APPLICATION OF KNOWLEDGE BASED ENGINEERING IN DESIGNING AUTOMOBILE RIM AND MACROS IN CATIA

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DECLARATION

I declare that this project report entitled “Application of Knowledge Based Engineering in Designing Automobile Rim and Macros in CATIA” is the result of my own work except as cited in the references

Signature : .....
Name : .....
Date : .....

APPROVAL

I hereby declare that I have read this project report and in my opinion this report is sufficient in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering (Automotive).

Signature : .....

Name of Supervisor : .....

Date : .....

To my beloved

mother and father

ABSTRACT

This project aim to help the engineers in design phase, to facilitate their daily task by applying Knowledge Based Engineering technique. It can cut the cost of production and also save time. In general, by applying Knowledge Based Engineering, the routine design time can be shortened and it promises minimal human errors as well. The main objective of this project is to construct a program in CATIA software to perform certain tasks during design phase by writing the coding/macros. There are various methods of writing macros by using various programming language. This thesis will describe the methods for achieving the objectives. Start with the study of the programming language in CATIA software. The study has been conducted for the selection of the language that will be used in creating the program. The design subject is the tyre rim for the automobile. The design process of the rim is produced by using Knowledgeware workbench. Basically, the function of the program is to automate the repetitive tasks and speed up design process. The generated macro captures images of CATPart/CATProduct and saved the images in desired file location. It automates repetitive, non-creative design tasks which allow the designer concentrate on the creative sessions. The coding of the program that has been constructed is successfully demonstrated in this report. The problems or challenges encountered are also included and how they are ovecomed for useful reference. This work may indeed become a significant starting point to significantly improve the efficiency of the design process.

ABSTRAK

Projek yang telah dijalankan ini bertujuan membantu jurutera di dalam fasa rekabentuk bagi memudahkan kerja seharian dengan mengaplikasikan teknik 'Knowledge Based Engineering'. Ianya mampu mengurangkan kos and menjimatkan masa. Secara umumnya, teknik ini akan memendekkan tempoh masa rekabentuk dan juga mengurangkan kesalahan dalam proses rekabentuk. Objektif utama projek ini adalah untuk mencipta program dalam perisian CATIA untuk menjalankan tugas-tugas tertentu semasa fasa rekabentuk dengan menggunakan CATIA macros. Terdapat banyak cara untuk menulis macros dengan menggunakan pelbagai bahasa pengaturcaraan. Tesis ini akan membincangkan langkah untuk mencapai objektif. Bermula dengan pembelajaran bahasa pengaturcaraan dalam perisian CATIA. Pembelajaran ini membantu untuk memilih bahasa yang akan digunakan untuk mencipta program untuk projek ini. Subjek rekabentuk adalah rim kereta. Proses rekabentuk telah dihasilkan dengan menggunakan fungsi 'Knowledgeware'. Tujuan program ini dicipta adalah untuk mengautomasikan tugas yang berulang dan mempercepatkan proses rekabentuk. Macro yang telah dihasilkan akan menangkap gambar CATPart/CATProduct dan menyimpan gambar tersebut di dalam lokasi fail yang dikehendaki. Ianya mengautomasikan tugas yang berulang dan membolehkan pereka bentuk untuk menumpukan kepada sesi rekabentuk kreatif. Macro yang telah ditulis telah berjaya ditunjukkan didalam tesis ini. Masalah yang timbul sepanjang projek dan cara mengatasinya telah dibincangkan. Hasil projek ini mampu menjadi titik permulaan yang penting untuk meningkatkan kualiti proses rekabentuk.

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LIST OF ABBEREVATIONS

KBE	Knowledge Based Engineering
AI	Artificial Intelligence
CAD	Computer Aided Design
CATIA	Computer Aided Three-dimensional Interactive Application
COM	Computer Object Model
CAM	Computer Aided Machining
OLE	Object Linking Embedding
VB	Visual Basic
VBA	Visual Basic for Application
KWA	Knowledgeware Advisor
PKT	Product Knowledge Template
CAA	Component Application Architecture

CHAPTER 1

INTRODUCTION

1.1 THE BACKGROUND

Knowledge based engineering or also known as KBE stands at the cross point of diverse vital disciplines, such as artificial intelligence (AI), computer aided design (CAD) and computer programming. Nowadays, it is hard to find scientific books or journal that can be related to KBE. In addition, this topic did not enter into the mainstream academic research yet. To date, knowledge based engineering may become the potential asset where it can save the engineering cost and reduce the time for the product to be developed. Based on the literature survey, knowledge based engineering is defined in many ways, and in order to relate to this final year project's topic, knowledge based engineering can be defined as a technology based on the use of dedicated software tools called KBE systems, which can record information and systematically reuse the product and process, with the final goal of reducing time and costs of product development by several means (Rocca 2012).

Main benefit in adopting KBE is highlighted by the sets of curves in Figure 1. As the definition of KBE states, one of the KBE methods is to automate repetitive, non-creative design tasks. Beside saving time and cut the cost, it also help the user to enhancing creativity which allows exploration to the design field (Verhagen et al. 2012).

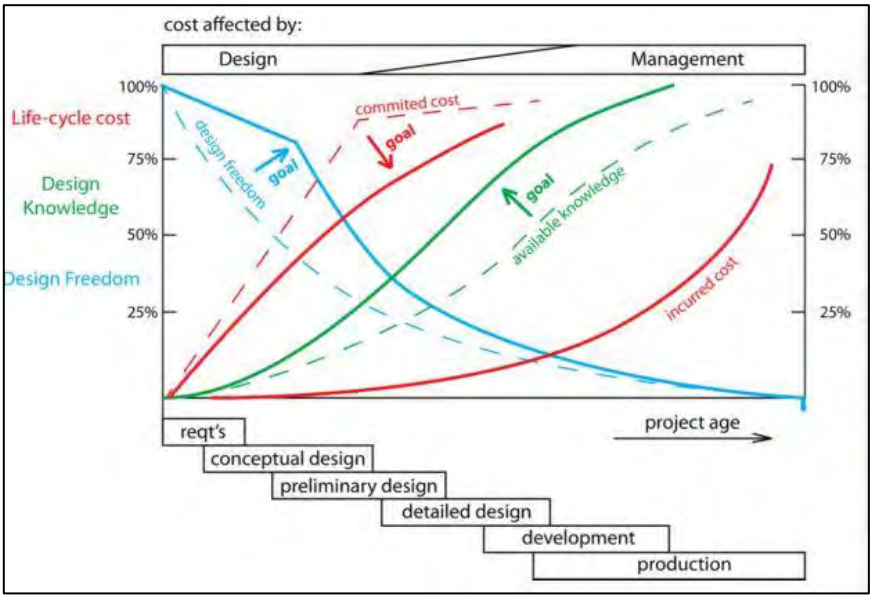

Figure 1 Product life-cycle cost, design knowledge related to design process (Verhagen et al. 2012)

Knowledgeware is not one specific CATIA workbench but several workbenches. The tools can be found on the standard toolbar in Part Design workbench. In short, Knowledgeware is a collection of tools that allow the designer or user to create or generate, manipulate the creations. CATIA Macros is one of the Automation which allows communication between several processes.

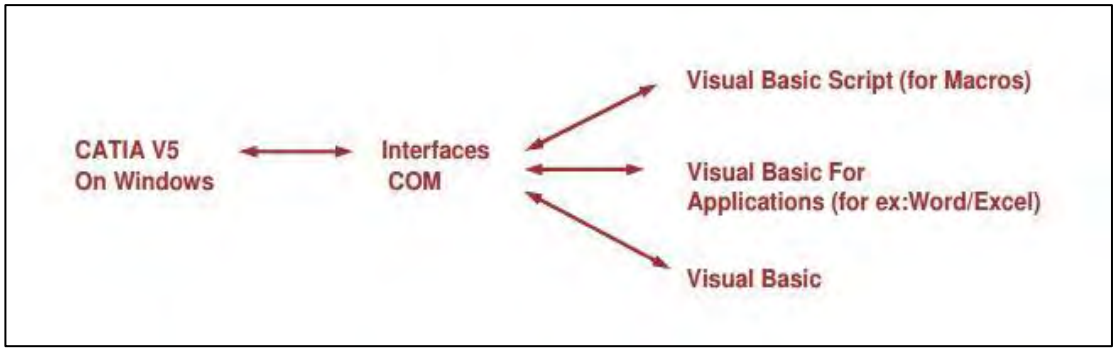

Figure 2 Interface of COM
(Source: Dassault System)

Based on Figure 2, COM (Component Object Model) is the Microsoft standard to share objects among several applications. Automation is Microsoft technology to use COM objects in an interpreted environment (Shahidi, 2012).

For writing script in CATIA, there are three possible languages;

1. Visual Basic
2. VBA
3. CATScript

1.2 THE PROBLEM STATEMENT

In an industrial environment, mostly in design department, automation is rarely applied in order to solve problems that may occur during compilation of data. Based on experience in an industry, some engineers complain that the huge set collection of data (CAT Drawing CAT Part, CAT Product) needs to be open one by one and it lead to time wasting and inefficient work. The design part for case example for this project is the automotive tyre rim. For designer to design 3 rims with different specifications, it would take a long time to complete and decrease the focus for creative design task. . Hence, this project focuses on solving problems that industrial engineer facing by creating macros that automate repetitive tasks and speed up the design processes

1.3 THE OBJECTIVES

The objectives of this project are as follows:

1. To investigate the language of scripting that applied in CATIA macros.
2. To apply Knowledge Based Engineering for designing process of the automotive rim.
3. To create a program that can facilitate or automate the capturing of multiple images (looping process) by using CATIA macros.
4. To save the images that has been captured into designated file or location.

1.4 SCOPE OF PROJECT

The scopes of this project are:

1. CATIA V5 software will be employed to run the program. It offers Macros toolbar where the script can be written.

1.5 GENERAL METHODOLOGY

The actions that need to be carried out to achieve the objectives in this project are listed as below:-

1. Learning the language
   Firstly, languages of macros that will be used for scripting need to be learnt. After selection of the language, the project can proceed to next phase to create macros.

2. Literature and Survey
   To appreciate the problems and challenges encountered by the engineers, questions will be asked for further improvisation of the project objectives.

3. Analysis and proposed solution
   Analysis will be presented on how the program will be executed by explaining the script line by line. Solutions will be proposed based on the analysis.

4. To demonstrate the application of the generated macros for example case.
   In this case, the automobile tyre rim will be design by applying Knowledgeware and the macro will run to test the result.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

This chapter will describe about the scientific studies on the project. Since this project was not an experimental in nature, it is very challenging to find relevant references from the journals or books that are related to this project. Nevertheless, some reviews were found which has helped to understand the project were collected and summarized. Therefore, reviews for this study were quoted from the websiteb internet, thesis and journal.

2.2 COMPUTER AIDED DESIGN (CAD) / KNOWLEDGE BASED ENGINEERING

For this project, CATIA software was used as the platform in order to address the problem. CATIA stands for Computer Aided Three-dimensional Interactive Application which is a multi-platform for CAD/CAM and CAE. It was developed by the French company, Dassault System and was written in the C++ programming language. CATIA was created in the late 1970s and early 1980s.

Knowledge Based Engineering system is a usage of proper computer software for obtaining and recycling knowledge on a product and process in a most integrated way. Therefore, it is especially beneficial in the event where the design is constantly develop. As stated by (Skarka 2007), creation of a good project depends largely on creativity of the designer himself. In other words, in order to have a good project, it is advised to devote, in optimal solution, much more that 20% of designing time for creative actions.

Furthermore, in order to make it possible, it is necessary to reduce time devoted to routine task significantly. One of the methods of aiding routine tasks is the usage of KBE technique.

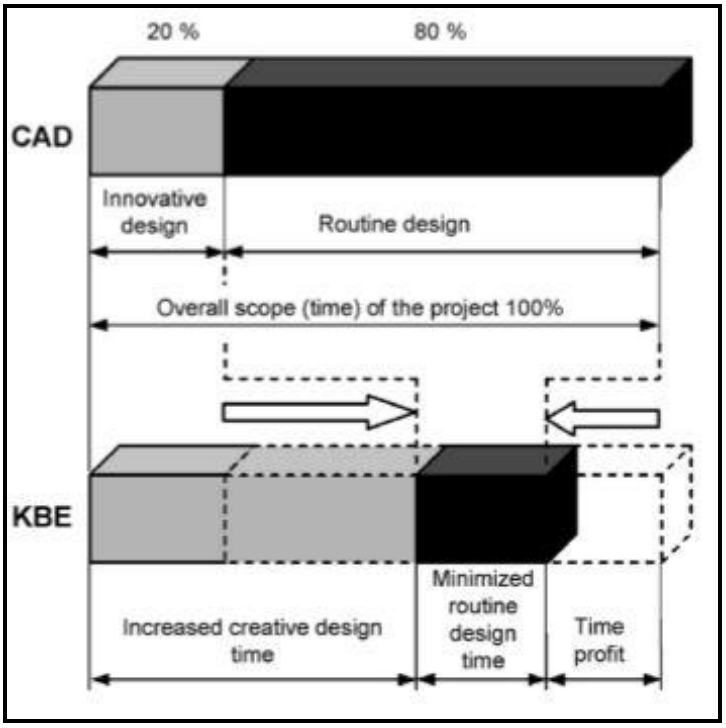

Figure 3 Influence of KBE usage on time of main design tasks.
(Skarka 2007)

2.2.1 SECONDARY DEVELOPMENT OF CATIA

CATIA V5 is an open system for developing innovative macros for special needs. Macro is a code that written in certain programming language which can create, analyse modify etc. In Computer Aided Manufacturing (CAM) operations, macros is useful for generating the product as desired by designer. Macros are developed using CATScript, VBScript or VBA script. For developing a macro in CATIA V5, all we need is the inputs, outputs and necessary supporting data from user.

According to (Du et al. 2014), there are a variety of ways to access the CATIA object in the secondary development of CATIA. The in-process application has access to CATIA object by VBScript, CATScript, or VBA script. The script that has been written by the designer will be recorded in CATIA macro and can be modified as desired by the designer. For the out-

process application, Component Object Model (COM) will accessed to CATIA object.

Based on (Kong et al. 2012), in the CATIA environment, when the macro executed, CATIA is in a non-activated and variables cannot be stored between calling macro. This relatively simple approach can be completed in the CATIA environment. In in-out process applications, CATIA and external applications run in different process address space.

2.2.2 CATSCRIPT

According to (Constantinovici 2013), access to the CATIA object model is provided using scripts in different ways depending on the operation system and on the application that can share their own objects with CATIA. CATScript is one of the CATIA programming languages that allow compability between the Unix Basic Script engine and the Windows VBScript engine. In addition, CATIA does not host the Basic Script engine on Unix since V5R7. Hence, it has only been kept for compability purpose. It is actually operated by a VBScript engine after exclusion of the typing information.

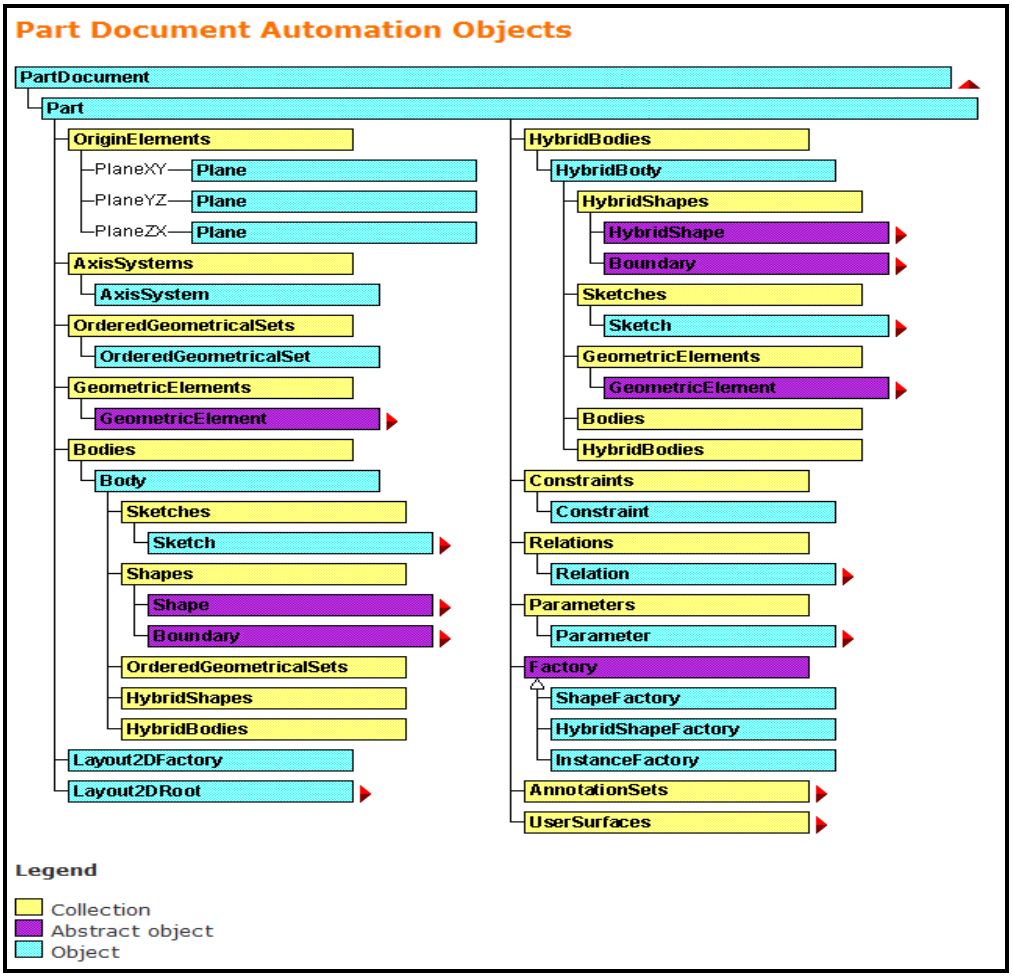

Figure 4 Part modelling object tree

(Constantinovici 2013)

In CATIA, the part document automation objects are functioning to create certain part model, for example, to define the plane of sketch, creating 3D model, are all under a tree as shown in the figure above. The part object can be developed using macro programming for customization or automation in CATIA V5.

2.3 KNOWLEDGEWARE REPRESENTATION IN CATIA SYSTEM

Knowledgeware in CATIA is a function of group that specialized in knowledge and allowing expanding design features with knowledge elements. It has submodules that focus on different type of knowledge. Some of the vital modules that Knowledgeware offers are:

- Knowledge advisor
- Knowledge expert
- Product knowledge template
- Business process knowledge template

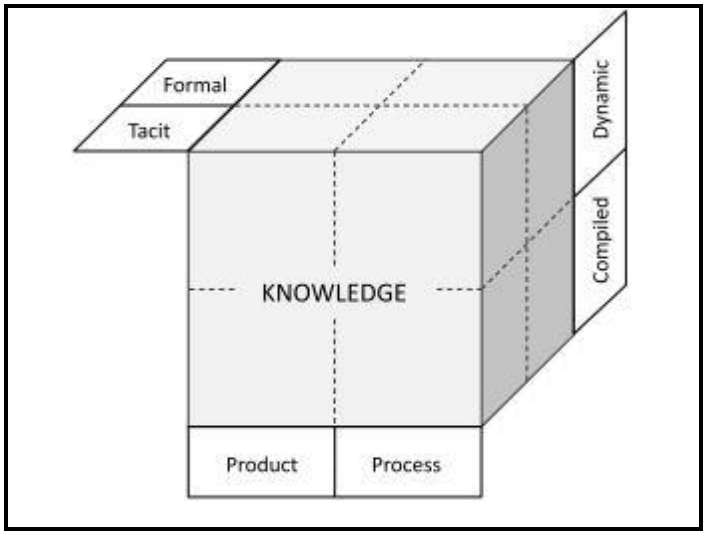

Figure 5 Classifying knowledge-the different dimensions.

(Source: (Chandrasegaran et al. 2013)

According to (Skarka 2007), these tools offers many utilities that permit automation of the choice of design features with the use of different forms of knowledge representation by ready-made tools:

- Parameters: simple tool to parameterize a component.
- Published parameters: defining relations.
- Formulas: to determine the features using formula.
- Design table: to define the product by dimensional families.
- Rules and check: to implement determining product features in provisional technique to match rules.
- Power copy: document template-enable reuse of entities of product.
- Reactions: which can add a behaviour to a component.
- Set of equations: to solve equations and inequalities.
- Scripts.

These functions allow to integrate the know-hows using the Knowledgeware language and to optimize them in order to facilitate technical decisions, reduce errors and automate design for maximum efficiency. Figure 6 below generally show the solution overview limited to KWA and PKT products.

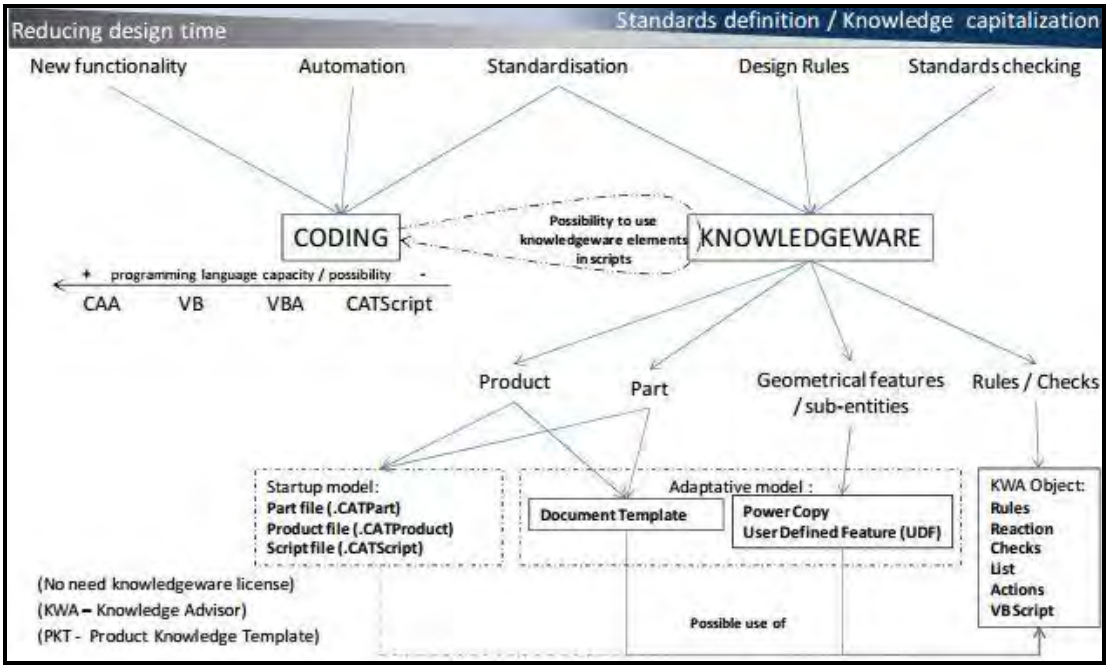

Figure 6 Knowledgeware solution overview

(Bodein 2009)

2.4 USING A MACRO TO THE FURTHER DEVELOPMENT

Users use macro recording a sequence of operation, automatically generated code and use certain language (VBScript/VBA/Catscript etc) as editing tools. It is customized interactive way. Automation interface can call “InputBox” and “MsgBox” function to get user input information and output.

According to (Kong et al. 2012), the users can apply more complex Visual Basic to define more input and output interfaces. An icon can be associated with macro in running, and be into CATIA display frames. The characteristics of using the macro to the further development of CATIA are convenient, not only

to achieve the desire function, but also to record the macro method and obtain the required procedures.

2.5 SUMMARY

In this chapter, literature review managed to enhance the knowledge about CATIA and its functions. Apart from that, after close reference to the journals, websites and books an increase in the understanding in CATIA macros and how it was achieved. The topic definitions and explanations that are related to the project can be found in this chapter.