THE APPLICATION OF KNOWLEDGE SYSTEM COMPONENTS IN CAD/CAM SYSTEM BASED ON XML LANGUAGE

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Abstract: The paper deals with specific application of knowledge system in CAM system Pro/ENGINEER Wildfire using XML language. The knowledge for NC sequences generation of single operations is implemented by means of XML language. The time of CL data preparation for control system is much shorter by this way and the manufacturing productivity is direct proportionally higher. The XML language is the standard for web application and for data entry, which is simple and clear for large clientele of users in whole production company structure and its co-operators.
Keywords: CAM system, knowledge system, XML (eXtensible Markup Language) Language

1. KNOWLEDGE ENGINEERING

At the present time, the knowledge engineering is considered as serious engineering discipline that is focused in applied applications of artificial intelligence. The specific software systems, i.e. knowledge systems, are the output of knowledge engineer work. It is the reason why the knowledge engineering is consider as specific field of software engineering.

The development trends of knowledge engineering influence the trends in whole software engineering. For example, the methodology of object oriented programming, the methodology of rapid prototyping penetrate into software engineering from the knowledge engineering.

The work phases of knowledge engineer who creates the knowledge system are sum up into five phases described at the fig. 1.

Scheme at the figure 1 can also present the evolution of knowledge system. It describes four different levels of knowledge representation in establishing system. The levels are, as follows:

1. Conceptual level
2. Formalization level
3. Program representation level
4. Implementation level

2. DESCRIPTION OF XML LANGUAGE

XML (eXtensible Markup Language) is similar to HTML formats for data registration as clear text. The principal idea is the filling of the text with tags that define single parts of text. Unlike the HTML, these tags are not strictly specified and each user can make its own tags according to its own requirements.

The text item can be in XML Language as follows:

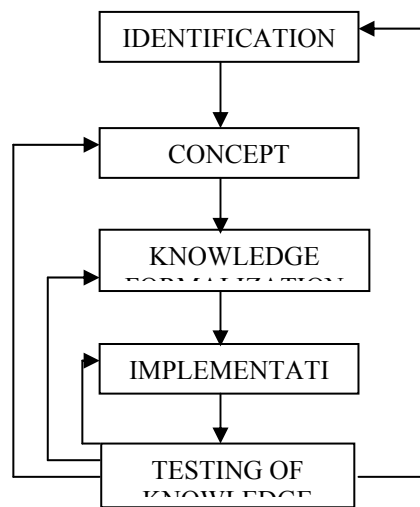<PRO_E_HOLE_CYCLE_TYPE type="int">**4**</PRO_E_HOLE_CYCLE_TYPE>

Fig. 1 *The work phases of knowledge engineer*

3. XML LANGUAGE APPLICATION IN CAD/CAM SYSTEM Pro/ENGINEER WILDFIRE

Solving the concrete part design and its follow manufacturing, we use knowledge of knowledge system establishment and XML Language. We follow the following steps:

1. Create 3D model.
2. Identification pattern element of model
3. Create database of material/ XML item
4. Create tool and sketch tool/ XML item
5. Saving and retrieving tool parameters/ XML item
6. Attaching CL Command(s) to tools/ XML item
7. Customizing tool manager layout/ XML item
8. Defining custom parameters for tools/ XML item
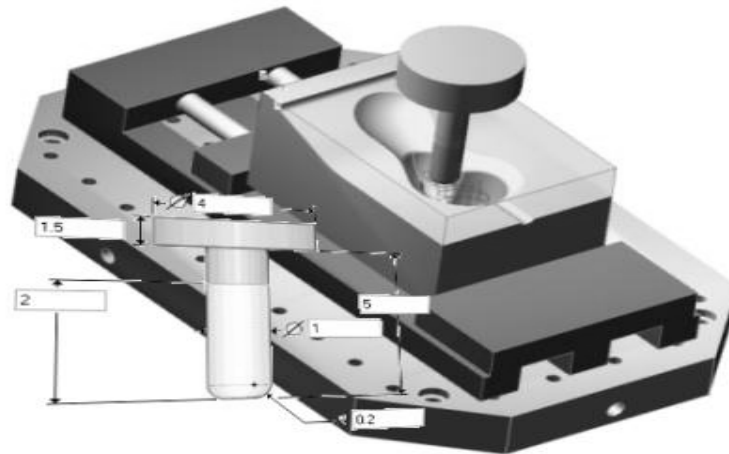9. Reading parameter from tool/ XML item

Fig. 2 *3D model of workpiece with tool*

At the fig. 2, 3D model of workpiece with tool that is modelled in Pro/ENGINEER Wildfire is described.

We make engineering design in module Part by modelling the 3D model of part. In Manufacturing module, we design technological process of manufacturing by CNC machine. It means that we define NC sequences for manufacturing. Our applied research focused in CAM field in Pro/ENGINEER system using the XML Language for communication between CAM system and some of knowledge system components.

In the table 1, the classic process of work in CAM system and the new process of work using XML Language are compared.

Table1. Classic and New Process using XML

Sequence number	Classic Process	New Process
1	Open Mfg Model	Open Mfg Model
2	Create Workpiece	Create Workpiece
3	Application operation/ Workcell	Identification pattern element, PartModel
4	Create Coordinate Systems/Ratrac Plane	Setting up Tools/Tools Parametres, XML
5	Create Fixtures	Manufacturing Parameters, XML
6	Definition Type NC Sequences	Reading parameter from tool,
7	Setting up Tools/Tool Parameters	Machined volumes and surface
8	Manufacturing Parameters	Visual machining process
9	Machined volumes and surface	
10	Visual machining process	

3.1 Creation of a standard tool

Standard tools also called parameter tools are Pro/NC supported tools with known geometry that can be described using an APT 7 type set of parameters. The resulting

geometry will be used by Pro/NC for tool path computation, material removal (when possible) and degauging.

The parameters for each tool are listed in the general panel of the tool description area using a graphic representation of the tool shape. Pro/NC proposes a list of predefined supported tool shapes.

These tool types correspond to the types of NC sequence performed in the workcell. The tool type, in turn, defines the tool's cross-section and, therefore, the set of parameters you have to specify for the tool.

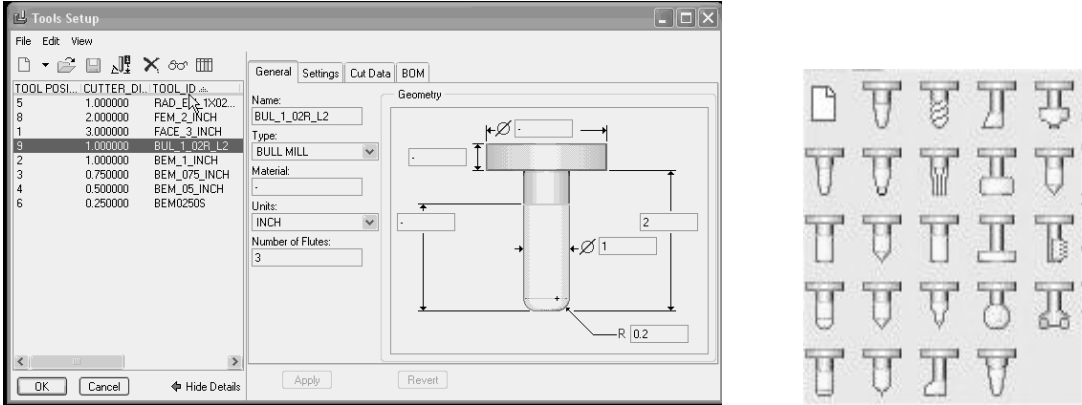At the fig. 3, tool selection and tool setting is shown.

Fig. 3 *Tool selection and tool setting*

3.2 Saving and retrieving tool parameters

Tool parameters can be saved in the form of ASCII text files; XML based, and then reuse these tools in a different manufacturing process without having to set them up from scratch. When you save a tool, the system stores its type and parameter values in a text file named .xml, where is the tool Name. In previous the default extension for the tool parameter files used to be .tpm. Old files with this extension will be recognized by the system as tool parameter files, that is, they will be retrievable. Whenever a tool parameter file is stored, however, it will now have the .xml extension.

The User Defined tabbed page lets us define custom parameters to be associated with the tool. The **mfg_custom_tool_param_file** config.pro option controls the appearance of this tabbed page. The User Defined tabbed page appears in the Tools Set up dialog box only if we set the value of the **mfg_custom_tool_param_file** configuration option to point to a valid user-defined parameter file with XML extension.

The example of text items in XML Language is shown at the fig. 4.

```xml
<?xml version="1.0" encoding="UTF-8"?>

<MfgSetupDocument>
    <DocType>PRO_NC_CUTTING_TOOL_SETUP</DocType>
    <DocTitle>Cutting Tool Information</DocTitle>
    <DateTime>2005-06-20T12:50:40</DateTime>
    <ApplicationInfo AppName="Pro/ENGINEER" AppVersion="C000"/>
    <ToolingSetup>
        <Tool Id="FEM_1_INCH" Type="MILLING" RefXmlId="encref_6">
            <Attr Name="UseOutline" Value="false" DataType="boolean"/>
            <Attr Name="ProLibraryTool" Value="false" DataType="boolean"/>
            <Attr Name="SketchTool" Value="false" DataType="boolean"/>
            <ModelInfo Type="Assembly" InstanceName="FEM_1_INCH"/>
            <MfgParam Name="LENGTH_UNITS" Value="INCH"/>
            <MfgParam Name="CUTTER_DIAM" Value="1"/>
            <MfgParam Name="CORNER_RADIUS" Value="0"/>
            <MfgParam Name="SIDE_ANGLE" Value="-"/>
            <MfgParam Name="LENGTH" Value="4"/>
            <MfgParam Name="NUM_OF_TEETH" Value="2"/>
            <MfgParam Name="TOOL_MATERIAL" Value="HSS"/>
            <MfgParam Name="GAUGE_X_LENGTH" Value="-"/>
            <MfgParam Name="GAUGE_Z_LENGTH" Value="-"/>
            <MfgParam Name="COMP_OVERSIZE" Value="-"/>
            <MfgParam Name="TOOL_LONG_FLAG" Value="NO"/>
            <MfgParam Name="HOLDER_DIA" Value="-"/>
            <MfgParam Name="HOLDER_LEN" Value="-"/>
            <MfgParam Name="COOLANT_OPTION" Value="OFF"/>
            <MfgParam Name="COOLANT_PRESSURE" Value="NONE"/>
            <MfgParam Name="SPINDLE_SENSE" Value="CW"/>
            <MfgParam Name="FLUTE_LENGTH" Value="-"/>
            <MfgParam Name="TOOL_COMMENT" Value="-"/>
        </Tool>
    </ToolingSetup>
</MfgSetupDocument>
```

Fig. 4 *Text items in XML Language*

3.3 Reading NC parameter from tool

NC sequence parameters can be initialized based on the tool selected for the NC sequence. There are two mechanism for this :

- Manually : In the Parameter Tree dialog, Use **use Edit > Copy From Tool** to copy tool parameters to the NC sequence. Click Edit > Copy From Tool > Misc to copy custom parameters from the tool to the NC sequence.
- Automatically : use the **mfg_param_auto_copy_from_tool** configuration option to have Pro/NC copy all, miscellaneous, or cutting parameters from the tool to the NC sequence.. However, Pro/NC copies automatically the tool parameters to the Param Tree dialog box only when we create an NC sequence, i.e. the first time we access the parameter tree dialog.
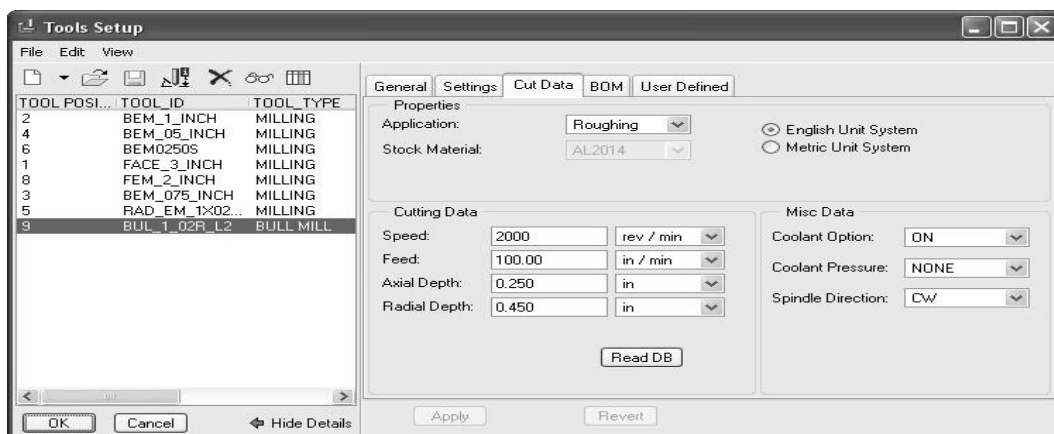

Fig. 5 *Tools Setup*

4. CONCLUSION

XML Language was used for communication between some components of knowledge system, which was established for selected sections and parameters, and single modules of CAD/CAM Pro/ENGINEER Wildfire system.

The mentioned language is simple, clear and easy to apply and in the main it is easy to learn for everyone who manages the Microsoft Office products. This fact made the many companies to use XML Language and its formats even though its simplicity cases large text items.

XML Language was suitable for our work for several reasons:
- its universal applicability,
- availability,
- usage for web diagnostic of CNC machines,
- communication with SQL and Oracl databases,
- communication with Microsoft Excel,
- good export from table formats into environment of user,
- communication with CAD/CAM systems (CATIA, Pro/Engineer, UG).

XML Language allows us to make simple text item of identified features for model and thereafter to assign the tool parameters and manufacturing conditions. Using the XML Language, we loaded concrete data into manufacturing process manager. If we will assign certain identification marks, we will be able to make selection of different variants of features and manufacturing process parameters considering the concrete manufacturing conditions. The repeated usage is also possible.

5. REFERENCES

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