Automatic Assembly Modeling for Product Variants using Parametric Modeling Concept

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Abstract - **Parametric modeling is strong tool for the industries mechanical product design. Current scenario of the market is competitive. To sustain in the market for company product time to the market have to be minimum. Companies existing product demands from the customer are to be provided quickly as soon as possible. Existing product requirement has same parametric features of components for different specification. Design and modeling of the product time is generally 60-70% of overall time of the product development. Design phase has lot of potential where time can be saved. Parametric modeling can be used for saving the modeling time. Knowledge based approach can be useful for saving the design time. Lot of repetitive calculation can be saved avoiding tedious work. Creo 2.0 software is selected having strong parameterization. Mechanical product selected is reduction type gearbox. Reduction type gearbox manufactured by many companies across the world. Algorithm defined for the automatic assembly of the single stage helical reduction type gearbox. Algorithm is implemented with Pro/program in the generic assembly of gearbox. User interface is developed with Excel VBA.**

Keywords: Creo 2.0, Parametric Modeling, Pro/program, Reduction Gearbox, Excel VBA.

I. INTRODUCTION

In this competitive environment, time to market for the mechanical product design becoming shorter. Enterprises over the world are using CAD software's according to their product instead of using pencil, drafter, eraser and large sheets. Nowadays, advancement is that industries are using the customized CAD software's for their products. For industries it is necessary to launch innovative products or existing products as quick as possible to sustain in the global competitive world. Generally 80% of time is waste in design process in whole product development cycle. Lot of time can be save in the design stage. Design stage have repeatable task such as modeling process, design calculations for the some mechanical products. This repetitive task can be captured and standardized[1]. Parameterization technique is suitable for this work. Gearbox is widely used for the transmission purpose because of the high efficiency of transmission, compactness, reliability. Globally lot of industries manufacturing gearboxes for their customer's according to their types and specification. Each specification of gearbox assembly has its own modeling approach. Parametric

modeling can used for different configuration of the same assembly.

1.1 Creo as Parametric modeling software

PTC Creo, previously known as Pro/E is parametric,3D CAD/CAM/CAE solution created by parametric technology corporation (PTC). It is used by many industries for design and manufacturing applications. Developed by Dr.Samules P. Geisberg in the mid 1980s, as ruled based constraints 3D CAD modeling system. Special importance is given to an unitary database management system. It has facility functions such as sketch, part, assembly, drawing, manufacturing processing, mechanism simulation and finite element analysis and automatic measurement [2]. Two-dimensional CAD software has disadvantage like cannot have all product information, stability, validity and general parametric modeling of complex models and losing information. This disadvantage is overcome by PTC in Pro/E. China is widely using the Pro/E for the electronics, mold design, household application and toys etc. design. Lot of publication is done on parametric modeling and design from Chinese authors. In parametric design, modification of certain parts of model will automatically does the modification of its related parts/features. In case of new product development, engineers just sketches out a contour of model without an settings. Final shapes can be obtained by relating the parameters of models with design calculation and optimization in relations. Thus user has saving time for modifications of models[3]. The biggest challenge is to start thinking parametrically. You must constantly ask "How could this change, and what could change with it?", and then learn to create parametric relationships that can make adjustments automatically when the changes occur [4].

II. LITERATURE REVIEW

Lot of research have been done in the area of parametric modeling. Ault H.K.[5] has discuss basics of parametric modeling. Monedro J. [6] describes the evolution of parametric technique. Macros is one type of tool which be used for remodeling. Macros is recording a script of commands and data values used to create model. Constraints term is closely related to parametric design. Yin C.G. et al.[7] have clearly expressed method for

modeling parametric constraints among different product's features. Traditional way of the gearbox design is available from standard handbook. Calculation has to be done and model the gearbox parts and assemble it in suitable 3D CAD software. New approach for the remodeling is by relating the parameters and constraints of gearbox. Proposed work can be implemented with the help of software development project related to CAD software. Parametric assemblies can be done by three approaches top-down, bottom-up, both. Bottom up approach allows user to modify the each part and assembly separately. Top-down approach good for any size assembly, is ideal for tens of thousands of parts [8]. Gulati V.[9] parametric modeling for jewellery in Visual Basics application with AutoCAD with help of Active automation is done. Algorithm is defined which can utilize geometric CAD entities (points, line, circle, etc.) and operations (extraction, revolution, union, intersection etc.). Zhao Y.[10] A software is developed by Visual Basic for three-dimensional heating furnace design with ActiveX-Automation technology in Solid Edge. Gao S.[11] ,Variational design is achieved by Excel. Firstly in Solid Edge component/part of 3-d model generated and the model parameters in the Excel for the drum parts. Zhou S. [12] & Shrivastava R.[13] have done use of Solid works and API for modeling bearing sections and rangoli patterns respectively. Cucovics s. et al[14], developed automatic determination of grinding tool profile for helical surfaces machining by using CATIA/ VB interface. Shah D. B.[15] developed an automated 3D modeling of flange coupling. If the power transmission capacity is changed the dimensions of flange coupling parts are changed, when the coupling parameters are changed the solid model of coupling is changed. The time required to draw 3D model of different dimensions of flange coupling is more. Instead of to draw the model of coupling for different input parameter shah integrates the Autodesk Inventor (3D modeling software) with Microsoft Excel spreadsheet. Trivedi R.D.[16], Integration of modeling CAD software Pro/E with Microsoft Excel spreadsheet is done. Excel spreadsheet is formed containing all features name and respective dimensions. Another Excel sheet containing bearing database is formed. Second sheet linked with first by VLOOKUP or HLOOKUP function. Excel sheet containing feature name linked with bearing model in Pro/E. Firstly generic model of bearing is modeled in Pro/E. This technique is more suitable and simple than any other techniques like VB interfacing, Pro/Toolkit etc. Peipei G. [17], parametric 3-d part model is established according to types of gear shaft. Visual interface is developed with help of Visual C++. Interface, database and Pro/E model is linked together to achieve rapid parametric design of gear shaft. Connection of each button in user input form with parametric modeling is done by writing program with corresponding function and variables. Zhongtu L. et al[18], developed CAD system which uses knowledge based design approach for mechanical design process. System developed for the Intesolid 2.0 which is parametric feature based modeling software.

Implementation of the research work done for design of two stage reduction gearbox. Design task network (DNT) automates nearly all the design process, reduces the time for design and redesign. Errors are also reduce by lowering human contribution. Wang J. et.al.[19], A knowledge involved in design process for hydraulic actuators is categorised into three layers, rules, cases and 3-dimensinal case model.

III. METHODOLOGY

3.1. Pro/Program: Pro/program is selected as programming languages for achieving our objective. So to jump to higher used of this languages to the gearbox assembly modeling. Basics of the Pro/Program can be understand in this sections and applying it to the model in next section of chapter. Each model in Pro/ENGINEER contains a listing of major design steps and parameters that can be edited to work as a program. By running the program, you change the model according to new design specifications. The program used for the editing is called "Pro/Program".
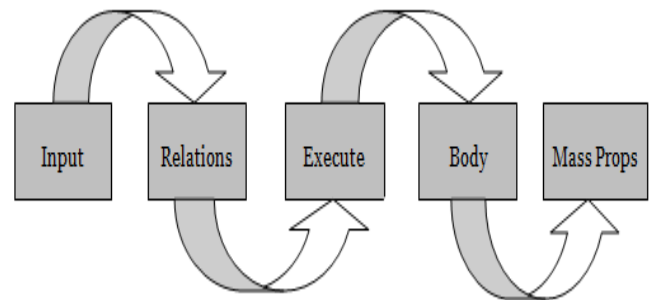

Fig-1: *Structure of Pro/Program*[20].

3.1.1. Input: Input statements provide values to automate your product. We can use three types of the inputs strings, number, Yes_No. Yes_No type of statements can be used in input structure which is useful for the include of any subpart in parts/assembly.

INPUT

INCLUDE_VALVE YES_NO

"IS VALVE TO BE INCLUDED IN THE MODEL"

END INPUT

Strings are useful of selecting the required material in case of parts or parts/subassemblies in case of the assembly. In case its use for material is written as follows.

INPUT

MATERIAL_CASING STRING

"ENTER THE MATERIAL OF CASING(STEEL/CAST IRON)

END INPUT

You can enter data using the following commands on the GET INPUT menu by current values, enter, read files.

Current values useful when the regeneration is not properly done. Change in feature operation is done. Valid CAD model can be generated with help of current values. Enter values has provision to select which parameters related to dimension of the CAD model have to change. Program can also uses input from a file. File name with its extension has to be type. Input values should be only one at one line.

3.1.2 Relations: Allows you to manipulate the inputs. Built in functions of Pro/E can be used as Math functions (+,-,*,/, Tan, SQRT, Log, etc.), IF/ELSEIF/ENDIF, Ceil, Floor, Lookup_inst.

RELATIONS

IF MATERIAL== 'C40'

SUT=450

ENDIF

IF MATERIAL=='STEEL'

SUT=480

ENDIF

3.1.3 Execute - These statements only works in assemblies. You can use as many as required execute statements. Execute statements are surrounded by IF/ENDIF to skip the particular execute statements according inputs from user. Execute statements allows an assembly to pass values to subassemblies or parts and run their programs. Acts as subroutines.

Table-1 Instances listing dimensions for rectangular box with hole.

Instance Name	d4	d6	d5	d7	d2	d3
Block	60	200	150	100	300	400
Block65_X_120	65	200	150	120	300	400
Block70_X_140	70	150	200	140	400	300
Block75_X_150	75	100	200	150	400	300
Block80_X_180	80	200	100	180	300	400

EXECUTE PART PINION

MOTOR_POWER = MOTOR_POWER

INPUT_SPEED = INPUT_SPEED

END EXECUTE

3.1.4 Body - The body is where the features or components are added. Pro/program can suppress or resume features / components. Surrounds the Add/End Add with If/End If. Pro/Program can replace components automatically. If the original program for the pats is as follows.

ADD PROTRUSION...

ADD HOLE...

ADD CUT....

END ADD

Modification in the body of the program can be done as follows.

ADD PROTRUSION...

IF D1 > D2

ADD HOLE..

END ADD

ENDIF

ADD CUT ...

END ADD

3.1.5 Mass property: Use of the Mass Prop statements to update mass properties each time the geometry changes. Use IF/ENDIF to classify which statements are updated.

MASSPROP

IF THICKNESS > 1

PART PLATEELSE

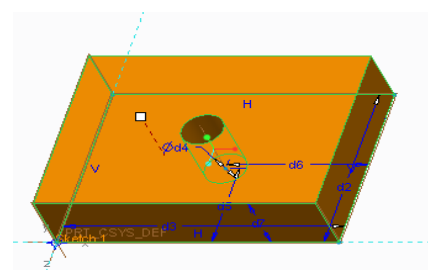ASSEMBLY BLOCK

ENDIF

END MASSPROP

3.2 Pro/Program and related concepts:

3.2.1 Replacing components in assembly:

Components in the assembly can be replaced by programmatically in the input section, programmatically in the relation section and family table by using the lookup_inst function. Family table driven component replacement is used for proposed work.

Table-2 Instances listing dimensions for cylinder.

Instance Name	d1	d0
Cylinder	100	60
Cyl65_X_120	120	65
Cyl70_X_140	140	70
Cyl75_X_150	150	75
Cyl80_X_180	180	80

(a) *Rectangular box with hole showing parameters.*
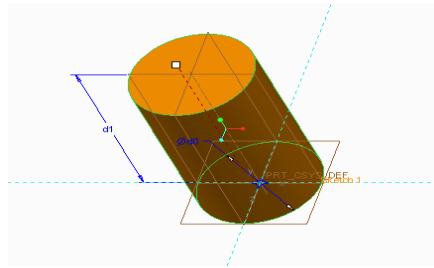
(b) Cylinder showing parameters.

Fig 2: Parameters name for dimension in model.

Pro/program of assembly generated is edited as follows :

INPUT

MODEL_BLOCK STRING

"Enter the type of model"

END INPUT

RELATIONS

MODEL_NAME = LOOKUP_INST("CYLINDER.PRT", 0,"D0",D4:0,"D1",D7:0)

END RELATIONS

ADD PART (MODEL_BLOCK)

INTERNAL COMPONENT ID 40

END ADD

ADD PART (MODEL_NAME)

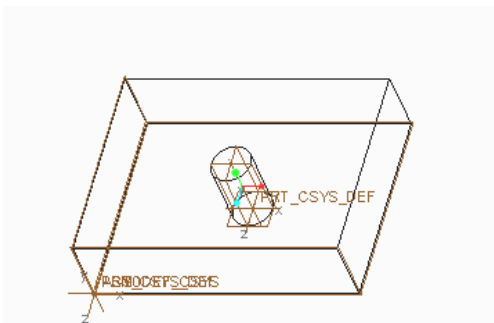INTERNAL COMPONENT ID 41

PARENTS = 40(#5)

END ADD

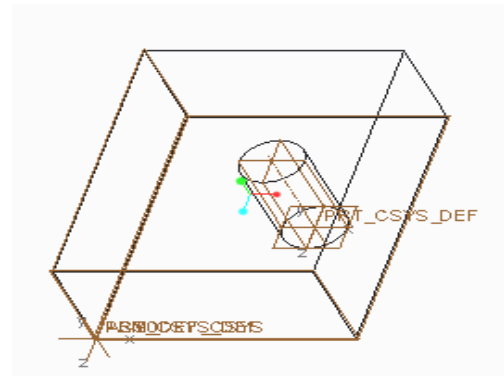Fig-3(a) Assembly model of cylinder and rectangular box with instances Cylinder.prt and Block.prt

Fig-3(b) Assembly model of cylinder and rectangular box with instances Cyl75_X_150.prt and Block75 X 150.prt

Fig-3: Replacing of assembly components automatically.

3.2.2 Feature suppression

Feature suppression can be done as follows which is required to implement is concepts in the hand of the gear. Implementation of this will ask the user which handed pinion you wants, i.e. left hand or right hand. Rectangular model is drawn and then two feature cut and add with drawing sketch on the upper plane of the rectangular box. First add is done on the upper side by taking some limit value. Add feature is suppress from the model tree. The model looks like nothing is added as shown in Fig.4. Then from same sketch the cut feature is added. Now add features is resume, figure shows change occurred. Pro/Program generated is now ready for the edit. If else statements are included in the body of the program.
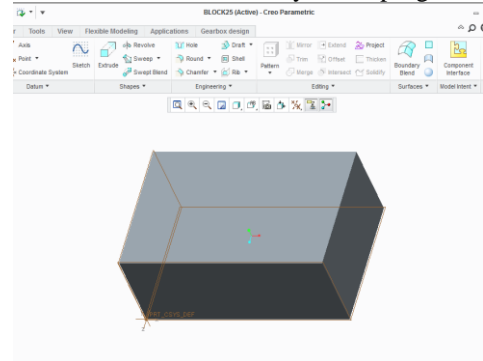
(a)Rectangular box with both feature are suppress.
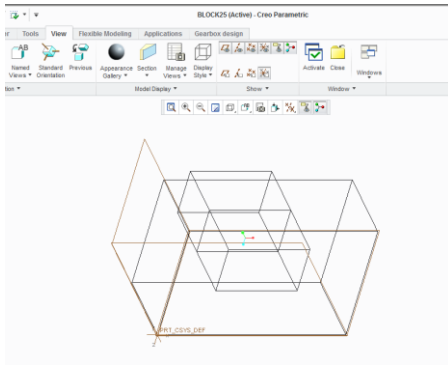
(b) Rectangular box with both feature are resume.

Fig- 4(a-b) Feature suppression.

INPUT

EXTRUDE YES_NO

"Do you want required sketch extrude"

CUT YES_NO

"Do you want required sketch cut"

END INPUT

IF EXTRUDE == YES

ADD FEATURE (initial number 8)

INTERNAL FEATURE ID 84

PARENTS = 76(#7)

PROTRUSION: Extrude

....

END ADD

END IF

IF CUT == YES

ADD FEATURE (initial number 9)

INTERNAL FEATURE ID 16

PARENTS = 76(#7)

CUT: Extrude ....

END ADD

END IF
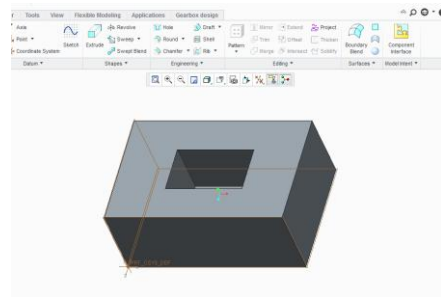
(c) Extrude feature suppress and cut resume
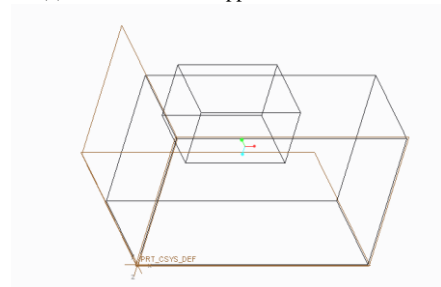
(d) Extrude feature resume and cut suppress.

Fig-4(c-d) Feature suppression

3.2.3 Constraints test for assembly modeling

When the design calculations are done in program of model, new parameters are calculated. New values of parameters like leads to change to related model parameters like gear size according to input speed. In assembly when size of gear and pinion changes the constraints given to them should kept them in meshing pair condition. Simple example for that taken as two rectangular box are assembled giving constrained as one plane will coincide with the other. The figure shows the assembly model of this two boxes. In all the cases when the sizes are changed the constraint defined remains same.
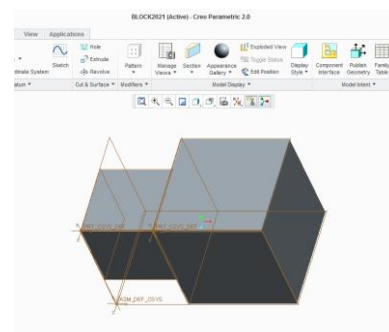
(a) Assembled model of two blocks
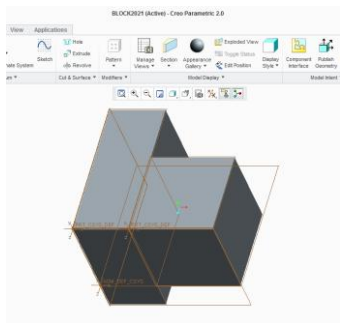
(b) assembled model when sizes changes programmatically
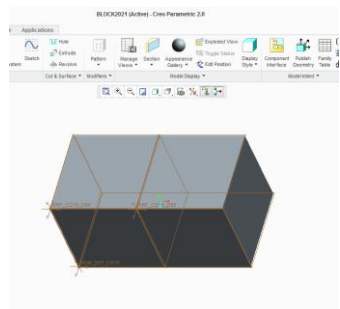
(c) assembled model when sizes changes programmatically to equal.

Fig-5: *Constraint test.*

3.3 Proposed schematic module:

Pro/program and parametric modeling concepts are cleared from the previous sections. Now implement it for the assembly of the single stage reduction type helix gearbox. To do the program one case study for the preparing assembly model is taken. This model will be the generic model. Every time when the requirement this there according to specification of the gearbox, open the generic model and gives inputs through the user interface developed. The whole process is done with help of Pro/Programs included in assembly model and individually in parts in Creo 2.0 and Excel VBA. The proposed module is as shown in the Fig.6.
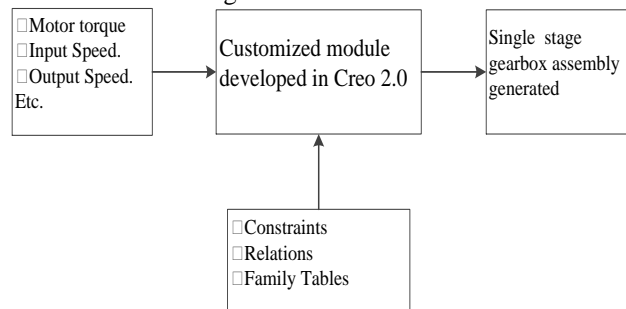
Fig- 6: Proposed schematic module in Creo 2.0.

3.4 Gearbox design

Driving machine - Electric motor, Driven machine - Belt conveyor, Motor power - 25 KW, Input speed - 1000 rpm, Output speed - 200 rpm [21] . Design calculation required are shown in the flowchart and model are generated according to design data book&[22]. This data is taken for preparation of generic assembly model of single stage gearbox. For assembly modeling of the other specification of the gearbox two examples are taken. (a) Driving machine - Electric motor, Driven machine - Belt conveyor, Motor power - 25 KW, Input speed - 1000 rpm, Output speed - 200 rpm. (b) Driving machine - Electric motor, Driven machine - Belt conveyor, Motor power - 25 KW, Input speed - 1000 rpm, Output speed - 200 rpm. Only specification of the gearbox are entered in the user interface.

START

MOTOR POWER, INPUT AND OUTPUT SPEED, DRIVEN AND DRIVING MACHIN, GEARS MATERIAL, SHAFT MATERIAL, TEETH ON PINION

σp*yp and σg*yg

If σp*yp < σg*yg

YES

NO

PINION IS WEAKER

GEAR IS WEAKER

WEAKER ELEMENT IS DESIGN FOR BENDING STRENGTH ACC. LEWIS EQUATION

A

A

GEAR DIMENSIONS BY FORMULES IN DESING DATA BOOK P.S.G

VERTICAL AND HORIZONTAL BENDING CALCULATION FOR SHAFT DIAMETER AND A.S.M.E STANDARDS

BEARING IS SELECTED FROM DESIGN DATA BOOK P.S.G BY BEARING LIFE CALCULATIONS

TAKE NEXT BEARING

IF L10CAL > L10TAKEN

NO

YES

SAFE

KEYS SELECTED FROM P.S.G DESIGN DATA BOOK

CASING DIMENSION AND NUT BOLT DIMENSION CALCULATED FROM EMPERICAL FORMULAS FROM IIT MADRAS NOTES

END

Fig-7: Flowchart for manual calculation of the single stage gearbox

3.5 User interface

User of the software generally not knowing the programs runs in background of the screen. Software should be user friendly so that time is save due to confusion. User interface is developed in the excel with help of the recording the macros. Macros are script recorded for the keyboard inputs. According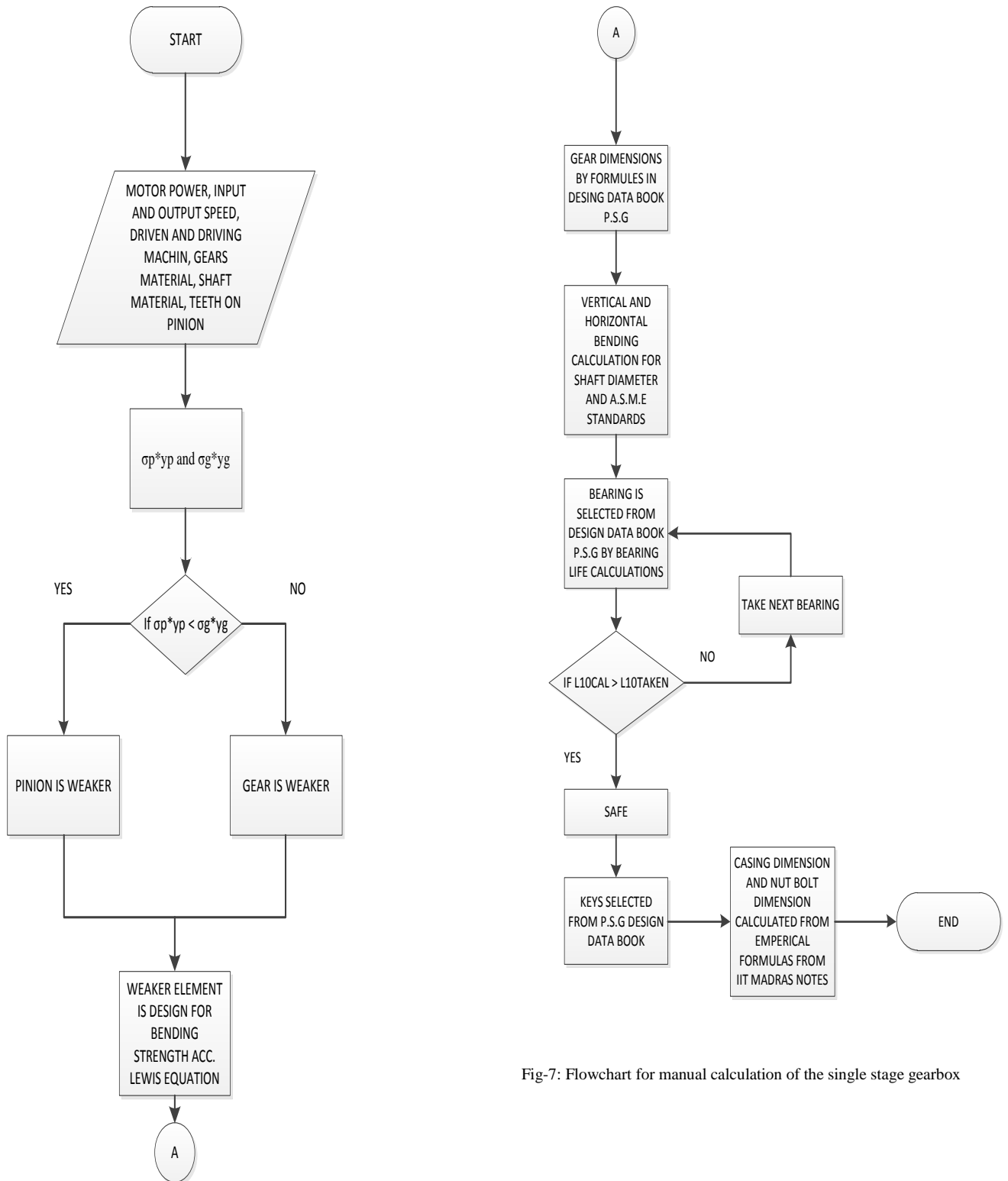 to the data required in the relation and body of the programs, the inputs are managed without user interface in it. User have to only select the options in drop-down menu and values in the boxes. Save button is press which replaces the old input values in the text file. Inputs from the excel is saved in the text tab delimited format. Format is as required in the Creo where the input from text file option is selected. User interface screen looks like as shown in fig.8.

USER INTERFACE FOR INPUT TO SINGLE STAGE REDUCTION TYPE HELICAL GEARBOX ASSEMBLY	
MOTOR_POWER	25
INPUT_SPEED	3600
OUTPUT_SPEED	80
Driving Machine	Electric Motor
Driven Machine	Belt conveyor
TYPE_OF_WHEEL_CUTTING	Vm≥ 20 m/s for precision wheels
V (INITIAL VELOCITY ASSUME)	20
PINION_MATERIAL	16Ni1Cr80
GEAR_MATERIAL	C50
FACTOR OF SAEFTY FOR GEAR	3
FACE WIDTH IS HOW MUCH TIMES MODULES	10
NO. OF TEETH ON PINION	18
HELIX ANGLE	23
NORMAL_PRESSURE_ANGLE	20
PRESSURE_ANGLE_TYPE	INVLOUTE

Fig-8: User interface developed in Excel macro enabled file.

IV. PROCEDURES FOR MODIFYING THE ASSEMBLY MODEL:

4.1 Working

Working of the module is shown in the fig. Module is divided into two parts as shown in Fig7. Firstly input Excel VBA developed in the excel with help of the macros. Inputs are selected from according the user gearbox specification like driving machine, driven machine from drop down list. The values are entered in the boxes like input speed, output speed etc. After successful entering of the inputs save button is pressed. This saves the changes in input text file written initially. Another button is pressed which launches the Creo 2.0 and directly open the generic assembly model of single stage gearbox initially modeled. Click on run button in the Creo 2.0. Pro/Program is run taking inputs from the text file. Finally assembly model of the Single stage helical gearbox is regenerated. The time required to do the things write from the design calculation to modeling and assembling tasks are dropped with this developed module.
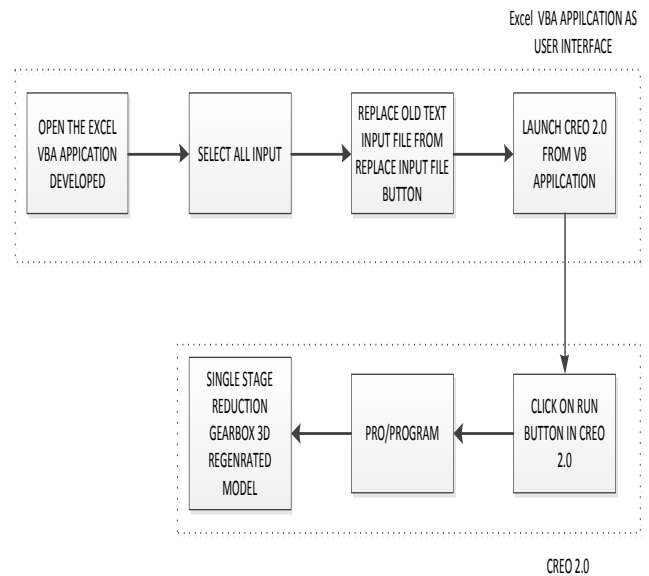
Fig-9: *Working of the Module.*

4.2 Pro/Program structural flow for single stage reduction gearbox:

Single stage helical reduction gearbox design procedure is implemented in the part program of assembly and assembly program with help of Pro/program. Standard parts are taken from family tables act as database. Structure of the Pro/Program is shown in the Fig.4.17 After getting input file to the program, module(m) calculations are done. Gears and pinion are regenerated. Standard parts like bearing, keys, nuts, bolts imported from the database library. Database library is made with help of family table for those components. Assembly modeling for the different specification according to customer needs in very short time can be done with help of above module developed in Creo 2.0. For separate design calculations and the modeling each parts of gearbox and assembling it requires nearly 4-5 days according to expertise in design and modeling. Same things are done in few minutes.

```
        START
          │
          ▼
   ┌──────────────┐
   / TEXT FILE    /
  /  FROM VB     /
 /   OUTPUT     /
          │
          ▼
   ┌──────────────┐
   │   MODULE     │
   │ CALCULATION  │
   │ ACCORDING TO │
   │ LEWIS EQUATION│
   └──────────────┘
          │
          ▼
   ┌──────────────┐
   │   PINION     │
   │ REGENERATED  │
   │ ACCORDING TO │
   │ INPUT DATA IN│
   │  TEXT FILE   │
   └──────────────┘
          │
          ▼
   ┌──────────────┐
   │    GEAR      │
   │ REGENERATED  │
   │ ACCORDING TO │
   │ INPUT DATA IN│
   │  TEXT FILE   │
   └──────────────┘
          │
          ▼
   ┌──────────────┐     ┌──────────────┐
   │SHAFT IMPORTED│ ◄── │ FAMILY TABLE │
   │ IN ASSEMBLY  │     │  OF SHAFT    │
   │AUTOMATICALLY │     └──────────────┘
   └──────────────┘
          │
          ▼
         ( A )
```
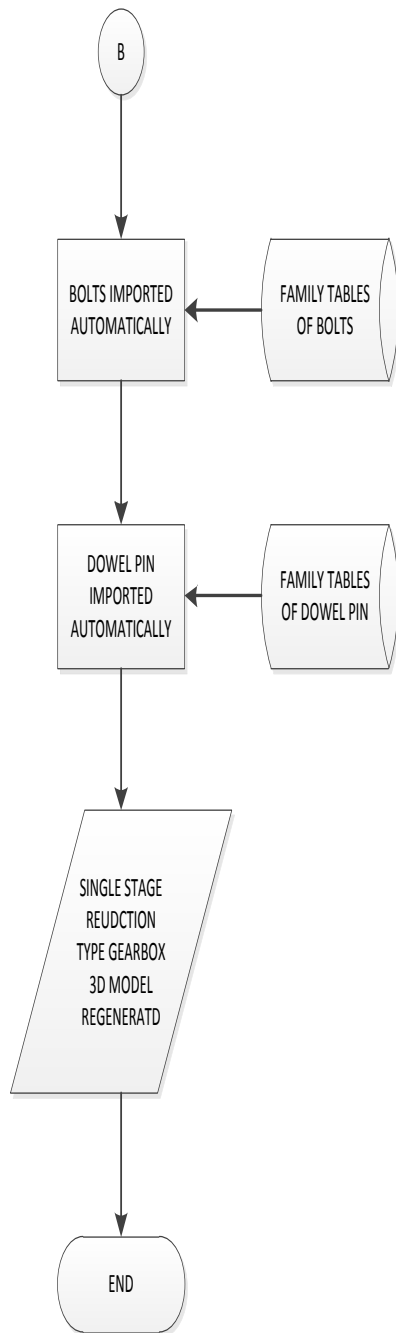
```
         ( A )
          │
          ▼
   ┌──────────────┐     ┌──────────────┐
   │KEYS IMPORTED IN│◄──│ FAMILY TABLE │
   │  ASSEMBLY    │     │   OF KEYS    │
   │AUTOMATICALLY │     └──────────────┘
   └──────────────┘
          │
          ▼
   ┌──────────────┐     ┌──────────────┐
   │BEARING IMPORTED│◄──│ FAMILY TABLE │
   │ IN ASSEMBLY  │     │  OF BEARING  │
   │AUTOMATICALLY │     └──────────────┘
   └──────────────┘
          │
          ▼
   ┌──────────────┐     ┌──────────────┐
   │   CASING     │     │  DATA FOR    │
   │ REGENERATED IN│◄── │  DIFFERENT   │
   │  ASSEMBLY    │     │  TYPES OF    │
   │AUTOMATICALLY │     │   CASING     │
   └──────────────┘     └──────────────┘
          │
          ▼
   ┌──────────────┐
   │ BEARING CAP  │
   │ REGENERATED  │
   │AUTOMATICALLY │
   └──────────────┘
          │
          ▼
   ┌──────────────┐     ┌──────────────┐
   │NUTS IMPORTED │ ──► │ FAMILY TABLES│
   │AUTOMATICALLY │     │   OF NUTS    │
   └──────────────┘     └──────────────┘
          │
          ▼
         ( B )
```

Fig-10: Pro/program structural flow for assembly of single stage gearbox.

product design and development. Less expertise on design and modeling of the mechanical product can do the job easily with few hours training. Current module is valid for reduction ratio up to 6:1. Same producer can be implemented for the multistage gearbox design and also for spur, bevel, worm and combination of them. In future parametric modeling is used analysis of mechanical components where in analysis iteration of the model parameters will be done till safe condition.

V. CONCLUSION

Parametric modeling concepts related to modeling and assembly is strong tool to be used in the industry. Concept of the parametric modeling, constraints and knowledge engineering for defining algorithm to select components in gearbox assembly is successfully formed. User interface in the excel is developed. Results obtained for the current work shown that lot of time is saved. Time saved can be unitized for the innovative development in the existing product and new product. Industries can used this type of customization in parametric CAD software for their

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