International Journal of Applied Engineering Research
ISSN 0973-4562 Volume 10, Number 6 (2015) pp. 16111-16120

http://www.ripublication.com

Knowledge – Based Parametric Modeling for Bolts, Nuts and Bearings using SolidWorks

E. Jayakiran Reddy[1,a,*], C.N.V. Sridhar[1,b], V. Pandu Rangadu[2,c]

[1] Department of Mechanical Engineering,
Annamacharya Institute of Technology and Sciences, Rajampet, Kadapa (Dist),
Andhra Pradesh, India – 516126
[2] Department of Mechanical Engineering, JNTUA College of Engineering,
Ananthapuramu, Andhra Pradesh, India – 515002
[a]ejkiran@gmail.com, +91-9963786665; [b]sridharcnv@yahoo.co.in, +91-9948149739;
[c]pandurangaduv@yahoo.com, +91-9440686695;

Abstract.

Generally the design of components involves numerous calculations. These calculations can be easily computed with an interactive computer program by considering the designers inputs. Knowledge Based System (KBS) enables the user to generate the required CAD model based on these computed values. In this research work an attempt has been made to develop a KBS for modeling of Bolts, Nuts and Bearings with the help of parametric modeling technique by integrating commercially available CAD package SolidWorks with Microsoft Access. Various product variants of bearings have been executed with the developed concept. Further it is proven that, this KBS can handle the modeling of assembled component.

Keywords: Knowledge Based System; Parametric Modeling; SolidWorks API; Computer Aided Systems; Bearing Design; Macro;

1 Introduction

CAD (Computer Aided Design) became as the inevitable design practice in the recent decades because of ease in 3D solid visualization, generation of drawing and documentation. However, modeling is a time consuming process because of lack of skilled CAD modeling professionals. This can be overcome by the parametric modeling technique as it regenerates the task in very less time in comparison with human. As the parametric modeling technique is dimension driven it facilitates automatic re-use of existing design process based on the results on engineering analysis. Additionally, parametric modeling technique is

highly efficient, good interactive and less time consuming in comparison with conventional 3D modeling techniques.

Bearing is a machine element which provides relative positing and rotational motion by reducing the friction between the moving parts. The designing process of bearing should consider free linear or rotational movement on the moving parts. Developing a part model for bearing includes the modeling of inner ring, outer ring and rollers. The parameter like width, inner ring diameter, outer ring diameter, number of rollers etc. are required for modeling the bearing. Similarly, the diameter is the key parameter for modeling of the Bolts and Nuts.

1.1 SolidWorks API

Application Programming Interface (API) is a tool which provides integration between different applications by providing a code in a programming language within another application [1]. SolidWorks API supports programming languages like Visual Basic (VB), Visual Studio and C++. With the help of SolidWorks API the automation of modeling, assembling of components is possible by writing specific codes. SolidWorks API is a tree like hierarchical structure which covers all the functions of the software. For improving the design efficiency of a component Bo, Qin, and Fang [2] developed a standard parts library using VB forms with SolidWorks API functions. Similarly, Tian and Liu [3] developed re-use software for designing the components by applying SolidWorks API.

1.2 Visual Basic fundamentals

Visual Basic (VB) is primary development programming language with which API can access graphical user interface (GUI) [4]. Additionally, VB is a successful engine for generating the macros in all Microsoft software. For this reason, VB has become a very useful tool for generating different programming codes for many applications. VB is the engine of Microsoft Access for building the database and which facilitates the programmer for efficient database control [5]. With the help of ActiveX DLL (AvtiveX dynamic link libraries) programmers can integrate VB with different windows applications and also can develop new menus, tools and toolbars into the application environment [6].

2 Literature Review

The area of parametric modeling has been attracting numerous researches with the focus on the relationship with solid modeling as it simplifies the complexity of the modeling by automating the commonly performed tasks using a predefined algorithm [7]. In the process of generating an efficient modeling process, Kong et al. [8] developed a SolidWorks API based 3D plastic injection mold design system using Visual C++ programming language. By controlling the database software remotely, Wang et al. [9] was able to change the existing design of the bottle by interfacing the CAD software with parametric sketching. Kuang et al. [10] has developed parametric part family system by interfacing Microsoft Excel with UniGraphics software with Visual C++ programming language. Chu et al. [11] developed a computer aided

parametric design system for 3D tire mold production in CATIA using CAA (API tool for CATIA). Ongkodjojo and Gunawan [12] demonstrated the technique to control the CAD model by manipulating and customizing the parameters associated with flat bed conveyor design. Bor et al. [13] developed a knowledge based parametric design system for drawing dies using Pro/E CAD software with parametric modeling technique. Naranje et al. [14] developed the system modules for the development of knowledge based system for automated design of deep drawing die for axisymmetric parts using AutoLISP language for AutoCAD software by utilizing VB 6.0 for creating user interface. A new expert system was developed by Hussein [15] for the design of computer aided blanking die on top of CATIA V5 using CATIA script, visual basic.

3 Methodology for Modeling Automation and Implementation

SolidWorks is one of the most powerful secondary generation tools for validating the parametric model of the component. SolidWorks reproduces all geometric data of the part in text data form. This text can be modified with API to make the part model driven according to the inputs of the user. Here Microsoft Access and SolidWorks are integrated for the transfer of data from database to SolidWorks for facilitating automatic and/ parametric modeling.

3.1 Preparing the Microsoft Access Database

As SolidWorks API supports Visual Basic language, Microsoft Access database management has been chosen as the database because both are from Microsoft Corporation. Three different databases have been prepared with all the respective relevant data for designing and modeling of the Bolts, Nuts and Bearings. For example, the database for Bearings contains outer ring diameter, inner ring diameter, width, average life, basic static load, basic dynamic load, maximum speed, corresponding SKF and/ ISI number etc. A sample database relating to deep groove ball bearing is shown in Fig. 1.The database has been adapted such that the user needs to give only minimum data, so that all the relevant feature dimensions will be retrieved from the database for respective calculations.

SKF_ID	Dia Max	Dia Min	Width	ISI_ID	Max_Speed
6200	30	10	9	10BC02	20000
6201	32	12	10	12BC02	20000
6202	35	15	11	15BC02	16000
6203	40	17	12	17BC02	16000
6204	47	20	14	20BC02	16000
6205	52	25	15	25BC02	13000
6206	62	30	16	30BC02	13000
6207	72	35	17	35BC02	10000
6208	80	40	18	40BC02	10000
6209	85	45	19	45BC02	8000
6210	90	50	20	50BC02	8000

Fig. 1. Microsoft Access Database of Deep Groove Ball Bearing

3.2 Developing a genetic model

For developing the generic model, macro file has been used. Macro is a code generated when any operation like mouse click, drawing a line, extruding a surface etc is done in SolidWorks screen. All the operations performed are stored in the form of code. Macro has the ability of recording and replaying the operation. The code generated in macro is in the form of a module of a Visual Basic Application (VBA). Macro uses the 3D coordinate system to store the dimensions and the position of any entity in the component.

First, a 3D model of the component with all the proportionate features and constrained dimensions has been generated while recording its macro file. Later, this recorded macro file is customized to fit into VB code of SolidWorks API for carrying out the corresponding modeling job of the component.

3.3 Generating the dimension specific model through Macros

As the macro is module of VBA, it can be linked to the database. The macro can be connected to the corresponding table of the component in the database for retrieving the dimensions based on the design calculations. Then macro is simply run to generate the component model. Similarly the respective macro files for 8 Bolts, 9 Nuts and 8 Bearings are generated in this research work (Bolts: Hexagonal Bolt, Square Bolt, T – Head Bolt, Cylindrical Head Bolt, Countersunk Bolt, Hook Bolt, Rag Bolt, Square Headed Bolt; Nuts: Hexagonal Nut, Square Nut, Cap Nut, Domed Cap Nut, Flanged Nut, Wing Nut, Castle Nut, Lock Nut, Slotted Nut; Bearings: Deep Groove Ball Bearing, Single Row Angular Contact Ball Bearing, Double Row Angular Contact Ball Bearing, Single Row Thrust Ball Bearing, Double Row Thrust Ball Bearing, Self Aligning Ball Bearing, Roller Bearing, Needle Bearing). The flow chart to modify the CAD model parameters using macro is given in the Fig. 2.

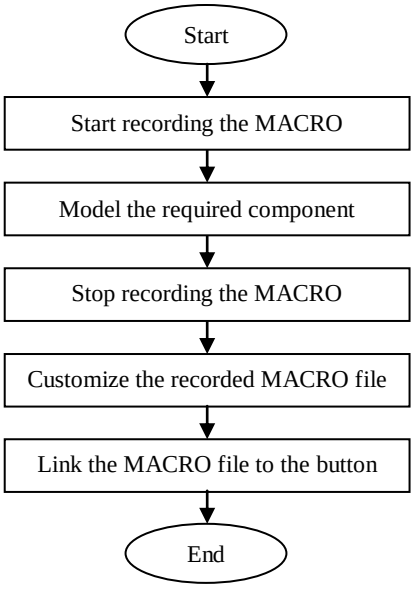

Fig. 2. Flow chart to modify the model

3.4 Creating SolidWorks buttons in user interface

After generating the respective macro files for bolts, nuts and bearings, buttons are generated in SolidWorks user interface with the function called 'New Macro Button'. These buttons are customized and linked with their respective macro along with image, tooltip and shortcut. Upon selecting these buttons the respective CAD model can be generated on the screen as per the code in linked macro files. In total 25 buttons (8 bolts, 9 nuts and 8 bearings) are generated in user SolidWorks interface as shown in the Fig. 3.

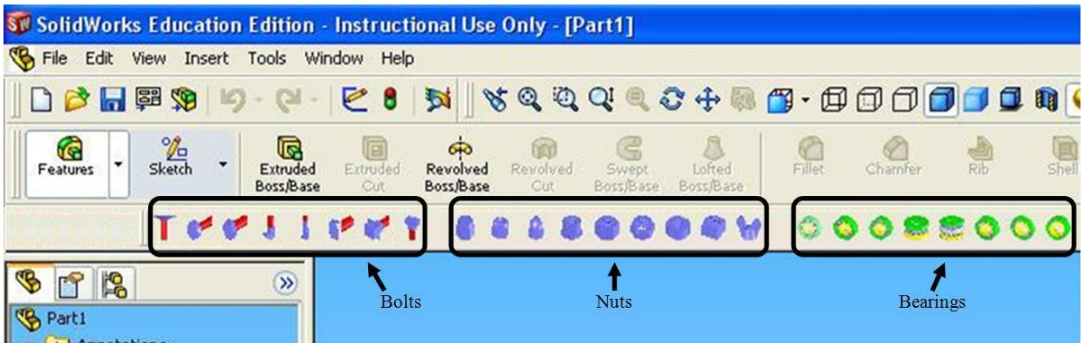

Fig. 3. Buttons for Bolts, Nuts and Bearings

3.5 Program development for design calculation

The developed KBS system in this research work can support the logical algorithms to do the respective design calculations. It is well-known fact that, in the process of designing, the dimensions of the bolts, nuts and bearings will change as per the working or loading condition but their geometry never changes. For that reason, it is tedious and time consuming to calculate the suitable dimensions for these parts whenever there is a change in working or loading conditions. In order to reduce the time consumption, a computer program is developed for design calculation purpose. The output from this program is linked with the SolidWorks software through the database in order to generate the CAD model with the dimension from the design calculations. Thus, the design and modeling work has been automated. Fig. 4 illustrates the detailed flow chart of the Bearing design process that is developed and followed in this research work.

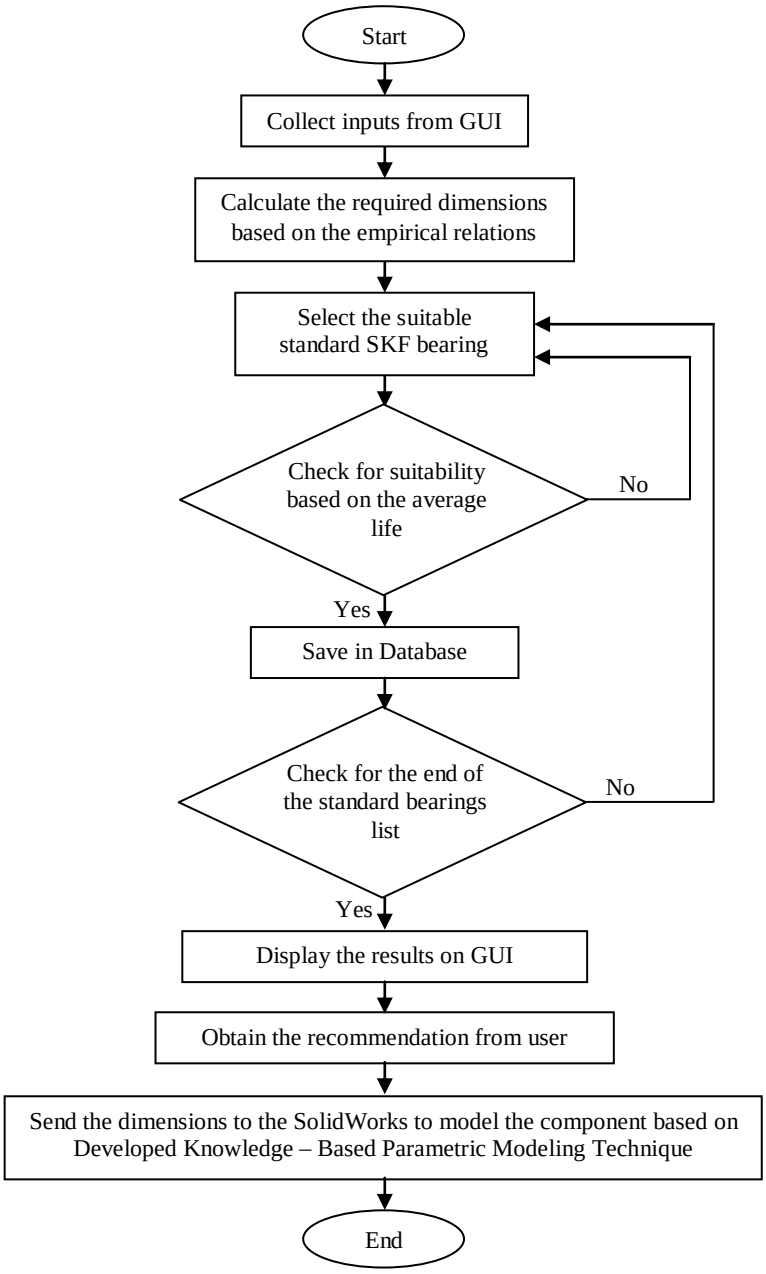

Fig. 4. Flow chart of the Bearing design process

For example, if the user wants to design a Deep groove ball bearing then the required input data like Radial Load, Axial Load, Speed and Average Life of the bearing to be designed will be submitted to the CAD System by the user through GUI (shown in Fig. 5) related to the Deep groove ball bearing button. After submitting the input data, the developed KBS will start computing the design calculations at the backend with the help of corresponding developed computer program. The result will be displayed in the GUI of Deep groove ball

bearing in combo box field as shown in Fig. 6. The number in the combo box is the SKF bearing standard number and they are sorted from best suitable to least suitable to model from top to bottom. By default the best suitable bearing will be modeled by the developed KBS. But, if the user wants to change the bearing to be modeled to some other, an option is also provided to do the same by selecting the required number in the combo box field. Later, upon the user recommendation, the dimensions of the selected bearing can be seen in the same GUI at the bottom portion as shown in the Fig 7. Soon after clicking the Model it button the developed KBS will generate CAD model as shown in the Fig. 8. As the bearings is an assembly consists of different parts like inner ring, outer ring, roller etc., at most care has been taken while developing this Knowledge – base to handle the assemble parts. Similarly, this Knowledge – base is adopted to model the Bolts and Nuts. The GUI of the Hexagonal Bolt and the Wing Nut are shown in the Fig. 9 and Fig. 10 respectively. KBS for Bolts and Nuts offers, two options to the user, Standard part modeling and customized part modeling. Standard part modeling option facilitates the user to model the standard part without any changes in the geometry or dimension. Whereas, the customized part modeling option allows the user to change the size of the shank if required. For the quick look, the output of generation of Hexagonal Bolt and Wing Nut CAD models are shown in the Fig. 11.

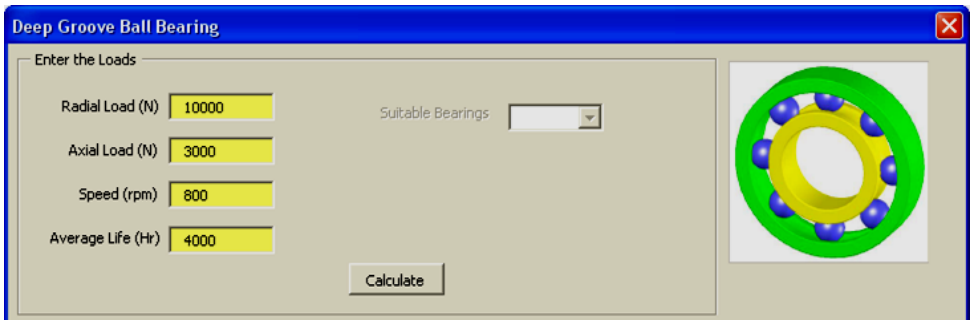

Fig. 5. GUI to collect the input data for the design of Deep Groove Ball Bearing

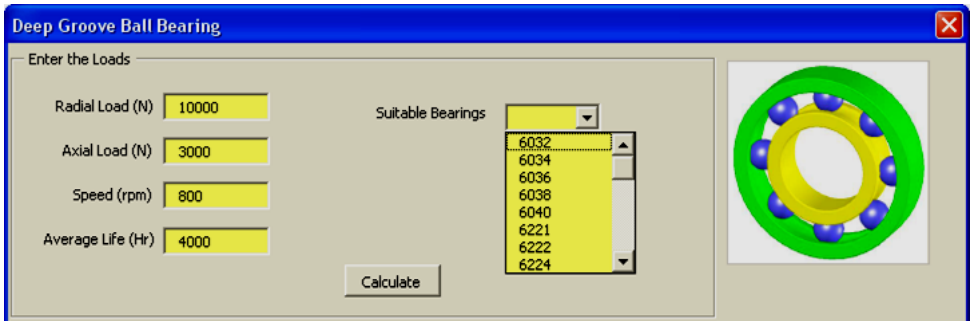

Fig. 6. Display of the suitable bearings to user

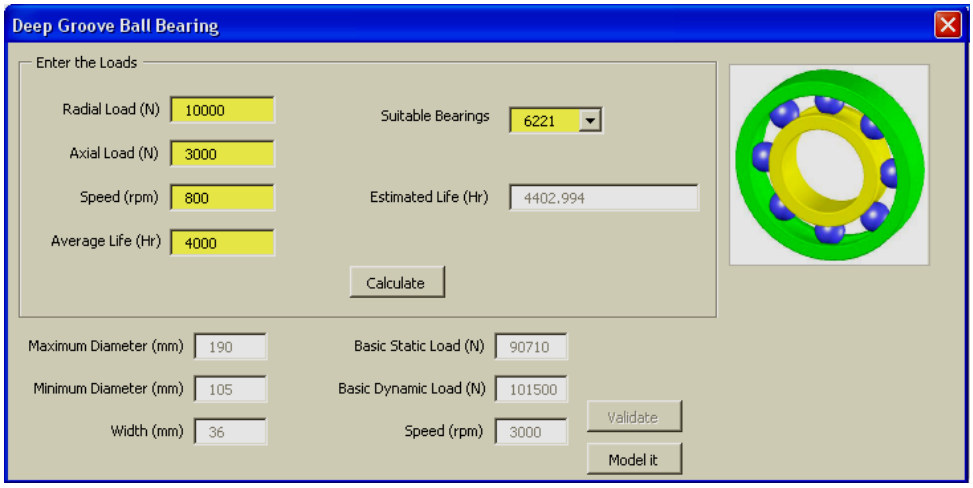

Fig. 7. Dimensions of the recommended bearing (at the bottom portion)

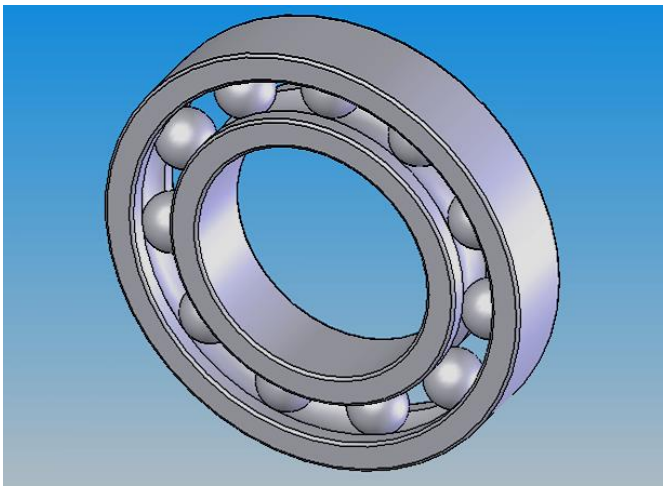

Fig 8. Generated CAD model of Deep Groove Ball Bearing.

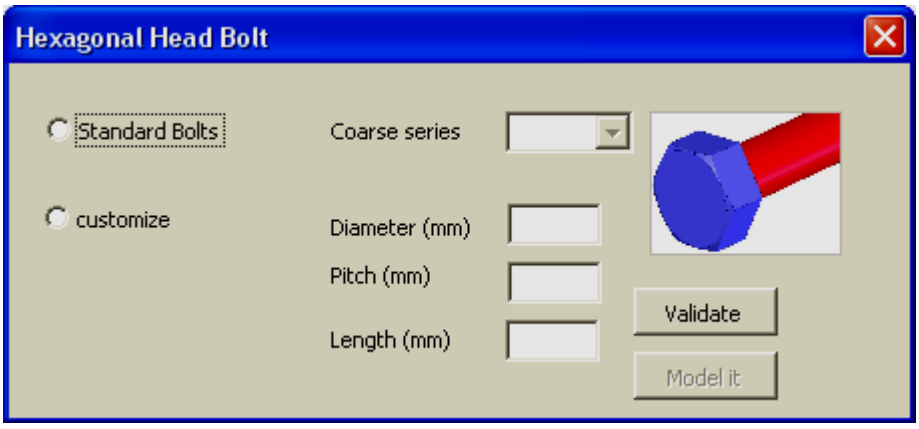

Fig 9. GUI of Hexagonal Bolt

Fig 10. GUI of Wing Nut.

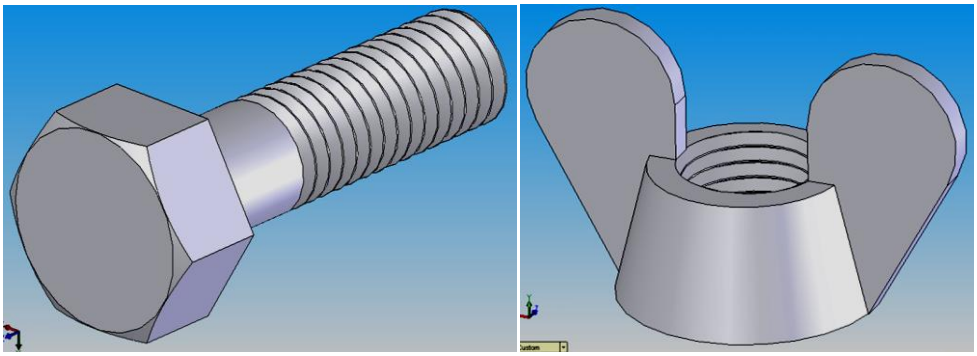

Fig 11. Generated CAD models of Bolt and Nut

4 Conclusion

Using the 3D parametric modeling method based on knowledge – based system of SolidWorks, the parametric model is established and the models can be updated easily according to different design requirements. For this purpose, VB 6 and macros commands are used. The approach of applying SolidWorks API was explained in details for standard parts and can be applied for similar cases. The system was tested on bolts, nuts and bearings. The developed approach results in saving time and cost when modeling standard components. The future work of this research can be aimed towards broadening the knowledge base for adding more standard components. Additionally, using this concept the models can further be preceded to the Analysis package or CAM package. Moreover, a comprehensive automated process can be achieved by better utilization of VB codes and macros to overcome the errors in order to make the process more efficient.

5 References

[1] S.P. Prince, R.G. Ryan, T. Mincer, 2005, "Common API: Using Visual Basic to Communicate between Engineering Design and Analytical Software Tools",

In: ASEE Annual Conference.

[2] S. Bo, Q. Guangtai, F. Yadong, 2011, "Research of standard parts library construction for SolidWorks by Visual Basic", In: Electronic and Mechanical Engineering and Information Technology (EMEIT) International Conference, pp.2651-2654.

[3] J. Tian, S. Liu, H. Fu, 2011, "CAD System Design on Standard Part Based on Software Reuse", In: Fourth International Symposium on Knowledge Acquisition and Modeling (KAM), pp.229-232.

[4] H.M. Deitel, P.J. Deitel, T.R. Nieto, 1999, "Visual Basic 6 how to program", Upper Saddle River, N.J: Prentice Hall.

[5] M.C. Kerman, R.L. Brown, 2000, "Computer programming fundamentals with applications in visual basic 6.0", Reading, Mass: Addison-Wesley.

[6] Uday Farhan, Simona O'Brien, Majid Tolouei Rad, 2012, "SolidWorks Secondary Development with Visual Basic 6 for an Automated Modular Fixture Assembly Approach", International Journal of Engineering, Vol. 6, No. 6, pp. 290-304.

[7] Jayakiran Reddy E, C. N. V. Sridhar, V. Pandu Rangadu, 2015, "Knowledge Based Engineering: Notion, Approaches and Future Trends", American Journal of Intelligent Systems, Vol. 5, Issue 1, pp. 1 – 17.

[8] Kong L, Fuh JYH, Lee KS, Liu XL, Ling LS, Zhang YF, Nee AYC, 2003, "A Windows-native 3D plastic injection mold design system", Journal of Materials Processing Technology, Vol. 139, Issues: 1-3, pp. 81–89.

[9] Song-Hao Wang, Steven Melendez, Chyi-Shyan Tsai, 2008, "Application of parametric sketching and associability in 3D CAD", Computer-Aided Design and Applications, Vol. 5, Number 6, pp: 822-830.

[10] Kuang Weihua, 2009, "Development of parametric Part Family system based on UG and Excel parameter form", In: Computer-Aided Industrial Design & Conceptual Design, IEEE 10th International Conference.

[11] Chu CH, Song MC, Luo CS, 2006, "Computer-aided parametric design for 3D tire mold production", Computers in Industry, Vol. 57, Issue 1, pp. 11–25.

[12] Ongkodjojo S, Gunawan H, 2006, "3D parametric modeling for product variants with Study case on flatbed conveyor", Petra Christian University, Yogyakarta.

[13] Bor-Tsuen Lin, Chian-Kun Chan, Jung-Ching Wang, 2008, "A knowledge-based parametric design system for drawing dies", The International Journal of Advanced Manufacturing Technology, Vol. 36, Issue 7-8, pp. 671-680.

[14] V. Naranje, S. Kumar, 2014, "A knowledge based system for automated design of deep drawing die for axisymmetric parts", Expert Systems with Applications, Vol. 41, pp. 1419–1431.

[15] Hussein H M A, 2014, "Computer Aided Blanking Die Design using CATIA", In: International Conference on Manufacture of Lightweight Components – ManuLight 2014, Procedia CIRP, Vo. 18, pp. 96-101.