TechniCom, Inc.

Solid Edge Version 19:

Solid Edge Version 19 focuses on global innovation, adding complete functionality for employing JT files, 3D annotation for conveying product manufacturing information, rapid motion simulation of assemblies, much improved AutoCAD drawing import, an easy-to-use animation capability, integration with NX CAM Express modules, additional sheet metal functions, and more.

This report reviews and comments on the major changes implemented in Solid Edge Version 19.

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Executive Summary

Introduction

Ray Kurland, the author of this paper, spent several days at Solid Edge headquarters in late June 2006, reviewing Version 19 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 Adrian Scholes, Director of Solid Edge Marketing, UGS, for an overview of Version 19. Scholes 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 Field Support Engineers for Solid Edge - Doug Stainbrook, Chuck Brashear, and Mark Thompson who demonstrated and explained in detail the major new functionality in Sold Edge Version 19.

Solid Edge continues to advance in the marketplace

The following sections discuss the detailed highlights of Version 19. Equally as important as having excellent technology is the marketplace acceptance of Solid Edge. 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, yet need easier to use and lower cost systems. Mid-market companies also need to participate in global markets and also to respond to rapidly changing economic and business requirements. Thus their tool needs are similar to those needed by large enterprises, but must be less complex for supporting smaller companies with commensurately smaller IT staff. To address these unique mid-market needs, UGS provides software specially tailored to integrate tightly with Solid Edge. These include Femap, Teamcenter Express, and the new NX CAM Express. Customers are "voting with their dollars" because Solid Edge revenue is growing at about two times the overall MCAD market. Femap is enormously popular; Teamcenter Express trials 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. As an example, this year UGS plans to increase by 50% the number of channel partners. Having reached 30% halfway through the year, this goal should be met well before year end 2006.

Highlights of Version 19

Solid Edge Version 19 focuses on global innovation networks - "companies working globally and interacting with each other to bring together many of the players, thus enabling the rapid exchange of ideas and designs that foster innovation." UGS - Solid Edge aims their software investment at this goal. The functionality aims to make this happen. In response to this requirement, Solid Edge Version 19 delivers 11 major advances, all focused around advancing design collaboration and improving user global innovation networks. These include the following:

• A new easy-to-use and fast running motion simulation for assemblies, with capabilities to quickly and easily define relationships between different motion drivers, such as gears, pulleys, hydraulic cylinders, and motors. Motion studies demonstrate the visual appearance of a product as well as how it functions in real time.
• An integrated and more flexible capability for animating exploded views and assembly/disassembly dynamic documentation. Enhanced tools provide for capturing, modifying and animating how parts are assembled and disassembled, allowing generation of videos for clearer manufacturing instructions, technical manuals and training materials.
• Direct use of JT files with no intermediate steps, as if a JT part were a native part.
• An enhanced workflow for importing AutoCAD drawings results in Solid Edge drawings that look the same as the AutoCAD drawing.
• PMI - product manufacturing information (3D annotation) allows conveying product and manufacturing information directly on a 3D solid model, perhaps eliminating the need for a drawing and simultaneously making it easier to interpret such annotations.
• A new set of NX CAM Express applications that automatically executes directly on Solid Edge models and puts the user into an NX CAM operating environment. Added to an already existing Solid Edge license, prices range from $6500 for 2-axis machining to $18,500 for a full 5-axis package, about on a par pricewise, with the competition.
• Enhancements to the outstanding Solid Edge Insight workgroup data management software, and improvements to Teamcenter Express, thus executing on the plan for a fully scalable design management system.
• Support for Microsoft Windows 64-bit OS architecture, introducing a 64-bit version for Solid Edge Classic and Foundation for those users creating massive assemblies, with both versions co-existing.
• The addition of considerable new design and

manufacturing functions (e.g., hems, gussets, etc.) to sheet metal.
• The addition of hundreds of additional customer-focused enhancements.

The following sections describe the major functions of Solid Edge Version 19 that reviewed in this paper:

• UI Improvements
• Explode and animate
• JT collaboration
• PMI
• Sheetmetal
• NX CAM Express integration

Conclusions

Solid Edge Version 19 offers a substantial upgrade from Version 18. We were particularly impressed with the explode-render-animate function which is completely new, the complete incorporation of the JT format, the sheetmetal extensions, PMI, and the incorporation and tight integration of NX CAM Express into the Velocity Series.

Animation and assembly explosions, completely new in Version 19, coupled with a new approach to simulation, promise to make these functions available for use by the typical user. Using an experienced operator, in less than an hour we built four highly sophisticated animation sequences involving movements, explosions, collapsing explosions and specific grouping of parts and visibility. These capabilities will prove to be particularly appealing for improving communication with downstream processes such as NC, visualizing how jigs and fixtures and parts are mounted for machining, assembly instructions, and possible disassembly instructions for repair. The animations, being directly derived from the part and assembly models, will be easier to maintain, and in most cases, will automatically update as the model geometry changes.

Already, the leading provider of sheet metal design software, Version 19 is loaded with additional goodies that continue to enhance Solid Edge's leadership. Gussets, added stamping lines, and complex hemming operations, all difficult to model in other systems, ease the ability to produce complete sheet metal designs.

PMI should prove particularly useful to those users adopting full 3D documentation, often proving simpler to interpret than drawings containing similar information. These, along with JT integration and the addition of integrated CAM to the velocity Series all enhance the comprehensive coverage of Solid Edge Version 19.

Description of the new functionality of Solid Edge Version 19

Our impressions of Solid Edge Version 19 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/SEV19. 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.

Dynamic preview, motion simulation added, and AutoCAD import improved

• The feature preview capability dramatically improves the ability to preview modeling changes. Prior to this release, Solid Edge displayed the wire frame of the about-to-be-created feature. Now it displays full-color shaded solid models of the proposed new feature.

• Version 19 introduces motion simulation for assemblies, which goes beyond using kinematic joints to simulate motion, as was done in the previous version. In Version 19, motion simulation is easier to set up and executes much faster. The new function uses UGS' D–Cubed component software technology, incorporating specific motion simulations, such as circular to circular, circular to linear, and linear to linear. The concept of a motor drives motion simulation and can control speed and direction of motion. Simplicity of set-up and speed of execution as well as new visualization options improve the ability to understand what is happening within the motion simulation.

• Solid Edge improved their AutoCAD conversion wizard so that the imported AutoCAD drawing file looks the same as the AutoCAD file.

Explode and animate

There has been extensive work done to improve animation using a timeline programming capability to control the animation and to organize dynamic assembly explosion capabilities.

Exploding an assembly not only allows for easier vi-

sualization, but uses the exploded paths to build and control later animations by time sequencing the explosion "events." Version 19 adds substantially more control into the explode capability of solid models. Explode adds a new pathfinder tab that allows significant control over the events of the explosion - events are defined as either a linear or rotational move.

In the images below, we show the explosion of an assembly by its sub-assemblies. That is, during the explode, the system groups the sub-assemblies together and moves each sub-assembly along a path using the assembly mating constraints that were defined when the assembly was built. Each "move" group and event is recorded within a new explosion pathfinder tab that allows such events and groups to easily be edited and used in animation sequences later.
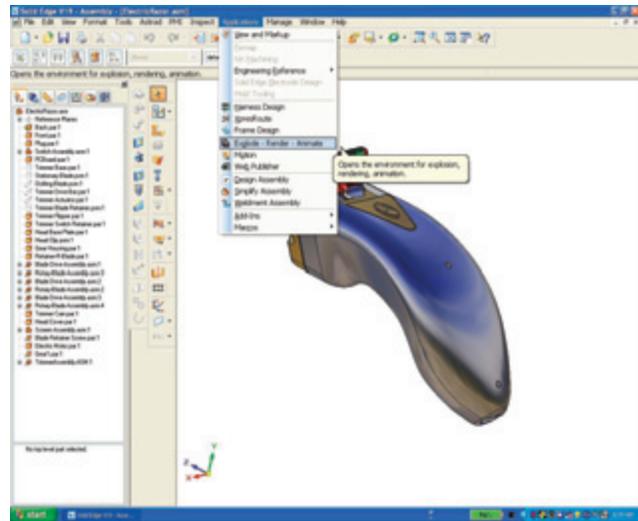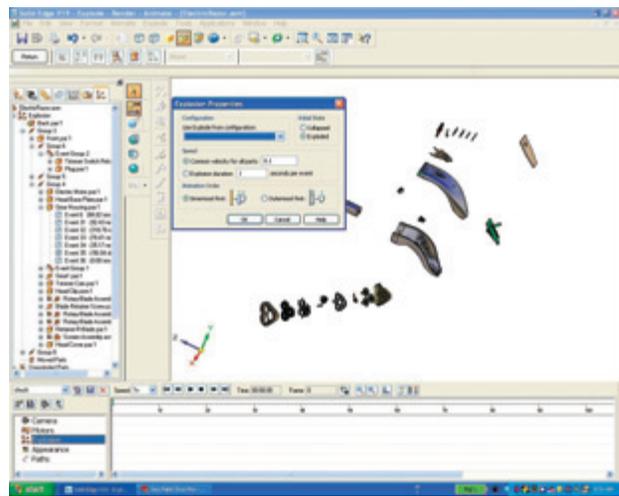

ERA02 - Changing from the assembly environment to "Explode-Render-Animate" application
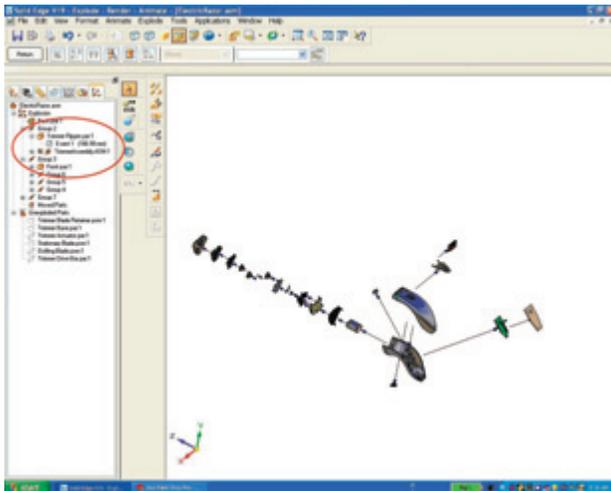
ERA03 - New explode pathfinder tab depicting exploded parts, unexploded parts, moved parts, explosion groups and events

*ERA01 - Adding an explosion configuration to the animation timeline*

Different types of groups are created under the new pathfinder tab. (These might be a bit easier to see at the website which has larger images). One such group is the direction that we are exploding. All of the components that were exploded in the same direction will be organized into a group; if we have events occurring at the same time, those will also be included within a group. An event would be a linear or rotational move. In the case of a pattern, the event would appear simultaneously, had we pre-defined it as a pattern during the build of the assembly. During the definition of the animation, the system allows synchronizing different events by grouping them such that their animation movements occur simultaneously. Controlling the direction of component movement is difficult in some systems -- not in Version 19 -- because flow lines (that define the movements) are 3D line segments that can be manipulated. For example, in one case, one of the flow lines was modified by adding additional jogs and a rotation, all of which were recorded in the explosion pathfinder.

Animation adds the ability to automatically create timelines using the explosion groups and events and places them into the animation editor, as shown below in image ERA01. The animation editor's timeline graph controls the animation duration and sequence of each event. Animation can create a movie file, which can, in turn, be edited using traditional software for movie editing to add such items as audio tracks and title frames.

In the case of the dough mixer example show below, we added the effect of motors and gears, then performed a motion simulation.
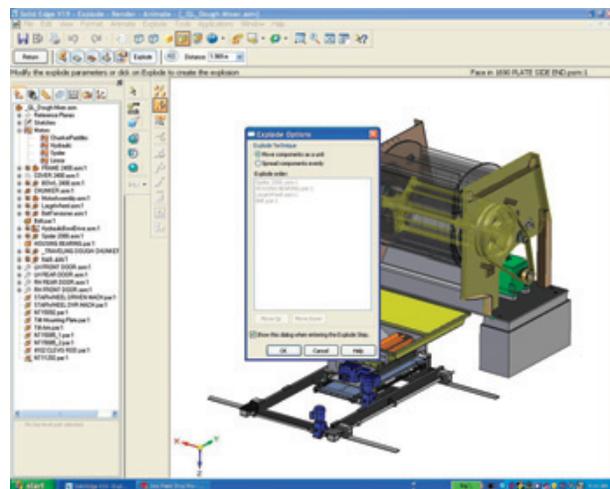
ERA04 - Motors and gears applied to Dough Mixer model

We simulated a motor by selecting a cylindrical object and defining the object as a rotational motor. When we chose to connect this via a gear to another object, the system automatically calculated the ratio based on the diameters of the two objects. Changing into the "explode-render-animate" environment for the dough mixer, allowed us to build a complete animation using the explosion "events" previously built to move the objects plus the motion simulation from the assembly environment, all acting to create the resulting animation. The animation editor allows adding explosions, changing the appearance of parts, adding motors, changing camera angles for viewing, and

defining specialized paths for components to follow, even a b-spline path. Parts also have appearance parameters that control whether and when along the timeline they display of not. A unique capability of Version 19 allows limiting the angular rotation of motors. Mirroring a motor motion makes it appear as if the motor were running in reverse. For more detail, we suggest readers visit the above referenced web site for an animation example containing many of these options.

Collaboration using JT

JT - perhaps the most widely used and well defined format for storing CAD system independent, lightweight representation of a model - provides for saving various levels of model details. Models can be saved with a basic tessellated format all the way to a full B-rep assembly structure, additional properties and attributes, and even the precise representation of the model. The author of the JT output determines the level of model precision to be saved.

In Version 19, Solid Edge accepts a JT file with no intermediate steps and can use a JT file as if it were a native part. In this way Version 19 supports multi-CAx representations. Currently, there are three opportunities available for viewing JT and mixed files. They are the JT Viewer, the Solid Edge Viewer (allows viewing Solid Edge assembly, part, sheet metal and draft files, as well as DXF and AutoCAD DWG files), and XpresReview (allows a combination of 2D and 3D data and other documents to be packaged into a single file and allows measurements, mark up, and section views).

PMI - Product Manufacturing Information (3D annotation)

PMI, based on the ASME Y14.41 standard, can convey product and manufacturing information on a 3D solid model. Rather than being restricted to drawings for annotation, PMI allows adding dimensions and annotations directly on the solid model. Such annotations are organized onto certain views.

PMI002 - Solid Edge PMI annotations can extract hole callout information and part property information directly from the 3D model. This includes tapped hole callouts, counterbore hole callouts, drilled hole callouts, etc.

Also, unique to Solid Edge is the ability to dimension and annotate section views of the model, as shown below in image PMI005.

PMI005 - This image shows a PMI Annotation, in which the name for each component callout is derived from the 3D model. This also shows a cut section. The callouts can be derived from an edge of the part or from the face of the part.

Model views can be used to control, and often clarify the display of annotations, Model views accompany and are part of the model, often easing collaboration. The designer / engineer receiving the model can easily access the model views and display the dimensions and annotations as organized by the originator. All dimensions and annotations are associated to the model. Changes to the model change the associated annotation. Furthermore, the intelligence built into a model can ease annotation placement. For example, in the instance of a hole callout, the system can automatically place the proper hole size and the hole type (such as counter–bored) from the original hole design by accessing the data stored when creating the hole using the hole command feature. This 3D annotation can eliminate the additional step of creating a drawing. PMI thus allows ready creation of preliminary information used for design reviews and collaboration. Using it instead of drawings for collaboration provides the opportunity to save considerable time by not requiring the production of rigidly defined drawings. For providing quick and easy design reviews, Solid Edge Version 19 does an excellent job of not only creating the PMI information, but also organizing such PMI information into views - making the presentation highly readable.
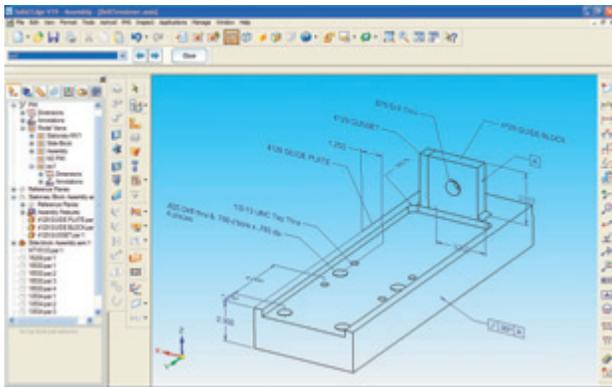
The above image illustrates the possibilities of displaying GD&T's on the solid model and depicts how much easier to understand the data is when displayed on the model.

According to Norm Crawford, a Tolerance Analysis and GD&T Consultant, the use of 3D GD&T and PMI results in significant reduction in time by eliminating the non-value added replication of information on drawings and provides greater clarity of design. With 3D GD&T, basic dimensions are not required nor are the other features that can fall under the realm of a title block tolerance specification. Instead, only the tolerance specifications are required since the CAD model is the "definition." A single datum symbol clearly defines the datum feature that is visually highlighted on the 3D model. As for detail and section

views, normally shown in wireframe on a drawing, any decent 3D viewer of a model can easily zoom in on details and dynamically section a part or assembly model. Thus providing an unlimited number of views instead of the limited details and sections on a drawing that a draftsperson assumes will be enough for someone else to fully interpret the design. As for clutter, a viewer can filter out the PMI to display only what a consumer wants to see. For example, during a review with a supplier, it is easy to turn off all dimensions except for the Datum features so that understanding and agreement on the datum features can be easily reached.

Sheet Metal

More functional changes to sheet metal are included in this release than in any previous release. Version 19 enhances the modeling capabilities of sheet metal and improves its manufacturing information, allowing easier creation of the sheet metal part. Version 19 adds extensive use of additional attributes, such as line styles so that punch and bending software can easily create the final part from flat patterns.

To illustrate some of Version 19's sheet metal capabilities, a sheet metal bracket will be added to attach a butterfly valve hydraulic cylinder, to be located on the upper part of the hopper in the image below.
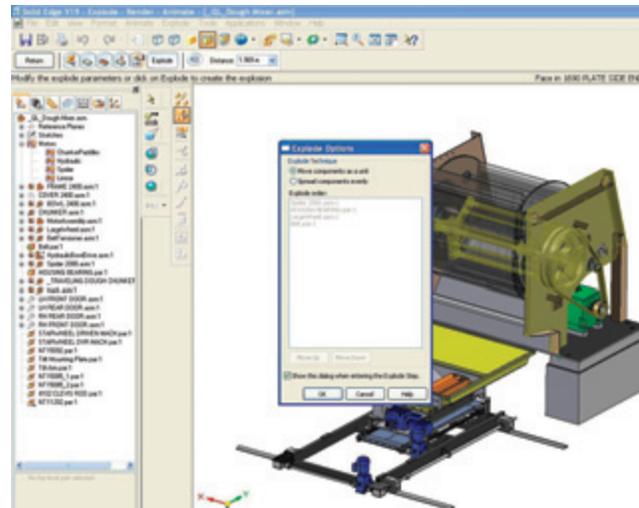

Building in the context of the assembly, we activate the assembly components we will refer to, dimming the rest. In the activated components, selecting some geometry allows us to design the new tab needed for the bracket, associative to the hopper. Completing the tab, a flange was constructed that matched the hopper angle, then used a new deformation feature to reinforce the flange - a gusset. Adding the gusset - difficult to construct without this feature - proved easy to do by selecting the deformation-gusset function from the sheet metal menu.

Various options are available when placing gussets, such as placing a number of equally spaced gussets along an edge. The image below depicts the result.
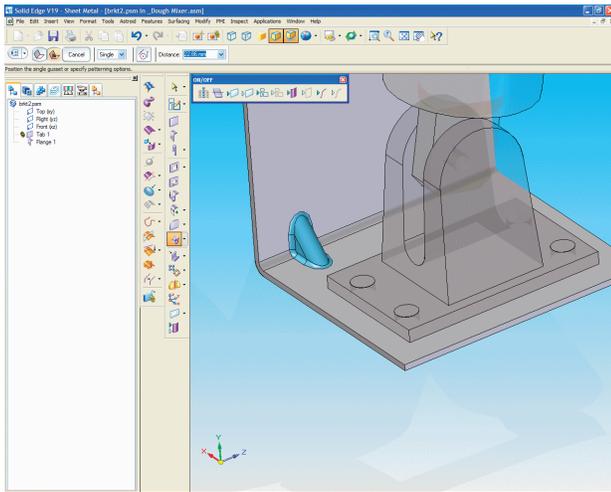

Gussets add strength to bends. This gusset was created on the bend by identifying the bend and specifying the number of gussets. The user can create user defined gussets using the gusset options dialog.

Another stiffening option, used for large sheet metal panels, shows the placement of and outputs the location of a cross-brake - slight intersecting bends.
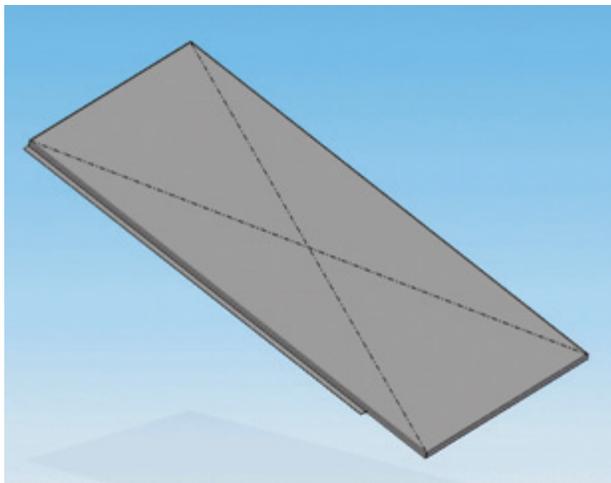
Cross brake lines shown on a sheet metal panel

In Version 18, Solid Edge added the ability to have a contour flange on a curve. Version 19 adds the ability to change the thickness dynamically. Version 19 also enables adding thin lines called triangulation lines onto conical bends that indicate to the manufacturing operation where the specific bends are to be placed to properly produce the desired conical bend.
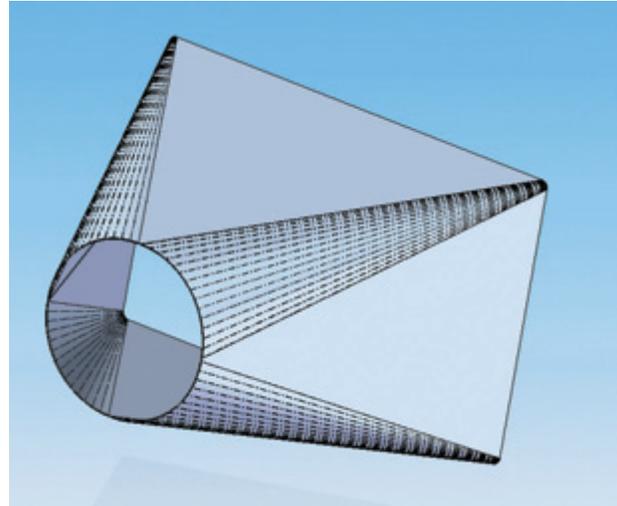
Triangulation lines showing where bends are required to be made to generate the correct fold.

Other functions further improve sheet metal. Version 19 now allows add different types of hems, such as those depicted in image SM003 below. Another allows the use of stencil fonts - important for sheet metal lettering. Also Version 19 allows producing bend tables process sheets allowing the brake operator to easily visualize the angles that need to be made, and the bend directions, either in 3D or a drawing.
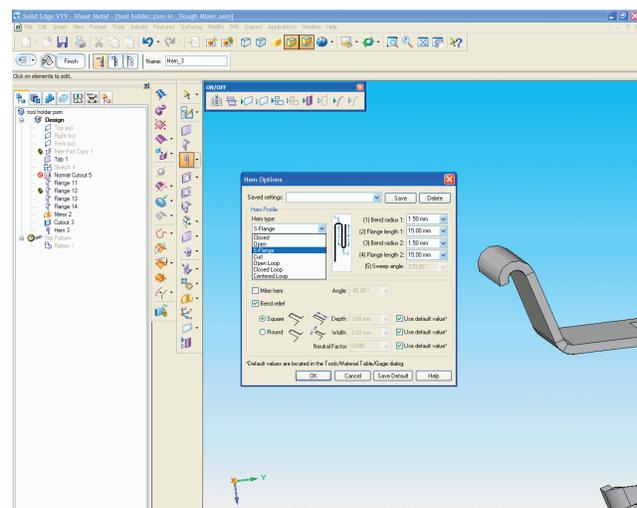
SM003 - Version 19 supports the creation of various types of "hems" in sheetmetal, offering several hem creation options, which the user can select using the dialog shown.

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 19 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/SEV19 .

About the author

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