Automated Design and Modelling of Knuckle Joint
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Abstract— This paper shows the methodology of developing an integrated application of a design process and generating model of a knuckle joint using a Visual Basic and the SolidWorks. The methodology concentrates on the making a Knowledge Base Engineering (KBE). Where KBE helps with the storing and reusing the data by the user and it shows developing Graphic User Interface (GUI) as a KBE tool for the standard design for developing the design process and modelling of a 3D model of a knuckle joint by standard empirical relations using SolidWorks. From that interaction the user can decreasing the designing time along with the modifications of reusing same data with other parameters and generating a 3D model with the resultant parameters. There by the designer can decrease the designing time. This paper mainly concentrates on the integration of two software’s Visual Basic and SolidWorks that could process could design and develop an accurate design of a knuckle joint.

Key words: Computer Aided Design, KBE, Standard Design process of knuckle joint, Parametric Modeling, Visual Basic, Solid Works API

I. INTRODUCTION

Machine design can be defined as the use of scientific principles, technical information and imagination in the description of a machine or a mechanical system to perform specific functions with maximum economy and efficiency. The main purpose of this automated design is to create an application which will satisfy the user requirements in an innovative way. Designer spends the most of their time in understanding the existing designs and dealing with the challenges associated with the design modifications and the improvements in those designs. Lots of engineering man hours is consumed by doing repetitive tasks of remodeling the existing designs. Knowledge Base Engineering (KBE) allows automation of repetitive design tasks while capturing, retaining and re-using the design knowledge. KBE is a system or process which collects and stores and organizes this knowledge and makes it available in the reusable form by providing computational support to the design process.

The objective of the present work is developing the methodology for automating the design process and to generate parametric assembly model in modeling software automatically. Which is not only support rapid geometry creation but also this application creates facilitates its design analysis. This paper describes the procedure for developing KBE tools in a Computer Aided Design (CAD) environment. In the present work, the associations with the design analysis and between various parts for a parametric assembly model are designed and developed. A user interface is to provide and facilitate the inputs and the developed KBE tool generates the response parameters through design evaluation using failure criteria. The final output is an optimized design process and mechanical part/assembly in the form of a CAD model. The proposed methodology is implemented using object oriented module known as Visual Basic and SolidWorks for the standard mechanical model.

II. KNOWLEDGE BASE ENGINEERING

Normally, type of KBE are used to be automate creation of geometry using set of expressions and thumb rules for the technology to the domain of manufacturing design and production. For the developing KBE tools like an application the relevant knowledge is identified, followed by theoretical knowledge and to covert it as codification. To reduce the time and the efforts required for repetitive design and modeling, the KBE system should have the following functions:

- Ability to store all kinds of knowledge effectively
- Capability to search efficiently
- Convenience to maintain and manage knowledge

Besides, it is developed as a reusable, generic, and generative. The resulting product or an application models based on the KBE system would include all the product, process and functional knowledge.

III. PROPOSED METHODOLOGY

The methodology for developing automated applications that are used by industry often differs depending on the company’s tasks to be automated. The proposed methodology involves following steps:

- By identify the parts and products functions and behaviors.
- Convert these functions and behaviors in terms of rules, associative expressions, design evaluation constraints (identification of knowledge).
- Managing those constraints, expressions and the rules in the form of database (knowledge management).
- Access this knowledge, expressions and evaluation criteria through program and user interface generation (knowledge acquisition and codification).

Further, in the present work the mechanical assemblies are designed based on either form or function. User interface is developed which contains five types of design methods i.e. based on (a) load (b) Design process and individual parts of (c) knuckle pin (d) single eye (e) fork end.

Normally, in the five approaches, the user feeds the load that the joint is supposed to bear and select the desired material such that stress values are given by standard data so that it can be taken or user can reenter the value so that by entering the command VB executes the program there by values are obtained. All the design forms are united in a userform as shown in fig 1 thereby from the selection of option by the user there relevant forms are to be generated.
A. Design by Load:
It’s a design to obtain a diameter for generating a 3D model in a solid works it requires a load and the material from the user it does a simple calculation programming (Fig.2). Next is step to generate model in a modelling software.

C. Knuckle Pin:
In the knuckle joint design knuckle pin plays a major role such that it had its own constraints so that to change of material and others so that design takes place. In this case the user enters the required values there by from the provided event program executes and obtain the resultant stress (Fig.4). Event also created whether the design is safe or not. If the user satisfies run an event to generate parameter so that from the command the 3D model is to be generated through the interface of visual basic to solid works.

B. Design Process:
It’s an empirical based approach that from the user inputs it executes the program that there by from the given load and the selected material and from the factor of safety there by it generates the diameter of the rod where other parameters of a joint based on the rod diameter such that from the standard empirical relations the event develops the other parameters such that the major parameters a shown in the textboxes (Fig.3). When the user satisfies the obtain diameter a 3D model is to be generated automatically by the solid works through the interfacing of design software to Visual basic.

D. Single Eye:
Design of a single eye had an individual event to generate and design of rod end. In this case the user enters the
required values of load and material there by from the provided event program executes and obtain the resultant stress. Event also created whether the design is safe or not (Fig.5). If the user satisfies run an event to generate parameter so that from the command the 3D model is to be generated through the interface of visual basic to solid works.

IV. CREATION OF DESIGN PROGRAMMING AND DATA BASE

A. Integrated Development Environment:

Visual Basic comes from its integrated development environment (IDE). IDE gives to user everything to create great applications, to write code for them, to test and fine-tune them, and, finally to produce executable files. These files are independent of environment and therefore can be delivered to customers for the execution on their machines.

In this work, the database of a material properties and standard design information of a knuckle joint is prepared in forms by using Visual Basic Environment. Where the user can select design process by selecting a command controls that are to be created in the IDE, from that the main form to be appear thereby user can select the given options to design and model of knuckle joint. The user should enter the constraints based on user requirement such that to design and model where in this case user had a flexibility to redesign if the obtained results does not satisfy the need, after the obtaining of results it means knuckle joint parameters the user had an option to generate a model in the solid works. If the user runs an event called generate which was provided in the form it automatically rebuilds the knuckle joint by the obtained parameters.

B. Parametric Modelling:

There are two types of modeling strategies, direct modeling and parametric modeling. The direct modeling quickly defines and captures only part geometry. Designers create geometry in direct modeling rather than building constraints, design rules and design intent into their models. In parametric modeling, designer anticipates and defines expressions, constraints and associativity, ensuring that any changes in design will necessity to change there by all the related geometries are to be designed by the base parameter. Direct modeling is suitable for all the designs where speed and flexibility is needed whereas parametric modeling is used where designer is required to meet certain design constraints, thumb rules and some manufacturing criteria. The present work uses parametric modeling strategy. To create the parametric model, parts functions in terms of geometric parameters are identified and those relations between them are established in terms of expressions. Those relations are converted in the form of expressions, rules and constraints. Thereafter these parameters, relations and constraints are optimized and then it is checked whether designed model is valid and fulfills all functions correctly for which it is designed.

C. Macro and API:

API is an abbreviation of Application Program Interface, is a set of routines, protocols, and tools for building software applications. A solid works macro is a shortcut to a task you perform repeatedly in solid works 3d cad software. It is a series of commands and actions like mouse pointer also that can be stored and run within solid works whenever you need to perform the task. You can record or build a macro in solid works, and then play the macro to automatically repeat the series of commands or actions. The quickest and easiest way to start programming with the SolidWorks API is to record a SolidWorks macro, which contains the SolidWorks API calls that correspond to the actions performed in the user interface. It can be used as an interface of visual basic and
the solid works software. The user can modify the macro in Microsoft Visual Basic for Applications (VBA) or Microsoft Visual Studio Tools for Applications (VSTA) to fit the user work site's needs.

D. Design Process and Integration:
Firstly there is a standard machine design process for the knuckle joint design process. It has been observed that how design analysis, design procedure of knuckle joint takes place from assumptions. Design analysis of a joint to be done from the Method of Failures, there are no. of failures that carried by a knuckle joint.

These Method of Failure to be generated as a standard formulas and from the failure of rod diameter is obtained and design of joint takes place by empirical relations where those empirical relations of a knuckle joint based on rod diameter so all parameters are obtained from it. From the Empirical Relations Dimensions are carried out from those the Modes of Failure have been tested and check whether the induced stress are less than allowable stress so that the design is to be safe. From that standard design method those formulas are carried out in the software Visual Basic. The designer also includes database linking functions and the user need to give inputs required for Graphical User Interface (GUI) and which it is to be generated in User Interface Style. In the Visual Basic GUI the designer can generate the program by using VB tools to create an event for forms. So that it carries a number of commands, labels, Textboxes etc., which are integrated with the design date to run that command to design process to be take place by filling certain constraints of the process.

Those resultant parameters which are to be carried out to the 3D model. From the creation of direct integration between the VB and the SolidWorks the developing of 3D model code by using the SolidWorks API. The developed code that is given as a command in the design form where that obtained resultant parameter diameter of rod is integrated with that command so the knuckle Joint model is developed in the solid works from the VB, therefore the code generation and the integration of the VB and SolidWorks to be done from the resultant parameters.

E. User Interface:
Form the developing of GUI the user can interact with the user form by identifying the constraints that are displayed in the form or in message box so that user constraints as a design constraints that are entered in the text box like load and selecting the required material and the factor of safety then to run the command to design stress and next command for resultant stresses for the part or a design based on selected form. There by user can generate knuckle joint parameters by running an event given there so that if the user satisfies with resultant parameters can generate the knuckle Joint model in the SolidWorks. The User interaction with the forms can be explained below.

1) User to run the VB file that shows the userform1 that contains five buttons those are knuckle joint proposed methodology those five are:
   - Based on Load
   - Design Process
   - Knuckle Pin
   - Single Eye
   - Fork End

2) Select from the given options and run the even there by forms are displayed based on user selection
3) All forms having some similar functions that requirement of constraints.
4) Based on load and the Design Process mainly focus on the parameters and the developing of model in SolidWorks.
5) Next 3 options are the events that are developed for the design process for the constraints to satisfy the needs of the customer to full fill its requirements.
6) Finally if the user satisfies with the parameters can run an event called Generate. This develops the model in the SolidWorks if the system having software in it.
Fig. 7: Components of Knuckle Joint Assembly (a) Knuckle Pin (b) Single Eye End (c) Collar pin (d) Collar (e) Double Eye End (f) Knuckle Joint Assembly

By following the above steps the user can design a knuckle joint assembly as shown in fig.8. In this (a) shows the design values of a knuckle joint and (b) shows the generated knuckle joint assembly in SolidWorks

Fig. 8: (a) Design Process form with values (b) Generated 3D model of a knuckle joint based on design parametric values.

V. CONCLUSION

This paper presents an application that support an automated approach of design and modelling of a knuckle joint. From the knowledge base engineering which was created by the designer in a computer aided design environment. It is to avoid repetitive work of redesigning and remodeling of a knuckle joint, from the computer aided design it improves design efficiency. This paper work provides a safe design and modeling of a knuckle joint by the graphic user interface. The approach of the designer is that if user predicts and defines relative, expressions and feature constraints in such a way that design could change automatically with the given updated data and all modifications to be done in the assembly / parts. This application is suitable where the engineer is given a criteria with standard design and specific constraints. In the similar way the engineer can develop a number of applications for automating a number of machine elements based on standard Design as a computer aided design.

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