

# Design Automation of Rigid Flange Coupling by interfacing Creo Parametric 2.0 with Ms-Excel



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## ABSTRACT

Today's customers demand a new product quickly and frequently. Thus bringing new product to the global market quickly and cost effectively is a major competitive advantage. Computer-Aided Design (CAD) has been a valuable tool in product development. Professional CAD softwares like Creo Parametric, CATIA, Unigraphics etc., enable us to create the virtual 3D models of a product which can be further used for structural/thermal analysis, sensitivity analysis and optimization, and then finally put into production. Normally we use CAD softwares answers "WHAT" the design is but not answers "WHY & HOW" the design is done that way as both mathematical calculations and modeling are done at remote level. But to bring product in market quickly, we have to automate the whole design process by integrating output of design process calculations to the CAD package. Due to such type of integration, we can save lot of time as well as reduce stress on the Design engineer. This is achieved by using an Application Program Interface (API). This results in considerable reduction in re modeling time, compared to feature creation through the traditional method. Also, more complex features would require sizable programs. In this paper the interface between PTC's Creo Parametric 2.0 and MS Excel programming is developed to automate the design process of flange coupling design. The applications developed as a result, successfully demonstrated automation of feature creation on Creo 2.0 through a series of simple inputs from the user.

*Keywords-*

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## I. INTRODUCTION

The objective of parametric solid modeling is referred to as Parametrics. Parametrics was first introduced in Parametric Technology's Pro/Engineer® and it has now become a critical capability of most major CAD packages today (Hoffmann and Kim, 2001). In a parametric system, solid models are created and manipulated through parameterized and user-modifiable definitions. A parametric model is defined by its attributes which could be input and/or structural parameters. An input parameter is a variable or relationship that defines a key dimension whereas a structural parameter describes topological and hierarchical similarities between instances (Loukipoudis, 1996). A large majority of mechanical parts and assemblies are not

designed from scratch but are modifications of previously manufactured parts. A parametric model is hence a re-usable model. Most major CAD systems use two approaches for parameterization – Interactive and Programmatic. An Interactive Approach involves using the native parametric capabilities within the CAD system to create flexible re-usable models. The model can be updated or modified from a common, shared parameter database. Once the model has been defined parametrically, the user allows the CAD program to regenerate or update the model with new values (Rohm, 2000). Using PTC's Creo Parametric CAD software the design automation task is accomplished by four ways-

- Pro/E PROGRAM Tool- with *Unidirectional Associativity*
- Creo API (Application Program Interfacing) Toolkit using C++ , VB etc.
- Pro/web.Link- HTML, Javascript
- MathCAD- *Bidirectional Associativity*

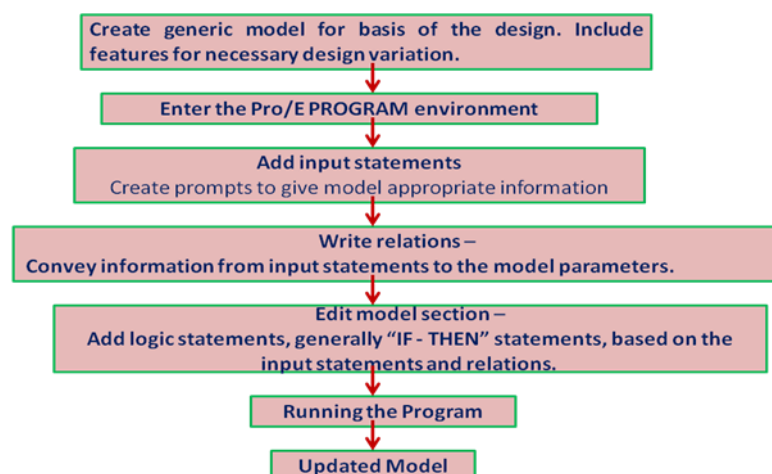
These programming environments serve different needs such as Customization of the Creo Foundation menu system , solid modeling and manufacturing feature creation, Assemblies, Drawing automation etc. Following are the some applications where we use programming tools-

1. Automating the creation of complex features
2. Automating the production of CREO deliverables, such as BOMs, drawings, and manufacturing operations
3. Improving product quality by performing design rule verification based on inputs from an external, knowledge-based system.
4. Automation of various drawings
5. Customization of the CREO-Foundation menu system
6. Collaborate between CREO applications

## 2. PRO /PROGRAM

The Pro/PROGRAM environment, support quick and relatively straightforward interactive graphical programming. It allows you to vary your design by incorporating user prompts into the regeneration cycle. Use Pro/PROGRAM to manually delete, reorder, and suppress features, modify dimensions, and pause the regeneration process to add additional features. The programming environment is simply CREO and *Microsoft Notepad* or *Word*. To invoke the Creo/PROGRAM environment, click *Model> Model intent>> Program...* from the model ribbon menu. To show or edit the program, one can click *Edit Design* from the PROGRAM menu.

The following steps involved in design automation using Pro/Program are-



1. Create generic model for basis of the design. Include features for necessary design variation.

## 2. Edit design changes –

- Add input statements - Create prompts to give model appropriate information. >> Parameter\_Name Parameter\_Type “prompt that is displayed in the message window”

- Write relations - Convey information from input statements to the model parameters. (Can also be added with Edit Rel option from the RELATIONS menu.)

- Edit model section - Add logic statements, generally “IF - THEN” statements, based on the input statements and relations.

3. Save and Exit from Editor. After exiting from the editor, the system checks your program for correct syntax. If it finds an error, you are required to correct it.

4. It asks you if you want to incorporate the changes into the design.

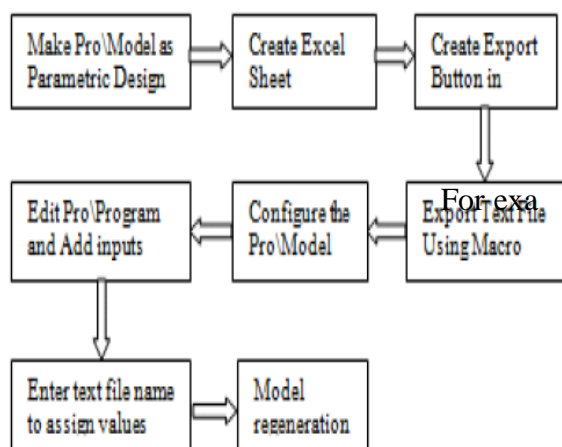
- Yes - Overwrites old program with new, edited version and executes the program.
- No - Uses old program for the model, but saves the edited version on the hard drive for future use. Saves file as “part\_name.pls”. The next time you edit the program, it asks you to specify which program to edit -- the current model or the saved version on the hard drive.

5. To run the program, simply regenerate the part. You will be prompted with the following choices upon regeneration:

- Enter - Specify new values for any or all of the prompts.
- Current Vals - Retain current values for the parameters.
- Read File - You can make a file containing all of the values for the prompts and then read it into CREO.

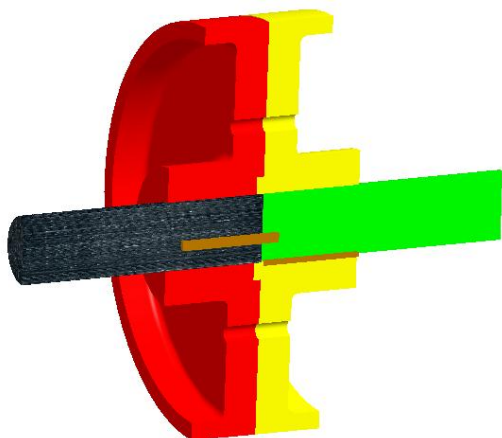
## 3. Integrating Creo 2.0 With Ms-Excel

To integrate the CAD assembly model with Ms-Excel file which consists the rigid flange coupling design calculations, the following steps are followed-



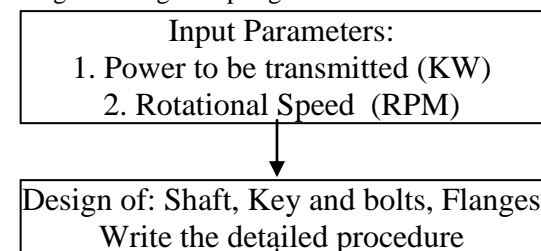
### Creating various part and assembly of parts in Creo

These parts (input shaft, output shaft, input flange, output flange and keys) are assembled in assembly mode. While execution, Creo ask respective input parameters for each part model.



### Create Excel sheet

Ms Excel is one of the best tool for programming. Following steps are followed to write program for the design of flange coupling.



Extracting design dimension of each part and transfer to Creo Parametric 2.0 should be done in rows as a comment in the text file. Knowing the power to be transmitted, Rotational Speed and Service Factor, calculations can be performed to determine if the design is acceptable. The spreadsheet prepared can not be directly read by Creo in excel format due to special characters and tables. Creo can read only text file (i.e. \*.txt), therefore we have to save excel file as *Tab delaminated text file*. In this format only texts, digits and tabs are stored and not any special formatting.

### Edit Pro-Program

The final dimension derived from excel file must be mapped to the dimensions assigned to each part in the zero. While mapping we should take care that input variables used for each excel output value must be same in zero relation. Each variables should be succeed by 'equal sign'('=').

For example if we are accepting shaft diameter as a variable named " SHAFT\_DIAMETER ", in excel spread sheet it should be written as-

SHAFT_LENGTH=	150
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And in Creo we read this variable in INPUT section

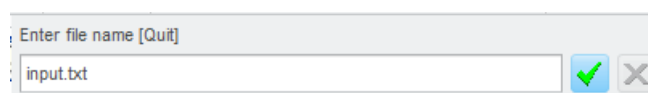
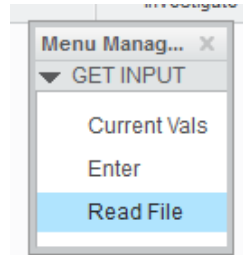
### INPUT

SHAFT\_LENGTH NUMBER

### END INPUT

The text file should and Creo parts and assembly must be saved in the same folder and set the working directory.

### Read Text file and Update Model



After reading all input values from the text file generated from Ms-Excel file, CAD model is regenerated and new updated model is created.

/*DESCRIPTION	SYMBOL	VALUE	UNIT
/*			
/*POWER TO BE TRANSMITTED	pow	50	Kw
/*ROTATIONAL SPEED	n	1000	rpm
/*SERVICE FACTOR	fact	2	—

Thus by doing the calculations for Power =50 kW, Rotational speed=1000 rpm and Service factor=2, all the parts are created in Creo using following values-

<b>SHAFT_LENGTH=</b>	<b>150</b>
<b>SHAFT_DIAMETER=</b>	<b>40</b>
<b>HUB_DIAMETER=</b>	<b>80</b>
<b>RECESS_DIAMETER=</b>	<b>60</b>
<b>LENGTH_OF_HUB=</b>	<b>60</b>
<b>THICKNESS_OF_RECESS=</b>	<b>5</b>
<b>OUTER_RIM_THICKNESS=</b>	<b>10</b>
<b>OUTER_RIM_WIDTH=</b>	<b>20</b>
<b>NO_OF_BOLTS=</b>	<b>4</b>
<b>BOLT_DIAMETER=</b>	<b>8</b>
<b>PCD=</b>	<b>120</b>
<b>KEY_WIDTH=</b>	<b>12</b>
<b>KEY_HEIGHT=</b>	<b>8</b>
<b>KEY_LENGTH=</b>	<b>60</b>

Thus by change only three input parameters all the design variables are calculated and thereafter model is updated with new design dimensions.

#### **4. Benefits of Integrating Creo Parametric with Ms-Excel**

1. Reducing the designing time ,designing cost as compared with traditional approach of design Iteration process, the values in an excel is easy to readable and it displays the inputs clearly based on the standards. Automation of design is possible Provide message feedback alerts on the excel if user enter invalid or incompatible values apart from standards.
2. Iterative process, through iteration in parametric design parameter, is required to obtain & the value of parameter is required. The value of parameter will be updated automatically in Creo after execution of MsExcel program
3. Reducing time to make CAD Model and Provide feedback(alert the user if invalid or incompatible values are entered)
4. Reduce designing cost very much compared to traditional method.

#### **CONCLUSION**

The parameterisation and relationship of model, the modular approach of the designing time to be reduced. This reduced time required to perform the desired operation to a great extent and subsequently, the lead time of design and designing cost is reduced. Using parameters and its relationship; one can use modular approach with significant reduction in designing time. These reduce time required to perform the operation to a great extent compared to traditional method. Subsequently, the lead time of design department is reduced as well as lesser designing cost. Interfacing of MsExcel with Creo (formerly Pro/Engineer) is more beneficial when iterative process along with more complex relationship is there between parameters of model. These applications for automation of design based on excel visual basic macro is simple to understand and implement.

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