AN INTERACTIVE KNOWLEDGE-BASED CAD SYSTEM FOR ROTATION PARTS

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ABSTRACT

This paper present a practical knowledge-based system, for rotation parts. The core of a CAD system is the, which makes use of graphics for product representation, databases for storing the product model and drives the peripherals for product presentation. The possibility of improvement CAD using the application based on object oriented method paper is shown in this paper. The application is linked to the CAD systems using the API commands. Originally the technique was aiming at automating a number of tasks a designer is performing and in particular the modelling of the product.

1. INTRODUCTION

Design automation is to automate conventional manual design process by use of computer or by extracting knowledge from knowledge base. This knowledge can be standard design procedure, past experience, manuals, charts, etc.
In this design knowledge, past experience, condition is stored in computer database or programmed so that it can be reuse again whenever needed.

Figure 1: Design Automation

2. APPLICATION PROGRAMMING INTERFACE (API)

API (Application Programming Interface) is software development technique for integrating two different software. With the help of API we can develop custom stand-alone windows executablefiles, for API programming we can use VB.NET, C#, Visual C++ languages. In this paper for developing software application, Microsoft visual studio is used. We integrated Microsoft visual studio with CAD package; this resulted in exchanging data between these two software. There are many CAD software packages having Application Programming Interface functionality. In this SolidWorks CAD package is used because it is user friendly, most important it supports Application Programming Interface functionality.

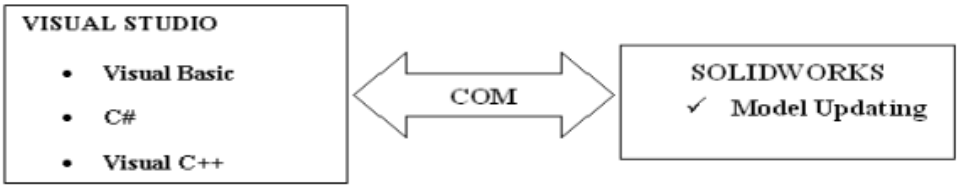

Figure 2: Application of Programming Interface

2. PARAMETRIC MODELING AUTOMATION ROBLEM DEFINITION

We can problem definition on:

- As discussed above winding machine having many mechanical components. Designs of these components depend on customer requirement. These design calculations for all components are repetitive and time consuming task.
- Also this design process is currently done on past experience; experienced personal are needed every time for designing..
- After design calculation are done, it is time consuming to make CAD models of machine part and its manufacturing drawing which also requires skill.

There are following two approaches for modelling automation
a) Master Model
b) Generative Model

The first method, which we call the master model method, uses a worst case model and drives that model. You don't create geometry. You don't create assemblies or drawings. You simply open an existing model and you drive it. The other method, known as generative modeling actually generates models and assemblies and drawings on the fly. There are certainly cases where both have their distinct advantages.
With the master model approach, you start off with a "worst case" mast er model. We use SolidWorks to suppress or delete features that we don't want, for example. That means that all of the features and components that you might need, have to be in the master model. Does this make the models more complex? Certainly, it does. But the programming for master models, and this includes programming in the traditional sense, where you're working with the SolidWorks API and when you're writing rules inside of a DriveWorks or a TactonWorks(Commercially available solution partner product) type of product.

3. SCREENSHOTS OF DEVELOPED APPLICATION

On the figure 3. show is development application make in program Microsoft Visual Studio, which conection basic 3D model rotation master part from Solid Works application and enable generation of new rotation part entering parameter values using API command.

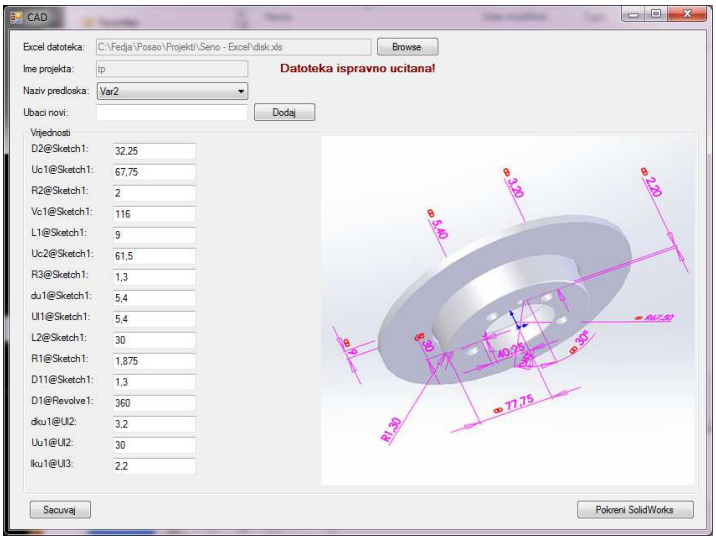

Figure 3: Graphical User Interface for General Input

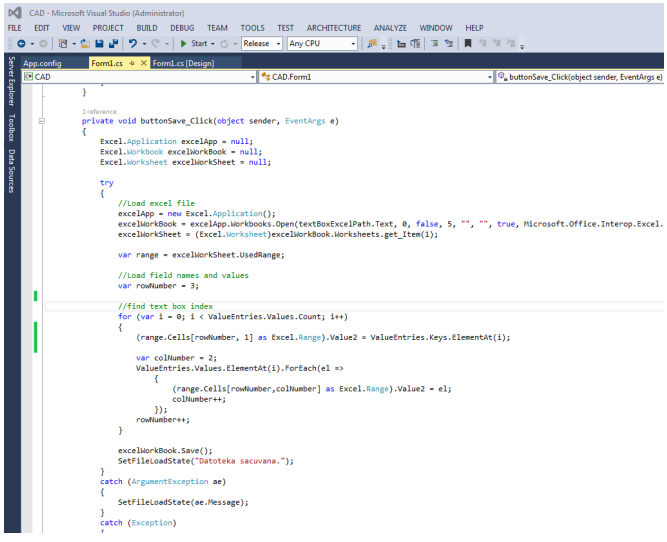

Figure 4: Visual C code for loading data in the excel file

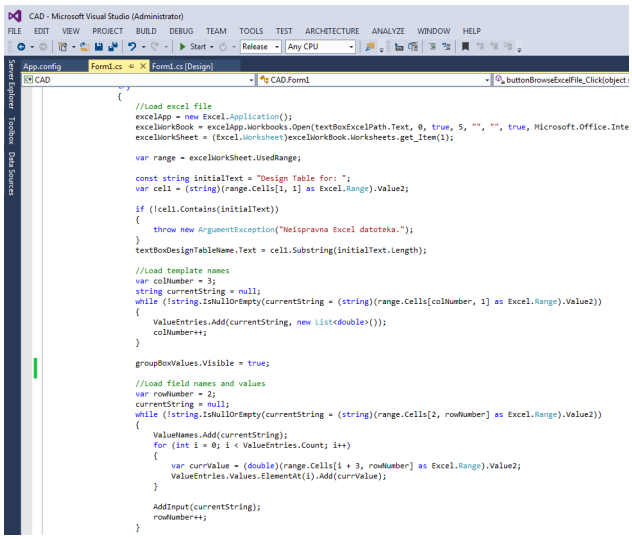

Figure 5: Visual C code for saving the data in the excel file and create new variants

In Figures 4 and 5 shows the code that manages the developed application and connection to the CAD software Solid Works. On figure 4 is show cod that allows entry of new data on a rotating part, and on figure 5 shows is code that allows entry of new data on a rotating base part.
The aim of the application is developed to allow the automatic obtaining new variants of the rotating part in Solid Works software.

4. CONCLUSION

The paper presented possibility of improvement CAD using the application based on object oriented method. The application is linked to the CAD systems using the API commands.
The work presented here is developing a design and drawing automation system for rotation parts, the approach presented in this thesis could be further enhanced and extended by considering the following aspects: Presented work shows design and drawing automation of rotation parts, this application can be developed with other design standards and other types of rotation parts. And further, it can be developed for the CAE automated application to solve and simulate the components.

5. LITERATURE

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