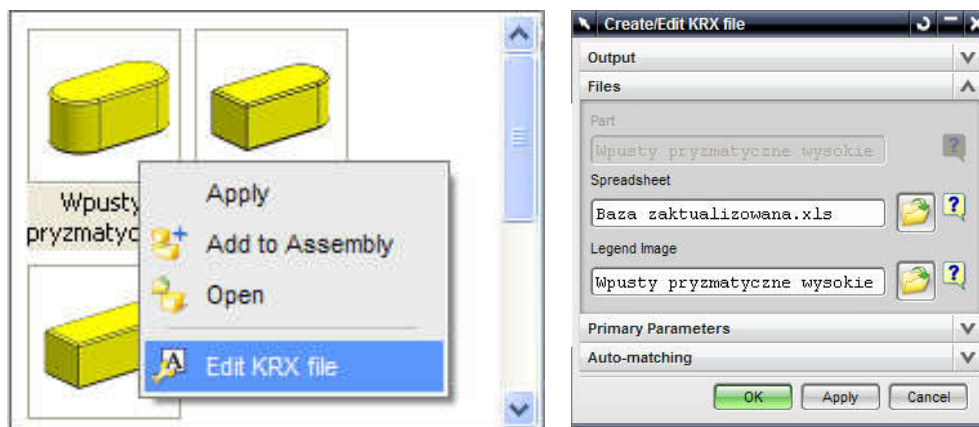

Pic. 7. Database update method

Firstly, on the exemplary shaft model, an inlet groove was created by applying the "Pocket" command. Its dimensions are of no importance for the time being, as these would be combined with the dimensions of the standard inlet, inserted from the database. The following stage of the preparations consists of selecting the distinctive features, which would be connected with the standard inlet. It would be executed with the "WAVE Geometry Linker" (pic. 8a) command. This option permits for retrieving the surface, points and curves of the selected element. Afterwards, in the "Part Navigator" window, new elements would appear, being an interpretation of the retrieved geometry (pic. 8b).
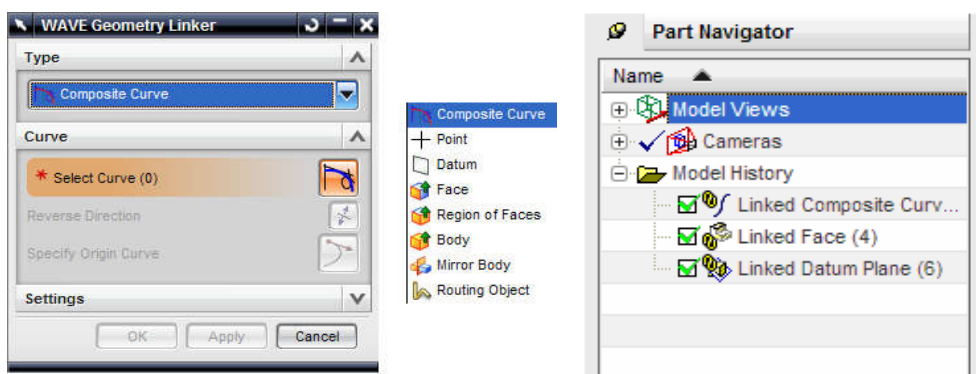

Pic. 8. Geometrical features linking stages: a) "WAVE Geometry Liner" command dialog box, b) "Part Navigator" window with clearly visible interpretation of the combined geometry

By application of the "Associativity Manager" command, one may define the appropriate relationships and links between elements in the assembly by matching them. The differences for the inlet not adjusted to the groove (pic. 9a) as well as for the components fully associated (pic. 9b) are visible on the presented comparison.
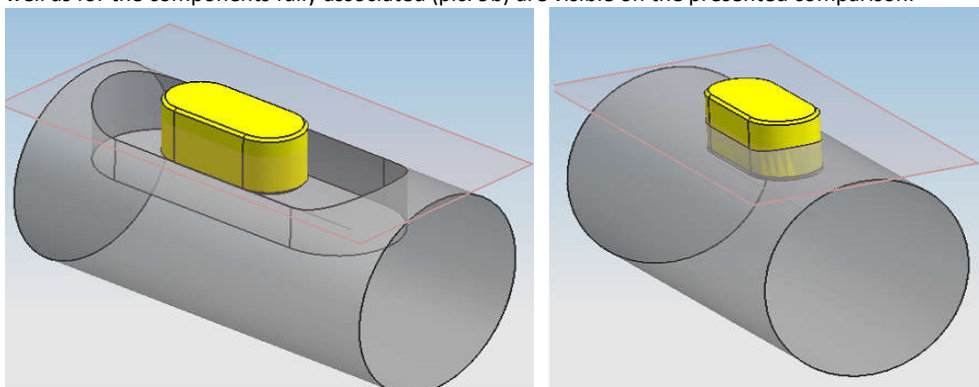
Pic. 9. Result of geometrical features association
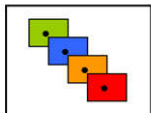
The key role in this connection belongs to the plane, as the "t1" parameter is determined on its basis. With a modification of the standard inlet by "Edit Reusable Component" option, the groove dimensions would be properly adjusted.

SUMMARY

The NX software belongs to the main representatives of the CAx systems. The possibilities of this tool are enormous and permit for the complete product management (from its concept to recycling). Fluent operation of the tool requires time and knowledge from a particular design field. In the article, the preparatory process of the standard elements database was presented, including bearings, washers, nuts, screws, bolts, inlets, edging elements. It should be underlined, that the most desired parts were considered and the study was limited only to the exemplary presentation of the selected component modelling method.

During processing the library elements, there were no issues that could not have been eliminated by deeper software cognition, therefore it may be stated, that the NX system permits for modelling even complex shapes. It should be noticed, that the models included in the library were attempted to be accurately mapped, accordingly to the geometry included in Polish norms. It was associated with profound recognition of the available functionalities of the system, in order to achieve a requested geometry. In consequence of a far-reaching parametrisation of the models, the library, rich in standard elements, along with its relatively small size, was achieved. It is possible to use the elements included in the library directly from the flash drive, however it is recommended to install the library into the hard disk of the computer. When designing the library, it was assured, that easy updates of the data included would be possible. The data being the series of types of the particular norms were saved in the Microsoft Excel spreadsheets, due to which it was possible to achieve the transparency and its simple edition. Naturally, further expansion of the library is possible as well. In the study, the modelling process of the elements typologically similar and constructional templates was explained in details, thanks to which the design process may be significantly automated. The modelling method and the amount of standard elements included in the present library may constitute a significant facilitation for the constructors using the NX software in their design work

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