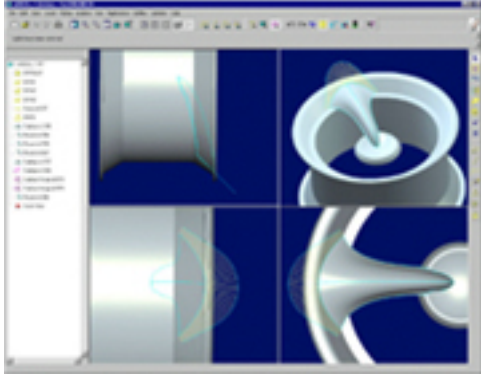


Software Review

Pro/ENGINEER 2001 - Beyond The Parametric To Interactive Surfacing

by Jeffrey Rowe, IDSA

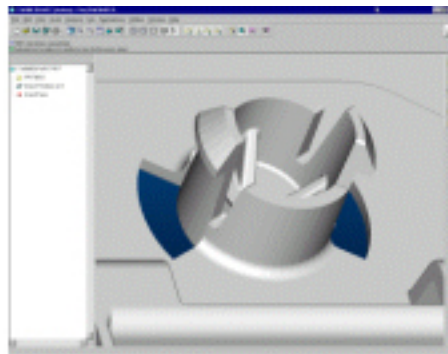


[Interactive Surface Design](#)

The convergence of design and engineering inside Pro/ENGINEER

There was a time (and not all that long ago) when there was no such thing as a parametric modeling package for digital mechanical design. PTC (formerly known to the world as Parametric Technology Corp.) changed that by introducing the first parametric (also called feature-based) solid modeling package, Pro/ENGINEER (or Pro/E - I know the company hates this reference, and I'll try and use it sparingly) in the early 90s. Parametric design really revolutionized mechanical design because models consist of combinations of features. Using commands, you can add, subtract, and intersect features to define and create a parametric model.

Features are the basic units of parametric models, and each feature has properties that define it. When a feature is created, applicable geometric constraints and size parameters are applied to it. The parametric modeling application stores the properties and creates the feature. If a change is made, the modeling application regenerates the feature according to the definition properties assigned to it.



With practice, solid features like this relatively complex barbed boss can be generated relatively quickly.

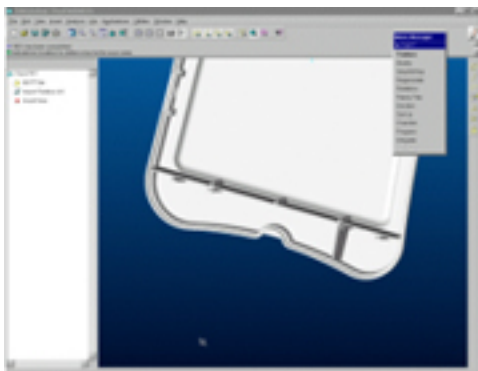
In parametric models, a model's features are defined by their sizes and relationships (also called parameters). These parameters are stored in the model and generate parts and assemblies and are what actually control the geometry. Two types of parameters drive a model's geometry:

- Size parameters that are a model's dimensions

- Geometric parameters (also called constraints) that define and maintain a model's geometric properties, such as tangency.

A parameter is analogous to a variable because it can change. Also, like a variable, a parameter can define other parameters. However, unlike a variable, a parameter always has a value assigned to it.

Although there are other modeling methods, probably parametric modeling's biggest advantage is the fact that models can be easily updated by changing the relationships and sizes of features that define the models. In other words, with parametric modeling you can change and update models that reflect changes more quickly than with other modeling methods.



A detail view of a Pro/E part model.

In addition to providing parametric modeling, Pro/ENGINEER also employs another unique twist; something PTC calls behavioral modeling. Basically, behavioral modeling allows specific requirements and desired functional behavior of a product design to automatically drive and adapt the parametric design. It lets you solve a variety of potential engineering problems, analyze the impact of design changes, and perform "what if" scenarios based on multiple objectives and design variables.

Pro/ENGINEER was the first widely available CAD application that exploited the power of parametrics. Since introducing the newest version of its flagship MCAD product, Pro/ENGINEER 2001, PTC has said that this latest version has had some significant improvements and enhancements in the areas of its user interface and surfacing.

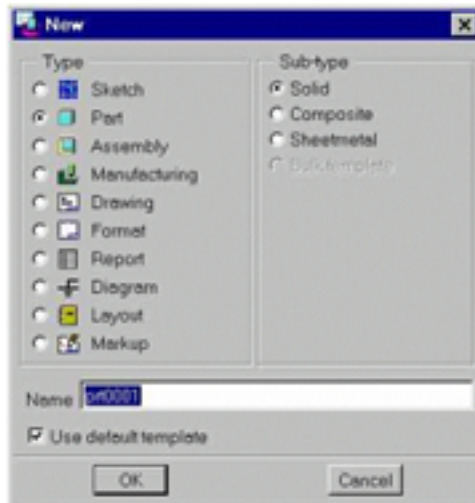
Although the sheer magnitude of Pro/E is huge, let's have a look at what's new and improved for the 2001 edition.

User Interface

Before we get into the UI, I'd like to briefly touch on installing Pro/ENGINEER, because I found it to be a bit daunting compared with most other MCAD applications I've dealt with lately. Even in a standalone platform environment you'll have to have an Ethernet card installed to install the software, as it keys your machine's ID according to the NIC. I'm all for anti-piracy schemes, but to load and use Pro/ENGINEER (or just about any PTC product), you need a CPU ID number, customer number, sales order number and/or configuration number. This all seems like a bit much to me, but I guess it works, because you don't hear or see much about pirated copies of Pro/E in North America.

With Pro/ENGINEER 2001, PTC claims to have "streamlined the user experience." Significant UI improvements actually began with Pro/ENGINEER 2000 and have continued with the 2001 version. If "streamlined" means less cluttered, then the claim is true, as the sheer quantity of icons on the screen at

any given time have been greatly reduced. Also, several of Pro/E's commands are actually icons that can be expanded as needed, such as New file`New Sketch. Although more efficient than in the past, my experience with Pro/ENGINEER 2001 required more picks and clicks than most other packages for invoking functions and commands while modeling, but in most cases, not that many more.



All projects in Pro/ENGINEER 2001 begin here by selecting the type of project you want to accomplish.

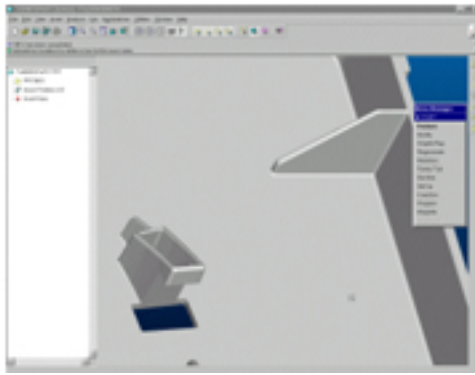
While not fully Windows compliant, the pull-down menus are gradually getting more of a Windows "feel." Significant here is the Insert pull-down menu, because it contains most of the commands normally displayed on cascading menus. As much as the UI has improved, however, it still has a definite "Unixy" feel to it. And while they have been reduced in number, there are some instances where there are too many cascading menus. These are relatively small things, but do not exactly make Pro/ENGINEER user friendly and easy to learn/use for users either new to CAD or users migrating from other CAD packages.

Finally, this version of Pro/ENGINEER takes fuller advantage of right-click menus. Because they are context and object sensitive, their main advantage is the fact that you can interact directly with features and parts without having to mess with (potentially) tier upon tier of cascading menus.

I consider a software product's documentation part of the user interface, or user experience, and I found the material supplied with Pro/E to be insufficient for many tasks, especially more complex ones; inconsistent, and in many cases cryptic. Also, the on-line help is 100% wall to wall text. At least a few graphics would be nice to illustrate some points. This documentation deficiency is another one of the things I feel makes Pro/E so intimidating to so many new users. A company called CADTRAIN (www.cadtrain.com) offers third-party computer-based and Web-based training for Pro/ENGINEER 2001, and I'd recommend you check it out.

Creating Parts and Building Assemblies

From the beginning, Pro/ENGINEER's part modeling capabilities have been renowned as extremely powerful, but they also had a reputation of being difficult and cumbersome. Why? Because formerly you quickly got bogged down with having to constantly refer to a myriad of menus and dialog boxes with relatively little direct interaction with your model. That's changing now, though, because you can directly interact with sketches and features like you never could before with previous versions of Pro/E. With what PTC calls Direct Modeling, you can now literally push and pull sketch curves and surfaces to create a desired shape or form.



Ribs and other mechanical features in plastic parts are added as a product is developed in Pro/ENGINEER 2001.

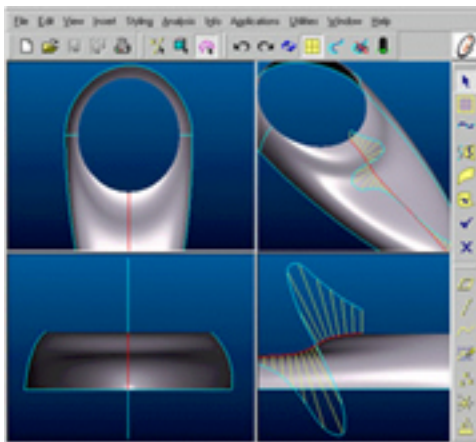
I created several plastic parts with ribs, tabs, and bosses and found Pro/E 2001 to be much easier for creating and modifying these parts than previous versions. However, I have to admit that virtually all of these parts could have been created with most mid-range MCAD applications, such as SolidWorks, in less time but equivalent results. Assemblies, though, are another story.

Modifying and regenerating parts and assemblies, especially complicated ones, used to be one of Pro/ENGINEER's least virtuous traits. This was addressed in a positive way, however, with Pro/E 2000i2 and the Intent Feature capability that lets you actually define what you want a feature to be and do through design intent, and not just strict constrained geometry. For example, you could apply a chamfer between solid bodies, not just edges. In addition to this, in Pro/ENGINEER 2001, Selective Regeneration lets you regenerate only specific portions of construction history, and not the entire tree, thus accelerating the regeneration of parts and assemblies that are modified.

Another nice capability related to part regeneration is that Regen Info that displayed information for a current step and operation has been superseded by Model Player that visually steps you through your model's construction history. Once viewed, you can modify specific features, repair any regeneration errors, and then continue with your design.

Interacting With Surfaces

What, a "creative" Pro/E tool? Yes, with the Interactive Surface Design Extension (ISDX), for the first time, Pro/ENGINEER can legitimately be associated with the words "style" and "styling." ISDX also provides a new design paradigm for Pro/E and a new user experience for Pro/E users as a tool dedicated to creating curves and surfaces. ASX with the Pro/E environment really combines two different types of modeling - freeform and parametric. Interestingly, the ISDX capabilities are invoked by creating a so-called super-feature from within Pro/ENGINEER called Style (Insert`Style). I found ISDX to be the best true sketching tool for Pro/ENGINEER 2001, because it had the most natural, intuitive feel of the sketching methods contained within Pro/E. Another thing that I liked about ASX is its four-view window that you can modify and customize as best suits your needs.



With Pro/ENGINEER's Interactive Surface Design Extension (ISDX), engineers and industrial-designers can use a number of surfacing techniques to create the style of a product by working directly on the model.

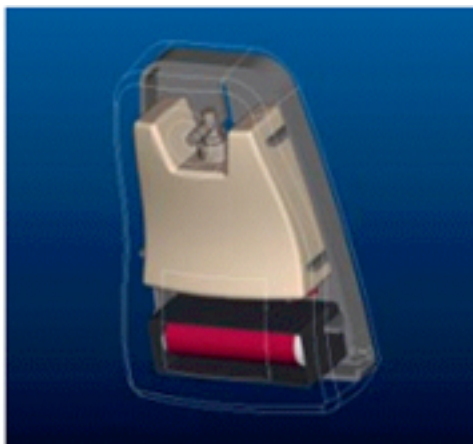
ISDX is actually an extension, or complement, to the Advanced Surface Extension (ASX) and with it you can create curves in either 2D (specifying the third dimension after creating the 2D shape) or directly in 3D. ISDX uses unique "soft points" for lighter and more flexible curves, making it relatively easy to establish relationships between curve points.

Curve and surface handles let you directly control levels of continuity between all geometry, while managing other relationships between curves and surfaces. The interactive part of ISDX lets you build up new super-features at the part level, as well as model curves and surfaces with as few or many constraints as you want or are required. Finally, I liked the ability to perform freeform blending between various types of Pro/ENGINEER features.

ISDX isn't going to challenge industrial design heavyweights, such as Alias/Wavefront's Studio Tools, but it's a good start for Pro/E's entree into the styling arena.

Design Optimization

For evaluating and exploring many design alternatives, and eventually arriving at an optimized design, Pro/E's Behavioral Modeling Extension (BMX) optimization feature is invaluable. It integrates a collective product definition through virtual prototypes from features of Pro/ENGINEER models. BMX automatically generates designs and runs analyses with its multi-disciplinary optimization tools.



In this example Behavioral Modeling is applied to the design of a cologne dispenser. ([View animation - 260 Kb](#))

The ability to optimize a design is largely dependent on results obtained from analysis features. Analysis parameters track areas of particular interest, such as surface area, pressure drop, or length. Once these are captured as analysis features, they can be used to optimize a design. So you're not forced to consider a zillion possible iterations, you specify a goal to maximize or minimize, design constraints to maintain, and design variables that can change. BMX then determines the design variables that have the most impact on the goal and constraints, varying them until the best design is found.

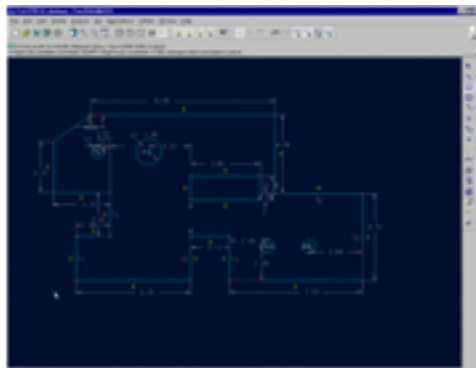
Once found, the optimization feature can then be created through the File menu. By including that feature, changes can be made and the goal and design constraints are checked with the optimization feature's place in the sequence. If the goal or design constraints are not satisfied, BMX determines the design variables that have the greatest impact, again, arriving at the best design.

Analysis results and output can be bi-directionally linked with Microsoft Excel for viewing and editing. Because this is a non-CAD interface, many more people can have access to the data than was previously possible with Pro/E's internal spreadsheet application, Pro/TABLE.

In the future, PTC plans on integrating BMX with its Windchill PDM offering for optimizing not just products, but processes (anything with variability), as well.

Detail Drawings

With all that was added or improved with regard to features and capabilities on the design side of Pro/E 2001, I was pleasantly surprised to see the amount of work that had been done on the drawing side. PTC is right in realizing that drawings are still a major medium and means for documenting and communicating designs.



As you sketch, Pro/ENGINEER automatically dimensions features as you go.

Within Pro/E, drawings are created with Pro/DETAIL. In Pro/ENGINEER 2001, the drafting tools have been redesigned and in many ways are similar to the UI found in Pro/E's 3D Sketcher, with automatic translation, rotation and scaling for drafted entities. 2D entities can be parametric and associative to model geometry, such as a bolt circle or intersection point. Drafted entities are parametrically attached to a model point in a manner analogous to how note leaders stay attached to geometry. Once created, 2D entities snap and stay connected to other drafted entities. Constraints that you can apply while sketching include horizontal, vertical, midpoint, parallel, perpendicular, and tangent.

System Requirements and Pricing

System Requirements:

- CPU: Intel Pentium Pro, II, III, or IV

- Windows 98/NT 4.0/2000
- 64 MB main memory
- 128 MB swap space
- 400 MB hard disk space
- Microsoft TCP/IP
- Ethernet network adapter
- OpenGL-supported graphics card

The following are purchase prices for the products required to run Pro/ENGINEER 2001 with ISDX:

- \$4,995 - Pro/ENGINEER 2001 Foundation
- \$4,995 -Advanced Surface Design Extension (ASX)
- \$3,995 - Interactive Surface Design Extension (ISDX)

You also can rent Pro/ENGINEER and several complementary MCAD products for 30 or 120 days. This program is handy for independent contractors, companies that experience peaks and valleys in their workflow, and users who need additional functionality found in options that they did not initially license.

The Bottom Line

There is no doubt that Pro/ENGINEER 2001 is a very capable and powerful design software package. As far as usability goes, it's somewhere in the middle - a little easier than CATIA, but more difficult than say, SolidWorks. Powerful, though it may be, Pro/ENGINEER takes a certain kind of person (technically adept and oriented) and a dedicated amount of effort (several months) to feel comfortable and productive with the product. And, although the product has made strides to improve the user interface and ease of use, it still has a ways to go in my book.

As important as it is to the company, PTC does not live by Pro/ENGINEER alone. The comprehensive Pro/E product line contains modules for just about every conceivable aspect of mechanical product development and product lifecycle management (PLM). These seemingly dissimilar products, ranging from design to analysis to machining and manufacturing work together with the Associative Topology Bus (ATB), PTC's solution for interactively linking the various products. The ATB somewhat ensures that these products look, feel, and behave, and allow provide user interaction in a similar manner. It's not perfect, but it seems to be working fairly well.

As far as Pro/E has come, and as improved Pro/ENGINEER 2001 is, I tend to question how truly committed PTC is in maintaining future development efforts at this level. To a certain extent, PTC is a company again in search of itself - what does it want to be when it grows up? Does it want to be an MCAD company through Pro/ENGINEER, a PDM company through Windchill, or a combination of the two? If it's the latter, can adequate resources be dedicated to adequately develop and market both sides of the coin or will one take precedence at the expense of the other? These are questions with no easy answers, but questions that will be answered loud and clear probably in the next few months. However, these questions do not prevent me from heartily recommending the Pro/ENGINEER family of products for heavy duty mechanical design and engineering.

Comments? Feedback? [Click here](#) to tell us what you think about this topic or if you have additional information you'd like to share on this subject!

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