

# Setting Up the Organisation Standard in CAD Modeling Environment

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## ABSTRACT

So many Mechanical industries in India are introducing IT in their manufacturing process to reduce the time lag of launch a new product into the market. Since, the most industries are using CAD software to prepare their product virtually instead of using the pencil, eraser and big drawing sheets. As we know that 80% time is waste in the design process. Also prepare drawing of part on such a huge sheet is quite difficult compared to using CAD software, because it reduces the time to prepare drawing of the part due to the availability of different features directly within the software. CAD software provides ease to prepare model virtually and also helpful to visualize the model before it is produced. Use of CAD software since reduces very much time in whole lifecycle of product. Most of the industries use the different kinds of CAD software according to their need. They were facing so many problems while designing the part also there can be possibility to introduce automation for the particular process. Those will become beneficial for the reduction of the design cycle time. In the paper I have decided to implement the customization of CAD software Creo Elements/Pro, formerly known as Pro/ENGINEER, an integrated 3D CAD/CAM/CAE solution created by Parametric Technology Corporation (PTC). This will be beneficial to reduce the time of designer while assembling a product.

Keyword: CAD customization, Pro programming, Pro toolkit

## INTRODUCTION

Creo Elements is widely used software in the industries for preparing their drawings and models. Since, I have decided to provide automation facility in this software for the reducing the time of designer while designing. In customization I have decided to deal with assembly section of the Creo Elements, as industry is spending huge amount of time in removing constraints manually for huge number of components assembly which can be minimized. To minimize it I am preparing the Pro Toolkit application which will automatically remove constraints of component and fix it with the help of batch mode.

### 1.1 Creo Elements

Pro/ENGINEER is the first commercial CAD system entirely based upon the feature-based design and parametric modeling philosophy. Creo Elements/Pro (formerly Pro/ENGINEER), PTC's parametric, integrated 3D CAD/CAM/CAE solution, is used by discrete manufacturers for mechanical engineering, design and manufacturing.

### 1.2 PTC Creo Elements Customization

Customization is defined as the configuration of products and services that meet customer's individual needs. Customization research has been developed in the fields of product design and operations management to develop ways in which goods and services can be adapted to individual standards. [3]

To customize Creo Elements various programming techniques are provided by PTC which are listed below:

- a) Pro Toolkit
- C programming based Application Programming Interface commonly known as API for Creo.
- b) J-Link -Java programming interface for Creo.
- c) Visual Basics Programming -Easy creation of user interfaces.
- d) Web Link-JavaScript programming interface for Creo.

### 1.3 Creo Elements/Pro TOOLKIT

Creo Elements/Pro TOOLKIT provides a large library of C functions to provide the external application safe and controlled access to the Creo Elements/Pro database and applications. Creo Elements/Pro TOOLKIT is the primary PTC application programmer's interface (API) for Creo Elements/Pro. [1]

Pro/Toolkit provides functions that enable communication with Pro/Engineer through a program written in the C language with the help of Microsoft Visual Studio C++ (VC++). Each Pro/Toolkit library C function performs an action on a specific type of object. These objects are items in the Pro/Engineer database, such as features and surfaces, etc. Hence, Pro/Toolkit can be used to access most Pro/Engineer operations through these C functions.

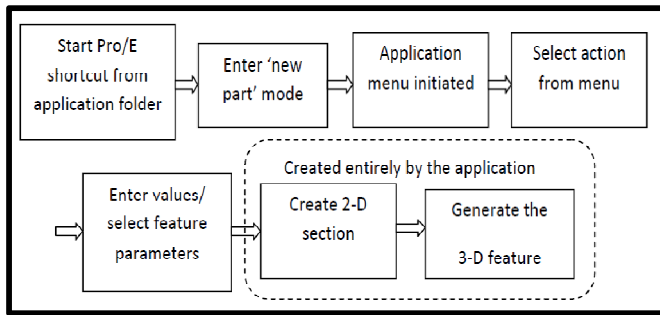


Figure. 1. Working of a Pro/Toolkit application [6]

An application that automates feature creation in Pro/Engineer first creates a two-dimensional section based on values input by the user and then performs the three-dimensional feature operation on it. Depending on the program parameters that define the sketch, the two dimensional section could be of any shape.

## II PROBLEM STATEMENT & PRAPOSED SOLUTIONS

The most of time consume of design engineer in setting up organization standard and modeling environment. We found some solution which stated below so that we can save lots of time to setting up organization standard and modeling environment.

So the following are the proposed solutions:

- 1) Defining standard BOM tables in drawing template with auto updating feature.
- 2) Modification with help of MS Excel.
- 3) Modification with help of Pro Programming.
- 4) Defining standard part with family table feature with help of MS Excel Integration.

So, we are providing a solution by setting up organization standard In a CAD modeling environment.

## III CREATION OF BOM TABLE USING REPEAT REGION

The Bill of Material (BOM) lists all parts and part parameters in the current assembly or assembly drawing. The BOM also comprises a breakdown and a summary of the report. The Breakdown section lists what is contained in the current assembly or part. The Summary section lists the total quantity of each part included in the assembly, and is the list of all the parts needed to build the assembly from the part level.

### 3.1 BOM Balloons

BOM balloons are circular callouts in an assembly drawing that show Bill of Materials information for each component of an assembly view. The information is derived from a report table repeat region that you also specify as a BOM balloon region. Before you can add BOM balloons, you must create the table, add the repeat region, enter the desired report symbols, and designate the BOM balloon

region. When you have done this, you can show BOM balloons on a selected assembly view

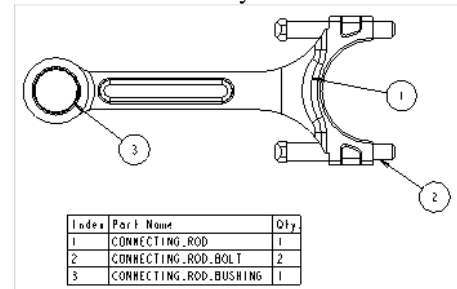


Figure. 2. BOM Balloons

BOM balloons usually show an index number, corresponding to a part name in the table. The report symbol for the index number is rpt.index. The report symbol for the part name is asm.mbr.name.

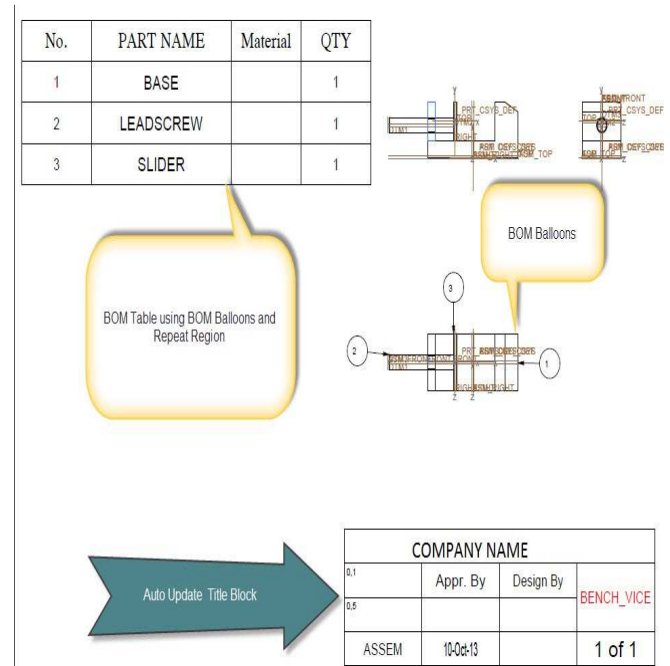


Figure 3 -Drawing Sheet With BOM Balloons Table.

## IV-MODIFICATION WITH HELP OF MS EXCEL.

The parametric nature of any component design makes the effective implementation of the designconcept with ease and simplicity using Pro/Engineer. Using parameters and its relationship, the parametricconcept is implemented for designing differentcomponents using Pro/Engineer software throughdifferent interface for automation of them. DetailMethodology of interfacing of Pro\Engineer with Excelsheet is reported in this chapter. Data from Excel istransferred to Pro/ENGINEER via Pro/Program touupdate the solid geometry. The key steps in setting up ProE for automation through an Excel file are:

- Create the model in ProE.
- Name dimensions.
- Edit the Pro/Program to include the dimensions named above.
- Create a tab-delimited text file with the dimensions named above.
- Create a trail file that regenerates the ProE model from the text file.
- Create an excel macro to run ProE using the trail file.

#### Step I:

In ProE:

- Create the model.
- Name the dimensions you want to change (by right-clicking a dimension and going to Properties).
- Edit the Pro/Program (in Wildfire 2.0 it is under Tools -> Program -> Edit Design):

INPUT

*Dimensionname1* number

*dimensionname2* number

*dimensionname3* number

END INPUT

#### Step II:

In Excel:

Create a macro to save a tab-delimited text file with the names of the dimensions on your ProE model that you want to modify (check out the excellent Profiles Magazine article on the entire process: Drive Your Pro/ENGINEER Model with a Spreadsheet). Enter values for the dimensions and save a tab-delimited file ("mydata.txt").

#### Step III:

In ProE:

If you have ProEngineer open, close it. Open it again, open the file you want to modify and perform all of the actions that you would like to automate (including Edit -> Regenerate -> From File, and select the text file you generated from Excel The code given in Annexure-II). End by closing the file and exiting ProEngineer. Open windows explorer, find the trail file that was created, and rename it to "mytrailfile.txt" or something similar.

#### Step IV:

In Excel:

Open the spreadsheet you created earlier. Create another macro (using the Visual Basic editor) that opens ProE in no graphics mode and run it from the trail file you created earlier. You are going to want to use the Visual Basic "Shell" command with the following:  
proe.exe -g:no\_graphics mytrailfile.txt You can also set the relative path using Visual Basic "ChDir". Whatever relative path you set will determine the directory that ProE opens with as the default working directory. ProE will save

trail files, etc, to this directory even if you change the working directory during your session. If the relative path is not the location of proe.exe, you will need to use the full file path name to proe.exe (and possibly also to mytrailfile.txt, although it is easiest just to put the trail file wherever you will be setting the relative path to).

## V. MODIFICATION WITH HELP OF PROPROGRAMMING.

### 5.1 About Pro/PROGRAM:

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. To enter the Pro/PROGRAM environment, click Tools > Program from the PART or ASSEMBLY menu. Initially, you can gain access to only a design listing that exists in the model. However, whenever you edit a listing, a file is created that contains the latest design specifications. At this point, two design listings exist for the same model, From Model and From File. After you successfully incorporate design changes in the model, From File is deleted, and only From Model is available. In those cases where a From File design listing exists, the WHICH DESIGN menu displays two commands:

- From Model—Retrieves a design listing built in the model.
- From File—Retrieves a design for a model from an existing file named assemblyname.als or partname.pls.

### 5.2 View the Model Design:

- From the PART or ASSEMBLY menu, click Program.
- Click Show Design or Edit Design from the PROGRAM menu to view the model design.
- If you choose Show Design, the program appears in an information window.
- If you choose Edit Design, the program appears under the system editor—usually in the start up window.
- A typical design listing may contain any of the following:
  - Input variables with their current values.
  - Relations.
  - IF-ELSE clauses
  - Lists of all the features, parts, or assemblies in the design, which, when enclosed within "IF condition... ELSE... END IF" statements, create alternate design branches.
  - EXECUTE statements (Assembly mode only).
  - INTERACT statements.
  - Feature suppression and order.
  - MASSPROP statement.

#### 5.2.1 Entering Input Variables :

When a model design has input variables, the system prompts you to enter their values whenever you regenerate the model or incorporate new instructions in the model. You

can enter data using the following commands on the GET INPUT menu:

- Current Values— When you run the program, it uses the current values without requesting your input.
- Enter—Enter new input values as prompted. Check boxes in the INPUT SEL menu control parameter selection. Pro/PROGRAM only prompts you to enter a new value for the checked parameters.
- Read File—When running a program, the system uses input from a file. Type a file name (including the extension, if any).

### 5.2.2 To Select or Modify Input Parameters:

- Click PART > Program, and then click PROGRAM > Edit Design. The system editor displays the current program for the model.
- Edit the program input list.
- Incorporate your changes into the model.
- Click GET INPUT > Enter.
- In the INPUT SEL menu, click the check boxes next to the input parameters for which you want to enter values, and then click Done Sel.
- Enter the values as prompted in the message area.
- Click PROGRAM > Done/Return.

### 5.3.3 Input from a File:

Instead of entering variables manually, you can enter them from a file located in the current directory using Read File. The input file must have one input per line, formatted as follows:

param\_name = value or expression

For example:

THICKNESS = 2.5

INCLUDE\_VALVE = YES

MATERIAL = "STEEL"

If you enter parameters from a file that contains fewer parameters than are called for in the INPUT statement, the system assumes current values for the missing parameter.If, on the contrary, the output file contains more variables than are needed for the execution, those parameters not pertaining to the program are disregarded. Because the program ignores those parameters that do not pertain to this particular program, you can create an input file that acts as a global source for a number of models.

## VI. PART FAMILIES WITH HELP OF MS EXCEL INTEGRATION

Partfamilies are especially convenient to use in assemblies. From a singlepart file, you can assemble multiple instances and multiple occurrences of each instance in multiple assemblies. If you have a Assembly license, instances from the same family can be exchanged with each other when you use the Edit ► Replace command. The Replace command on the Edit menu lets you replace a component of the manufacturing assembly, for example, a reference part, by a member of the same part family. You can create NC

sequences for one member of the family, and then generate appropriate CL data for other members by replacing the design model and regenerating the manufacturing model. When you replace a design model and regenerate the work piece, the NC sequences and material removal (where applicable) are updated according to the new model After an instance of a family has been assembled, it can be replaced automatically with any other instance in the same family, if the replacement instance has all of the necessary features for assembling (that is, no features used in the assembly process are suppressed).

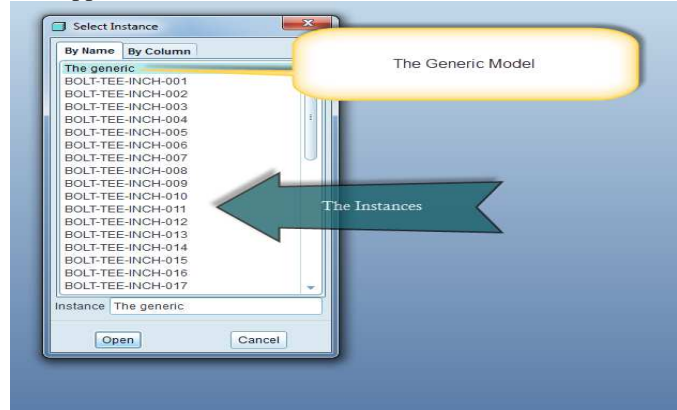


Figure. 3. Part Families with generic and instances

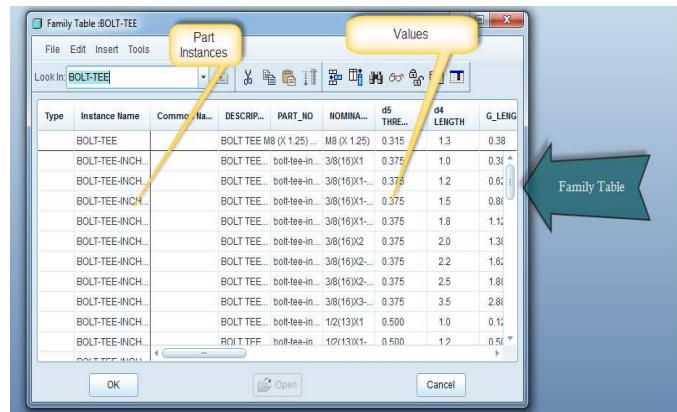


Figure. 4. Part Families Table with Values of Instances

## V. CONCLUSIONS:

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 C Language with Pro/Engineer is more beneficial when iterative process along with more complex relationship is there between parameters of model. This application for feature is simple to understand and implement. Also with auto update BOM table need not create bom table every time and need not to give part no by this automation it can automatically created. There is 90% time saving with this automation.

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