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Automatic Assembly of Mechanical Joint Based on Extraction of Dimensional Data and Geometric Information From A 3D CAD Model

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Abstract- In this paper, describes how the automation of assembly of mechanical joint, in CATIA can be approached, by extracting geometrical and topological information from CAD part models by means of Ms-Excel as a database and with the visual basic as event driven programming. A three dimensional (3D) components of the assembly were collected from customer. The part models are rich source of dimensions and other related geometric information which is later used in assembling the parts digitally and thus a generating a new product. The information extracted from the part models have a greater significance in interfacing CAD with process planning, CAM, material requirement planning, etc., Among the extracted data, the information related to the features present in the part are adding more and more applications can be used. The developed algorithm is achieved based on Visual Basic language using API of CATIA CAD software.

Keywords – ***Automation of Assembly, CATIA V5 R20, Ms- Excel database, Visual Basic 6.0***

1. INTRODUCTION

CATIA (Computer Aided Three dimensional Interactive Application) is a multi-platform CAD/CAM/CAE commercial software suite developed by French company Dassault Systems and it is marketed world-wide by IBM. CATIA is the world's leading CAD/CAM/CAE software. CATIA V5 is an open system for developing advanced macros for special needs. A macro is a piece of code (written certain programming language) which groups a set of operations that define a certain task. These macros may be useful for creating, analysing, measuring, modifying, translating, optimizing surfaces, solids, and Wireframes, more. Macros are also useful for assembly operations, CAM operations and all multidisciplinary applications. Macros are developed using VBScripts and VBA. In order to develop a macro in CATIA V5 all we need is the inputs, outputs and necessary supporting data from the user.

In most of the engineering designs the product of interest is a composition of parts, formed into an assembly, as most of the product cannot be generated from a single part. Modelling and representing assemblies as well as analyzing assemblies are all relevant issues to geometric modelling. Assembly modelling is actually an extension of part modelling. In order to properly assemble the parts, the spatial positions and hierarchical relationships among the parts and the mating conditions between the parts must be specified explicitly by the designer. After the creation of an assembly it is analyzed. Assembly analysis includes interference checking, mass properties. Any design modifications performed in the part models will be automatically reflected in the assembly as well.

2. SYSTEM ARCHITECTURE

The System Architecture indicates the complete block diagram of the assembly flow process developed in this paper as shown in Figure 1.

The assembly automation can be obtained by using geometric model from customer and database as Excel and controlled by Visual basic as event driven program. The geometric and topological information of the components can be stored in excel database for assembly process in terms of base part, second part, feature type recognition, mating condition, orientation identification.

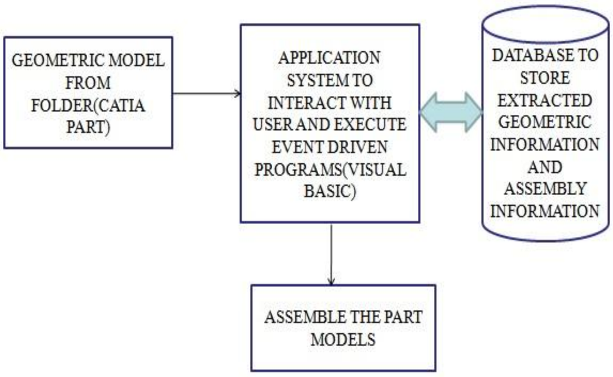


Figure 1.System Architecture-Block diagram

3. EXTRACTION OF DATA FROM A CAD MODEL

It is possible to extract various dimension and geometrical information from a CATIA part model using two main techniques. They are

i. By converting the CATPart file into STEP format, which is a neutral text file format from which the required information can be extracted by reading the file for the required data.

ii. By using the macro programming technique.

In this paper, the second method of using macro programming is employed to extract the required data from the part model.

3.1 Macro programming technique

Macro programming technique is one, which is capable of accessing the model tree of the part model, and extracts the required data. This is achieved with the use of inbuilt functions on a hierarchy of objects with which the part model itself is generated. For this purpose, the program codes can be generated in macros editor, which is inbuilt within CATIA, or using Microsoft Visual Basic code editor [2]. As visual programming is more interactive with the user, the Visual Basic code editor is used in this project and is interfaced with the user using interactive forms. Visual Basic programs are event driven programs in which the user interfacing forms consist of various controls and when a desired event is performed on these controls the corresponding program modules will be executed.

In CATIA the part objects, which are used for developing part model i.e. three dimensional objects are structured under a tree as shown in Figure 2. As and when needed the part object can be extracted with the macro programming for customization or automation of CATIA V5. The objects are created in a hierarchical way and while extracting the data we can investigate through the same object hierarchy and just apply the related methods and properties on these objects and store the extracted data in a separate database.

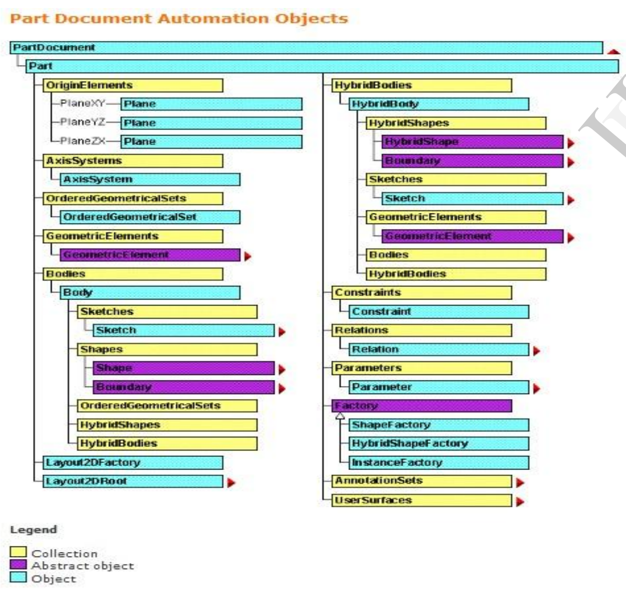


Figure 2.Part Modelling Object Tree

3.1.1 Methodology used to extract data from models First all the part models from which data to be extracted should be collected from the customer as folder as shown in Figure 3. Then the user interfacing form is form is created using Visual Basic with all essential controls to effectively interact with the user. The code required to extract the data from part model is added under suitable control's desired event. The code is executed to extract all the essential data. The extracted data is stored in a Microsoft excel and the database is stored under suitable name as shown in Table 1.

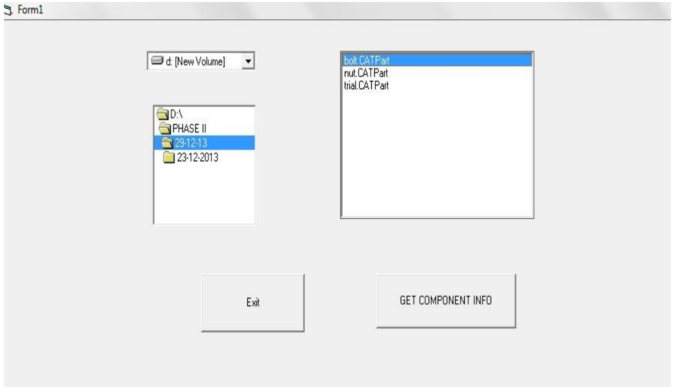


Figure 3.Components from customer (Case study Bolt and Nut)

Table 1.Data extracted from Parts

	A	B	C	D	E	F	G	H
1	PartBody							
2		Pad.1						
3			padlength	20				
4			feature_originx	0				
5			feature_originy	0				
6			feature_originz	0				
7			parallel_plane	PlaneXY				
8			feature_direction	0,0,1				
9					Line.1			
10						start x coordinate	0	
11						start y coordiante	0	
12						end x coordinate	100	
13						end y coordinate	-2.18792E-14	
14						length	100	
15					Line.2			
16						start x coordinate	100	
17						start y coordiante	-2.18792E-14	
18						end x coordinate	100	
19						end y coordinate	80	
20						length	80	
21					Line.4			
22						start x coordinate	1.45536E-14	
23						start y coordiante	80	
24						end x coordinate	0	
25						end y coordinate	0	
26						length	80	
27					Point.1			
28						x coordinate	100	
29						y coordinate	-2.18792E-14	
30					Point.2			
31						x coordinate	100	
32						y coordinate	80	
33					Point.3			
34						x coordinate	1.45536E-14	
35						y coordinate	80	
36					Circle.1			
37						circle direction	anticlockwise	

Sheet1 Sheet2 Sheet3

4. CURRENT TECHNIQUE USED FOR ASSEMBLY

The assembly design module of CATIA requires constraint to be specified manually. The various constraints used in the assembler includes

i) Coincidence constraint: This is for the coincidence of axis of two mating parts.

ii) Surface contact constraint: The part surfaces which are having direct contact or constrained with 'surface contact constraint'

iii) Angular constraint: This is used to specify the angular relation between two components.

iv) Fixing constraint: The base component is fixed and with respect to that base part remaining components are assembled.

In case of complicated product models with more number of parts it requires a lot of manual interactions (zooming, selection of axis, surfaces, edges, faces) and more amount of time. The application of constraints for each mating part, selecting the reference planes and other activities required for an assembly have potential for automation [1]. If intelligence is added to computer, through the CAAP system, to perform the assembly activities, hitherto done manually by the designer, then enormous amount of time can be saved for a designer.

4.1 Assembly of parts using the data extracted from the part models

It is also possible to assemble the part models using the data extracted from each of those parts. When parts are assembled using this technique it highly reduces the user interactions and the total time of assembly. Although the technique used in this paper demand little user interactions to input the mating axis and plane details, it can also be automated completely by using the dimensional data of the mating features and analyzing the positive and negative nature of the features.

In CATIA the product objects, which are used for developing product are structured under a tree as shown in Figure 4. As and when needed the product object can be extracted with the macro programming for automation of assembly using extracted data in excel in CATIA V5.

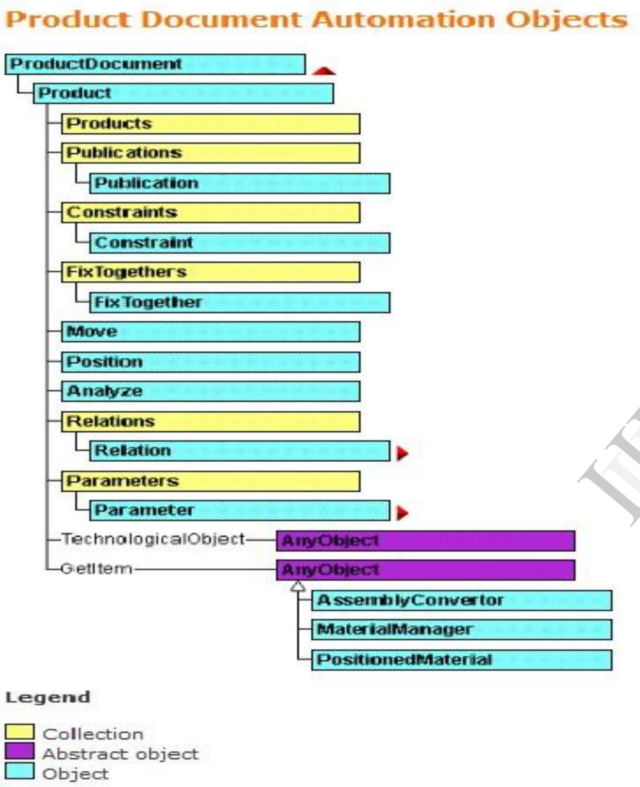


Figure 4.Product Object tree

4.1.1 Methodology for assembling parts using the data extracted from the part models

The biggest (or) a part which has maximum number of features are required for ease of assembly and faster operation.So,depending on the number of features in each part from database, the parts are sorted in descending order and the first in the sorted list is taken as the base part.

Positive features are protruded features of different sketches. Negative features are depression of different sketches. Here checking of each and every feature of the base part with every feature of the second part requires more amount of time and increases complexity of program. In order to reduce the time, only positive features of base part are checked with the negative features of the second part. Similarly the negative features of base part are checked with the positive features of the second part as shown in Figure 5.

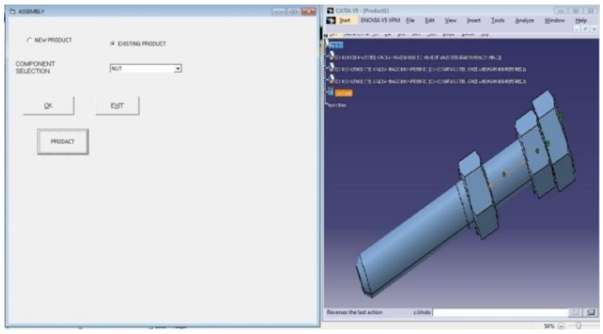
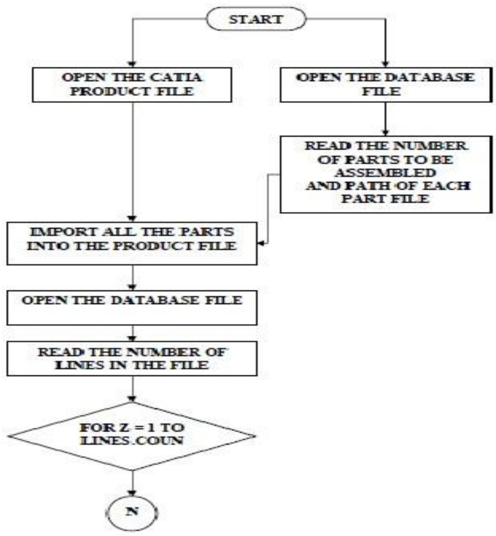


Figure 5.Product Obtained by Assembly automation

4.1.2 Checking the possible mating condition of parts from database

Length between the two co-ordinates (points), angle between two line, radius of circle, pad (or) pocket length are considered for checking the mating possibility. Exactly two sub assemblies are joined at each assembly task. After parts have been put together, they remain together until the end of the assembly process.

If the mating condition is satisfied, both part models are collected in the assembly design module of CATIA.The flowchart for assembling the parts as shown in Figure 6.



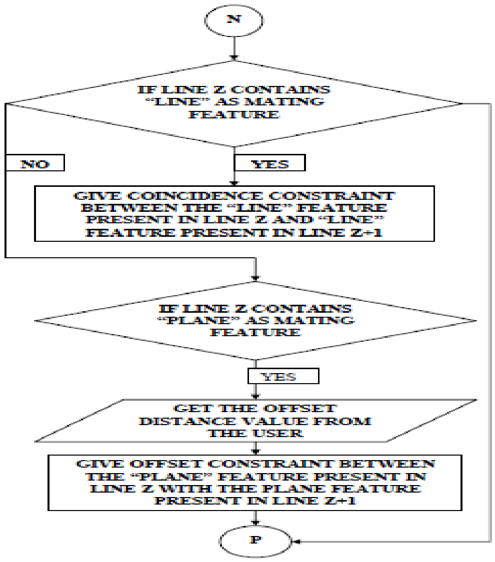


Figure 6.Flowchart for Assembling the Parts

4.1.3 Advantage of assembling parts using the data extracted

The major advantage of assembling by using the data extracted is that we can record the details of the mating features. Most of these details could not be obtained from the model tree directly and this can also be used in several applications like: i) Collision detection,
ii) Assembly direction detection, etc.

5. CONCLUSION

The three dimensional mechanical joint components has been created using CATIA has been collected from customer and it has been converted as macros in visual basic. This paper developed a program for automating the assembly process for Mechanical Joint. Selection of Parts and assembly instructions based on visual basic and its macros of CATIA functions are incorporated. The approach of applying Catia API was explained in details for assembly automation and can be applied for similar cases. Thus, this automation / customization will increase productivity of the designer with increase in quality of design which in turn reduces lead time for assembly of mechanical joint.

6. REFERENCES

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