

Real time object customization in CAD system

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ABSTRACT

Object customization in Computer Aided Design (CAD) is a method used to modify the sketch parameters and change the model geometries. This method is one of important features in part modelling which empowered CAD user to simply modify their product. The conventional method of modifying CAD model is usually relied on the manual editing. This paper outlines a development of program than can be integrated in CAD/CAM system for real time object customization. It uses Visual Basic (VB) programming with custom Graphical User Interface (GUI) generated in NX10 (CAD/CAM software) interface. The main contribution of this research is a tool for real time object customization that capable to assist the modification of 3D model. In order to develop the tool, 3D model modification instructions are translated into programming codes by using Journaling tools in NX interface. Editable parameters are identified based on the model shapes. The related codes are reconstructed, modified and linked to the functional GUI. This allows user to simply modify the model shapes in real time with minimum process steps. The performance of the method is evaluated in terms of the reduction of process steps and time in modifying the 3D model in CAD system. All the findings and details of approach are presented.

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1. INTRODUCTION

CAD has been traditionally used to assist in engineering design and modeling for product representation, analysis and manufacturing [1]. Manufacturing industries over the world are using CAD to speed up their product development instead of relying on manual methods. One of the contributions of modern 3D CAD to accelerate the product development process is the ability to reuse and make alterations to existing model in efficient and relatively easy way [2].

Many researchers have identified the ability to re-customize previous design as an essential factor in modern engineering and product development scenarios [3-5]. In the process of customizing the existing design, designer needs to deal with a lot of repetitive tasks which time consuming and lead to mistakes [6]. Generally, design phase corresponding partially for time waste in product development process [7-9]. Process of product customization by using CAD is one of the common processes in design phase [10]. In order to reduce the development time and minimize error, turning to automation is the best approach to speed up cycle time from product design to manufacture.

Object customization in CAD can be carried out by using the parametric modelling technique [11]. Basically, the geometry is mainly controlled by non-geometric features called parameters [12], which can be defined by dimensional, geometric or algebraic constraint. Parametric CAD enables user to do rapid alteration of existing models by simply editing the values of respective parameters. Modification of object will happen right after the user finish to set the parameter value. This technique comes in forms of graphical

user interface (GUI) where the value of the parameter needs to be filled up by the user. GUI allows interaction between user and the computer to accomplish desired goals [13]. The object can be modified but it requires user to know the parameters value and run the program before the modification tool place in the model interface. The drawback of this technique is the modification towards object only happened after user finish to set the parameter value which make this modification process is not in real time.

An example of this issue can be observed in customization works related to furniture industries. Most of the customer would require furniture that can fit perfectly with the available space. Hence, designer needs to deal with a lot of modifications in term of size, additional compartments and shapes. To execute these tasks manually will be time consuming as it requires several repeated cycles before the final design is achieved. In order to resolve this issue, the modification instructions used in CAD can be translated into programming language which leads to the automation in modifying the model. The codes generated can be exploited and modified, then link with the custom GUI to allow real time object customization in the modelling stage. Providing a freedom to customers to control certain design parameters will boost their excitement and feeling in purchasing the product.

In recent years, object customization technique became important studies among researchers and most of them are focusing on the parametric modelling technique to facilitate object customization. Therefore, this paper intended to extend the capabilities of this technique further by integrate with real time abilities to assist the modelling tasks and product visualization.

1.1. Development in Part Modelling

Several studies conducted previously have noted several methodologies to enhance the system and shorten the product development stage. S. Siddesh *et.al* [14] has proposed a new technique to generate CAD models automatically. There are two methods formulated, one is by writing a complete program manually and another one is the utilization of macro tool in the modelling software. The aim of this development is to shift from manual modelling technique to automation modelling process. CATIA has been used as a platform for their proposed model. A. Lad *et.al* [15] developed a software application for product design using SolidWorks CAD package which supports the Application Programming Interface (API). The software is able to automatically update the CAD model by formulating past design experience, knowledge and procedure in computer database. This information is exchange repetitively by integrating Microsoft Visual Studio with SolidWorks CAD. Manipulating the input parameter in Visual Basic will change and affect the output in SolidWorks interface. Another development involves a computer-aided systems for automatic modelling of compound die [16]. The developed software is able to generate 2-dimensional and 3-dimensional drawings for all major die components and die assembly automatically in the drawing editor of AutoCAD software. Parametric modelling technique has been used extensively to execute the operation. The result has successfully simplified the method of modelling for compound dies.

Study conducted by T. Chen *et.al* [17] has established VB and SolidWorks-based 3D Fixture component library as a plug-in function in SolidWorks menu. The system aims to simplify the translation of 2D to 3D-drawing through the developed GUI. It allows users to select the standard parts through SolidWorks menu and then the application program will automatically query all the data required for the drawing. This program has been created by integrating SolidWorks and Visual basic Programming software through parametric modelling technique. Another development proposed a method to read the values from MS Excel file using macros and generate solid model in CATIA V5 [18]. This technique will automatically send the calculated model parameter values from MS Excel to CATIA V5 using macros and later produce the solid model in CATIA interface. The automation is achieved by using macros in CATIA V5 and Visual Basic software. The macros are written/recorded in visual basic programming language. By using this technique, the repetitive tasks in designing component can be reduced. M. Hafiz *et.al* [19] and M. Zahid *et.al* [20] through their research has developed the functional GUI to simplify the simulation of process planning and reduce the dependency on user's experience while developing the machining program in NX CAM system. The developed GUI uses the parametric modelling methods.

To sum up the review, there are several attempts have been made to assist the modeling tasks in CAD system. The assisting tools are developed and work in different CAD system for instance SolidWorks, CATIA and AutoCAD. It is important to note that most of the development shared a similar methodology to assist the modelling tasks by implementing the parametric modelling technique. User needs to key in the required parameters in the system through specified GUI before the instruction being executed. In most cases, the translation process took sometimes before the changes are visible in CAD interface. Currently, very limited development claimed the modelling process can be executed in real time based as the input information changed. Therefore, the present work is an attempt to propose a methodology for a real time object customization in CAD interface by implementing parametric modelling technique.

2. RESEARCH METHOD

The modelling task focused in this study is based on the modification of object through several parameters that will affect their shapes and sizes. The integration between NX 10 and VB is explored through the NX Open Tools. The tool provides all the programming instructions used to assist the modification of model which later can improve the automation and fusion of the tasks. The programming codes generated are customized and integrate with several commands to provide the real time capabilities. The customization work involves four main steps distributed between the model in NX system and VB programming. Figure 1 summarizes the process flow of the methodology employed in this study.

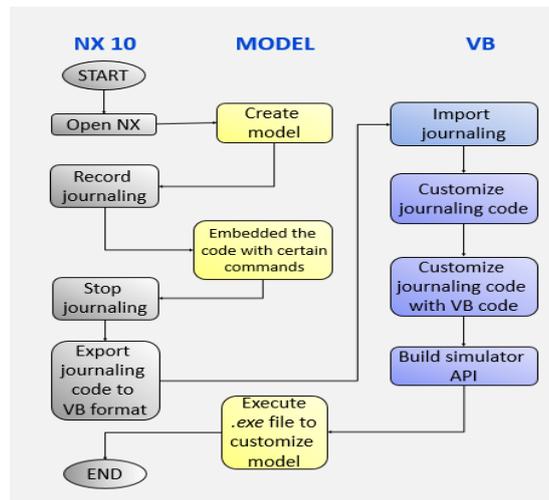


Figure 1. Overview of system flow chart

2.1. Part modelling

The development of the methodology employed two examples of product that can be modified due to the technical requirements or user preferences. The first model is shaft coupling. In the real application, the hole diameter in this coupling part is varies based on the specific technical requirements [21]. In order to accommodate with this requirement, the GUI is developed to control the parameters that reflect the holes and visualize the changes in CAD interface in real time. Figure 2 highlights the hole features of the coupling that can be customized using the developed program.

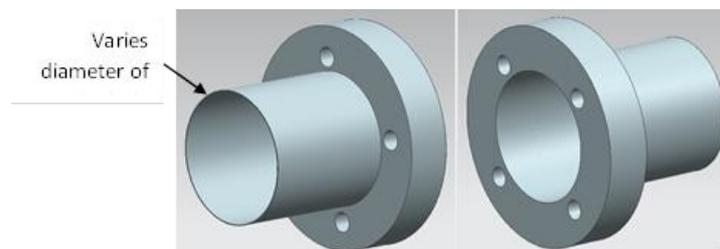


Figure 2. 3D drawing of coupling model

2.2. Journal record and translation

Journal function in NX10 is an important tool to provide basic programming codes based on the specific instructions executed to modify the part model [22]. The codes can be exploited further and modified through VB.net software. The translated codes come with the section headings which allow user to understand the operations it deals with.

For the shaft coupling model, the parameter that need to be controlled is the size of the centre hole and this can be made through the developed GUI. To enable this, new parameter value must be modified in

the specific code that will reflect the geometry changes in the system interface. For example, the new diameter of the centre hole needs to be changed on the code. During the process of hole size modification, journal will record the instruction and generate the code as exemplified in Figure 4.

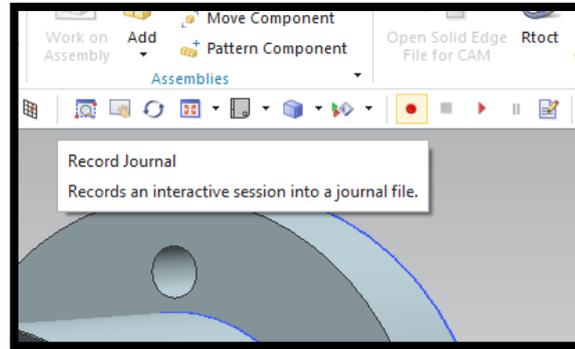


Figure 3. Recording journal button

```
Dim markId3 As NXOpen.Session.UndoMarkId
markId3 = theSession.SetUndoMark(NXOpen.Session.MarkVisibility.Invisible, "Radial Dimension")
theSession.DeleteUndoMark(markId3, Nothing)

Dim markId4 As NXOpen.Session.UndoMarkId
markId4 = theSession.SetUndoMark(NXOpen.Session.MarkVisibility.Invisible, "Radial Dimension")
sketchRadialDimensionBuilder1.Driving.ExpressionValue.RightHandSide = 50
```

Figure 4. Captured code

The highlighted code in Figure 4 which ‘sketchRadialDimensionBuilder1’ represents a variable of the diameter value for the center hole which set to 50mm. this variable need to be find in the journal recorded. In order to allow user to control this parameter, the value is replaced with the variable function ‘HScrollBarNew.Value’ as illustrated in Figure 5. This variable represents the scroll function available in VB software for customization process. In order to communicate with the system user, a slider function is attached in the program GUI. It allows user to modify the size by simply move the slider which directly reflects the model. The developed program will copy the new value of hole sizes and exported to NX through the API. By implementing this method, the changes of hole dimension values will simultaneously reflect the model appeared in NX interface.

```
Dim markId3 As NXOpen.Session.UndoMarkId
markId3 = theSession.SetUndoMark(NXOpen.Session.MarkVisibility.Invisible, "Radial Dimension")
theSession.DeleteUndoMark(markId3, Nothing)

Dim markId4 As NXOpen.Session.UndoMarkId
markId4 = theSession.SetUndoMark(NXOpen.Session.MarkVisibility.Invisible, "Radial Dimension")
sketchRadialDimensionBuilder1.Driving.ExpressionValue.RightHandSide = HScrollBarNew.Value
```

Figure 5. Variable replaced

2.3. GUI design

In order to integrate recorded journal with the function of GUI in VB programming, extra code need to be added in the programming section to activate the recorded journal in VB programming [23]. This extra

code called header code that act as declaration for each function use to integrate both software between NX and VB [24]. Thus, allows the functionality of the GUI to be executed in the NX session during customization process. Figure 6 show the series of code functions that need to be declared in the global declaration in the programming section. Blue words stand for reserve word for every function available in VB software [25]. After the extra code has been added, customization toward GUI started with creating the new windows form. The rest of other functions such as button, slider and label are plotted on the windows form as shown in Figure 7. Journal recorded is then combined with desired function to control the modification as for example slider function that used to customize the circle diameter.

```
Option Strict Off
Imports System
Imports NXOpen
Imports NXOpenUI
Imports Microsoft.Office.Interop
Imports System.Collections
Imports System.Collections.Specialized
Imports System.IO
Imports System.Xml
Imports System.Math
Imports System.Collections.Generic
Imports NXOpen.UF
Imports NXOpen.CAM
Imports System.Windows.Forms
```

Figure 6. Header code

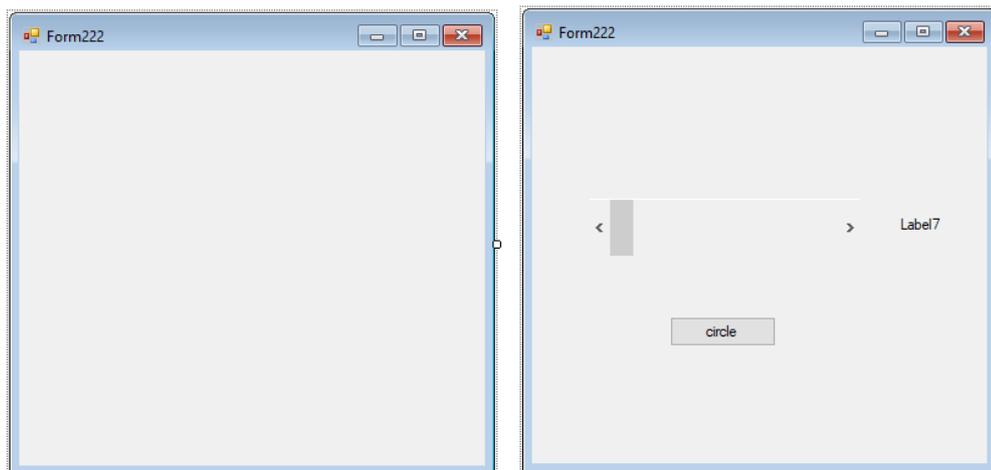


Figure 7. Windows form

2.4. Execution of *exe. file*

The GUI of the program is designed by using standard tools in Visual Basic studio with several functions to cater object customization. One of the important control tools is a horizontal slider which serves to provide input for the model parameter. This slider allows user to modify the geometry shapes within the permitted level based on the part specifications. Label function on the GUI provides the value of parameter simultaneously with the movement of slider. Meanwhile, the finish button used to close the program once user satisfies with the changes. In order to execute the program, user needs to open the model in NX interface or simply use the keyboard shortform 'Ctrl + U' and find the file '.exe' in specific folder. Then, the GUI as illustrated in Figure 8 will be appeared and ready to use.

Figure 9 visualizes the program in the interface with the part model to be modified. In this example, the model will update simultaneously when the slider button moves. Once the expected diameter is achieved, the model can be saved by selecting the finish button. During the modification process, user can observe a live preview of the changes occurred in the model.

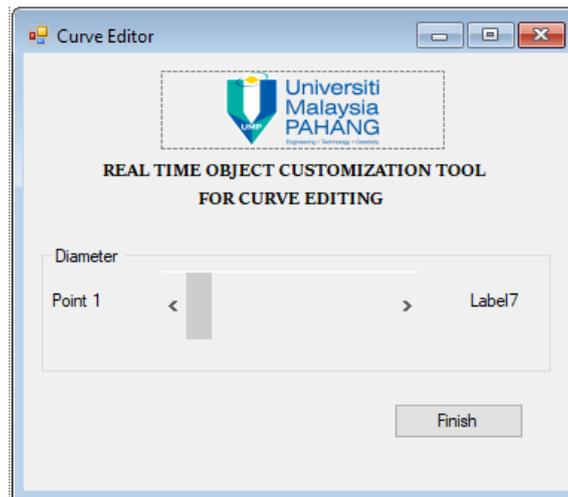


Figure 8. GUI designed by using available tools in Visual Basic studio

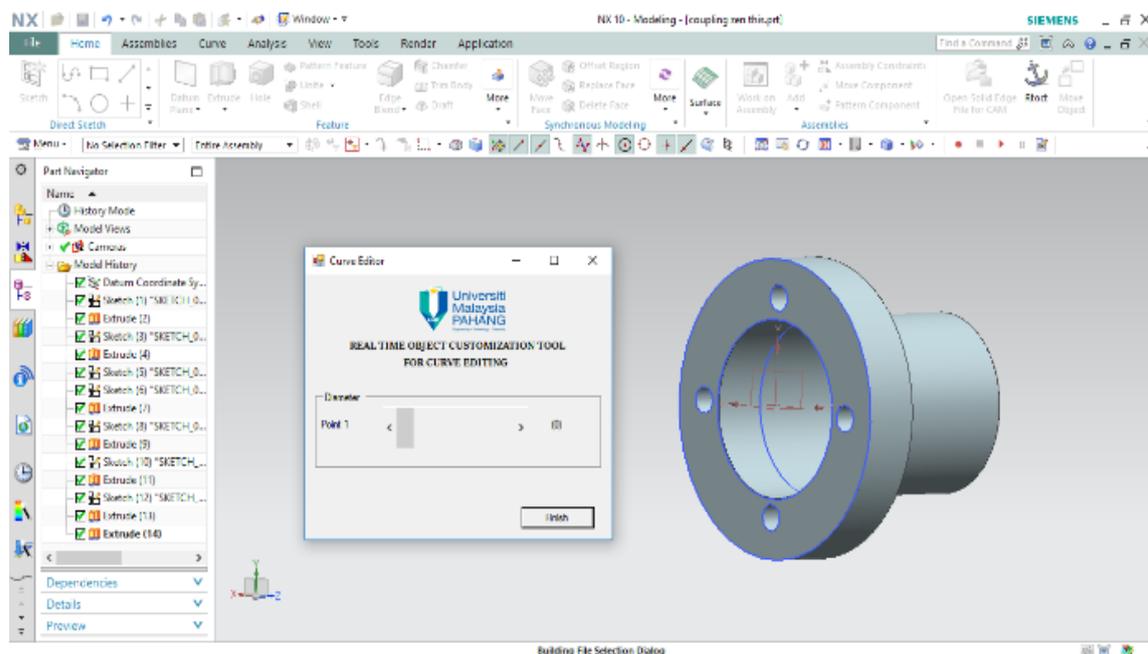


Figure 9. Developed GUI is activated in NX10 system

3. RESULTS AND ANALYSIS

The developed program for real time object customization is verified based on different level of CAD users, classified as expert and beginner users. Expert users represent by the people with more than 3 years of CAD experience while the rest below 3 years CAD experience are considered as beginner users. There are two models developed to validate the program, first is the coupling model and another is a guitar model. The customization involves in the first model are related to the modification of hole sizes. This process is visualized in Figure 8 which indicates the changes of hole diameter in the model.

For the second model, the customization is related to the changing of the B-spline curve shapes exhibited in the guitar. This modification is an example for the product that needs to be designed based on the customer preferences. The center curve of the guitar can be adjusted to fit with the user body. To execute this, the curve is modified based on the control points of B-spline surfaces. Therefore, the GUI contained several sliding cursors to adjust and change the shapes.

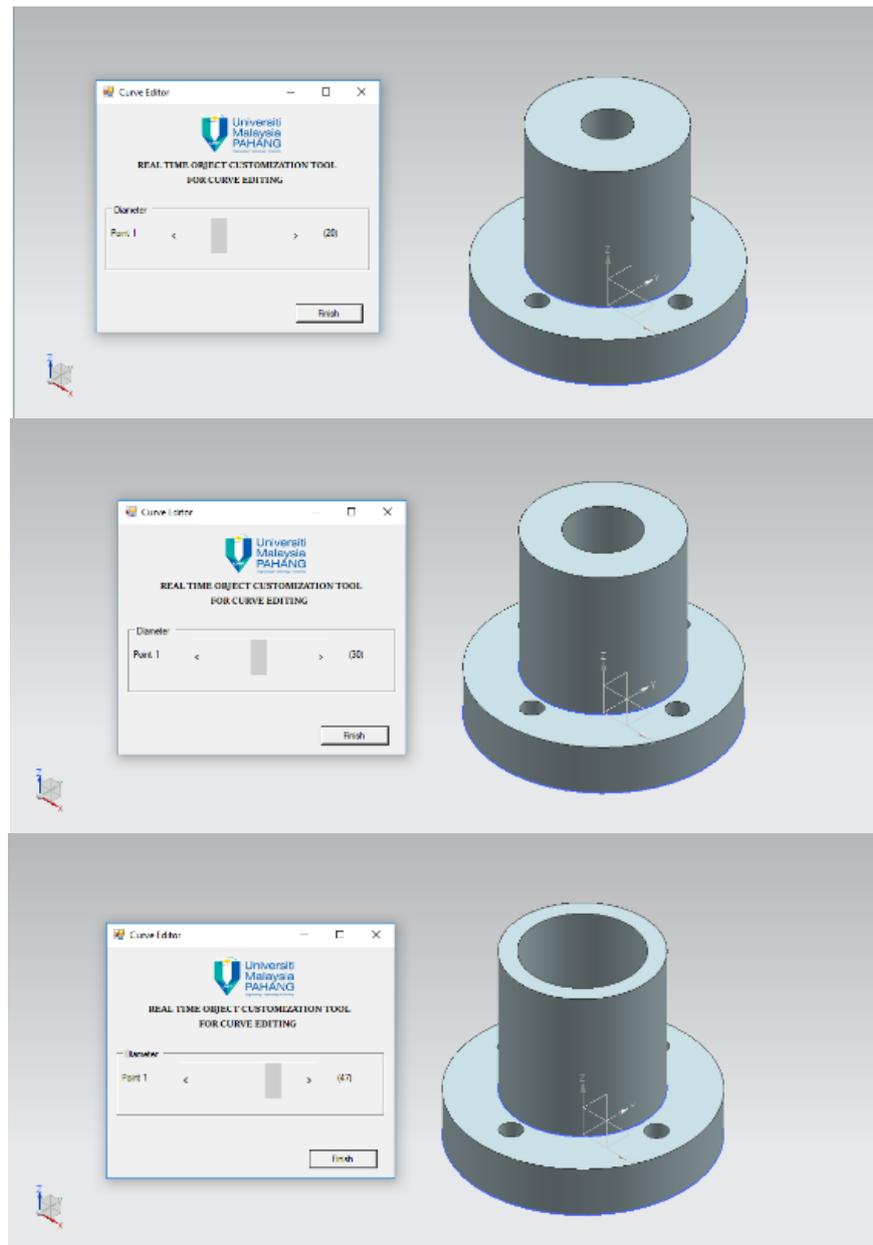


Figure 10. Customization on the circle dimension

Table 1 and 2 summarized the result in term of time required to modify the models. This time study is taken based on two categories user that relied on their experience in CAD. There are two sections of recorded time based in manual approach and the proposed approach. In general, the result indicates the time saving can be achieved by implementing the developed program for model modification. Using manual approach, a lot of time is spent on searching and selecting necessary tools to customize the model.

Depending on the complexity of the model, the time taken for modification would probably extend in the manual approach. On the other hand, by implementing the proposed approach, the steps taken to modify the model can be minimized roughly from 5 to 3 steps of instruction in CAD interface. In manual approach, the 5 steps involve re-open sketch, search for target area, select target area, define new parameter and save the change. Meanwhile, using the developed tool, user only needs to call the customization program, define new parameter and save the changes.

In the guitar model, the developed program manages to achieve 84% of the time saving for expert user and 43% for beginner user in customizing the shapes of the model. Since the model contained several B-spline curves, to manually modify the geometry is considered a quite complex task and need a skills CAD

user to deal with the process. However, by using the program, the steps for customization have been simplified and even beginner user can easily play with the tool. Based on the data in Table 1 and 2, the time saving efficiency are ranged from 55% up to 94% for Table 1 and 42% up to 84% for Table 2 which indicate the effectiveness of the program developed to assist model modification in CAD interface.

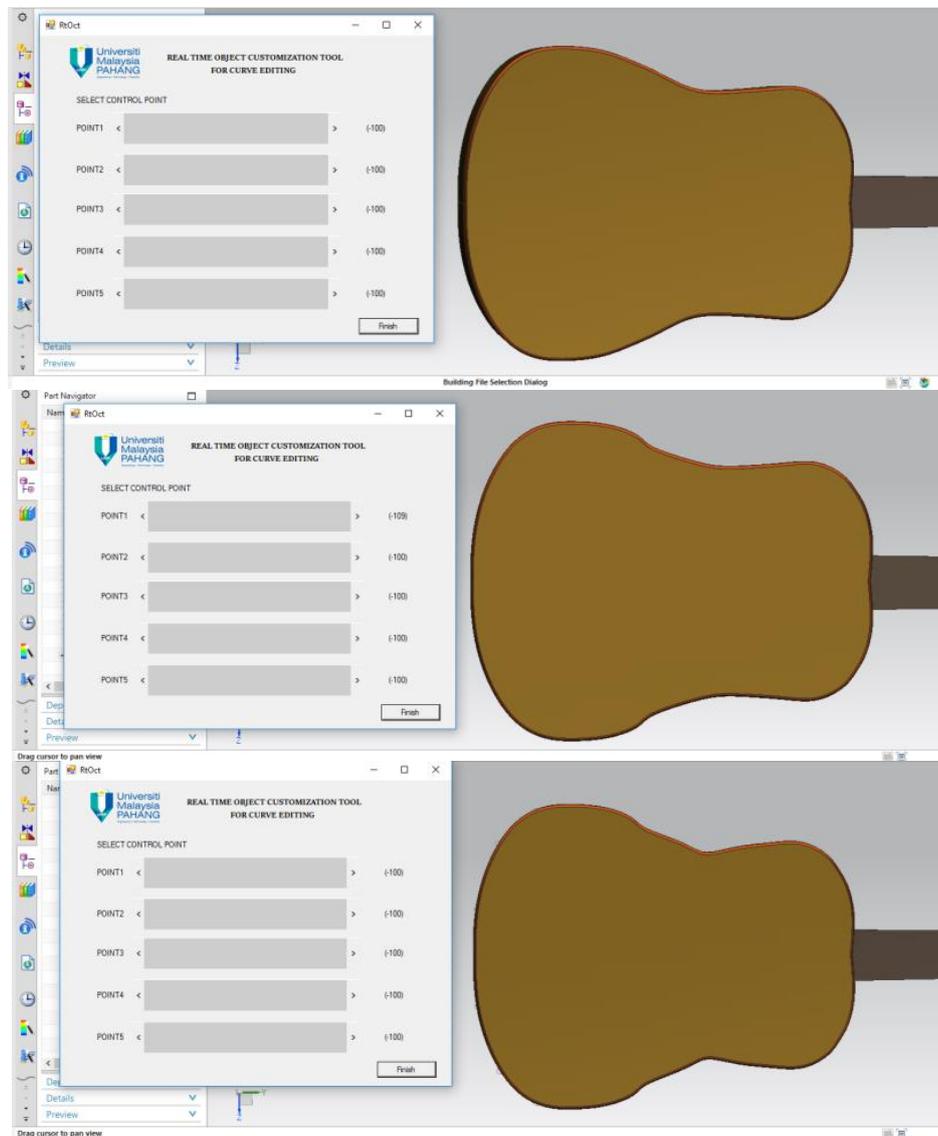


Figure 11. Customization on the B-spline control point

Table 1. Experimental results for coupling model

Coupling model	Time recorded in manual approach (sec)	Time recorded using the program (sec)	Time saving (%)
Expert 1	40	18	55
Expert 2	45	20	56
Expert 3	37	17	54
Beginner 1	283	20	93
Beginner 2	300	18	94
Beginner 3	295	17	94

Table 2. Experimental results for guitar model

Coupling model	Time recorded in manual approach (sec)	Time recorded using the program (sec)	Time saving (%)
Expert 1	30	17	43
Expert 2	35	19	45
Expert 3	29	18	38
Beginner 1	120	19	84
Beginner 2	138	20	86
Beginner 3	119	20	83

4. CONCLUSION

The work described in this paper suggest an alternative methodology for real time object customization in a CAD system. The developed program operated by control and manipulating the instructions used to modify the CAD model through Visual Basic programming. Instead of editing the model manually, the developed program realized a real time customization where it allows user to adjust and change the parameters on the GUI which simultaneously modify the shapes in CAD interface. There are a lot of way to utilize the developed program. One of the significant contributions is that its empowered customer to deal with the customization of any product within the permissible specification of the part. As a conclusion, the developed program managed to assist CAD model customization tasks by reducing the time and steps taken in the modification process. Future work will cover on the remove stickiness process which allows the GUI to be used towards any sketch in NX CAD software for object customization purpose.

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