NX5 Review

Jeffrey Rowe, "CAD/CAM/CAE solution from Siemens PLM Software is primed for production," Cadalyst, Nov 1, 2007

As you well know, there are many, many MCAD options available these days. Making a decision about the best MCAD application for your needs has never been an easy task, and in some ways, it's more difficult than ever. When it comes to MCAD packages, the terms midrange and high-end really don't apply anymore. In the past, though, NX was considered high-end, meaning it was expensive and difficult to master. With NX 5, however, this is no longer the case. In fact, I am quite impressed with how usable NX 5 is, considering its complexity and depth of features.

NX 5 is a major release for Siemens PLM Software (formerly UGS PLM Software), and it has more than 400 enhancements. Because there is so much ground to cover, *Cadalyst* decided to split the review into two parts. In Part 1, I will cover some of the basic aspects of NX 5 with regard to what makes this release so important, getting started with sketching and parts modeling and using concurrent and collaborative design practices. Next month, in Part 2, I'll cover assemblies, drawings, additional NX modules, and Teamcenter. Even with a two-part review article, NX 5 has so many new and improved features and capabilities that it will be tough to cover anything beyond the highlights.

Beyond discussing pure functionality, a secondary focus of this two-part review will be to dispel some of the misconceptions and misperceptions about the NX product line with regard to ease (or difficulty) of use and price. The core technology for NX 5 has changed dramatically for the better, thanks to improvements in the user interface (UI) and its functional capabilities, which together result in a better user experience and increased productivity. NX 5 is affordable as well.

I received NX 5 preloaded and configured for this review on an HP Compaq nw8000 notebook computer directly from Siemens PLM Software, so I could not provide an Installation grade on the NX 5 report card. The use of this system is significant, however, because even though the notebook was at least two years old, the software's performance on it was quite good.

Getting Started

If you have used NX in the past (my last experience was with NX 3), the first thing you'll notice upon launch is a much-improved UI. Admittedly, many of these UI changes may appeal more to new users, but experienced users will get the hang of most of them in relatively short order. The new UI likely will make users of all experience levels more efficient and productive.

As part of my software review, I looked at and considered product documentation as part of the user interface and experience. The online system for NX 5 is a weakness because it is somewhat incomplete and difficult to navigate.

Although NX 5 is far from the only product that features a customizable interface, it is configurable and role-based, meaning that users can create their own role and customize the NX interface to suit their work requirements and level of experience (**figure 1**). Users begin with a default role that is closest to their needs, such as Machinery or Consumer Products. Next, they decide which applications (such as Sheet Metal, Drafting, etc.) they need and customize the toolbars and dialog boxes for those applications. In the Role Properties dialog box, users can assign new roles; for example, the name of a particular task and the NX applications they want to associate with that task. Users can add an image to be displayed with a role name as well.
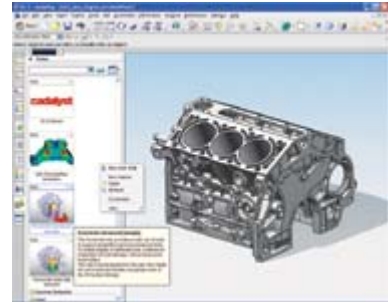

Figure 1. NX 5's customizable interface lets users create a job role that best suits their work requirements and level of experience.

Most importantly, NX 5 is able to accommodate users of all skill levels and myriad workflows, significant factors that few competitors can claim.

Direct Modeling

Before I discuss my experiences with NX 5, I want to take a brief look at the software's Direct Modeling (really, direct model editing) technology. It's pretty significant because it offers both history- and nonhistory-based approaches to design.

Most parametric modelers — Autodesk Inventor, Pro/ENGINEER, and SolidWorks — rely on feature history, but some do not. Notable nonhistory-based modeling packages include software from CoCreate, IronCAD, Kubotek, and SpaceClaim.

Although a history-based approach is what most parametric modelers use, it's not always the best way to go. History-based systems use a history tree to track and replay 2D profiles and modeling steps for generating and modifying 3D geometry. The main drawback of these systems is that each step in the history is dependent upon the profile and modeling step that came before it.

On the other hand, a nonhistory-based approach can be suited to manufacturers that make a lot of changes late in the design process. They are not bound by the constraints and complications that a history tree can impose. In addition, a nonhistory-based system can make data importation easier because users don't have to worry about the overhead and burden of dealing with a history tree from a different system. Neutral file formats, such as IGES and STEP, are literally native data to NX 5 and eliminate the problems associated with model data originating from other systems.

Depending on your requirements, with NX 5 you can have it either way — history or nonhistory based — and this is what Direct Modeling is all about. So, although the nonhistory-based

approach might not be for everybody, it definitely has its place. Of course, the opposite is also true. That's why, based on your perspective, a nonhistory-based approach can be both positive and negative. With NX 5, you have the freedom to choose the best approach for you.

Sketching and Part Modeling

As with virtually all MCAD packages, parts begin with sketches, and NX 5 really is no different. The sketcher is an NX tool you use to create 2D geometry within a part (**figure 2**). Each sketch is a named collection of 2D curves and points on a specified plane.
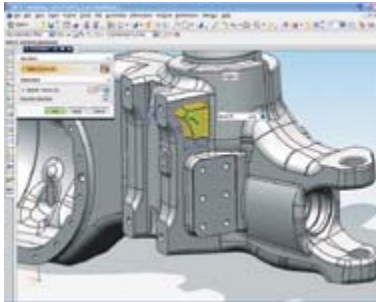

Figure 2. Features, such as the extrude shown in this graphic, are relatively easy to create with the NX sketcher tool.

NX sketcher tools let you capture design intent through geometric and dimensional relationships (collectively called *constraints*) to create parameter-driven designs that can be updated later. The sketcher evaluates constraints on the fly to ensure that they are complete and conflict free. A fully constrained sketch has as many constraints as there are degrees of freedom in the sketch so that there is no ambiguity in the final shape. The sketcher also lets users create as many, or as few, constraints as a design requires.

NX's sketcher can freehand a sketch and dimension an outline of curves. Users then can sweep a sketch using Extrude or Revolved Body tools to create a solid or sheet body. They can refine the sketch later to precisely represent an object by editing the dimensions and creating relationships between geometric entities. Editing a sketch dimension not only modifies the geometry of the sketch, it also modifies the body created from that sketch.

Users can position a feature — a hole, a groove, or any user-defined feature — relative to the geometry on a model by using positioning dimensions. The feature then is associated with that geometry and will maintain those associations whenever the model is edited. Users can edit the position of the feature by changing the values of the positioning dimensions as well. If the model is edited later, the associated drawing and dimensions are updated automatically.

Other NX tools operate directly on solid objects created within the Modeling environment without translating the solid body. For example, users can perform drawing creation, engineering analysis, and numerically controlled machining functions by accessing the appropriate application from within the Modeling environment. Finally, as a design evolves, models can be updated either automatically or manually.

Figure 3. NX Industrial Design

I'm an industrial designer, so I'm interested in creating curvy, freeform shapes. NX offers surface modeling and analysis tools specifically for computer-aided industrial design (CAID). Out of the box, NX 5 by itself is fairly well suited for industrial design, although the software also features tools specifically intended for industrial/conceptual designers who develop products with complex surfaces for the automotive and aerospace industries. The tools in NX Industrial Design include all of the basic conceptual stage options for creating and visualizing proposed designs, as well as additional tools for producing complex surfaces (**figure 3**).

(formerly known as Shape Studio) is well suited for creating some fairly advanced surfaces, as shown in this design (left) for a shower control panel. A final rendering (right) of the shower control panel includes a hose in a realistic environment.

Although it can produce some outstanding surfaces, getting to the final solution can be somewhat of a challenge. I found that some of the curve-creation tools required trial and error to arrive at the shape and form I wanted.

Overall, once I got beyond some nuances, creating sketches and parts was about on par with other MCAD packages — not really any easier, but definitely not as difficult as many have been led to believe.

Concurrent Design and Collaboration

Figure 4. Collaboration is an important aspect of NX, including the ability to measure features or entire parts for design review discussions.

The core of Teamcenter data management is included in all NX bundles, with capabilities for collaboration being especially important (**figure 4**). Core data management is included with NX, but more advanced Teamcenter capabilities are available as options at an additional expense. For example, Teamcenter Community is a Web-based collaboration utility. Built on Microsoft SharePoint Services, Teamcenter Community is a conferencing system integrated into Teamcenter in an environment that can be accessed directly from within NX 5. With it, users can package and send models to project participants and perform online design reviews.

So, that's it for Part 1. I could have covered many more topics, but *Cadalyst* has only so much space. I actually could have written an entire article about each of the topics covered in the subsections. NX 5 has many new and interesting features and capabilities waiting to be discovered and used. Next time, I'll cover assemblies, drawings, and additional modules (including Teamcenter). I'll also summarize my overall experience with and impression of NX 5.

With more than 400 enhancements, there is no doubt that NX 5 is a major release for Siemens PLM Software (formerly UGS). In fact, it was so major that *Cadalyst* chose to split the review into two parts.

In Part 1 last month, I covered some of the basic aspects of NX 5 with regard to what makes this release so significant, getting started with sketching and part modeling, as well as using

collaborative design practices. In Part 2, I'll cover assemblies, visualization, and drawings. Even though I was given the opportunity to review NX 5 in two parts, its many enhancements make it difficult to cover much beyond the highlights of this release.

So, let's get started with assemblies and move on other significant aspects of NX 5.

Handling Assemblies

For many MCAD packages, large-assembly (thousands of parts) performance is what separates the men from the boys. NX 5's significant architectural enhancements are the basis for major improvements in large-assembly modeling. To further improve assembly performance, the Siemens JT data format was integrated into NX 5. JT simplifies the pervasive multi-CAD environment that most manufacturers now deal with and offers lightweight assembly design functions for faceted assembly representations that improve performance when precise solid geometry isn't required, such as in design reviews **(figure 1)**. To a large extent, it is the JT format that improves NX 5's large-assembly capacity and performance while reducing memory usage and rendering time. Siemens PLM Software claims that test cases with NX 5 have demonstrated at least a 60% reduction in memory usage and at least a 65% increase in display-frame rates for typical large assemblies. All in all, these represent pretty significant performance gains.
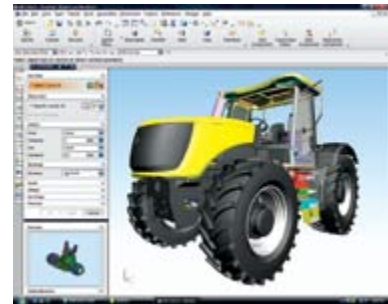

Figure 1. NX 5 introduces major architectural changes that dramatically improve design functions and performance for large assemblies. NX assembly modeling uses Siemens PLM Software s JT data format and Direct Modeling technology for lightweight, high-performance visualization and multi-CAD collaboration.

In NX 5, assembly part files point to geometry and features in the subordinate parts, rather than creating duplicate copies of those objects at each level in the assembly. This technique not only minimizes the size of assembly parts files but also provides high levels of associativity. This enables users to modify the geometry of one component so that all assemblies that use the same component automatically reflect the change. These relationships affect assemblies and other associated objects such as drawings, tool paths, and CAE meshes.

It's common practice in assemblies for a particular part to be used in many places. Each usage is referred to as an *instance*, and the file containing the actual geometry for the component is called the *component part*. For example, a car assembly may have two axle subassemblies, each of which has two wheel parts. In this example, we have four wheel instances and two axle subassembly instances but only two component parts (one wheel and one axle).

There are several approaches to assembly modeling, and with NX designers aren't limited to any one style. They can create individual part models and then later add them to assemblies (bottom-up), or they can create parts directly at the assembly level (top-down assembly creation). Additionally, users can start by using a top-down method and then switch back and forth between bottom-up and top-down modeling, depending on their specific needs. This versatile approach helps NX fit into a wide variety of workflows.

To expedite assembly design, multiple parts can be loaded simultaneously. Load options in the NX Assembly Navigator load implicitly or explicitly as a result of being used by some other loaded subassembly. The Assembly Navigator also lets users display information and manipulate the assembly for selecting, hiding, or suppressing assembly components. Additionally, loaded parts don't have to belong to the same assembly. The part currently displayed in the graphics window is called the *displayed part.* You can make edits in parallel to several parts by switching the displayed part back and forth between those parts.
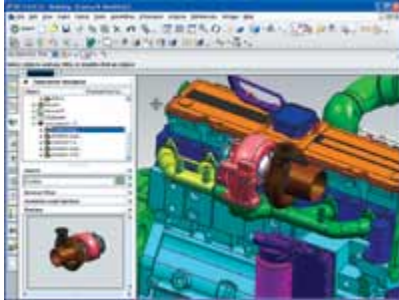
Digital Prototyping and Visualization

Figure 2. NX 5 embeds Teamcenter engineering process management to support faster searching of managed data and to promote reuse. The integration also delivers efficiency gains for typical design tasks.

The digital mockup (DMU) capabilities embedded in NX 5 are inherited from Teamcenter Visualization and are useful for design and review processes **(figure 2)**. The big advantage is that overhead for data management is eliminated because only one file is required for this process, not the typical two — one for design and one for mockup purposes. DMU is a technology also frequently called digital, or virtual, prototyping. This technology allows users to design and configure complex products and validate their designs, theoretically eliminating the need to build a physical model.

NX 5 has a capability called Active Mockup, an active technology that unites the ability to visualize an assembly mockup with the ability to measure, analyze, simulate, design, and redesign it **(figure 3)**. Active Mockup lets users create a multi-CAD digital mockup and then visualize and modify it. This capability gives immediate feedback in the context of the assembly. Lightweight 3D models with multiple levels of detail using lightweight data structures, such as JT, let you visualize, analyze, and interact directly with product design data.

Figure 3. NX 5 Active Mockup supports true design in context, even for massive, multi-CAD assembly models. It improves large assembly performance and unites the review and redesign processes. This image depicts a dynamic cross-section of a tractor assembly model.

Drawings and 3D Annotation

Although the dream of the paperless office has been proposed for many years, it still proves elusive, and the fact remains that drawings aren't going away anytime soon. Drawings are created in NX's drafting environment, and NX 5 provides easy-to-navigate user interface (UI) elements for producing engineering drawings directly from 3D models or assembly parts, as well as from 2D parts. The drawings are associative to models, meaning that a drawing reflects changes in a model as a design progresses. This ability ensures that drawings are current with regard to assembly models or individual modeling piece parts. Although these capabilities aren't new or unique, they do make NX 5 easier to use.

The drafting environment includes drag-and-drop annotation, the ability to edit annotation values, and the ability to delete drafting objects; helper lines (dotted lines) that visually assist you in aligning annotations and drawing views; and the ability to select member views directly on a drawing and to move and align a view. Drafting options are accessible from Edit and Insert Menus, toolbars, and Resource tabs. You can also drag and drop template files from Resource bar options to automatically create parts lists, tabular notes, and drawings.
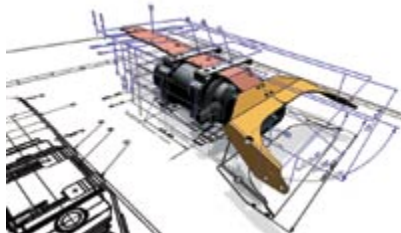

Figure 4. NX 5 lets users create 3D annotations such as PMI for capturing complete design intent, enabling information reuse, and further automating the drawing creation process.

NX 5 lets you create 3D annotations, also known as product and manufacturing information (PMI), for capturing design intent, encouraging information reuse, and automating drawing creation **(figure 4)**. New inheritance methods let you specify which PMI annotations will be inherited on 2D drawing views, eliminating the need to duplicate this step when creating drawings.

The Associated Objects option in the drafting environment associates geometry to a PMI annotation. Designers can use this option to capture design intent by specifying the portions of the model to which the PMI pertains. For example, if you have a note that specifies a particular finish for several faces but you only want the note to point to one of the faces, you can attach the note to that face and use the Associated Objects option to associate all the faces to the note. If you select the note, then all its associated faces highlight in the graphics window.

Some Final Thoughts

By itself, NX 5 is a very capable MCAD application. The universal and more specialized improvements make NX 5 a very significant release. Although the core NX 5 package is very comprehensive, myriad optional modules are available for NX 5 for just about any aspect of CAD, CAM, or CAE. Although virtually no one could use them all, I can't really think of any design and design-related task that couldn't be performed using them.

NX 5 is a heavy-duty MCAD package that can do some pretty heavy lifting. By that I mean that it can handle just about all manner of mechanical and mechatronic design. It's also a package that users are very unlikely to outgrow. In fact, with additional modules, it can grow with users if their needs and requirements become more demanding and sophisticated.

I have to admit that with the new UI and the sequence of the workflow, I felt like I was doing more than just modeling parts and assemblies — I was *designing* them. Overall, I was pleasantly surprised at how relatively painless the learning curve was, thanks largely to the more contemporary UI that is much more Windows compliant.

NX 5 is a broad application that spans the large space from design to production. The company put a lot of time and effort in filling this space, and it shows.

This is the first major release of NX since UGS was acquired by Siemens. Admittedly, virtually all of the functionality for NX 5 was decided and developed before the acquisition was completed. How much direction and influence Siemens will have in the future is unknown, but NX 5 is certainly an MCAD candidate worthy of serious consideration.