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3D Parametric Modeling for Product Variants Using Case Study on Inner Ring of Spherical Roller Bearing

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Abstract

3D solid modeling has been widely used because of ease in visualization, generation of manufacturing drawing and adaptability compared to traditional 2D drafting. However modeling process is a very time consuming and many draftsmen do not have the skills of performing such task. Parametric modeling makes this generating task easy and less time consuming because basic features and relations between them are in constant fashion for a specific product. In this paper an attempt has been made to integrate the commercially available package Pro/E with Microsoft Excel spreadsheet for 3D parametric modeling. Various product variants of the inner ring of spherical roller bearing have been executed by parametric designing concept in Pro/Engineer Wildfire. Bearing has been selected for this study because it is a commonly used mechanical component for supporting and transmitting relative motion.

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Keywords: : Parametric Modeling, Modeling Automation, Spherical Roller Bearing, Excel Analysis

Nomenclature

CAD	Computer Aided Design
Pro/E	Pro/Engineer
MS	Microsoft
UG	Uni Graphics
API	Application Programming Interface

1. Introduction

Bearing is a mechanical component used to provide relative positioning and rotational freedom while transmitting load or motion between two parts. Bearing is used wherever relative motion between two parts occur especially in automobiles, textile machineries, power plant equipments, agriculture machineries, etc. Spherical roller bearings are mainly used in automobile engines, driveshaft, heavy machinery, sewing machines and many others. Generating a part model for bearing includes model generation of inner ring, outer ring, cage and roller. Part modeling of the inner ring of bearing requires parameters like inner diameter, outer diameter, width, track radius, fillet dimensions, etc. These parameters change as per bearing designation and this change can be automated by Microsoft Excel and Pro/E integration. This integrated Excel file is also used to store all bearings' data in a single database.

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2. Literature review

Many research attempts have been made in the area of parametric modeling. The attempts mainly focused on parametric relations in solid modeling software. Wang et al. [1] made the design change automation possible by parametric sketching and design associability in 3D CAD for bottle design by interfacing between 3D CAD model and dimension database. They mainly focused on controlling the design remotely by changing a text file or database software. Thakkar and Patel [2] defined the technique for interfacing through MS Excel and C language by the case study of pulley design. Ongkodjojo and Gunawan [3] explored the technique of how to get full control on model by managing and driving the parameters attached to it by customizable flat bed conveyor design. They demonstrated the technique of developing product variants based on a single template model. Zheng and Chen [5] described the technique of spur gear generation with the tools like Relations, Parameters and Program in Pro/E for design modification. They mainly concentrated their work on exploring the tools like Relation and Program for secondary design generations. Jason [6] described the tool Pro/Program and procedure to modify the program for changing the part model. Fan et al. [7] automated the development of standard parts library by Family table tool in Pro/E interfaced by Visual C++ and Pro/Toolkit. They concluded that the parts having standard dimensions and simple geometric features adoption of Family Table tool could be more beneficial. Kuang [8] built standard parts with UG NX and MS Excel by Visual C++ interfacing.

3. Methodology for Modeling Automation

Pro/Program is a powerful secondary generation tool to validate parametric design of the component. Pro/Program reflects all parameters and geometric data of the part in a text data form. This data can be modified to add new feature, delete existing feature, suppress the feature and change the dimension of the feature. A proper user interface (API) can directly modify the Pro/Program and the part model can be driven according to the user input in user interfacing. Here Microsoft Excel and pro/E are integrated by Excel Analysis tool which acts as an interfacing media. Excel Analysis tool transfers the spreadsheet data to the Pro/E database. These data can be related to the part's features' data by 'Relations' tool. Thus transferred spreadsheet data can be related to the feature's data i.e. part dimensions.

3.1. Prepare an Excel spreadsheet

An Excel spreadsheet which contains all feature names and respective dimensions of the part for the particular bearing designation has been prepared. Another spreadsheet which contains all bearings' database has been prepared. The Excel spreadsheet has been customized such that the user has to input only bearing designation and all relevant feature dimensions would be extracted from the database which is created in another spreadsheet. This customization has been carried out using VLOOKUP and HLOOKUP functions of Microsoft Excel to communicate data between different spreadsheets.

	A	B	C
1		ENTER BEARING DESIGNATION	SRB_1
2			
3		FEATURE	DIMENSION (mm)
4		WIDTH	80
5		INNER_RAD	79
6		OUTER_RAD	102
7		TRACK_RAD	131
8		X	41
9		Y	228
10		FILLET_1	1
11		FILLET_2	4
12			

Figure 1: Excel Spreadsheet

3.2. Generate a generic model

A 3D model of the component with all correct features dimensions and proper constraints between features have been generated as shown in the figure 2. This model can be called as generic model and it would be changed as per new bearing designation entered in the Excel spreadsheet. Pro/E would give the feature dimension name as per its standard when a part model is created. Various features of inner ring have been renamed as previously described in Excel spreadsheet as shown in the figure 3. This procedure simplifies the identifications of the features of the inner ring.

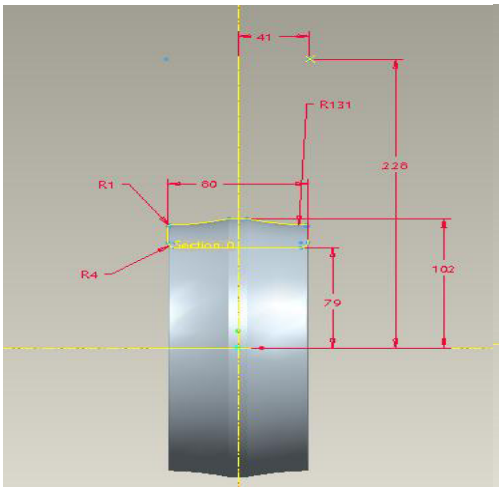

Figure 2: Generic Model

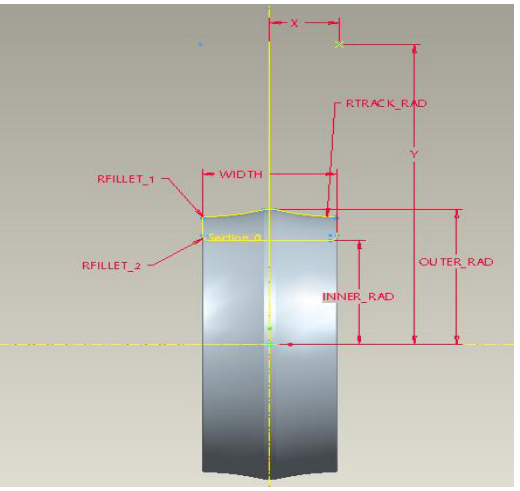

Figure 3: Renamed Model

3.3. Transferring data from Excel to Pro/E

An Excel Analysis from External Analysis feature have been selected using Analysis menu in Pro/E as shown in figure 4 to transfer data from Excel spreadsheet to Pro/E.

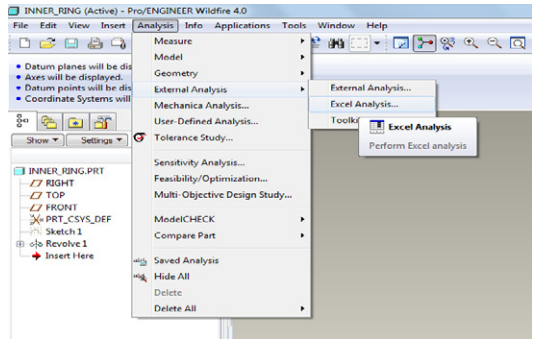

Figure 4: Excel Analysis in Pro/E

A dialogue box would open, in which the previously generated Excel spreadsheet have been loaded using 'Load File' icon. The way of communication Excel to Pro/E has been selected. The 'Output cells' icon in dialogue box have been clicked and the cell range containing dimensions of all features from the Excel spreadsheet have been picked as shown in the figure 5. After the 'Compute' icon has been clicked, selected cells from Excel spreadsheet have been transferred to Pro/E. By clicking 'Add Feature' icon, selected values would be stored in the Pro / E database. The proper name for this integration before closing Excel analysis dialogue box has been given as shown in figure 6.

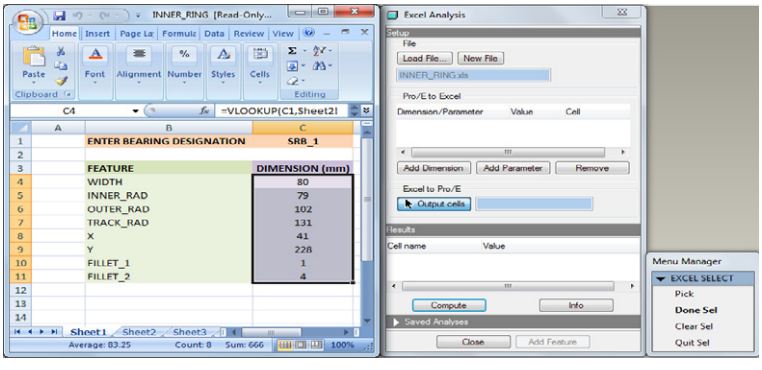

Figure 5: Cell Selection from Excel Spreadsheet

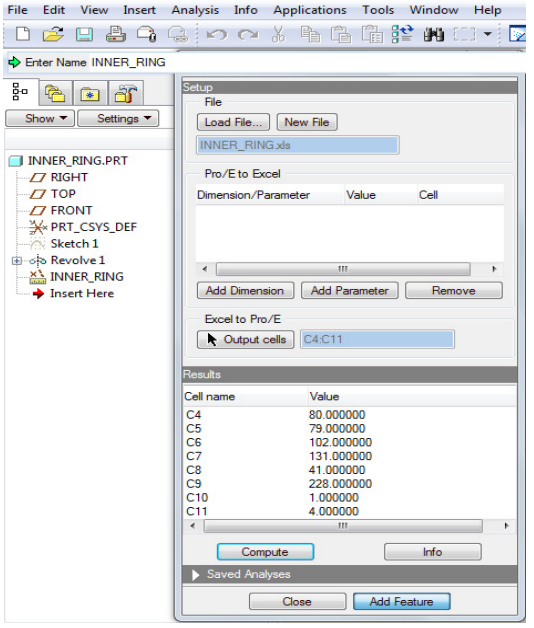

Figure 6: Integration of Excel to Pro/E

3.4. Relating data with part's features

To relate data of Pro/E with Excel spreadsheet, from 'Tools' menu 'Relations' icon has been selected. In Relations window, 'From List' icon from Insert menu has been clicked. In 'Select Parameter' window, 'Feature' object type from 'Look in' icon has been selected as shown in the figure 7. Values of each cell selected from the Excel spreadsheet with a particular alphanumeric name would pop up by selecting the previously defined integration name from the modal tree. Same has been shown in the figure 8.

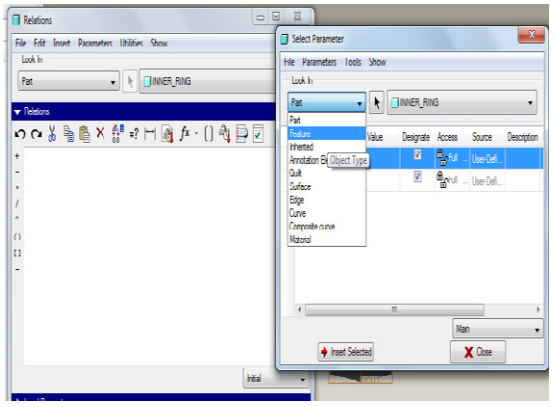

Figure 7: Feature Relations

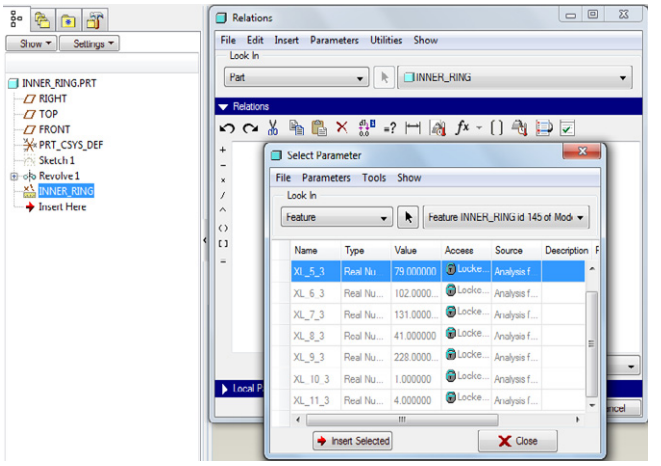

Figure 8: Feature with Dimension

By clicking 'Insert Selected' icon, highlighted name would open in 'Relations' window. This name has to be related with adequate feature's name given in renamed model. Similarly all features of the part with a corresponding cell address have been related as shown in the figure 9. All relations have been validated using 'Verify' icon in 'Utilities' menu which makes Excel and Pro/E completely integrated.

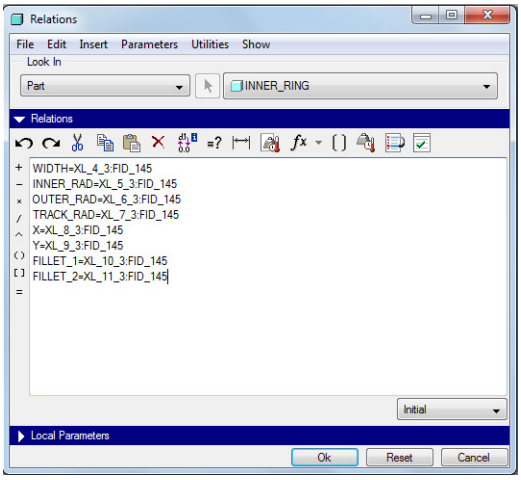

Figure 9: Relations with Corresponding Feature

4. Procedure for Modifying the Model

The basic procedure to modify the model as per bearing designation has been given in flow chart as shown in the figure 10. 2D drawing of this updated model can be generated by creating Drawing macro using 'Tools' menu as shown in the figure 11.

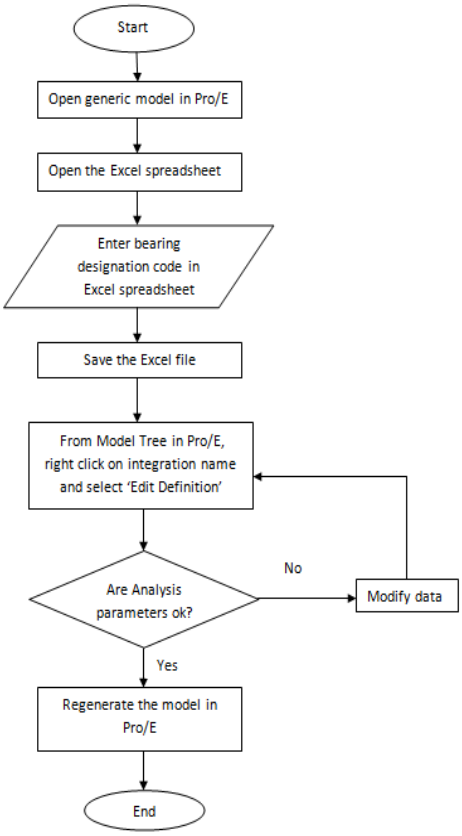

Figure 10: Flow Chart to Modify the Model

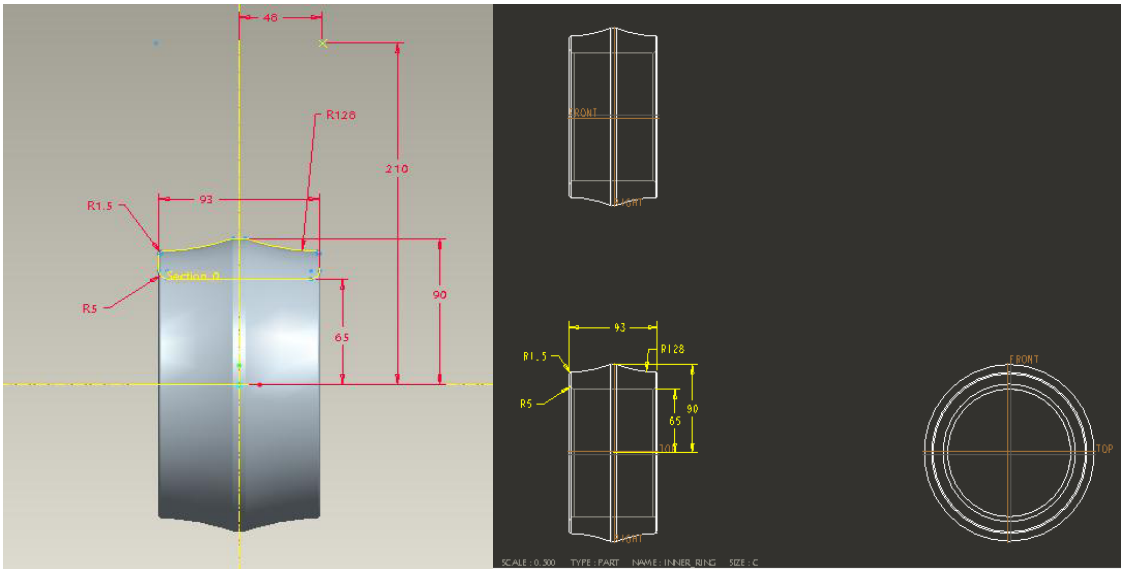

Figure 11: Updated Model and 2D Drawing of Updated Model

Conclusion

A generic model and a related database sheet in Microsoft Excel have been generated. Then Excel data have been transferred into Pro/E by Excel Analysis tool. All these transferred data are related to the respective features of the part by 'Relations' tool. Thus at the end Excel datasheet is linked with the Pro/E model. User can update the model just by modifying the sheet. This takes comparatively very less time to generate complex part models with respect to generating them individually. The present study demonstrates the modeling automation technique for the inner ring of spherical roller bearing. This technique is more suitable and simpler than any other techniques like VB interfacing, Pro/Toolkit, etc., when the relations between features of a part are of constant fashion. This automation can further be proceeded by exporting this model to the Analysis package or CAM package. Analysis task and NC program generation for manufacturing would be made easy by using 3D parametric modeling in combination with the Analysis package or CAM package respectively.

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