What's new in Solid Edge ST3?

Benefits

Enhancements to fulfill synchronous technology's vision
- Accelerated model creation
- Faster ECO processing
- Improved imported data re-use

Enhanced simulation
- Coverage for more design scenarios
- Less need for physical prototyping

Improved collaboration
- Solid Edge Insight
- Teamcenter Express

Customer-requested improvements
- Modeling enhancements
- Application enhancements
- Drafting enhancements
- Usability enhancments

Summary

When Solid Edge® with synchronous technology software was launched in June 2008, it represented a major turning point for 3D design with a new modeling paradigm that combined parametric design with explicit modeling. In its latest release, Solid Edge ST3 continues to:
- Fulfill the vision of synchronous technology by accelerating model creation, speeding ECOs and improving re-use with imported data
- Provide simulation enhancements that cover a wider range of design scenarios, further reducing the need for physical prototypes
- Build on Insight™ and Teamcenter® Express software's integrated data management solutions by opening collaboration to a wider range of CAD and non-CAD users
- Address thousands of customer requests across modeling, applications, drafting, and usability

Fulfilling the vision of synchronous technology

Solid Edge ST3 fulfills the vision of synchronous technology by continuing to help users accelerate design. Solid Edge ST3 enables users to pre-plan less, speed ECOs by eliminating model regenerations and improve imported data re-use with non-native modeling. This latest release extends synchronous technology into assembly design, integrates ordered features with synchronous modeling and migrates manufacturing drawing dimensions to an associated 3D model.

Synchronous assembly design with all assembly applications

In Solid Edge ST3, synchronous modeling has been integrated into the assembly design environment, giving designers access to all assembly
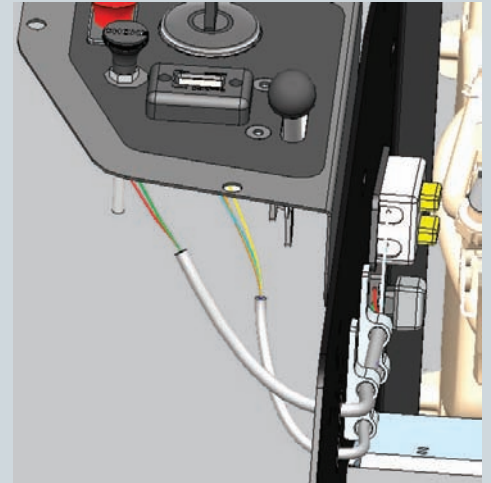

Figure 1. Synchronous modeling has been brought into assembly design, giving designers complete access to all assembly process-specific applications.

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applications for complete digital prototyping. Designers can model parts in the context of an assembly and immediately develop piping runs, structural frames and wiring harnesses using these proven process-specific applications. For complete assembly design, assembly features and families of assemblies can be used across models with synchronous or ordered features. Figure 1 shows a collection of parts with synchronous features that are used in conjunction with assembly applications. A quick drag of synchronous features is used to redefine hole locations and trigger an update to the wire harness.

Synchronous assembly design with part-to-part associativity
Synchronous assembly modeling now offers complete part-to-part associativity. Designers can make one part drive another before, during, or after the modeling process. Even imported parts can drive or be driven from other components without design pre-planning or remodeling. A new Create Inter-Part Relationships command automatically recognizes similar geometry between synchronous or imported geometry and creates a relationship linking them together. For example, several holes in the mounting plate can be automatically related to their counterparts in the bearing mount so that a change to one updates the others (see Figure 2). Users can selectively
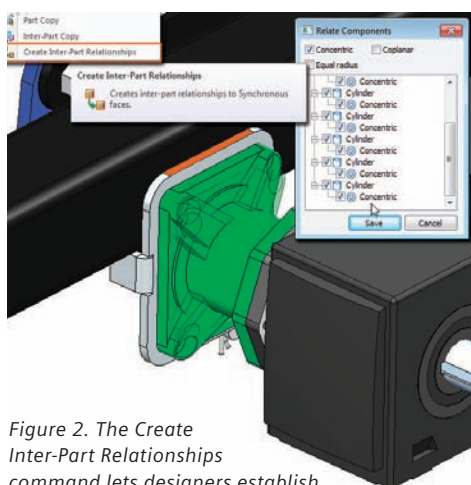

Figure 2. The Create Inter-Part Relationships command lets designers establish part to part associativity before, during or after the modeing procees, facilitating automated designs without pre-planning.
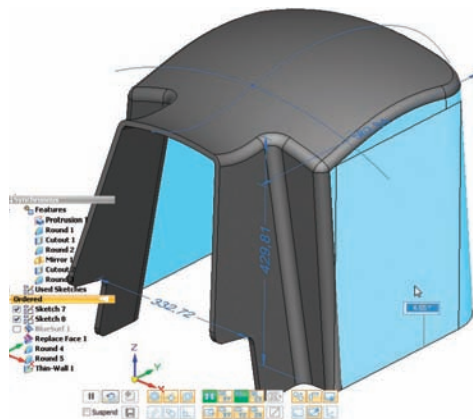

Figure 3, Ordered operations, such as a stylized surface, are added to a synchronous design. Precise edits can be made by altering dimensions or dragging faces; BlueDots connecting surface guide curves can be manipulated further eliminating the need to understand a model's construction before modification.

choose what relationships are captured (enabling control) or apply all relationships to make highly automated changes across multiple parts.

Integrated part modeling environment
Solid Edge ST3 now provides an integrated design environment that lets users add ordered features to synchronous designs for unlimited modeling capabilities. For model creation, synchronous technology forms the design foundation and allows users to add ordered features for process-type steps such as cast/machine operations or even surfacing for creating stylized designs. Edits to synchronous features are based on a synchronous solve for fast flexible modifications; any added ordered steps update automatically.

Figure 3 shows the taper being modified with a synchronous edit and ordered features, such as the surface adapt to the new shape.

In Solid Edge ST3, a single file can contain both synchronous and ordered features. This lets designers use existing methods while leveraging synchronous features with greater flexibility. Models can now contain both synchronous and ordered

features, enabling designers to selectively move ordered features to synchronous. This capability preserves established processes while leveraging the power of synchronous when needed. For example, the features defining the base shape of a part can be moved to synchronous for faster more flexible modeling, but simplified representations, subsequent machining operations, surfacing features, or complex blends can remain in an ordered mode. In Figure 4, to simplify rotating the vertical supports, the features originally created with order features, can be moved to synchronous for a quick drag. Machining operations, such as rounds or holes applied to the synchronous model, can remain as an ordered operation.
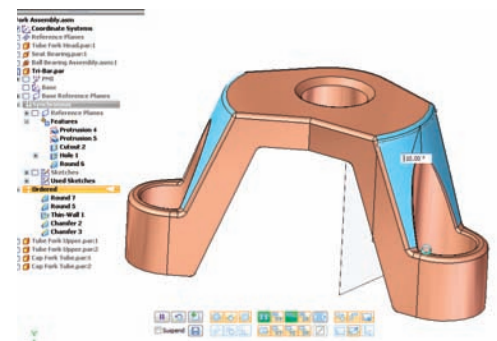

Figure 4. Designers can selectively move the features defining the base shape to synchronous for faster modeling, but keep some features in ordered mode.

Migrating 2D drawing dimensions to 3D models Solid Edge ST3 can take manufacturing dimensions defined on a 2D drawing and transform them into editable 3D driving dimensions on associated imported 3D models. Key manufacturing dimensions, such as an overall height, remain the driver, not the interim model or sketch dimensions. The Create 3D command transfers dimensions from drawing views (such as orthogonal, auxiliary and detail) to its 3D model. Newly added dimensions are ready for edits and geometric intent "lost" in translation is maintained with Live Rules, preserving design intent. Users simply open a 2D drawing, specify a view and identify a 3D model. Solid Edge ST3 applies the 2D dimensions to 3D, allowing immediate 3D edits.

Engineering simulation for more users New capabilities in Solid Edge Simulation lets designers validate more scenarios, thereby reducing physical prototyping. These capabilities also enable designers to get faster results and speed development times, as well as leverage fast flexible synchronous modeling and more refinements to improve product quality.
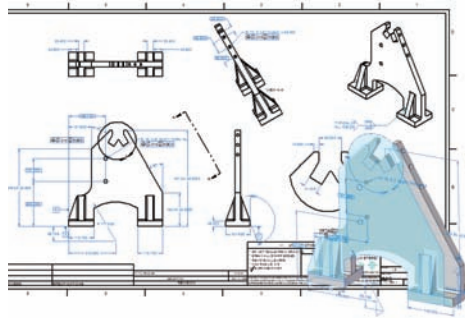

Figure 5. Solid Edge ST3 enables designers to better leverage 3D models from suppliers or customers by creating editable 3D driving dimensions to imported models from associated 2D drawings.
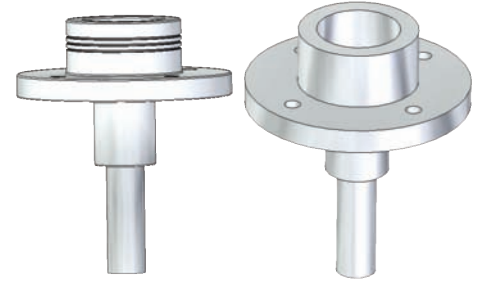

Figure 6. Model simplification removes unnecessary details from simulation studies and, together with synchronous edits, allows you to do more design refinements, which improves quality and reduces costs.

Improving analysis definition In ST3, Solid Edge Simulation includes additional constraints and loads to simulate more design scenarios. New capabilities facilitate torque loads, bearing loads, bolt connectors and custom constraints. The integrated design environment brings simulation to more users, by enabling existing models to use simulation directly. Model simplification is available to synchronous designs, offering faster results without compromising accuracy.

Design refinements can be now done in a fast flexible manner using synchronous technology. Driving dimensions can be

applied to a 3D model or within 2D Live Sections, while Live Rules maintains design intent.

Other exciting enhancements include improved post processing display with clipping planes. This enables designers to see what's happening "inside" the model. For automating common simulation tasks, full API support is now available. For replicating similar design studies, loads, constraints, and mesh controls can be copied or dragged to other studies. Simulation reporting has also been enhanced so that reports can be written to Microsoft Word (.doc and .docx) and Adobe (.pdf) document formats.

Spotlight on:
Modern Mechanical Fab, Inc.

Results:
- PE bills reduced by 50%
- Engineering collaboration up by 25%
- Material costs down by 10%

"Once we added Solid Edge Simulation, the bill from our outsourced professional engineer went down."
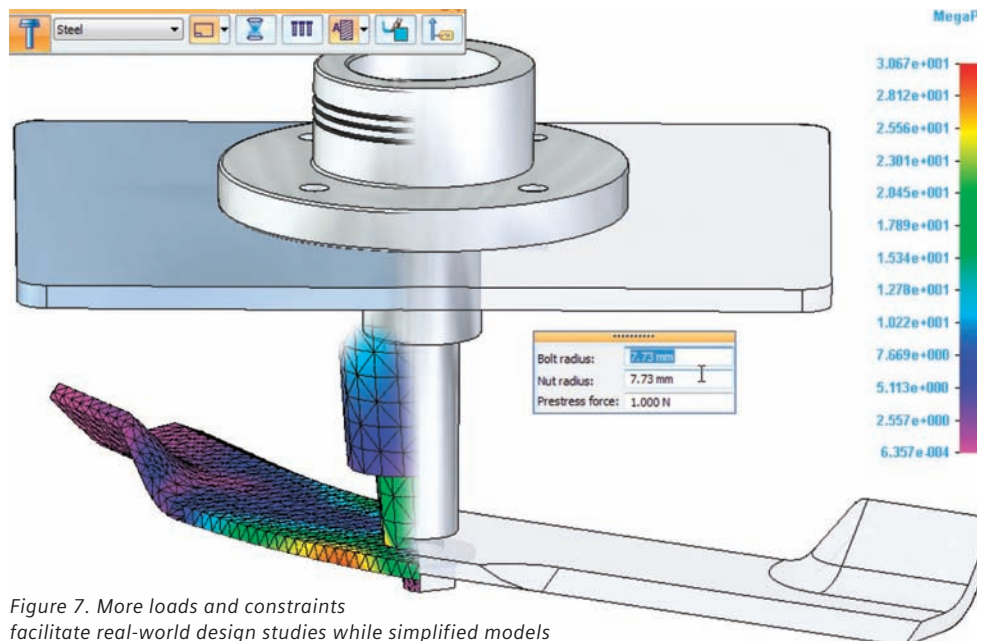John Fillion
Mechanical Engineer

Figure 7. More loads and constraints facilitate real-world design studies while simplified models speed results. Synchronous technology enables fast design refinements.

Improved collaboration through Insight and Teamcenter Express
Solid Edge ST3 continues to build in its scalable PDM solution to meet the needs of its customers regardless of company size. Our easy-to-implement solutions include capabilities for file vaulting, searching, complete workflow management and design data integration with other business processes.

Solid Edge Insight product data management with Microsoft SharePoint 2010 Solid Edge Insight is a totally integrated data management system offering transparent PDM capabilities using SharePoint. First released in 2001, it was the first SharePoint-based CAD data management solution. Today, it has thousands of users worldwide. Insight has proven to be effective in managing design data with minimal overhead.

Spotlight on:
Reading Bakery Systems

Results:
• Design productivity up 63%
• 2D drawing production time reduced from 3.2 hours to 1.2 hours

"Insight permits the re-use of design data to a much greater extent because we can find existing information and can trust it".
Michael Cox
Information Technology Manager

Solid Edge Insight now works with SharePoint 2010, taking advantage of many new capabilities. SharePoint includes built-in collaboration tools that enable users to share information with manufacturing, customers and suppliers. An integrated task and project management system is available to Insight users. This enables managers keep track of projects directly in SharePoint using actual CAD files. Another exciting SharePoint 2010 capability provides business reporting tools that can leverage the information within CAD data through Insight. By using Solid Edge models, users can estimate costs and analyze product makeup with PowerPivot, an integrated Excel-based pivot table.
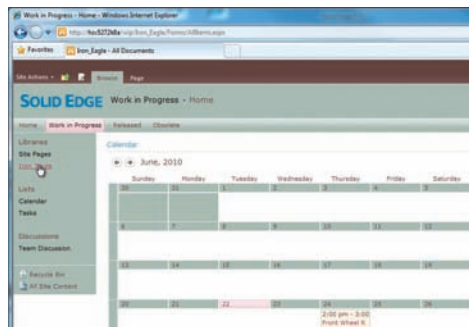

Figure 8. In ST3, Solid Edge Insight supports the latest SharePoint 2010 offering, thereby facilitating improved collaboration.

Teamcenter Express preconfigured cPDM Teamcenter Express is a preconfigured collaborative product data management (cPDM) environment that offers easy implementation for a fast return on investment. Teamcenter Express supports the multi-CAD, multi-site needs of today's manufacturing organizations.

The Structure Editor within Teamcenter Express has now been made into a standalone application that enables fast development of a product structure or bill of materials (BOM) outside of the CAD system. This tool can be a run by non-designers, such as product configuration specialists, managers, and sales engineers. The product structure can be easily converted into a Solid Edge assembly at any time and incorporate physical or virtual components.

Delivering thousands of customer requests across the entire product
Solid Edge ST3 adds depth and robustness by addressing thousands of customer enhancements across all areas of the product including its modeling, applications, drafting, and usability capabilities. Users will see increased productivity improvements that range from existing design methods to advanced synchronous technology. The following subsections discuss just a few of the top enhancements.

Assembly design It is now possible establish component item numbers at the assembly level and automatically populate the parts lists during drawing creation. You can specify item numbers early in the
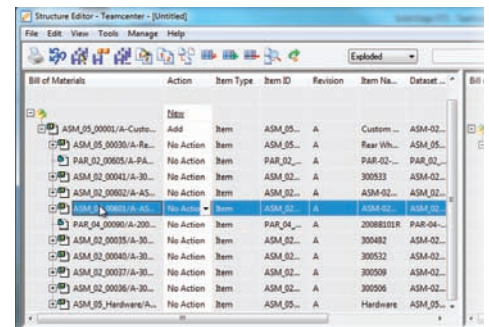

Figure 9. Managers can easily develop a product structure outside of Solid Edge with a stand-alone product structure editor.

design process. This enables designers to better identify key components on drawings. Part-to-part links are now shown in the assembly pathfinder, giving you the ability to quickly understand inter-part relationships even if files are opened out of context. You can also freeze and unfreeze inter-part links to test different design concepts.
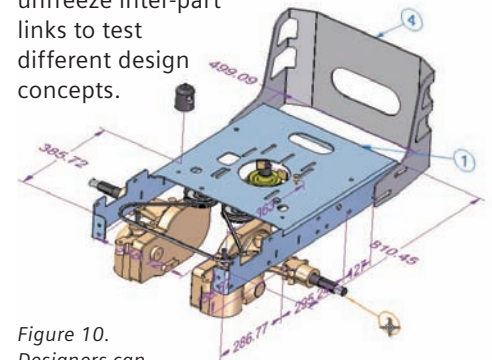

Figure 10. Designers can now specify item numbers in Solid Edge assemblies that propagate through to drafting.

Assembly display configurations are now table-driven and assembly configurations are available in the part or sheet metal environments.
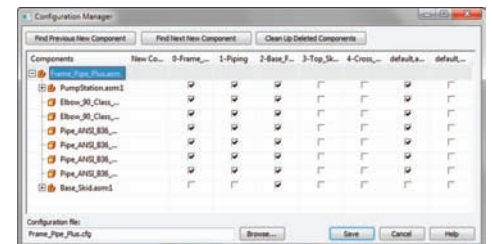

Figure 11. A flexible new configuration manager saves time setting and adjusting assembly display; it allows multiple display configurations to be applied at once.

Part modeling For synchronous modeling users, Family of Parts is now supported. Designers can specify member overrides for Live Rules, dimension values and 3D model relationships.

In addition, a new option in Live Rules provides locking geometry to the base reference planes, keeping planar faces co-planar with the base planes and cylinders and cones coaxial with the base axes. A new command for offsetting an edge on a face is available to help users divide surfaces or create tangent hold lines for advanced blending. Improvements to the 3D measure command are available to facilitate better reporting, more graphical feedback and cumulative and multi-leg measuring.
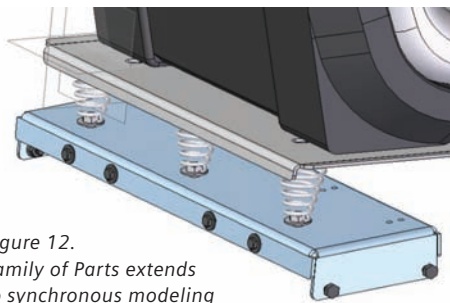

Figure 12.
Family of Parts extends
to synchronous modeling
with specific overrides for Live
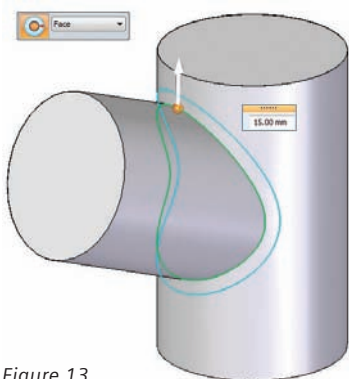*Rules, dimension values and model relationships.*

Figure 13.
Designers now can
offset and edge to simplify tangent
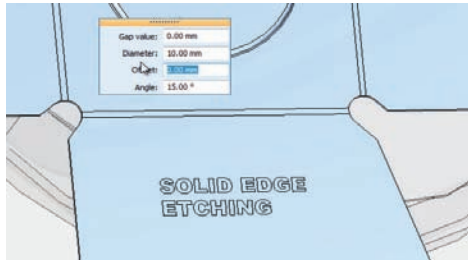*hold line creation.*

Figure 14. More close corner options are available such as these U shapes. The Etch command lets designers burn part numbers in models during laser cutting operations.

Sheet metal design New close corner options in Solid Edge ST3 are available for sheet metal designers including U, V, square, and mitered corners support, which lets users create just about any corner type. When designers need to write text on a sheet metal model, they can use the new Etch command to write part numbers or instructions on faces that can be burned in by laser cutting machines. Etched text will translate to DXF as a stick font and on a separate layer for easy identification by manufacturing software. Material libraries have also been enhanced to better link a material gage to the material type.

PMI dimensions Users can now directly edit the value of product manufacturing information (PMI) dimensions during placement and lock these values to speed workflows. These dimensions can be placed to silhouette points, tangent points and virtual vertexes and used to drive geometry, giving users total control over model shapes and sizes. Driving dimensions can also be placed to geometric midpoints for added control.

Assembly applications Frame design now lets you use any key point to position cross sections; sketches can be rotated or flipped. Frame elements can now be patterned to reduce design time. A new option is available to control how faces of tangent and collinear frames are merged. Piping designers can now specify pipe gradient for defining a slight rise or fall in the elevation of a drain or steam pipe. Pipe fittings can be graphically rotated during placement or edits giving users total flexibility in defining the orientation of valves, flanges and other components. Piping elements can be patterned, further reducing design time.

Figure 16. Slight pipe gradients for drainage are among many piping updates.

The Weldment design application includes several enhancements. Welds that follow sharp corners, such as fillet and stitch, now give a smoother more realistic bead around those edges. The display of weld beads has been enhanced within Teamcenter.
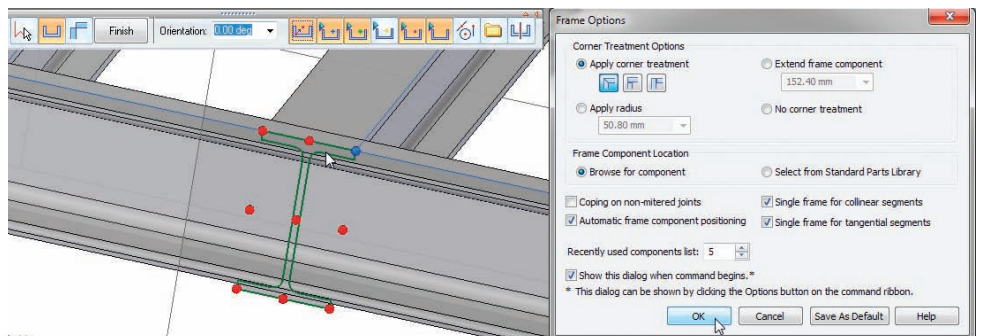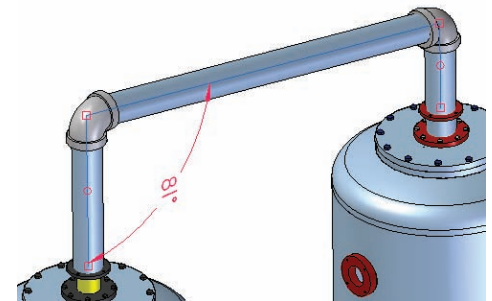

Figure 15. New frame options provide more design flexibility for merging frame members; they can be positioned at pre-defined anchor points at any geometric key point.
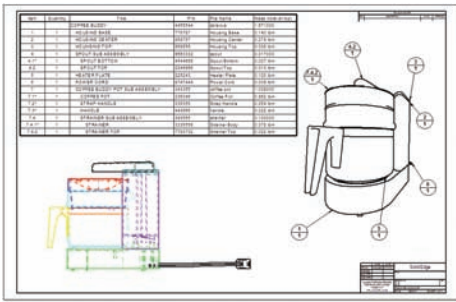
Figure 17. Draft now includes perspective views. Line color can be automatically driven from the 3D model color. Parts lists can have column totals and be indented to quickly identify sub assemblies and their components.

Figure 18. Solid Edge ST3 user interface updates, such as a transparent PathFinder and redial menus , improve user productivity.

Draft Draft creation in Solid Edge ST3 includes many new enhancements starting with parts lists. Designers can modify the BOM to report a fully exploded or indented parts list, and the order can be set to match the assembly structure. Columns can be totaled to show assemblies weights or other key summaries. Automating detailing of assemblies has been improved as retrieve dimensions from assemblies is now supported. The visible line color in drawing views can now be derived from 3D model color, giving users quick identification to key parts.

Perspective drawing views are now available letting designers create more visually appealing isometric views. For getting a quick report on a distance, diameter or angle, measuring in 2D has been added. This new capability will also be available in Solid Edge 2D Drafting – a free product. Drawing files can be shared across languages while correctly displaying all text and annotations; zone support during drawing view creation has been added, as well as many more capabilities.
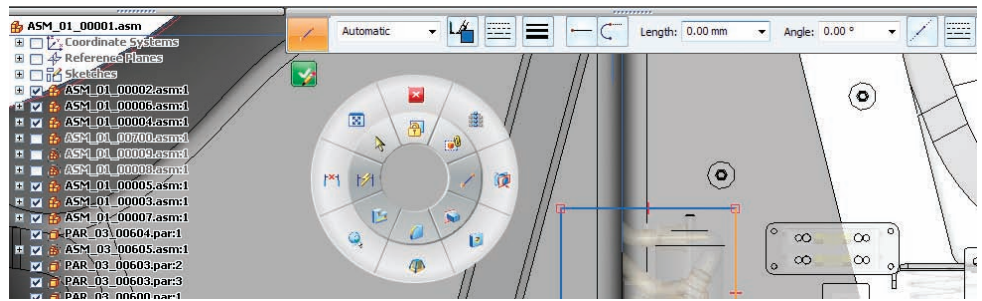
Usability Improving the usability of Solid Edge always receives top priority and Solid Edge ST3 is no different. The Command Ribbon bar can now be customized, letting users define the placement of their most useful commands. A new customizable Radial menu gives quick access to commonly used commands. The command bar has a horizontal format option that provides the look and feel of the famous SmartStep ribbon bar, which frees up space in PathFinder.

The display for PathFinder can now be shown in transparent mode, freeing up the display for Windows 7 users. Synchronous users now have quick access to the options in Live Rules with a new Heads-up display that appears transparently on screen whenever a synchronous edit is performed. Many other new capabilities also facilitate faster performance. Performance tests for part view manipulations can be up to 4x faster while assembly can be up to 10x faster.

Conclusion
Solid Edge ST3 continues to fulfill the vision of synchronous technology; it covers more scenarios with Simulation and opens collaboration to a wider range of users. By addressing thousands of customer requests, Solid Edge ST3 facilitates an excellent balance between innovation and the product's ability to meet the needs of its user community.