Automated CAD Modelling of Mechanical Components

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Abstract: Most of the manufacturing industries produce similar components where the designer has to model similar products again and again which is not only time consuming but also introduces operator fatigue and reduction in the productivity of the designer. Different activities of the CAD division can be automated in order to free the operator/designer from the repetitive tasks and perform other important functions such as analysis which cannot be automated. In the present work it has been successfully demonstrated, how the geometric CAD modeling can completely be automated reducing the chances of error and increasing the overall efficiency of the design process.

Keywords: *GUI, Application Programming Interface (API), Geometric CAD modeling*

1 Introduction

During the last few decades CAD software are extensively used to increase the productivity of the designer, improve the quality of design and communication through documentation and to create database for manufacturing. This is due to their benefits such as lesser design lead times i.e. lesser time to market, easy modifications, improved quality and productivity, less number of design iterations, etc. A variety of CAD software are available in the market made more and more generalised so as to fit the purpose of wide range of industries. But it is difficult to find CAD software which will meet all the requirements of particular industry. Hence there must be some facility in the CAD software so as to model them to satisfy the specific need of the industry. This can be done by customizing the CAD software according to the application programming interface (API) in which various objects of the CAD software can be used in the program and application which is termed as 'Automation'.

1.1 Limitations of CAD geometric modeling

CAD softwares have made their place in the industry due to many advantages they have. However, they suffer from following limitations.

- Skilled manpower required
- Time consuming modelling processes
- Higher cost of software as well as higher wages of the operator
- Chances of human errors and drafting errors are not eliminated.

1.2 Alternative Methods

In the present scenario the above problems are tackled by one of the alternative solutions as stated below [1].

- By storing the master file for each type of component and then modifying it as per requirement
- By using the 'Variable table' provided by the CAD software
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- By using the part family option if provided by the CAD software

1.3 Limitations of existing methods

All the methods discussed above, suffer few of the drawbacks as discussed below.

- A graphical master file is required to be stored: The biggest limitation of all above mentioned methods is that, all of them require at least one model file to be stored. The graphical nature of these model files requires more of the memory space to store. This problem more prominent when number of variations in the part goes on increasing goes on increasing.
- Shape changes cannot be accommodated: Though above methods are effective in case of dimensional changes, they are not at all useful to accommodate the shape variation in the components. Some of the features can definitely be suppressed or unsuppressed and few shape changes can be achieved but upto a certain extent. These methods are not available for assembly and drafting. By none of the above methods, repetitive drafting can be eliminated.
- Skill required cannot be completely eliminated: All the above-mentioned methods still require good knowledge of the CAD software rather greater knowledge and understanding of the CAD software that required for straight forward modelling. None of these methods can reduce the skill requirement. The wages are directly linked with the skill requirement and hence cannot prove useful for considerable reduction in cost.
- Error may creep in due to mistakes in inputting data from catalogues: The changes in the dimensions are to be achieved by changing the values manually wherein thee new values for the required model are generally available from the different catalogues (manufacturing catalogues) available in the design department. Errors may creep is due to mistakes in inputting data from catalogues. There is no any foolproof system to avoid any such mistakes and such errors may not be identified and thus the model created may be a faulty model.
- Accidental changes in master files are possible: This is one of the risky limitations of all the three methods. As the master files are in soft form only and are directly available for the user. There are also some changes of accidental changes in the master file. If such changes are saved this may cause major problem, as the data related to master file itself is lost forever. The change may or may not be repairable.
- Being soft in nature may get corrupt or accidentally be deleted: The model files stored are also softcopies only so they may be any time get damaged. The files may accidentally delete. Thus, the master file is required to be created again and reducing the reliability of the method.

1.4 Part Libraries and e-drawings

Every design calls for standard part such as bolts, screws, nuts, or linear actuators. Modelling these components repetitively takes a lot of time. There are various part libraries according to the different standards. Once installed, these part libraries can be viewed with the Library Browser so that parts can be inserted into your drawings [2].

On insertion, each part is converted into a block which can be re-inserted many times. Block, a group of entities can be inserted into the same drawing more than once with different attributes locations, scales and rotation angle. Such an instance of a block is usually called an "insert". Inserts have attributes just like entities and layers. An entity that is part of an insert can have its own attributes or share the attributes of the insert. The power of inserts is that you can modify the block once and all inserts will be updated accordingly. Blocks can contain useful text, dimensions and reference notes [3].

Many software provide most commonly required models on the internet in the form of e-drawings for almost all CAD software. e-drawings are the premier 2D and 3D design communication tool for internal and external design teams which deliver a rich collaborate tool set and enable to speed up the design process. [4]. The electronic data format of e-drawings files is highly compressed to enable easy transmission via email. e-drawings files are created through the use of the e-drawings publisher add-in, which works as a plug-in to many popular CAD products. Once created, an e-drawings file may be sent to anyone via email and support file formats such as CATIA V5 (.CATPart), SOLIDWORKS native files and templates (.sldprt, .slddrw, .sldasm, .prtdot, .drwdot and .asmdot), DXF/DWG files (.dxf, .dwg), ProEngineer files (.prt, .prt, .xpr, .asm, .xas), e-drawings native files (.edrw, .eprt, .easm, .edrwx, .eprtx, .easmx).

Even the above alternatives have drawbacks of size i.e. require high memory due to graphical nature of the files. The size of a typical e-drawings software is in the range of 10 to 12 GB and that of an e-drawing file is around 15 to 20 MB. It takes much time to download the files from the internet which may be comparatively costlier affair. The market price of an e-drawing software is about 1000$ which is very costly [5].

2 Problem definition

As the CAD modelling techniques become more and more advanced, there is a need to complete the product modelling and design changes faster than ever. Updating the assemblies having hundreds of sub-assemblies and parts manually in 3D modelling software is very complicated & time consuming. The CAD models that have large number of parameters and high memory required more graphics and time to process the data. To reduce the development time, minimizing the errors and introduce technologies faster to the market, many companies have been turning more and more to automation process [6].

Automation is a set of technologies that results in operation of systems and machines without significant human intervention by saving the time and also achieving performance superior to manual operation. By using automation it is possible to achieve increase in productivity, quality and robustness along with reduction in time, labour costs and other expenses. The concept of automating the different activities of the design department can better be demonstrated by taking some product [7]. The present study is to automate the different activities of the design department for the basic mechanical components.

3 Methodology

A novel approach of Application Programming Interface (API) is used in the present study to achieve the above stated objective. API serves as an interface between different software programs and facilitates their interaction, similar to the way the user interface facilitates interaction between humans and computers [7]. An API expresses a software component in terms of its operations, inputs, outputs, and underlying types, defining functionalities that are independent of their respective implementations, which allows definitions and implementations to vary without compromising the interface. A good API makes it easier to develop a program by providing all the building blocks, which are then put together by the programmer [8]. An API may be for a web based system, operating system, or database system, and it provides facilities to develop applications for that system using a given programming language. APIs often come in the form of a library that includes specifications for routines, data structures, object classes and variables. The API for CATIA is called Component Application Architecture (CAA) and it is written in C, C++, Java and Visual Basic [9]. Originally the API available for CAD software was developed with computer programming languages such as FORTRAN, ALGOL, AutoLISP for AutoCAD, GRIP for UniGraphics, ProProgramming for Pro-E and advanced languages such as C, C++, Visual Basic 6.0, Visual C++, etc. for UniGraphics, CATIA, SolidEdge, SolidWorks, etc.) In the present work, the API for CAD modelling software (CATIA V5) used is Visual Basic 6.0. It is interlinked within the CATIA tools option through VB Editor or it can be in the form of external software. The 'macro' used for automated modelling is coded in VB Script [10].

To simplify the inputs from the user and to minimize the skill requirement from the operator, a simple VB form is designed which takes input from the user in simple text format like the overall size, capacity etc. The sample form for user input is shown in fig. 1 below.
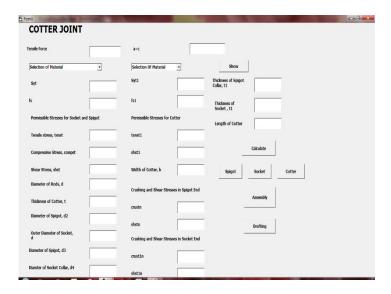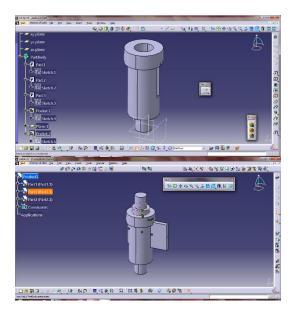

Fig. 1. User Interface

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5 Conclusions

The present studypropose a method of using an automated process for modelling commonly required mechanical components and subassembly. Following advantages can be observed with the proposed method.

- Skill required for the CAD modelling is greatly reduced. The user is never required to use a CAD software. The user has only the interface in the form of a VB form.
- Cost of design process is greatly reduced as the skilled operator is not required.
- As the complete modelling process is automatic it takes place in very few time. Ideally it takes about 2-3 minutes for modelling of a small subassembly like knuckle joint with all its components and subassembly.
- The modes are backed with the design calculations so that the final design is ensured that it is safe.
- Graphical memory is not used. Thus saving cost on memory requirement.
- As the model is generated where and when required, the transportation of the files becomes very easy. A very complicated model, which otherwise may require fe MBs of space, can also be saved through a text file with almost few kilo bites.
- Modeling errors are 100% eliminated.
- e-models and e-drawings can very easily be saved and trasfered with very few KB files even on mail.
- Overall, it can be said that, in coming future this method may prove to be the best method of modelling commonly required components.

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