Integration of Pro\Engineer with Excel and C Language for Design Automation

Ashish M. Thakkar

P.G. Student, Dept. of Mechanical Engg.
B.V.M. Engineering College
V.V.Nagar, Anand, Gujarat, India

Yashavant D. Patel

Associate Professor, Dept of Mechanical Engg,
A.D.Patel Institute of Techonology.
New V.V.Nagar, Anand, Gujarat, India

Abstract— **The parametric nature of any component design makes the effective implementation of the design concept with ease and simplicity using Pro/Engineer. Using parameters and its relationship, the parametric concept is implemented for designing different components using Pro/Engineer software through different interface for automation of them. Detail Methodology of interfacing of Pro\Engineer with Excel sheet is reported in this paper. Data from Excel is transferred to Pro/ENGINEER via Pro/Program to update the solid geometry. The C-Language interfacing with Pro/Engineer is also carried out here, which improves the product design capability significantly.**

Keywords-Pro\Engineer, Parametric, Interface

I. INTRODUCTION

Pro\Engineer is one of the most widely implemented CAD package in Industry. It enable creation of complex three dimensional models of a product which can be used for analysis purposes to optimize the design and then put into introduction. Pro\Engineer is the industry's first successful parametric 3D CAD modeling system. The parametric modeling approach uses parameters, dimensions, features and relationships to capture intended product behavior. Pro\Engineer provides a complete set of design and manufacturing capabilities on one, integral, scalable platform.

A parametric model is defined by its attributes which could be input parameters, An input parameter is a variable or relationship that defines a key dimension A parametric model is re-usable model because modification can be implemented on model. Mostly two approaches used for parameterization- Interfacing and Programming.

An interactive approach involves parametric capabilities within system to create flexible re-usable models. The model can be updated from a common, hared parameter data. Once the Pro\Model has been defined parametrically, the user allows the system to regenerate model with new values. so here for that interfacing of Pro\e is made with Excel Sheet.

A programming approach involves usage of programming language like C language to develop parametric models. For that Interfacing of Pro\Engineer is made with c language to take advantage of parametric concept. This will reduce the complexity of modeling. It will also helpful to increase design capability.

II. SET UP FOR INTERFACING BETWEEN PRO\ENGINEER AND C LANGUAGE

In order to make sure the compatibility between Pro\Engineer and C program, one must have to set following variables:

- To link and execute a Pro/ENGINEER User Program on a Windows NT based workstation, one must have already installed a certified compiler such as Microsoft's Visual Studio.
- After installing the certified compiler you must set the following environment variables under the "Environment" area within the "System" icon under the Windows NT "Control Panel": "Variable"=NT_COMPILER "Value" = CL (command line compiler for Microsoft's Visual C++)
- Update or create environment variables **LIB** and **PATH** with value to full path to respective platform specific lib and bin folders.
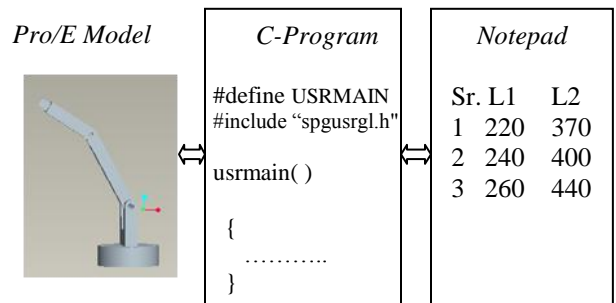
Interaction between Pro/E, C and file operations with editor

The illustrated interfacing helps a lot for various level of iterative process in safe design calculation for deriving a required dimension along with file operations for next higher or lower value selection from any editor improves the design productivity significantly without any error for any industry.

III. METHOD FOR PRO\ENGINEER INTERFACE WITH EXCEL SHEET

The detailed methodology of interfacing with excel is presented herewith. Pulley is taken as case study because design of pulley is depends on some of basic parameters. To interlink Pro\E with Excel sheet, following proedure must be followed by user:

(i) Create Pro\Model of Pulley

Make Pro\Model as per design by geometric and Parametric constraints.Fig shows 3-D model of Pulley.
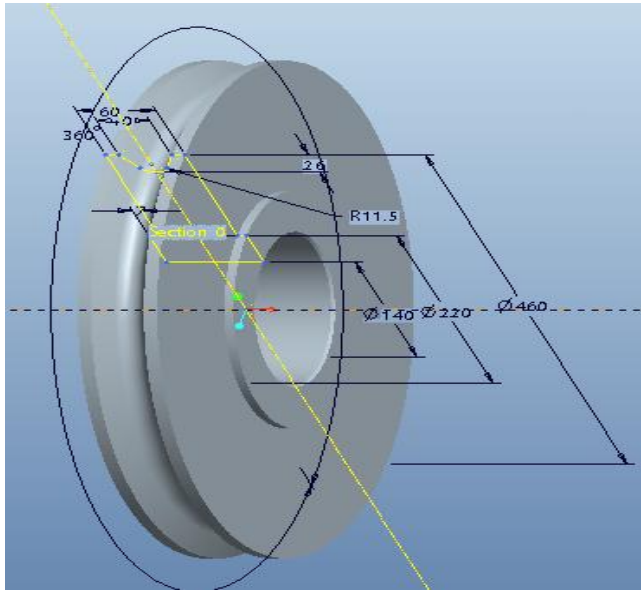

Figure:-1 Pro\Model of Pulley

(ii) *Create an excel sheet to act as interface*

Make new excel sheet and add input areas for pulley: OD, THK, BORE, W, R, L Also add EXPORT button in workbook. The dimensions are derived from standards. The cells are constrained using DATA\Validation in Excel.

	A	B	C	D	E	F
1						
2	OD	=	573			
3	HD	=	250			
4	BD	=	160			
5	THK	=	60		EXPORT	
6	HW	=	74			
7	W	=	48			
8	R	=	13.5			
9	L	=	36.5			
10						

Figure:-2 Excel sheet with EXPORT button

(iii) Export a text file using visual C-program

To use data in PRO\E, data will be exported as a text file. Using File\Save As and pick the file type text. A excel sheet macro will help automate the export process.Macro checks for and removes any existing data text file exports the active sheet to text file(C:\pulley\pulley1.txt), and checks for and removes the spreadsheet file before saving it again.Creat a new push button on the sheet using the Forms toolbar, and assigning above macro to the button. Click the button, data will be exported to "c:\pulley\pulley1.txt"

(iv) Configuring Pro\Model

In Pro\Model of Pulley, drive the dimensions as per parameters to pass the values. For that rename the dimensions to appropriate variable names. Use Info\ Switch Dimensions to display the dimensions names.
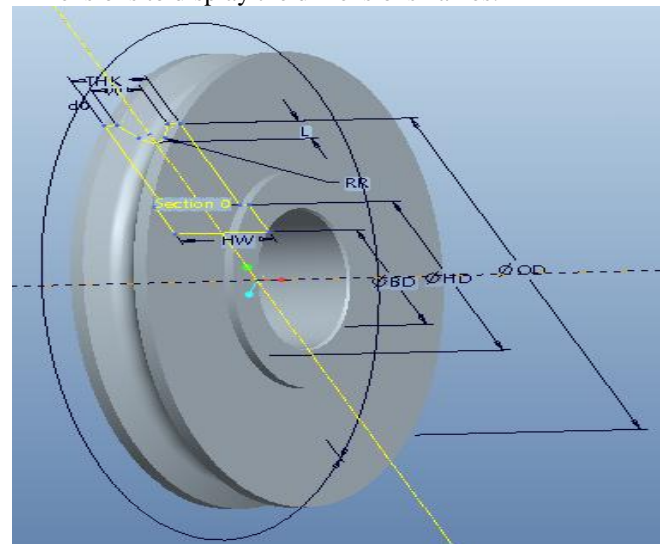

Figure:-3 Pro\Model with its dimension renamed.

(v) *Edit and Use Pro\Program*

To edit program, Tools\ Program\Edit Design\From File. The editor displays the pro\program.Then add lines for each variables, stating the name:

```
INPUT
OD
HD
BD
THK
HW
W
R
L
END INPUT
```

This allows Pro\Model to accept the inputs to drive geometry.Now save the file. Then yes (to incorporate changes). Now menu manager deflects the changes in the program with the program as below.Select read file.Enter

file name"c:\pulley\pulley1.txt". Hence geometry will be generated as per data in text file.

Figure:-4 Menu Manager

The time study of the model making of pulley is carried out to know the advantage. Using this one can reduce time up to 80% compare to traditional method.

IV. METHOD FOR PRO\ENGINEER INTERFACE WITH C PROGRAM.

Before one begin interfacing of pro\e and c language, one must have set all required variable shown above. Here to show the methodology of interfacing of c language and pro\e, Single Forged Hook is parametrically design by using I.S. Design as a case study. Because one know that design of hook is mainly depends upon two parameters Bed diameter (C)and Shank Diameter(G) which will change as per change in lifting capacity. One must do the following procedure to do interfacing:

(i) Create Pro\ Model

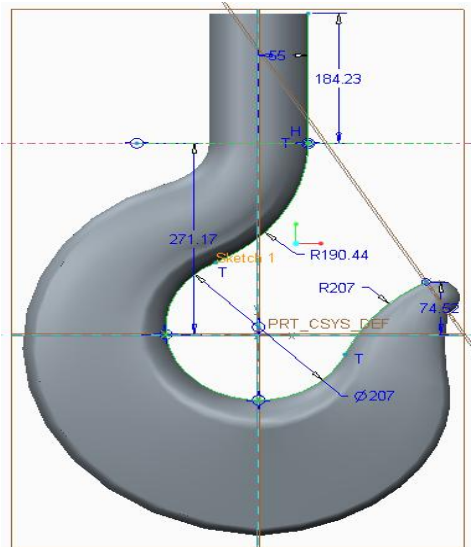Make PRO\Model as per I.S. design by using parametric relationship and constraints

Figure:- 5 Parametric Pro\Model of Single Forged Hook.

(ii) Create user program in C language

To make a new C program; Click on Tools\Relations. Relations dialog box open then go to Utilities\User Prog\Edit. Enter program name without the .c extension. An editor window opens. Make a user program in C language and save the program.

(iii) *Link C program*

To link C program, again click Utilities\ User Prog\Link. The system starts compiling the program. If the system detects errors in program, it prints message "Errors in compiling and linking" in the startup window. If error is there then edit the program first and then again link program the same program. After linking program, system will prints message "Compiling and Linking Successfully" in the startup window.

(iv) *Run C program*

To run this program, again click Utilities\User Prog\Run. The C program begins executing. Output will be written to the start up window. The program prompts user for necessary inputs. Enter the value of C and G in the opening editor window as per I.S. 3815-1969. Table is given below which shows different value of C and G.

TABLE:- 1 PARAMETRIC VALUES OF C AND G AS PER I.S.3815-1969

SAFE LOAD(TONNES)	C	G
5	85	45
10	119	60
20	169	85
25	189	100
32	207	110

(v) *Regenerate Pro\Model*

Regenerate the hook model to update pro\model as per promoted input by user. Updated model is shown below after executing new inputs:

The time study of the model making of Single Forged hook is carried out to understand the use and importance of this interface for industrial applications. The results are displayed in tabular format as under:

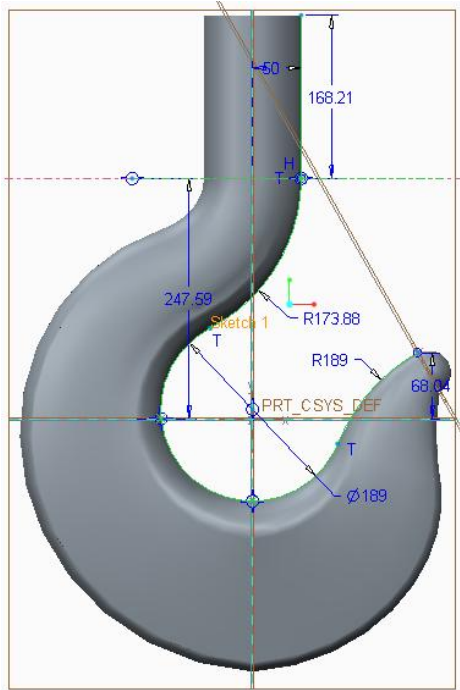| Sr. No.	Time for a new model	Time for new using interfacing	% time saving
1	30 minutes	3 minutes	90%

Figure:-6 Regenerated Pro\Model of Single Forged Hook

V.   STEPS FOR INTERFACING PRO\ENGINEER AND EXCELSHEET

VI.   STEPS FOR INTERFACING PRO\ENGINEER AND C LANGUAGE

VII.   ADVANTAGES OF INTERFACING OF PRO\E

- Iterative process, through iteration in parametric design parameter, is required to obtain & the value of parameter is required & this value of parameter will be updated automatically in Pro\E after execution of C-program.

- Reducing time to make Pro\Model

- Provide feedback(alert the user if invalid or incompatible values are entered)

- Present values in an easy-to-read format (display inputs and outputs clearly.)

- Design Automation is possible.

- Reduce designing cost very much compared to traditional method.

VIII.   CONCLUSION

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.
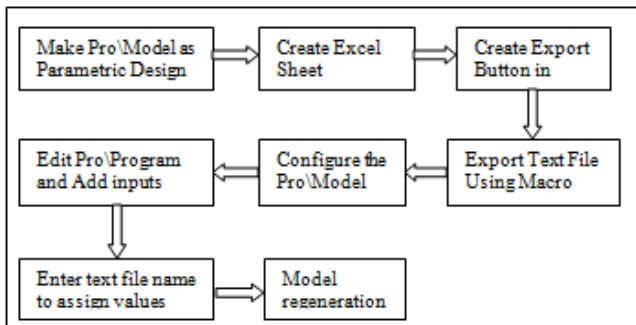
ACKNOWLEDGMENT

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