

ENGINEERING AUTOMATION

JANUARY 2001

REPORT

www.eareport.com

The Newsletter for Engineering Management

Reprinted from
**Engineering
Automation
Report,**
January, 2001,
Volume Ten,
Issue One.
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SolidWorks 2001 Unveils New Heads-Up User Interface

By Joe Greco and the Cyon Research staff

The mid-range MCAD market was essentially born when SolidWorks and rival Solid Edge were announced in 1995. Since then, both products have been updated at an impressive rate. With Solid Edge 9 just introduced (see November 2000 *EARreport*) it was easy to guess that the new version of SolidWorks would be along soon. Sure enough, SolidWorks 2001—the ninth release in five years—is almost here. In early December, we saw a beta version.

User Interface

Users assign varying importance to various aspect of a program, yet most place ease-of-use high on their “must have” list. The developers of most MCAD programs add lots of new functionality to each release, but often relegate user interface changes to minor improvements such as “we moved the Import command from the Insert menu to the File menu.”

While SolidWorks is widely considered quite easy to use, SolidWorks Corporation decided to totally revamp their program’s user interface, a change they refer to as “Heads-up User

Interaction.” Following a trend begun in SolidWorks 99, they’ve eliminated almost all dialog boxes. For example, when extruding entities, you can now assign values directly on screen, adjacent to the entities being extruded. The PropertyManager has now been implemented throughout the program, and context-sensitive callouts have been added. The latter appear within the model or drawing display to assist the designer. Context-sensitive callouts include symbolic sketch relation callouts as well as descriptive and editable feature callouts that display the appropriate design information, allowing the user to better understand and more easily modify designs. These callouts display information such as relationships on sketch entities, labels for feature inputs, or a method for changing feature parameters.

Other changes to the interface also increase productivity. For example, when placing a dimension in the sketcher, you no longer need to leave the dimension mode to edit or relocate the dimension. More Drag Handles have also been added to SolidWorks features to allow users to make edits without relying solely on the dialog entry of parameters. When sketching, Graphical Constraint Callouts clearly show

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which relationships exist on each sketch entity. These callouts have graphical icons for each relationship and are also color-coded based on the solve state for each.

The new Confirmation Corner provides a simple graphical indicator that allows the user to continue or exit a given task. It's a nice change from having to hunt for icons.

3D Modeling

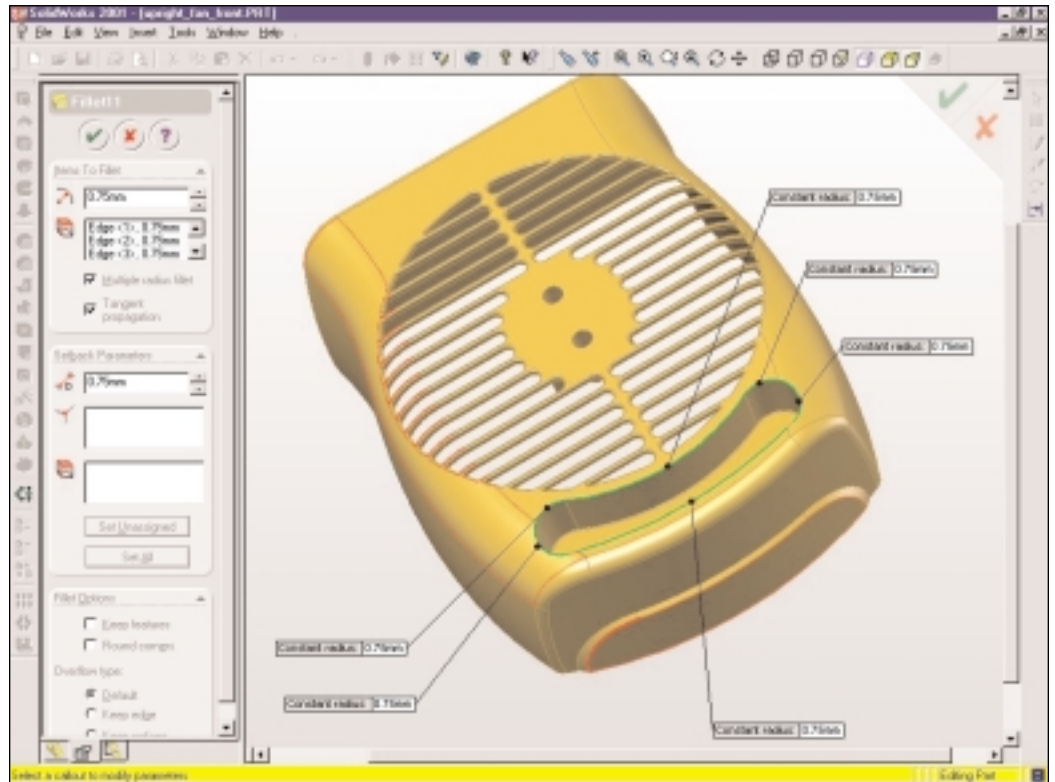
The program's 3D modeling has been improved on several levels. Tube and pipe shapes are now easier

to create, because SolidWorks 2001 allows concentric loops within sketches. (Multiple independent loops within one sketch are still not allowed.) Similarly, thin lofts and sweeps now only require a single step.

Though SolidWorks isn't a player in the realm of class-A surfacing, it is getting better. For example, SolidWorks 2001 now provides dynamic (interactive) tangency control over blended surfaces. Holes can also be created perpendicular to any surface—even curved surfaces. SolidWorks 2001 will also support N-sided patch surfaces, a capability derived from its big brother CATIA. (This feature will not make the first build on SolidWorks 2001, but will be delivered with a later build.)

Sheet Metal

Although SolidWorks has long had powerful sheet metal capabilities, it has often been overshadowed in this area by Solid Edge. The reason is a difference in design philosophy. Solid Edge implemented sheet metal in a traditional fashion—where parts are built from the bottom up, with bends, tabs, flanges and such added as the part was



SolidWorks 2001's new Heads-Up User Interactions includes an updated PropertyManager, as well as Context-Sensitive Callouts that show information such as relationships of sketch entities and labels for feature parameters.

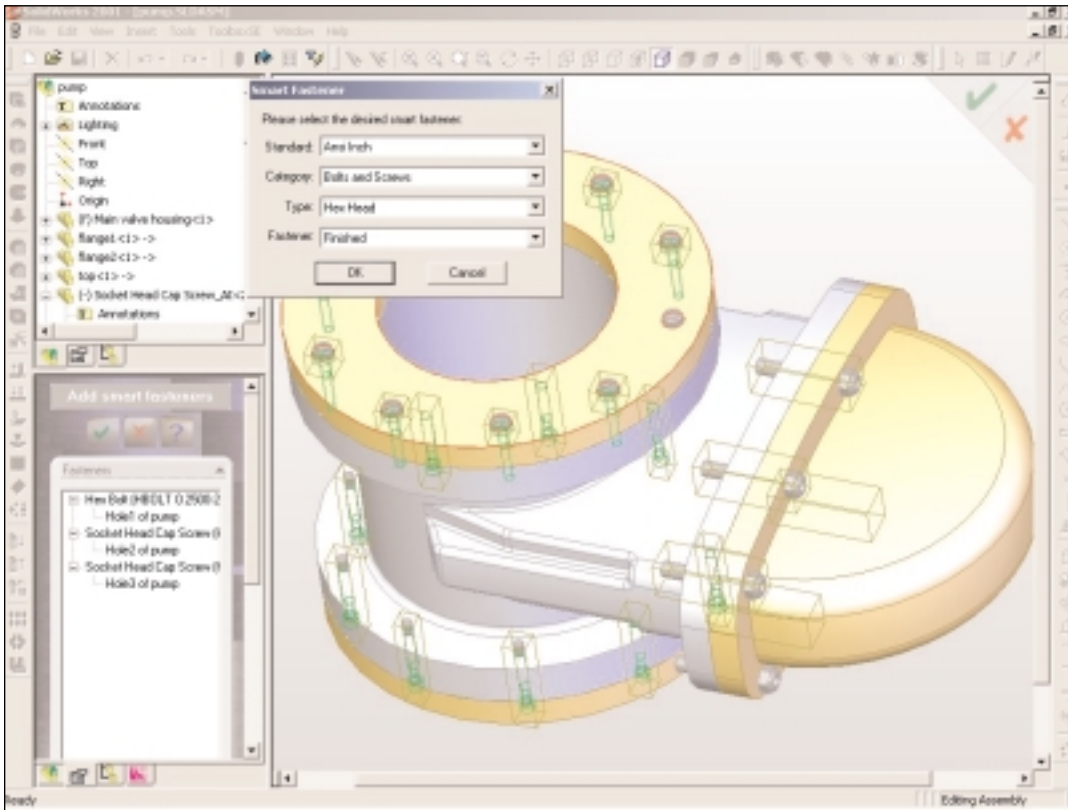
built. SolidWorks took a different path, with more of a solid-modeling type approach. Both approaches have their proponents. People who design a lot of sheet metal have tended to like the Solid Edge approach. People who design sheet metal mostly within the context of a larger project have tended to like the SolidWorks approach. With the 2001 release, SolidWorks includes support for the more traditional bottom-up approach. It's a significant improvement that many users should appreciate.

Overall, the sheet metal functionality in SolidWorks 2001 has seen quite a bit of attention, including the addition of long-awaited options such as oblong bend reliefs and special corner conditions. This version includes the capability to selectively unbend a feature, perform operations such as notching or punching across this flat bend, and then bend the feature back to its original shape. Users no longer need to flatten the entire part to perform such operations, saving time and increasing performance. SolidWorks 2001 also adds Excel-based bend tables.

Assembly Modeling

SolidWorks 2001 includes the capability to

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Smart Fasteners can select correctly sized bolts, holes, washers, and nuts, based on international or company standards, and assemble them in sequence automatically, saving time both finding and placing parts.

associative and configurable, so if the bolt holes change size, SolidWorks will notify the user, and ask if the fastener should be changed. The fastener library is based on international standards, but can be customized for individual company standards. Much of the Smart Fastener technology comes from CIMLOGIC, which SolidWorks recently acquired. Designers working with cam shapes will appreciate a new assembly mate type in SolidWorks

intelligently mirror components of an assembly. The program attempts to resolve which parts need to be mirrored and which only need to occur as an instance, thus reducing the number of unique parts. Consider, as an example, the suspension on a car. Certain parts, such as shock absorbers, are instances, while other parts, such as control arms, are mirrored. Users can also interactively edit what the program has done. For those users who deal with mirror image assemblies, this capability is substantial.

Another powerful new assembly capability is SolidWorks' new Smart Part Technology. Smart Parts are automatic (in terms of insertion, orientation, and size), associative (with notification based on change), configurable (iterations or families can be set beforehand based on standards), and extensible (they can be applied to many applications through the SolidWorks API).

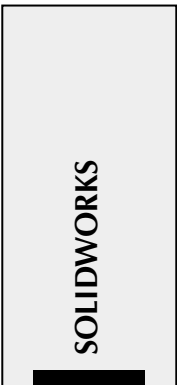
The first implementation of Smart Part Technology is Smart Fasteners. Using this technology, SolidWorks 2001 can automatically (with user control) apply stacks of bolts, washers and nuts to an entire assembly. Smart Fasteners are fully

2001 that supports ruled spline surfaces. This makes it possible to mate to cam-type profiles, and create motion based on these conditions. Mating conditions can now also be changed, which is particularly useful during the design stage.

Flexible sub-assemblies, another new enhancement, provides increased functionality when designing assemblies, particularly when conducting motion studies. For example, in an assembly, a sub-assembly of a piston would have previously been treated as a rigid part, thus preventing movement. Now, with the ability to make sub-assemblies flexible, the mates at the sub-assembly level can be solved to allow movement between other components that are mated to the piston.

2D Drafting

Drafting has been a major focus for the last few SolidWorks releases, and continues to be in this release. Users can now create multiple projected views with the same command, create section views



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from existing section views, and display connection lines from detail views to the parent view. While SolidWorks has long had the ability to show 3D movement of parts, that movement can now be conveyed in 2D as well, thanks to the program's new Alternate Position View. For example, users can now show the two extreme positions of a moving lever arm, one as a dashed line and the other using a solid line, in a single 2D view.

SolidWorks 2001 also incorporates Visio's extensive library of electrical components, thus allowing users to add these 2D-only elements a lot more easily. Other 2D improvements include a new DXF/DWG wizard, with a WYSIWYG preview and sheet size manipulation, so dimensions look exactly as they did in AutoCAD.

Other Improvements

For collaboration, SolidWorks 2001 includes add-ons that had formerly been available separately, including 3D Meeting, eDrawings, and the ability to publish live web

pages with 3D Instant Website. SolidWorks 2001 also supports Web Folders, which let multiple users store, retrieve, and reference parts, assemblies, or drawings, as well as other file formats on the Internet as easily as if they were located on their local PC.

SolidWorks 2001 has improved rendering capabilities, and now includes the same version of VBA (Visual Basic for Applications) that is found in Microsoft Office, offering a new level of customizability. The new release is rounded out with a new HTML-based help system, which includes online tutorials, a design portfolio with how-to instructions, and a complete glossary.

The 2001 release is another significant step forward for an already impressive CAD program. SolidWorks Corp. continues to show its commitment to strengthening both the capability and the ease-of use of SolidWorks, even in areas that many would argue were already very good. ■

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**ENGINEERING
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REPORT**

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