

# TechniCom

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## Solid Edge Version 20:

Solid Edge Version 20 focuses on modeling with massive assemblies, managed collaboration, and engineering productivity.

This report reviews and comments on the major changes implemented in Solid Edge Version 20.

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# Executive Summary

## Introduction

Ray Kurland, the author of this paper, spent several days at Solid Edge headquarters in late April 2007, reviewing Version 20 of Solid Edge. Ray met with Solid Edge management and technical experts, the goal being to develop this paper summarizing the most important new functions of the software and why they are important for prospective users.

Ray met with Bruce Boes, Vice President UGS Velocity Series Marketing, Kris Kasprzak, Director of Solid Edge Marketing, and Russell Brook, Solid Edge Product Marketing Manager, for an overview of Version 20. Kasprzak has primary responsibility for developing the marketing needs for each new version of Solid Edge while ensuring that the software meets customer needs and improves Solid Edge's competitive position. Ray also met with Doug Stainbrook, Field Support Engineer for Solid Edge, who elegantly demonstrated and explained in detail the major new functionality in Solid Edge Version 20.

## Solid Edge continues to move along at a rapid pace

While the following sections discuss the detailed technical highlights of Version 20, equally as important is marketplace acceptance of Solid Edge and UGS' other Velocity applications, all aimed at the mid market. On all fronts the software meets with success. UGS leads the way in understanding that mid-market companies have many of the same needs as their larger brethren. Mid-market companies also need to participate in global markets and respond to rapidly changing economic and business requirements, similar to large enterprises. Their tool needs are similar to those needed by large enterprises, but need to be easier to use and operate with little or no IT staff. To address these unique mid-market needs UGS provides software specially tailored to integrate tightly with Solid Edge, and if needed, provide associativity with UGS' NX. Femap (for analysis), Teamcenter Express (for PLM), and NX CAM Express (for machining) represent the balance of the Velocity applications and also interoperate with other CAD software, but not quite as tightly as with Solid Edge. Customers are "voting with their dollars" because the Velocity revenue is growing at about two times the overall MCAD market. Femap is enormously popular; Teamcenter Express early installs predict large growth; and, NX CAM Express brings a highly capable application into the mid-market with an attractive price point along with extensive capability.

The UGS Velocity Series software for the mid-market consists of Solid Edge, NX CAM Express, Femap, and Teamcenter Express; all modular, yet integrated, fully

native Microsoft, delivered with predefined "best in class" process templates, and they're all associative with each other. The velocity organization has restructured these products so they are a better fit for the mid-market. Along with deep integration within PDM, this restructuring includes new user interfaces (UI), different capabilities for mid market companies, an easier installation process, and Windows native operation. Solid Edge V20 also supports the new Microsoft Vista operating system and adds more support for 64-bit processors. A ten-day implementation plan package provides for the installation of Teamcenter Express, its initial operation, and users and administrators trained.

## Highlights of Version 20

Solid Edge Version 20 focuses on massive assembly modeling, managed collaboration, and engineering productivity.

- Solid Edge Version 20 delivers eleven major modeling advances, all focused around making working with massive assemblies (more than 100,000 parts) easier and more productive.
- A new dedicated Structure Editor provides an interface for re-using existing assemblies to create new projects in a managed Teamcenter environment. Included with the Solid Edge Embedded Client, Structure Editor allows cloning existing assemblies, revising, saving-as or re-using portions of the assembly tree, allowing ready repurposing of data within a managed environment.
- Solid Edge V20 adds function that makes migrating from AutoCAD easier, including XREFS and color 7, while its stand alone drafting environment should prove more friendly to maintaining drawings.
- Additional translator functionality improves Solid Edge's interface to external systems, including a new CATIA V5 bidirectional translator, updates supporting Pro/E Wildfire 3, and support for STL files as input for viewing.
- More than 170 other enhancements are also included in this release.

The following section "Description of the new functionality of Solid Edge Version 20" describes in more detail, not only the major functions of Solid Edge Version 20 that this paper reviews, but also our reaction to these items.

## Conclusions

Overall this is a very impressive release. Many of the major modeling functions added to V20 focus on improving the ability to work with massive assemblies. This is a tough problem to solve, often taxing the

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resources of even the fastest desktop processors. Solid Edge V20, in our opinion, has made substantial progress in this area and now has leading edge capabilities. Some of the new functions that make this possible are the following:

- Zones modeling - provides performance improvements by allowing users to define and work within small defined areas of large assemblies.
- Assembly auto(mated) constraints allow rapid constraining of imported assemblies for motion studies, performing in minutes, what would normally take hours to do manually.
- Component grouping allows better design control, yet does not effect the how the assembly is physically constructed. Simplifying the pathfinder (history tree) will yield more productive modeling from a simpler presentation.
- Drawing review mode - uses Solid Edge's independent (of the 3D master model) drawing view to quickly view drawings of large assemblies. This offers a complete visualization of massive assemblies in seconds, not hours, and is only possible because of Solid Edge's unique method of storing independent drawing data.
- If users need to make the full assembly active, assemblies can automatically unload "unused" components to simplify them and improve their performance "on the fly." Solid Edge automatically unloads (inactivates) parts not recently used, thus improving overall system performance by better utilizing memory.
- Dynamic preview of direct edits for component modeling shows what a change would be before it is made and committed, saving valuable design time.
- Tabulated drawings introduce a unique ability to gather and connect master model data to generate automatic tabulated drawing views for families of parts.
- Goal seek, feature grouping, feature library helpers, and the variable rule editor all act to improve engineering productivity. Goal seek presents a simple solution to a vexing problem, eliminating trial and error solutions in many cases. Feature library helpers makes feature libraries many times more useful by adding clear explanations of how features are to be placed - simple but powerful.

Beside the above modeling functionality, Solid Edge, as a the principal component of the UGS velocity program, is buttressed by the other Velocity components, offering users advanced mid market functions in analysis, data management, and machining, all associative and supporting a master model concept.

New translators and an even better AutoCAD migration allow more companies an easy way to migrate to Solid Edge. These all enhance the comprehensive coverage of Solid Edge Version 20. Users considering upgrading or migrating to an advanced 3D system should definitely put Solid Edge V20 high on their list.

Users will be able to tell by reviewing the descriptions below, that taken as a whole, UGS and Solid Edge have made concomitant massive advances to the "art" of dealing with such large amounts of data entailing equally massive amounts of compute resources.

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# Description of the new functionality of Solid Edge Version 20

Our impressions of Solid Edge Version 20 derive from discussions and demonstrations with Solid Edge Field Support Engineers and executives. We were able to interact closely during demonstrations and discussions of the key functions discussed below.

Since this is a very visual business, a few screen images are included below to expand on the text. This should enable the reader of this paper to obtain an introductory “feel” for the functions. More images are on our website at <http://www.technicom.com/SEV20>. In case these descriptions whet your appetite for seeing more details of the software, contact your appropriate Solid Edge representative or reseller. Solid Edge contact information is at the end of this paper. You can also contact the author with feedback at [rayk@technicom.com](mailto:rayk@technicom.com).

## Assembly auto(mated) constraints allow rapid constraining of imported assemblies for motion studies

The assembly auto constraining capability (also called the relationship assistant) would be used to add constraints to imported geometry for the purpose of possible motion studies. Solid Edge built geometry would already contain assembly constraints. Usually this would be a tedious process. However, using the relationship on the imported geometry model shown below (01.png) within approximately 5 minutes, we were able to add in the 48+ relationships rather than at least an hour or more building relationships by manually mating and align parts in the assembly. By appropriately grouping and asking the system to automatically compute the possible mating relationships, the system proposed, using a series of dialogue boxes, the relationships that it found. The user could choose to accept or not accept specific relationships. In our case, we accepted most of them. In the case of the piston rods we chose not to accept the proposed mates because we wanted to allow them to move. The sequence we followed was to remove the grounded parts (ones that would not move), then ask the system to suggest axial alignments, then to suggest mate and planar alignments. **The system has excellent capabilities for automating this function.** We have not found this capability in any other system, and it has proved to be very effective in migrating legacy data or working with imported assemblies.

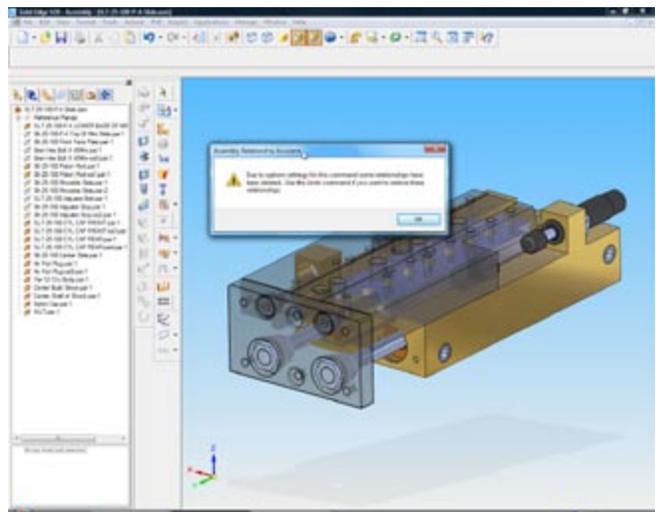


Image 1 (02.png) - A warning message to the user before the system removes the default relationships (grounds in this case).

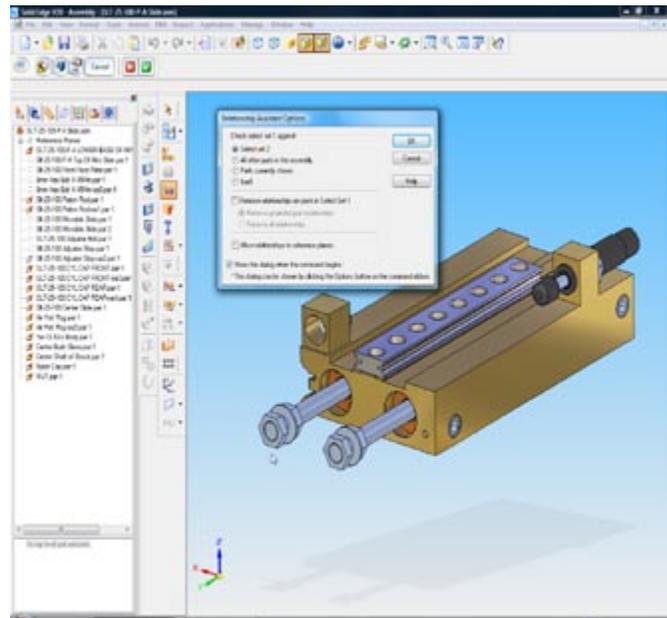


Image 2 (03.png) - We have now hidden the moveable half of the pneumatic cylinder (shown in light blue in the previous image) to begin making changes on the bottom half. To start we begin by specifying two sets of parts to constrain against each other. The first select set will be the block (in gold). The second set will be the rest of the parts (slider, pistons, bushings, and bolts).

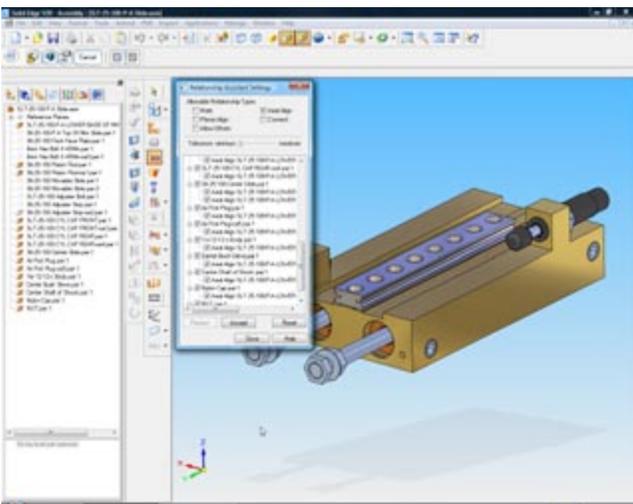


Image 3 (04.png) - Using the relationship assistant in stages makes it easier to see the results. First, we set the option to only detect Axial alignments between the two select sets. The window in the image above depicts the results of the possible axial constraints. Any relationship that has a check will be applied to the assembly after selecting accept. Note that only axial relationships have been proposed.

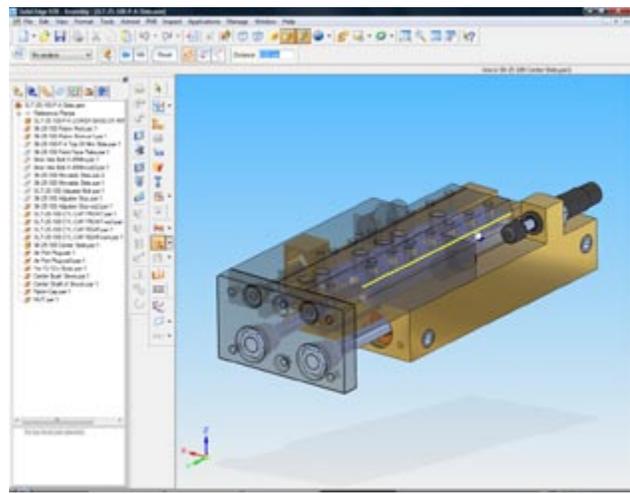


Image 5 (06b.png) - After Auto-constraining all of the parts, the movement of the pneumatic slide can be simulated using the "move" command and the entire assembly moves as expected.

**Dynamic preview of direct edits offers a complete visualization of the proposed change**

The next capability is dynamic preview in direct editing. Direct editing has been in Solid Edge since Version 18. The new capability in Version 20 is the dynamic preview, allowing a complete visualization of the change during the process, rather than inputting a numeric change value and then observing the result. This is very slick.

Using the ball valve shown in Image 6, selecting a small portion of the part, we initiated a significant angular change on one of the faces. All related fillets and other portions of the geometry stayed connected, as expected. In this example, when we reached a geometric condition that wasn't possible, the system presented us with a warning. This is much more effective in making reliable edits.

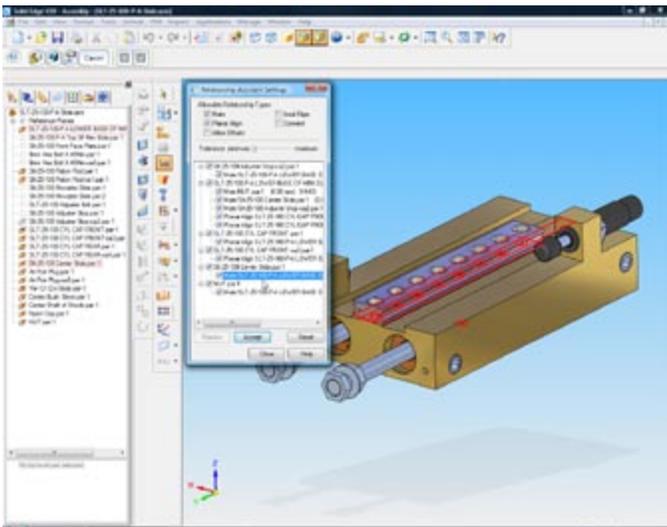


Image 4 (05.png) - Selecting a relationship from the Relationship Assistant automatically highlights the faces in the graphics window to show the user where the relationship will be placed. In this case the system has been asked to propose mates and planar alignments.

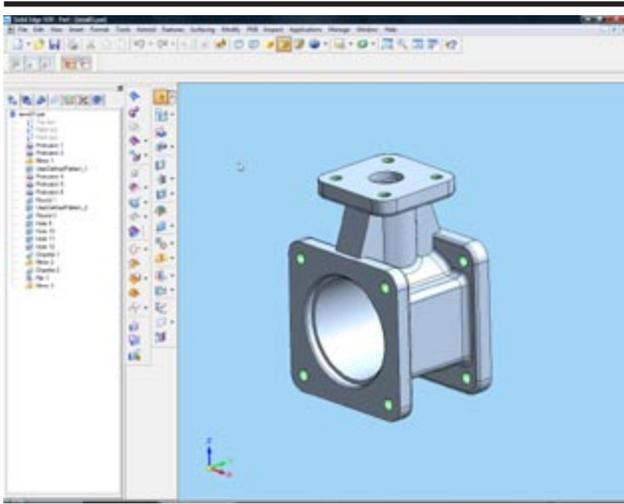


Image 6 (07.png) - This is the "before" image showing the Valve body, before modifying using Direct Edit.

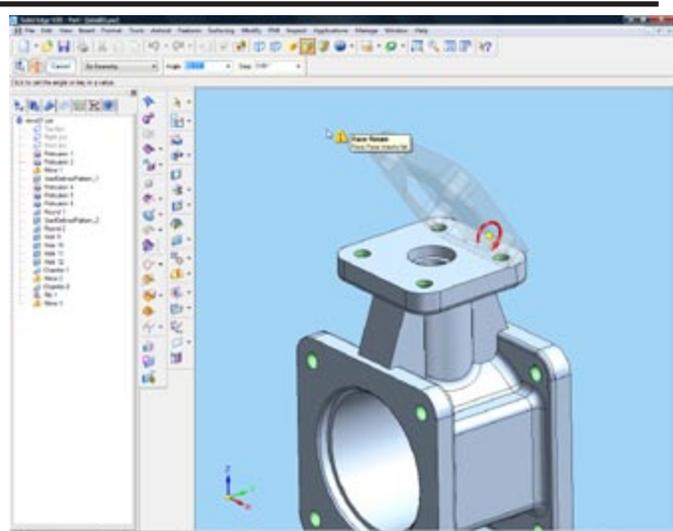


Image 8 (09.png) - Dynamic Preview in Direct Edit even gives an indication when the users edit will cause features to fail. Note that the dynamic preview disappears and a warning exclamation mark is displayed. Excellent feedback to the user!

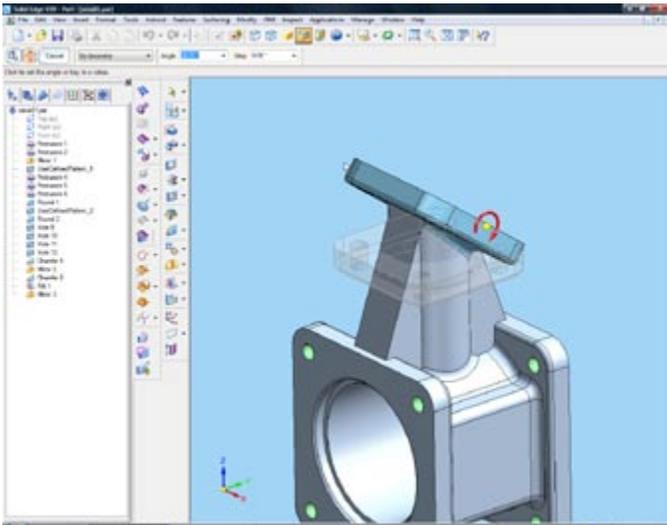


Image 7 (08.png) - This image shows the Dynamic Preview for (new in V20) Direct Edit. The user can see exactly what the result of the direct Edit will look like before committing to the change. In this example we used "Rotate Face" and selected the features shown in light blue to rotate about an edge. **Note that all the adjacent faces and rounds update to the change, even in preview mode.**

## Zones (Modeling within) - provides performance improvements by allowing users to define and work within small defined areas of large assemblies

In this portion of the demonstration, we focused on some of the massive assembly design tools by looking at some common workflows found in plant equipment design and layout. These include zone modeling, tools to copy, move and rotate an array of existing subassemblies within an assembly, and component grouping.

Zone modeling enhances interactive performance when working with extremely large assemblies. When opening an assembly, it can be opened in full mode, lightweight mode, or simplified mode. The zone modeling capability goes a step further to improve performance by allowing the user to define geographic areas (zones) that contain components or subassemblies. When opening an assembly, the user has the capability to specify a number of options for the assembly.

Users can create a zone by selecting parts to define the zone. In this case, the outer limits of the parts define the zone box, which is a 3D box showing the geographic space of the zone. When viewing a zone, parts can be viewed depending on whether they're inside the zone or partially inside the zone giving the user more display control.

What's important here is that by being able to define zones, each user only needs to open the zone they are responsible for, hence speeding overall system

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performance when working with extremely large assemblies. By defining a zone, every time the zone is opened, Solid Edge knows which parts are in the zone and therefore does not have to open additional component not in the zone. This powerful capability will save design time. Additionally, zones can easily be switched when users decide to work on different portions of a massive assembly. As a result, working with massive assemblies becomes much more manageable and easy to do.

Image 9 shows the new zone capability in the File Open dialog for opening a large assembly by pre-defined Zones. In this case we are responsible for the design of the Conveyor; only the components within the Conveyor zone were brought into memory; not the entire assembly.

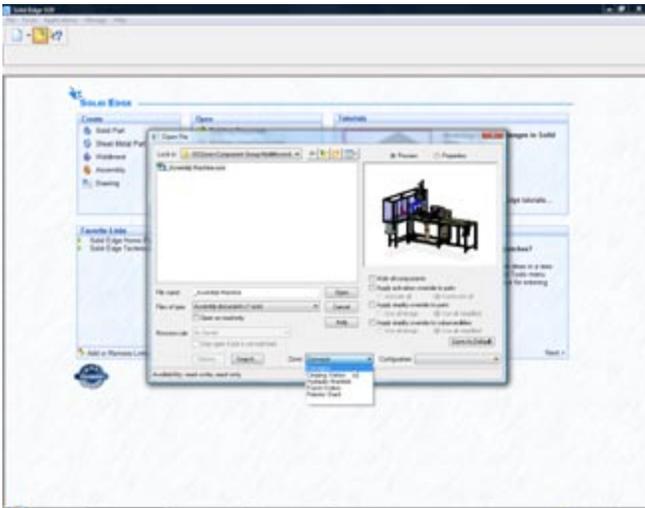


Image 9 (10.png) - New open dialog allows a Zone option.

Other types of zone definitions are possible. Another is a zone definition that includes everything either fully or partly within the zone box. Users can change the size of the zone box by selecting a face of the box and moving it to a new keypoint of a model. A zone can also be defined by selecting components that are to be at the extreme edges of the zone box in all 6 directions (to define height, width, and length).

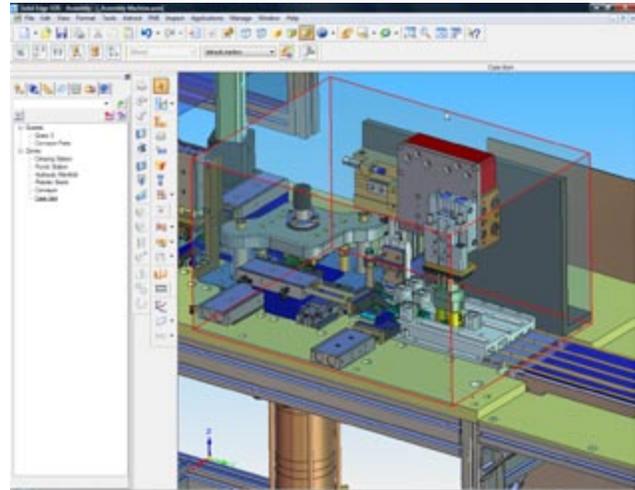
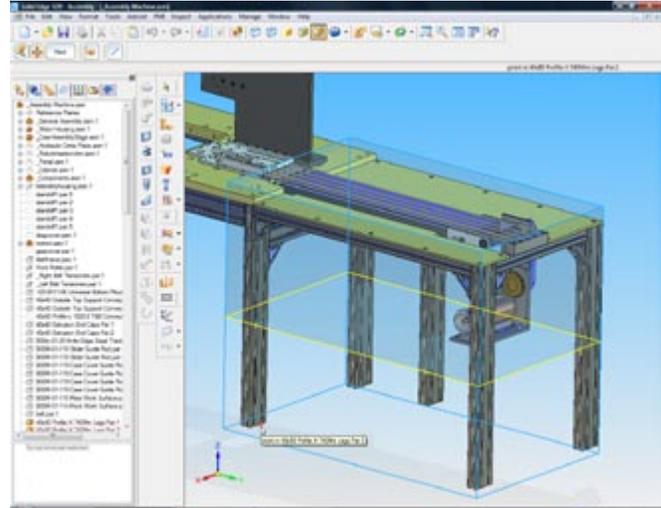


Image 11 (13.png) and Image 12 (14c.png) - Show Zone Boxes defined using different techniques.

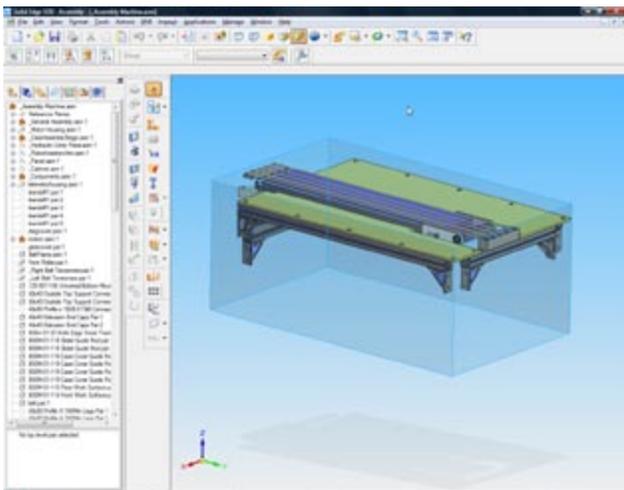


Image 10 (11.png) - Shows the result after File Open. In this image we can see the Zone was defined by everything INSIDE the Zone Box.

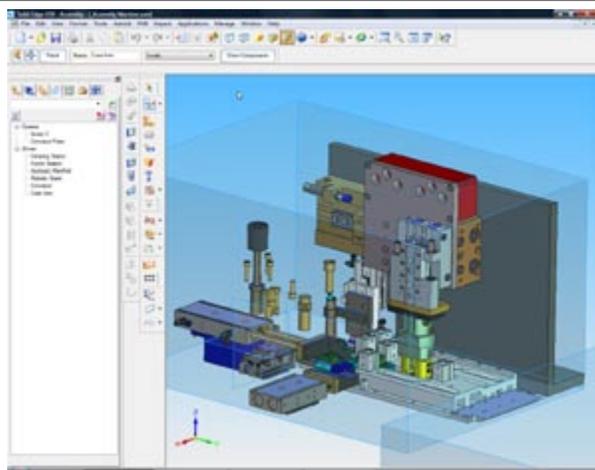


Image 13 (15.png) - Result of "Show Components" command with the option set to INSIDE.

### Component grouping allows better design control, yet does not effect how the assembly is physically constructed

The ability to group components further improves massive assembly design. This technique allows the user to group components under a single convenient node while leaving the bill of material and assembly structure intact. Component grouping has several advantages such as compacting large trees, simplifying inputs to pattern operations composed of multiple parts, and organizing common parts such as bolts into a single node for quick show and hide.

### Component positioning eases machine and factory layouts

To improve the modeling of machine and factory layouts, component groups in Solid Edge can be selected for operations such as move, copy, or rotate while maintaining internal mating relationships. Internal relationships can optionally be deleted. **Building digital versions of very large machines, frequently requires duplicating many subassemblies to form the final configurations and assemblies. Using this well thought out method for replicating and building groups of assemblies should save an enormous amount of design time.**

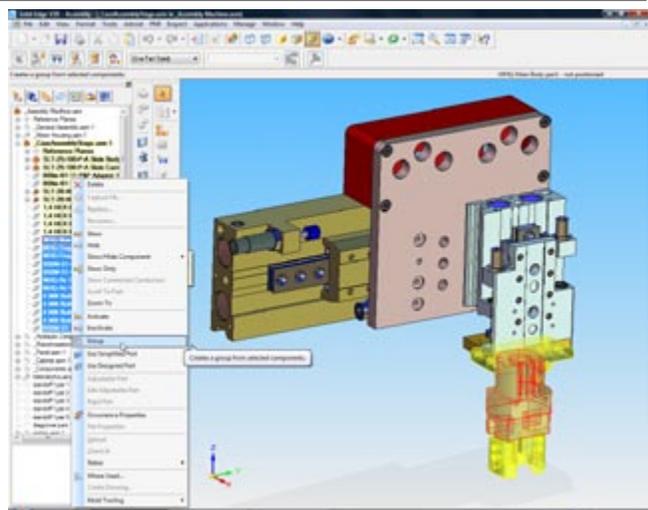


Image 14 (16.png) - This image shows creating an Assembly Group of the Gripper components. Assembly grouping enables us to collect a group of parts under a single node for selection for things like Activate, Inactivate, Show, Hide, and the new Move Multiple Parts command. These parts might not make sense to group into a subassembly (like all similar bolts), yet we can collect them in a group for selection. **Groups are invisible to the Draft environment, so the BOM structure is unaffected.** It can also simplify the feature tree (pathfinder).

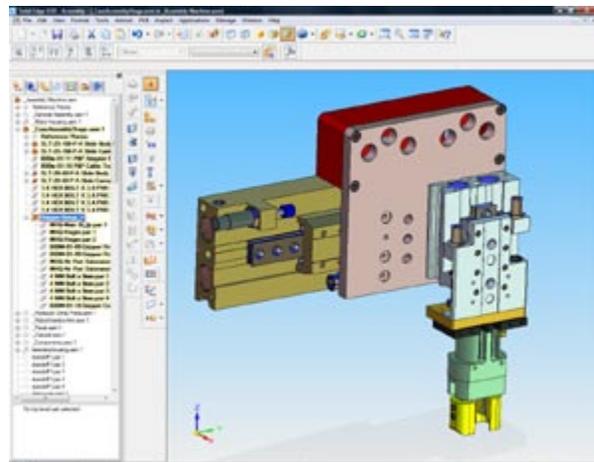


Image 15 (17.png) - This shows the result in the Assembly Pathfinder (in the left window) after Grouping the components.

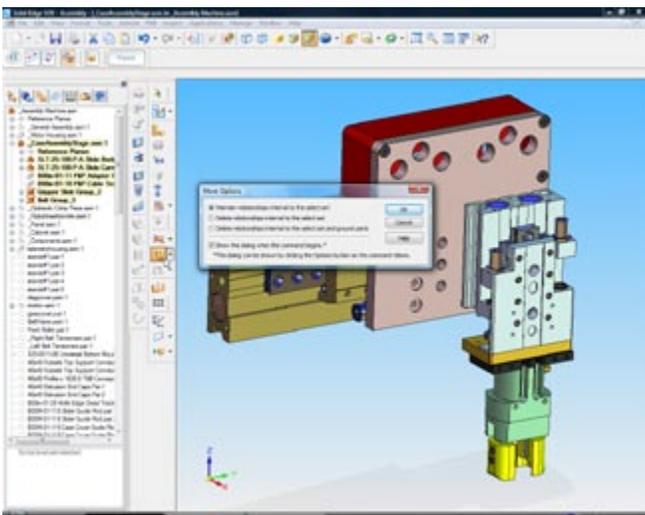


Image 16 (19.png) - For the new Move Multiple Parts for Machine and Factory Layout, the user is presented with the option to maintain relationships internal to the select set. In this case we will use this command to COPY a Group of components to a new location. This command can be used for Copy, Move, Rotate and Array components. Note how the group simplifies the selection for the move.

The position of the select set can be defined by dragging, Coordinates, Vectors, or by point-to-point selection. In this case we copy the Group from one center point to another. The slide components in Assembly Group\_4 can be dragged to simulate motion because we chose the option to maintain relationships internal to the select set (as shown in Image 16).

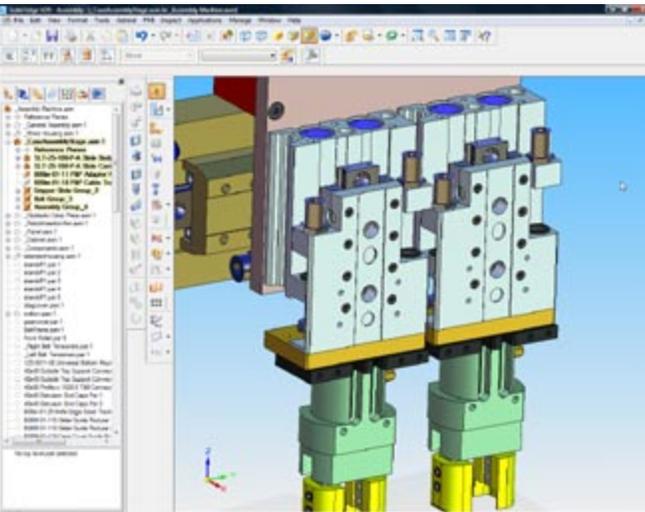


Image 17 (21.png) - This image shows the resulting Group that is created.

**Drawing review mode - uses Solid Edge's independent (of the 3D master model) drawing view to quickly view drawings of large assemblies**

Solid Edge 2D Drafting is architected differently than most other systems and for some very good reasons. The system is designed to store an independent copy of the 2D drawing that was derived from a 3D model. This important fact allows drawings to be retained and opened even if the 3D model is not found or worse, inadvertently deleted.

An additional advantage of this architecture and a new feature in Solid Edge v20 is the ability to open 2D drawings instantly. Because the 2D geometry is saved with the drawing, Solid Edge designers were able to optimize the open times significantly by opening drawings in inactive mode. In our test case, we measured drawing opening times at approximately 6 seconds. This speed is expected to be about the same, even for drawings of massive assemblies. When comparing drawing open times of active versus in active, it might take 20 minutes to open a drawing of a multi-thousand part assembly in active mode as compared to 6 seconds for opening up a drawing in inactive mode.

Are drawings opened in inactive mode useful for production work? Very much so. You can add dimensions and annotations, add and remove balloons associated with the bill of material, view and print. In essence, users can print, review, annotate, and dimension.

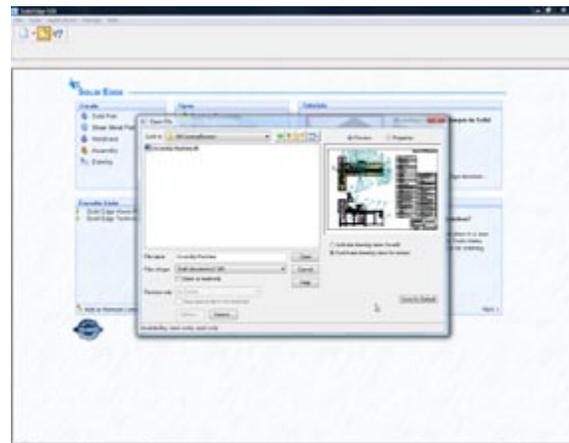


Image 18 (22.png) - This image shows the new file open option for a Draft file to open the drawing Inactive. This means the actual assembly model does not need to be loaded into memory to open the Draft file for review, dimensioning, annotating (including ballooning the assembly components) and/or printing. The result is very fast opening of a drawing file regardless of the size of the assembly detailed on the drawing. Drawings opened with the

file inactive show an "INACTIVE" watermark on the drawing. This watermark does NOT print.

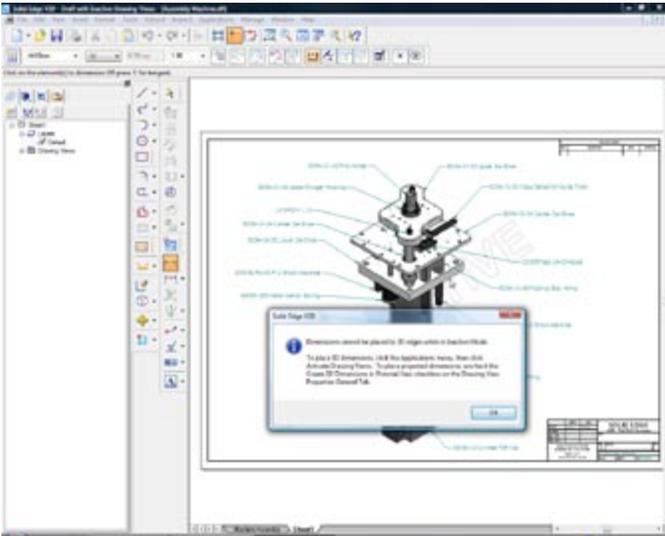


Image 19 (24.png) - If the user tries to do something that requires the user to activate the drawing, a dialog appears giving instructions for activating the drawing views. In this case we were trying to place a dimension on an isometric drawing view. Drawing views can be easily activated or inactivated by an option in the Application pull-down menu.

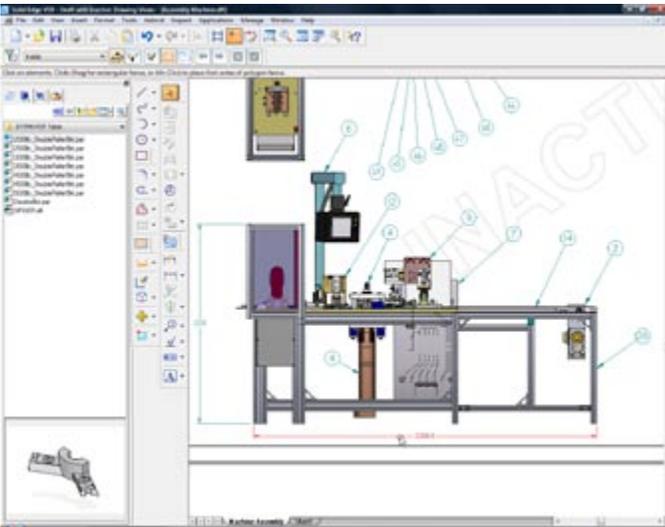


Image 20 (25.png) - This image shows placing over-all dimensions on an Inactive drawing.

### Massive assemblies can automatically unload "unused" components

Massive assemblies, as defined by the executives at Solid Edge, are assemblies of over 100,000 parts. Solid Edge has the capability to work with assemblies of much larger size. Pushing the ability to work with such large assemblies beyond that formerly thought feasible, resulted in creative thinking on the part of the Solid Edge software developers. Assemblies with large parts counts are typically dependent on the available amount of physical memory. Users can always add more memory, but now can also set an option in Solid Edge V20 to automatically unload "unused" components, which turns active parts into lightweight parts automatically, and frees precious system resources. Unused components are those that have not been used in any way. Should the user need to perform an operation that requires an active part, users can activate parts on demand. Image 21 shows the user interface of the option and how the user can set the time duration used by the unloading process.

Other operational alternatives also impact performance when working with massive assembly modeling. Zones improve open times by working within design envelopes; automatic unloading restores system resources; component grouping and positioning commands simplify massive assembly modeling; opening drawings of massive assemblies, for detailing, reviewing and printing all expand on Solid Edge's unique ability in massive assembly modeling.

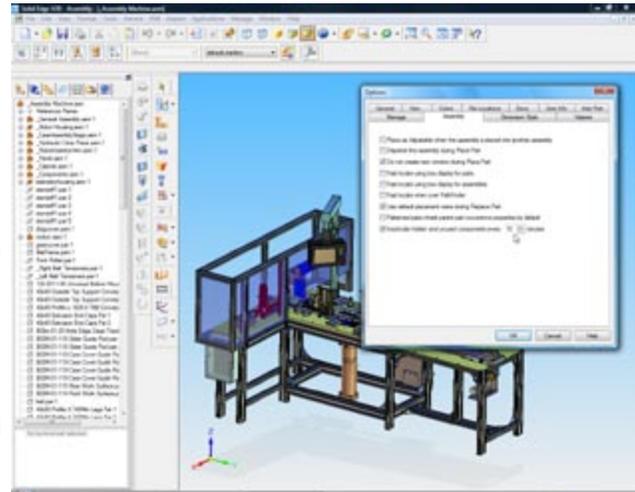


Image 21 (26.png) - This image shows the option to automatically inactivate (make lightweight) any parts that are hidden or that have not been touched by the user in a set increment of time.

**Tabulated drawings introduce a unique ability to gather and connect master model data to generate automatic tabulated drawing views for families of parts**

New for V20, tabulated drawings allow the ready placement of a family of parts table onto a drawing as a tabulated drawing. The family of parts table that exists on the master model facilitates this. To build a tabulated drawing we create a drawing by selecting the family of parts master model or any one of the members from one of the families. Once the views were generated, the new command “family of parts table” stepped us through the process. The resulting table placed on the drawing could be easily modified in terms of size and location.

A common dimensioning scheme is to overwrite critical dimension values with the actual variable name. Typically dimensions would be listed as “A”, “B”, or any convenient notation, and to facilitate this workflow, Solid Edge has options allowing selected dimensions to show the variable name. This technique is unique in the industry. Any other approach, such as manually placing the table on the drawing, would be more work and error-prone. The real value is that the table is fully associative with the family of parts model. Changing the family of parts table in the master model, automatically updates the tabulation on the drawing.

**A very impressive approach to building tabulated drawings, we particularly liked the ease of connecting to the master model variable table, how the drawing dimensions become the appropriate variable names, and the associativity with the model.**

which is the master part for a Family of Parts. The dialog shows the variables that are modified for each family member. (A, B, D, and H)

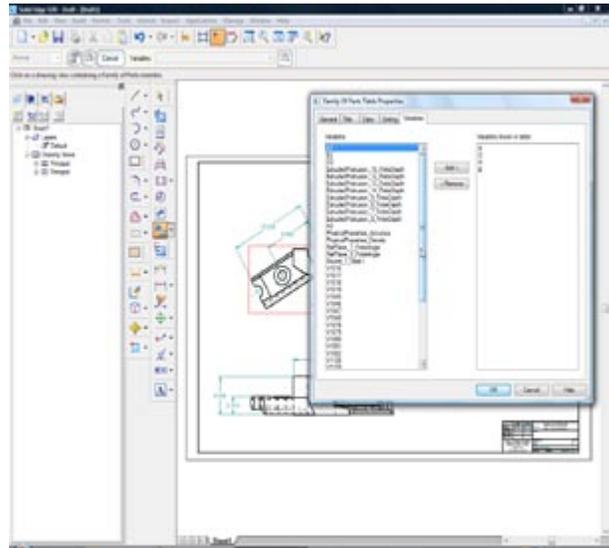


Image 23 (42.png) - When the user selects the command for creating a Family of Parts Table, they are presented with this dialog to define what variables will be displayed in the table. By default it will automatically include all of the variables that are varying from the variable table of the master part. (A, B, D, and H).

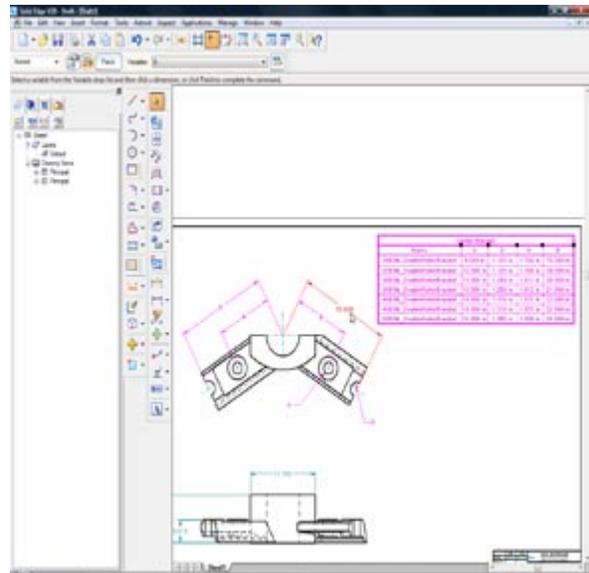


Image 24 (43.png) - Once the Family of Parts table has been placed, the user can assign the table variables to a specified dimension, changing the fixed dimension to the related variable name shown in the tabulation. This completes the operation of creating a tabulated drawing.

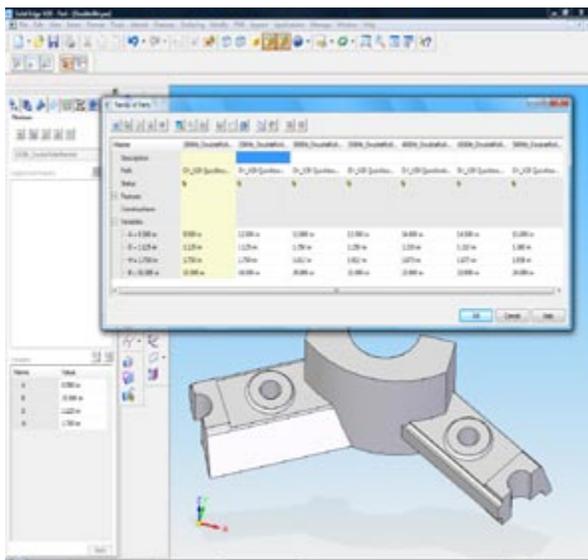


Image 22 (40.png) - This image shows the bracket

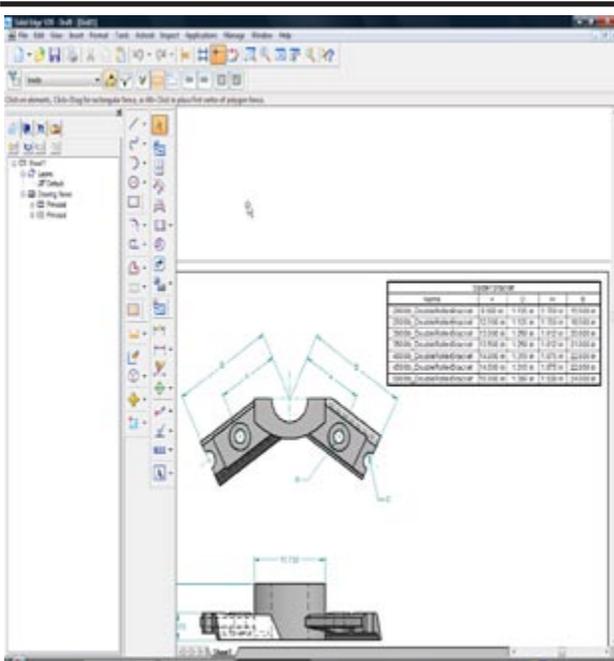


Image 25 (45.png) - Shows the final result of the Family of Parts Table and the tabulated drawing view

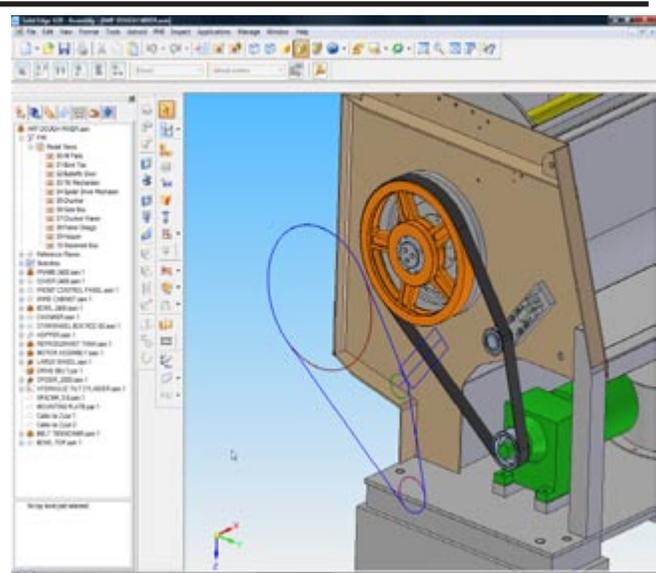


Image 26 (27.png) - This image shows the layout of the belt and pulleys. The tensioner pulley location is being driven by the assembly sketch. This is Solid Edge's existing hybrid 2D/3D method for laying out an assembly.

**Goal seek, feature grouping, feature library helpers, and the variable rule editor all act to improve the usability**

Goal seek allows performing two-dimensional what-if engineering calculations with a combination of 2D parametric geometry, mathematical formulas, variables and part properties. Using a sketch layout, users can let the system compute any one of the dimensions while varying (or Goal Seeking) another parameter. Some common uses are to determine pulley locations with fixed belt lengths, solving for areas when varying a boundary dimension and a host of other practical engineering scenarios. While calculations can be made by hand or done in a spreadsheet, Solid Edge allows users to graphically layout and solve complex scenarios. 2D sketch geometry can be used to drive both the size and position of 3D components. An example is shown below in Image 26, where a tensioner pulley is located according to a specific belt length.

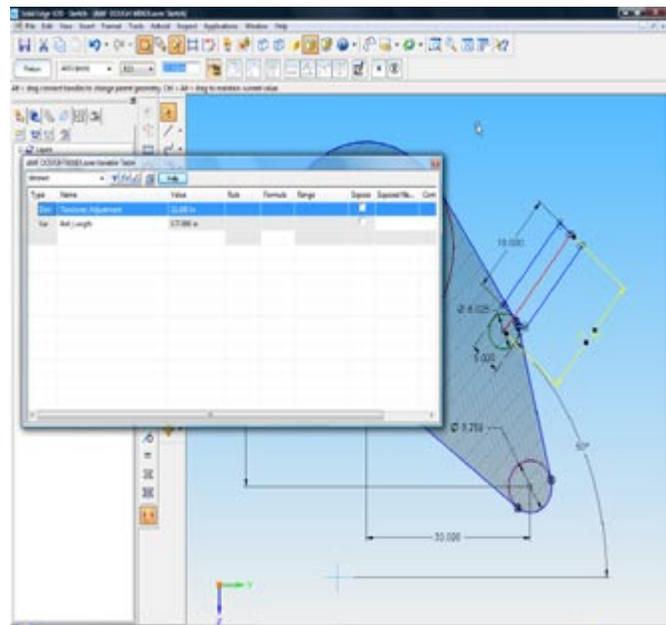
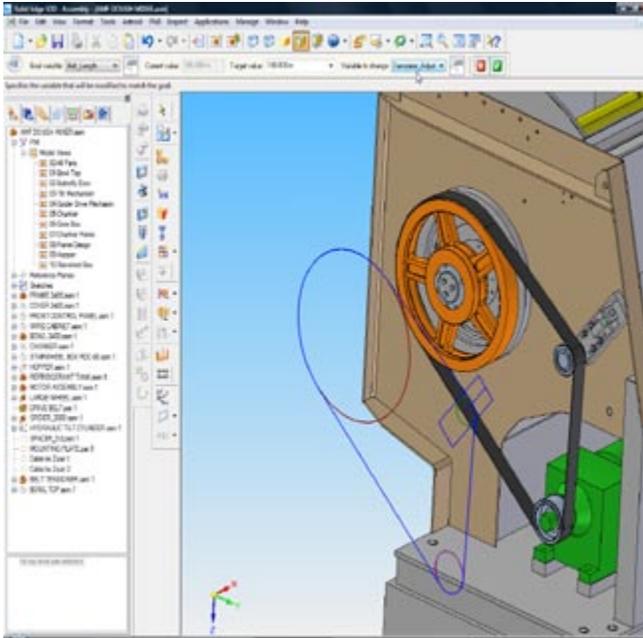


Image 27 (32.png) - The area command in the sketch view queries the area inside the belt and adds a perimeter calculation to the Variable table. In the variable table the Perimeter variable has been renamed to Belt Length to present a more easily understood name to the user. The Tensioner Adjustment variable controls the placement of the tensioner pulley and thus the belt length.

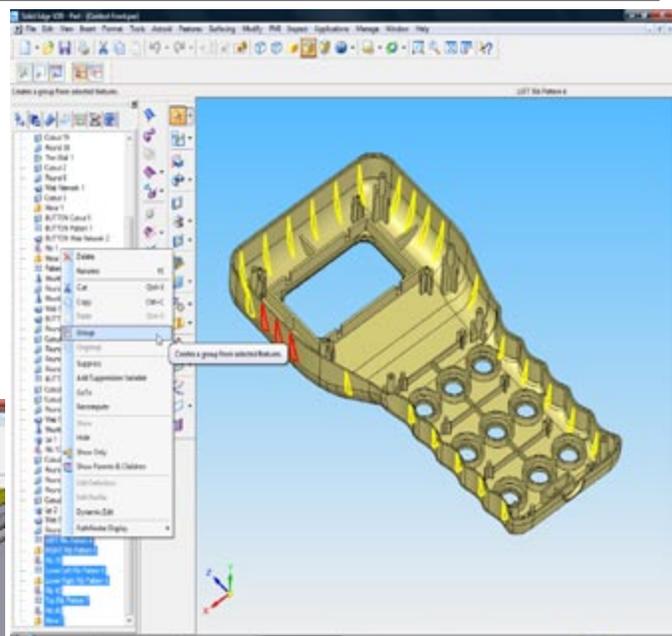
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The Goal Seek command allows the user to select a Goal Variable (such as Belt Length) which is a driven dimension, and set a target value for that variable. In the example shown here, Solid Edge adjusts a dimension or variable to reach the target value so that the belt length achieves a target value of 180.00 inches. To do so, the system seeks the final result by calculating new values for the Tensioner Adjustment dimension to achieve the belt length goal of 180.00 inches. After the pulley moves to the correct location, the belt model adjusts to the change based on the constraints built into the assembly sketch.

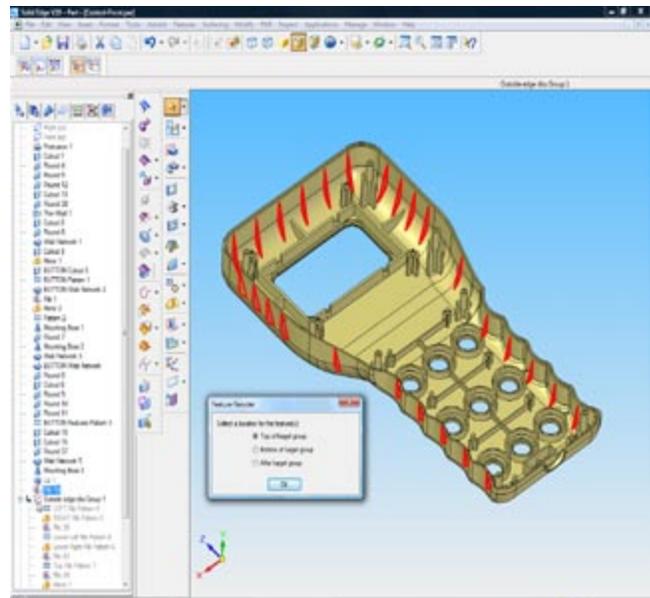


*Image 28 (33.png) - Showing the final position of the tensioner. Note the difference between this image and Image 26.*

**Feature grouping** allows grouping part and sheetmetal features together as a way to organize and simplify a model history tree. In the history tree, each group can be contracted into its name only (similar to a subassembly) or expanded into its elements. Grouping multiple features into a feature group allows the group to act as a single detail to speed up other commands that can use multiple features as inputs. For example, users can create a group of features and pattern, mirror, or create a feature library from the group. During group creation, the system does some validation checking to assure the grouped features maintain their child/parent rules. In the images below (Image 29 - Image 33), note that the many ribs and button holes are ideal candidates for feature grouping and for creating useful feature libraries. Groups can also be deleted, suppressed, shown or hidden.



*Image 29 (34.png) - This image shows Feature Grouping in Version 20. This enables the user to collect similar features under a single node in the feature tree. This can be used for a single selection for feature reordering, feature suppression, or to simply shorten the feature tree for clarification purposes.*



*Image 30 (35.png) - In this example we are reordering the Rib group up the feature tree. When reordering the user must decide where the feature is to be placed, keeping in mind the history of the features.*

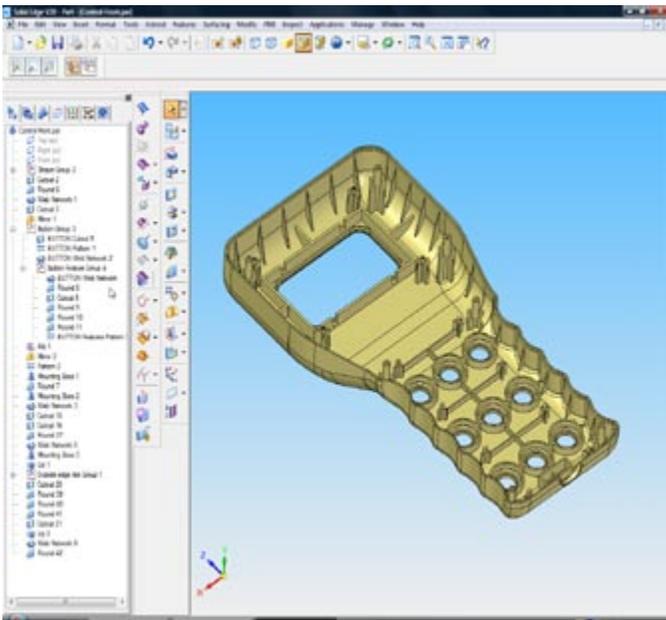


Image 31 (36.png) - This image shows nested Feature Groups

Feature library helpers guide a user on how to place a feature from the Feature Library, thus improving the ability to understand what references or constraints a library feature needs when being placed. During feature library creation, the author can define the prompts used to guide users as to how to use the library feature. After a library feature is created, it's easy to forget how to use it—especially complex libraries, but the new feature helper removes the burden of remembering how to place a library feature. Images 32 through 34 clearly depict this sequence. **Such a simple enhancement should make feature libraries many times more useful.**

A new *variable rule editor* allows the user to set either a range or a discrete list of valid dimensional parameters. Often models that are to undergo changes have limits on hole positions, overall sizes, or unique values. By adding ranges to the variable table Solid Edge now limits the size a user could make a part. Attempting to enter values outside the range prompts a message that lists the valid range. Upon dismissing the message, the part went back to the last valid size. Discrete values worked in a similar manner. After establishing a list of values (10mm, 15mm, and 20mm) only those values could be used for dimensional edits. To help users choose only those values, the list was presented on the actual dimensional value key-in field.

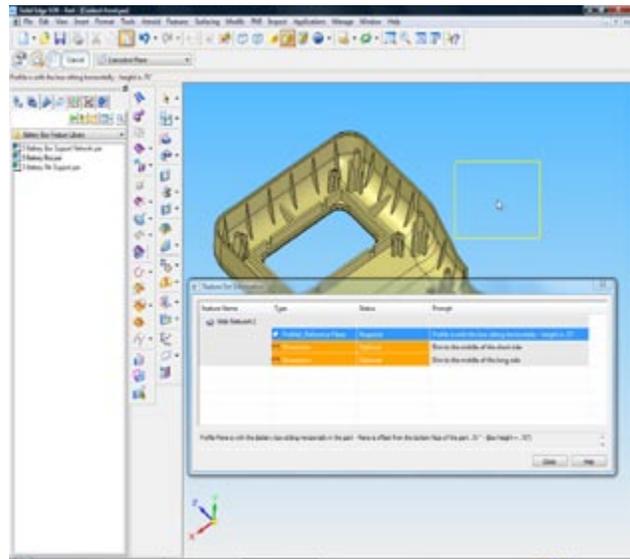


Image 32 (37.png) - Feature Library Helpers can now guide a user on how to place a Feature Library. Descriptions can be added for each placement requirement including any dimensions used to precisely locate the features. Note the helpful descriptions used in the prompt, as added by the author of the feature library item.

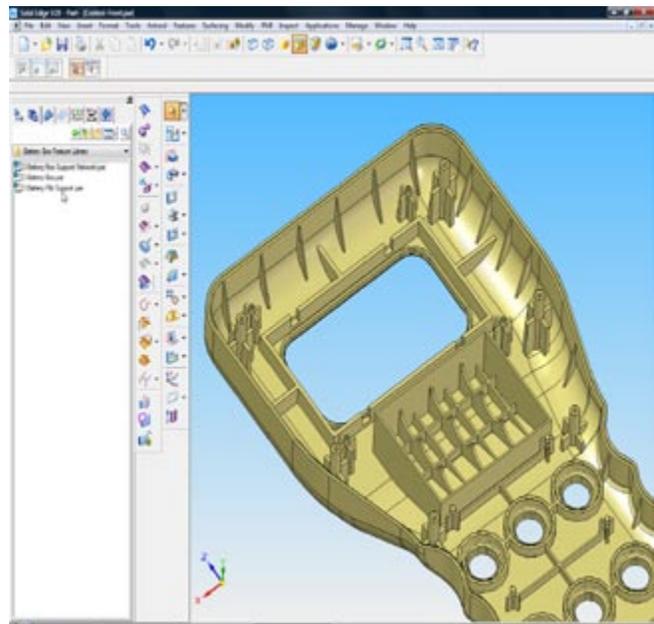


Image 34 (39.png) - This image shows the result after placing the three features from the Feature Library, greatly aided by the feature library help text.

## Conclusions

Our conclusions are in the Executive Summary at the beginning of this paper.

## About this paper

This paper summarizes the most important new functions of the Solid Edge Version 20 and why they are important for prospective users. The impressions and conclusions are solely those of the author, an independent analyst and consultant in the MCAD industry.

During the process of our review we collected many images of the software's new functions. We have placed those, most of which are not included in this paper because of brevity, on our web site along with explanations. You can access the website at <http://www.technicom.com/SEV20> .

## About the author

Raymond Kurland is president of TechniCom and its principal consultant and editor. His firm specializes in analyzing MCAD and PLM systems and has been involved in reviewing and comparing such software since 1987. Ray frequently consults with both vendors and users. He can be reached at [rayk@technicom.com](mailto:rayk@technicom.com).

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