Design Process Automation Support through Knowledge Base Engineering

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***Abstract*—This paper shows the methodology of developing an application of Knowledge Based Engineering (KBE) to automate the task of repetitive designs, while reusing and modifying the existing designs in a Computer Aided Design (CAD) environment. The presented work describes the procedure for developing KBE tools for standard parametric mechanical /engineering parts and assemblies. With the proposed methodology and the developed KBE tool, it is possible to achieve direct interaction between the user and the geometric model so as to simplify the design process so as to avoid redesign and remodeling of the product. The proposed methodology was implemented through Knowledge Fusion (KF), a Siemens NX module.**

***Index Terms*—Computer Aided Design, KBE, Knowledge Fusion, Parametric**

I. Introduction

THE main purpose of the 'Design' is to create things which will satisfy certain requirements of a user in an innovative way. Designer spends most of their time in understanding the existing designs and dealing with the challenges associated with the modifications and improvements in the designs. Lots of engineering man hours is consumed 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 [1]. KBE is a system or process which collects stores and organizes this knowledge and makes it available in the reusable form by providing computational support to the design process.

The need of KBE arouse due to problems in current knowledge structure, which is disorganized. Existing databases, designs, thumb rules, etc. are not maintained in order, which makes the reusability difficult. Current knowledge structure is not secure also, as the databases that are not prepared centrally are susceptible to changes by other designers and loose reliability. Capturing knowledge is essential, but capturing the knowledge that is not necessary would lead to confusion and decrease effective reusability.

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KBE systems solve all the problems which are indicated above. KBE allows designers to devote more time and effort in creative and innovative work and makes design process simpler. KBE system works where traditional Computer Aided Design (CAD) system fails because KBE systems ask the question "why does this need to be done?" not just "What needs to be done?" KBE system automatically extracts necessary knowledge from every stage of design cycle, and makes this knowledge available for remaining design cycles. KBE system allows capturing and organization of knowledge in an efficient manner. KBE system helps to capture rationale and intent behind the design. It also helps to identify whether design constraint is violated or not. With the application of KBE, it is possible to analyze the possibility of manufacturing any part, knowing whether design is optimal or there are other alternative solutions available [2]-[4].

KBE helps to increase the efficiency of designer's work by enhancing the level of automation in the design process [5]. Fabio et al. [2] proposed methodology which defines direct interaction between the designer and the CAD model for the design automation of automotive painting defects inspection tunnel using Siemens NX Knowledge fusion (KF) software. Tong et al. [6] had presented a novel dynamic modeling wizard for 3D standard part library. Hou et al. [7] proposed knowledge-based rapid response design system for the key parts and components of machine tool based on the UniGraphics and Teamcenter Engineering software. Ma et al. [8] presents a methodology of associating either the parts that are not defined geometrically or geometrical entities that define the part interfaces. The association is done automatically as well as through generic user interface and are both geometric and non-geometric in nature.

The objective of the present work is to develop methodology for automatically generating parametric assembly model which not only support rapid geometry creation but also facilitates its design and analysis. This paper describes the procedure for developing KBE tools in a CAD environment. In the present work, the associations between various parts for a parametric assembly model are developed. A user interface is provided to 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 of a mechanical assembly in the form of a CAD model. The proposed methodology is implemented using object oriented module known as Knowledge Fusion (KF) of Siemens NX for the standard mechanical assemblies.

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II. KNOWLEDGE BASE TOOL

Traditionally, KBE systems are used to automate creation of geometry using set of thumb rules and expressions. For developing KBE tools the relevant knowledge is identified, followed by acquisition of knowledge and its codification. To reduce the time and efforts required for repetitive 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 [7]

Besides, it is developed as reusable, generic, and generative. The resulting product models based on the KBE system would include all product, process and functional knowledge [9]. The procedure adopted to develop KBE tools is shown in Fig.1.

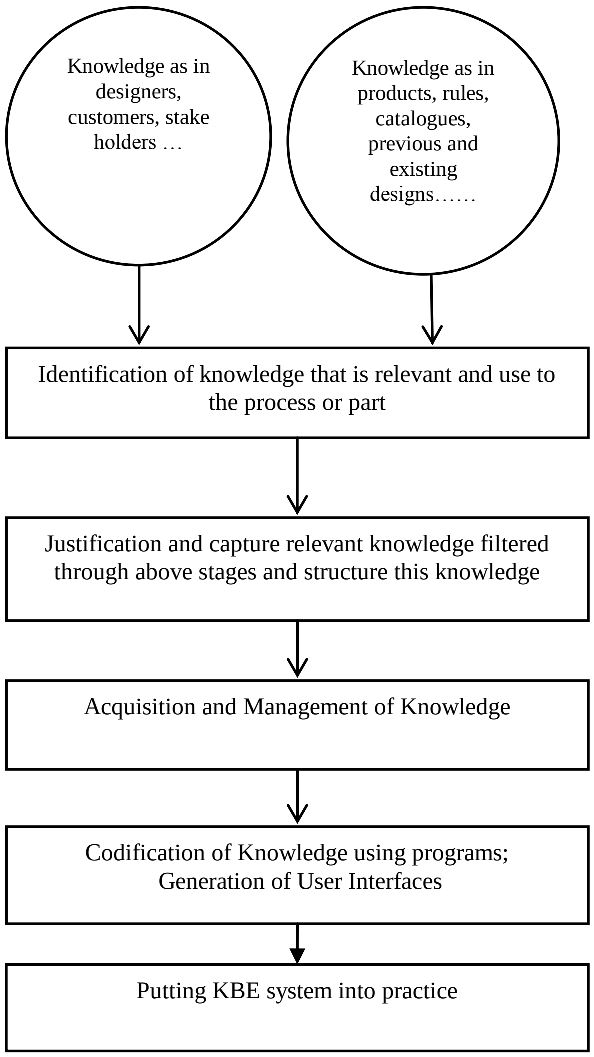

Fig.1. Generic procedure to develop KBE system

III. PROPOSED METHODOLOGY

The methodology for developing automation applications used by industry often differs depending on the company's tasks to be automated. The proposed methodology involves following steps (Fig.2):

- Identify parts and products functions and behaviors
- Convert these functions and behaviors in terms of rules, associative expressions, design evaluation constraints (identification of knowledge)
- Manage these constraints, expressions and rules in the form of database or spreadsheet (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 two types of design methods i.e. based on (a) load and (b) diameter. In the first approach, the user feeds the load that the joint is supposed to bear and select the desired material. Then the program calculates diameter of the rod taking care of all possible failure possibilities and generates parametric assembly model of the product in a CAD environment. In the second approach, the user is supposed to input the desired diameter of the rod for which the knuckle joint is to be designed. The user is also supposed select the material of the joint. The KBE application then evaluates the load which the joint can bear. In both the approaches, the CAD model of the assembly is generated and the output parameters are displayed in the user interface.

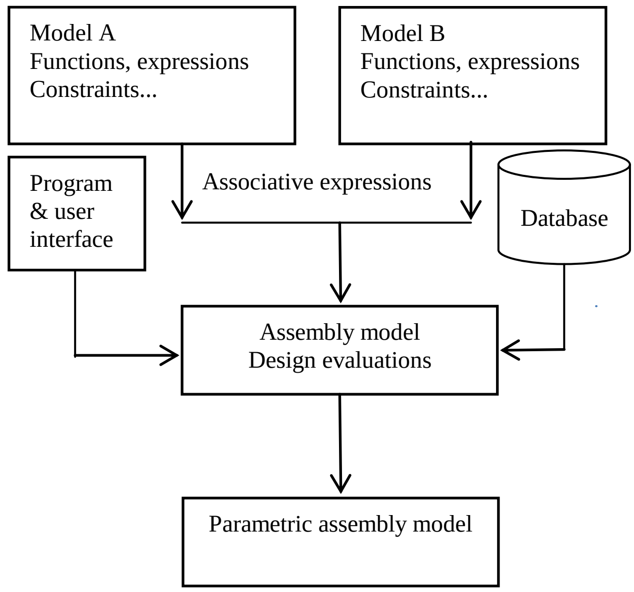

Fig.2. Proposed Methodology

IV. DEVELOPED WORK

A. *Parametric modeling of parts*

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, user anticipates and defines expressions, constraints and associativity, ensuring that any changes in design will necessitate changes in all the related geometries. 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

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criteria [10]. The present work uses parametric modeling strategy. To create parametric model, product or parts functions in terms of geometric parameters are identified and relations between them are established. These 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 [11].

The implementation of the procedure for developing parametric assembly is done by taking example of knuckle joint. Knuckle joint is used to connect two rods under tensile load. The first step in creation of parametric assembly is to model all the parts in CAD environment. Major components of knuckle joint are shown in Fig.3 along with assembly. All the parts are parametrically modeled using expressions. Dimensions of all the parts are defined in terms of diameter of rod. Considering all possible modes of failure of knuckle joints i.e. tensile failure of rods, shear failure of pin, crushing failure of pin and bending failure of pin, safe diameter of rod is determined. Expressions are used to create link between parameters of individual parts. Interpart expressions are used to create expressions among different components of assembly, so that changes in one component of assembly will reflect changes in all the other parts of assembly which are linked to it.

B. *Creation of assembly database*

In this work, the database of a material and strength information is prepared in Microsoft Access. Database is prepared centrally and saved in a secured folder so as to avoid accidental and unnecessary modifications by the user. Database consists of Data Source Name (DSN), different materials, values of strength for each material, queries, and data types of parameters as shown in Table I. The KF program is prepared to link database to user interface styler and assembly. It contains data source name, queries, Open Data Base Connectivity (ODBC) functions, variables and data types. The driven parameter table created in the database must be similar to the CAD model feature parameters. It should be consistent with the CAD modeling methods [6]. For example, if hole feature is modeled by extruding a circle and then applying Boolean operation subtract, the corresponding driven parameter should be the diameter of that circle and depth of extrusion. Similarly, if hole is created by direct command then driven parameters are diameter of hole and depth of hole. While defining parameters care must be taken that the same parameters names should also be present in the program of knuckle joint assembly otherwise undesired model results when the program is called.

C. *Code Generation and user interface*

After modeling and database creation, the next step is to convert all the expressions into programming code. In this work, KF program for knuckle joint assembly is developed in notepad. It includes all the parameters of part models which are linked by interpart expression. It also includes database linking functions and inputs required for graphical user interface which is generated in User Interface Styler. Graphical user interface is the most convenient and preferred way to create human-machine interface. Siemens NX allows its user to create GUI with .dlg extension for any application. It allows the users to create dialog boxes which can be linked to any KF application. GUI of knuckle joint assembly is shown in Fig.4 along with input for program and resource editor of user interface styler used to create dialog box.

TABLE I
MATERIAL PROPERTIES FOR KNUCKLE JOINT

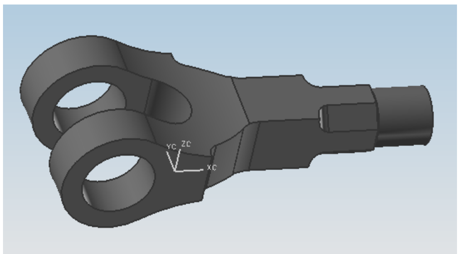
Material	Strength (MPa)
30C8	400
40C8	380
45C8	380
C20	241
C30	350
C25	450
FG 300	300
FG 350	350
30C8	400
40C8	380
45C8	380
C20	241
C30	350
C25	450
FG 300	300

D. *Linking code, user interface, database to CAD model*

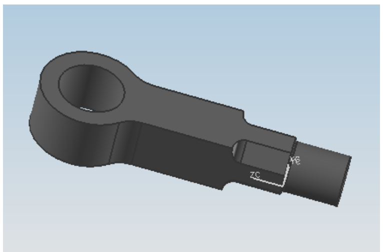
After developing CAD model by identifying feature constraints, rules and expressions along with database and user interface, it is necessary to define a link between them. In this work only one CAD system is used whether for modeling, code generation and GUI creation. The required steps for linking database, code and user interface to CAD assembly model are as follows:

1. Microsoft Access database is linked through the open database connectivity (ODBC). It provide standard interface to connect KF applications to many data sources. KF application connects database by executing SQL statements using standard library of functions. NX connects KF program to database through functions namely ug_odbc_database and fetches record of database through function ug_odbc_recordset.
2. Database is linked to KF application and CAD model through following steps (Fig.5.)
3. Code is written in notepad with .dfa extension and is linked to CAD model and GUI through DFA manager that is available in KF module of NX (Fig.6).
4. After linking code and database, GUI is called through the KF toolbar and the user input load constraint and other input parameters. Program calculates all the output parameters by checking against all failure criteria and generate CAD model of knuckle joint assembly in CAD environment (Fig.7).

KBE framework of knuckle joint assembly is shown in Fig.8.



(a)



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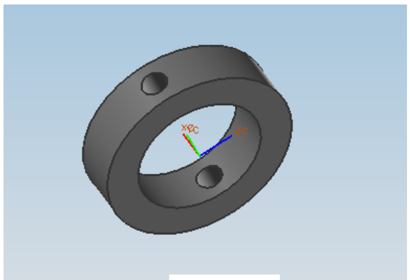

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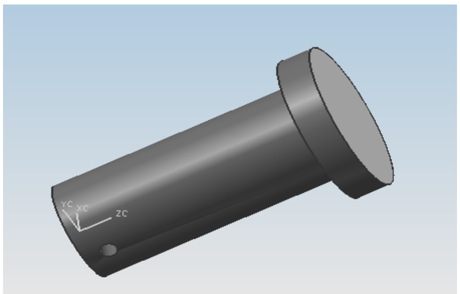

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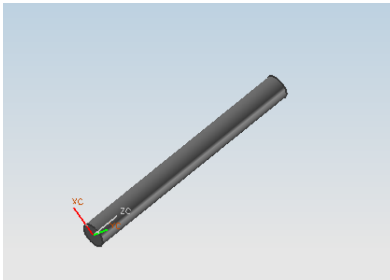

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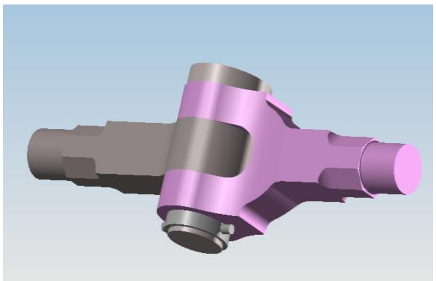

(f)

Fig.3. Components of Knuckle Joint Assembly (a) Double Eye End (b) Single Eye End (c) Collar (d) Knuckle Pin (e) Pin (f) Knuckle Joint Assembly

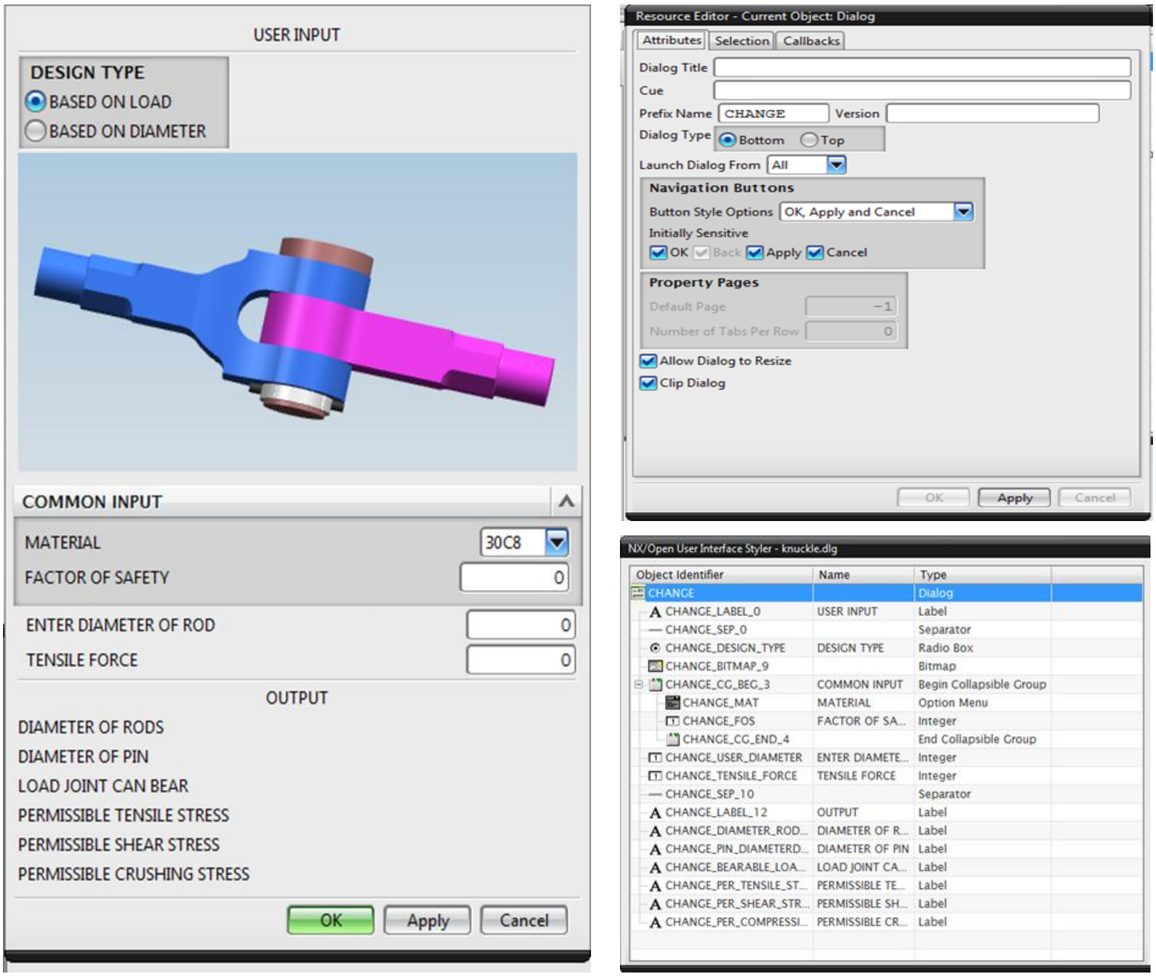

Fig.4. GUI of knuckle joint assembly

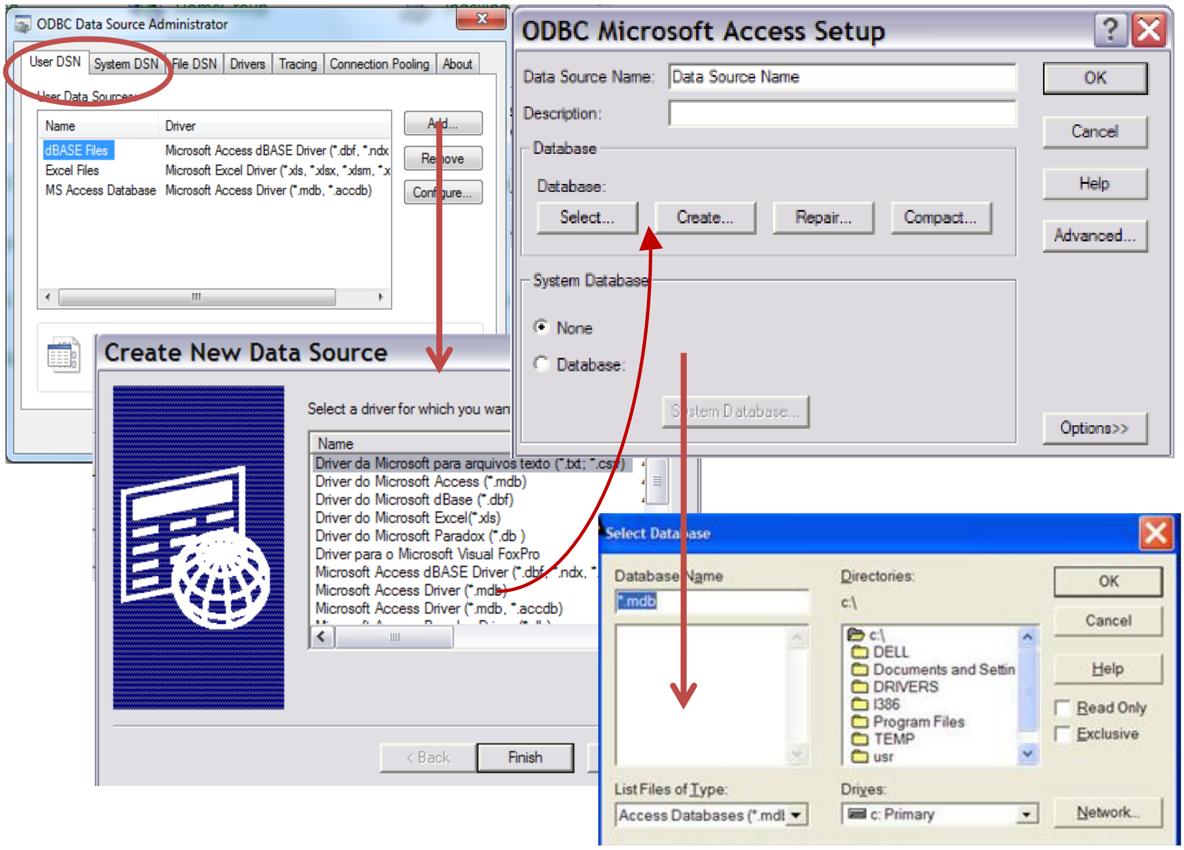

Fig.5. Linking of database to KF application and CAD model

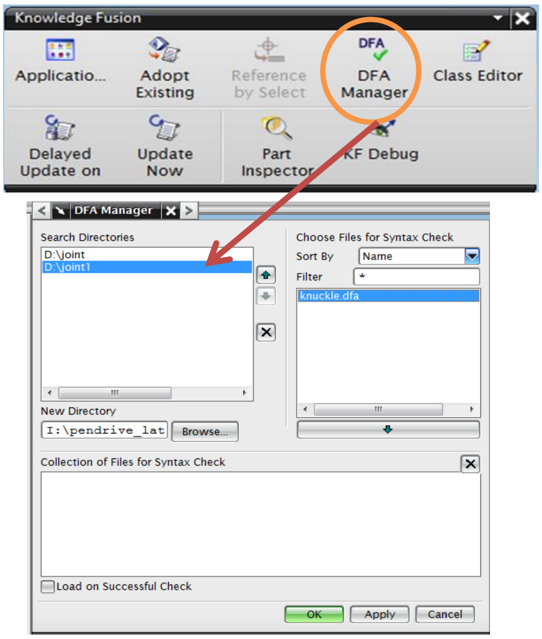

Fig.6. KF program linked through DFA manager to CAD model

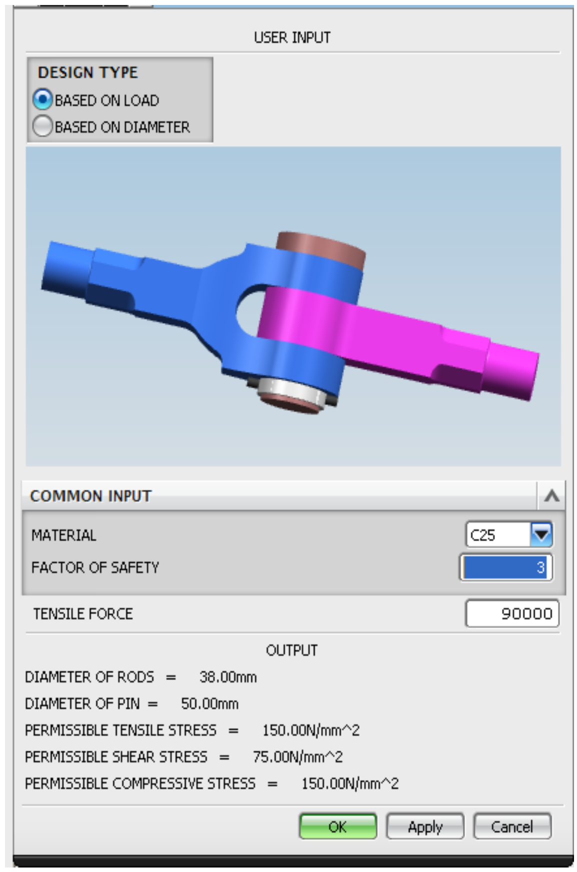

Fig.7. Output parameters by checking all failure criteria

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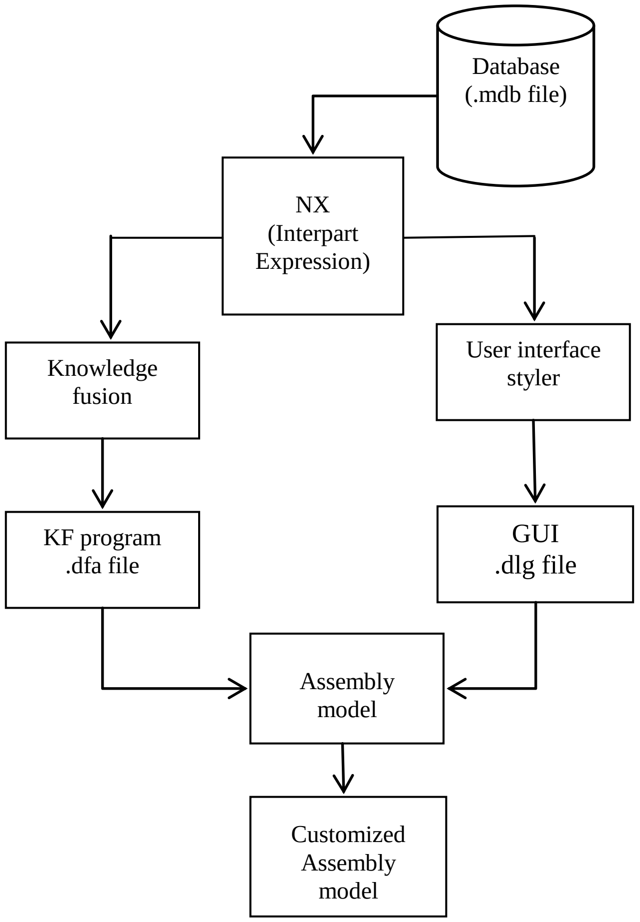

Fig.8. Knowledge base system of knuckle joint assembly

V. CONCLUSION

This paper presents an automated approach to develop knowledge base parametric assembly in CAD environment so as to avoid repetitive work of remodelling and eliminate redesign of an engineering product. The purpose is to improve the design efficiency. The present work provides graphical user interface for safe design of knuckle joint assembly so that novice user can use the proposed 3D CAD tool easily. The present work provides "engineer it" approach to the designer. User predicts and defines relations, expressions and feature constraints in such a way that any design change would automatically update the necessary modifications in all related geometries in predefined manner. This approach is suitable where the engineer is given strict criteria to meet certain design and company specific constraints. Every modeling strategy has its own advantages and requirements. Industries are adopting both direct and parametric modeling in order to achieve the creativity, quality and innovation to stay ahead in competition.

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