



August 10, 2000

The eighth meeting of the Madison Area SolidWorks® Users' Group (MASWUG) was held on August 10th, 2000 at UW-Madison Memorial Union. 71 people attended from 40 companies and organizations. This tied our largest attendance to date. I personally think it was due to the location (Union Terrace), the promise of beer during the social hour and the great topics local users were presenting (not necessarily in that order).

Introduction - George Skupniewicz, Nicolet Instrument Corp.

General Announcements

- ❑ UW-Madison Computer Aided Engineering Center officially hosted the meeting
- ❑ Election of MASWUG Steering Committee=
- ❑ Committee includes:
 - Dave Strait of Lunar Corporation: Co-Chairperson
 - Kent Ziemer of American Superconductor: Co-Chairperson
 - Ron Pulvermacher of Matrix Product Development: Co-Secretary
 - Mike Alex of Venus Graphics: Co- Secretary
 - George Skupniewicz and Jeff Wong Advisors

A vote of hands was taken and the new steering committee was voted into office.

The next meeting will be facilitated by the new Steering Committee
MASWUG Trial Web Page www.geocities.com/maswug

SolidWorks career opportunities

Location: Madison, WI – seeking senior software engineers to develop web enabled applications – check SolidWorks website: SolidWorks World Conference- Sunday-Wednesday, February 11-14, 2001 at Disney's Coronado Springs Convention Center, Orlando FL
<http://www.solidworks.com/swworld/>



Presentations

Gary Shruck – Mayville Metal Products

Topic: "FeatureWorks and Pro/E converter"

Demonstrate the import and conversion of a Pro-E form .prt into SolidWorks to create a form tool. This form tool can then be used within SolidWorks parts. This operation can be performed using the standard Pro-E converter with SW 2000 on Pro-E parts prior to version 2000i. With file versions of 2000i and after, the Pro-E file will need to be converted to an .iges or .sat file, then brought into SolidWorks. With this last procedure, the FeatureWorks module may help, but is not necessary.

Carie Locher - Impact Engineering Solutions

Topic: "Using Envelopes for top-down modeling"

Demonstrate how to use an envelope as a 3D layout. The envelope will show how the assembly should look when complete -- simplistically. With such a tool available, I will show how to mate components directly to the envelope and apply most of the external references to the envelope as well. This approach was created to reduce "Top-down assembly confusion", improve communication between SW users, and not have a phantom part show up in the bill of materials
Jeff Wong – Paradigm Designs Inc. Topic: "Adobe Acrobat and shrinking file sizes"

Demonstration showed how Adobe Acrobat can archive SolidWorks drawing files (or other applications with 2D data) to 1/100 the size and the new file is readable by the free program called Adobe Acrobat reader (already loaded on almost all computers, both PC and Mac). See attached spreadsheet.

Jeff Setzer – Graphic Systems Corp. Topic: "SolidWorks to Adobe Illustrator"

In order to get vector artwork out of SolidWorks, you can use either of 2 methods.

First, you can use DXF. This is self-explanatory...save your drawing out of SolidWorks as DXF and use it in whatever package you need. If you want to tweak the artwork, Adobe Illustrator version 8.0 and later can read DXF. The



only problem with DXF is that splines come out to be a whole bunch of really short line segments.

For those that want more precise data, you can get your artwork out of SolidWorks using Postscript. Since there is no "save drawing as Postscript" function in SolidWorks, the way to do it is to PRINT the drawing to a Postscript printer, which is set to write to a file instead of an actual device port. You will most likely need to tweak the artwork in a program like Adobe Illustrator -- it can read Postscript files if you bring it in as "parsed EPS." Be sure that the print driver you set up to create Postscript files is, in fact, a Postscript driver. I have had the best luck using the generic Adobe Postscript driver from adobe.com, but you could try more device-specific drivers like HP and Tektronix.

Jim Boda – Inspire Design Group, LLC

Example: "Probe handle with multiple lofts"

Kent Ziemer – American Superconductor

Topic: "SolidWorks and the Internet"

See Links

Mike Alex – Venus Graphics and Christine Shorey – Nicolet Instruments (on vacation)

Topic: "What's Cool SolidWorks 2000 & Forgotten Features"

New Tips for SW 2000

If viewing mates with a split feature tree or just trying to use the eye tab to view mates does not work. Make sure the check box under 'system options/general' is checked to turn property manager on.

If you have a 3 button mouse or a wheel mouse that does not work as it is explained in the what's new book check out Microsoft's web page: <http://www.microsoft.com/> . Then click downloads, choose the mouse you have (IntelliPoint mouse for the wheel mouse) and click find it. If you have any problems with the IntelliPoint driver try going to your start button/control panel/mouse/wheel tab and click the troubleshooter button.



Can't find your part? Is it gone in the assembly and in the part itself, but the feature manager tree does not say it's suppressed or hidden? Right click on the first feature in the feature manager tree and choose 'show solid bodies'. The new hide solid bodies feature is cool for working with surfaces, but can easily be selected instead of hide part. This has been a bit confusing for us since there is no visual change to the part icon in the feature manager tree after this has been done. (Enhancement request?)

Now you can break or lock in-context parts. However be careful of the difference. You can unlock the relationships but you cannot un-break them. Note the suffixes: -> means that it has an in-context reference that is solved, ->? means that it has an out-of-context reference that is not solved (open the assembly that it is trying to reference to solve and update the feature), ->* means that the reference is locked and ->x means that the reference is broken.

I have not used fast draft too much (I had hoped to by the meeting), but it seems to be a great time saver. Just a word of caution though, you cannot un-fast draft a drawing so check it out before applying it to your drawings.

Want to dimension to the center of a line or lock the center of two lines together with a relationship? Select the midpoint of the line by right clicking on it and choosing select midpoint.

Check out the new "file locations" menu under the options menu/system options. You can have multiple places to store your templates as well as your blocks and palette features, parts and forming tool. I suggest using a common folder on the network to help build a library of these files.

Default for 3 standard views in drawings changed. You will need to reset your options if you want it the way SolidWorks used to default to after installation. Right click on the sheet ('sheet1') in the feature manager tree and select properties, there is a box called "type of projection" that you can choose first or third angle. You should be able to save this setting in your template or sheet format file. (should check this out before the meeting to see which it is)

When creating a revolved part from half of its profile, you may want to have the dimension be the diameter of the part after it is revolved rather than the radius. You used to have to place a dimension from you line to the center line and then right click on to select properties and change it to diameter. Now you can just



move your cursor to the opposite side of the centerline before you place it and SolidWorks will display it as a diameter.

A really nice enhancement is how SolidWorks now fills in the current distance when doing a distance mate. This also works when doing an angular mate as well.

Forgotten Features

A fast way to create an offset plane is to hold down the control key while selecting and dragging the plane you want to offset from to its new desired position. SolidWorks will automatically fill in the distance and you can always change it to be a specific value.

For me the dangling dimension color (olive greenish brown color) is hard to see at times so I change it to a brighter orange to help flag myself there is a problem. You'll find this options menu/system options/colors/dangling dimensions.

Save clicks. When adding a dimension, don't forget you can right click to lock the dimension in the direction (horizontal, vertical, etc...) you want it be before placing it in the location. This also works for angular dimensions and the diameter dimensions for a revolved part as we discussed earlier.

If you do not use single click command mode because you want to be able to do a command multiple time occasional, be aware that you can double click on a button to stay in that function.

To rotate your part about a point axis or line, click the rotate button (or right click rotate view), double click on a line, axis or point and click and drag to rotate as normal.

To create a virtual sharp corner in a drawing use the control key to select the two lines and click the insert point button. If this does not work check your option menu/system options/sketch to see if display virtual sharp is checked.

Do you hate those annoying messages you get when you're trying to save an assembly with read-only parts in it that you don't want to save? Go to your



options menu/system options/external references and check the box that says, "Don't prompt to save read-only references documents (discard changes)".

When multiple people are working on an assembly don't forget about the reload option. This is also what you need to use when you want to replace a component in assembly with another of the same name but in a different directory. This is of course if the assembly is already open and you don't want to re-open it and use the references button. There are four ways to reload a file: reload it as read-only, reload as read-write, reload file to replace it with another existing file with read-only access or reload file to replace it with another existing file with read-write access. Remember however that if you want save the current part, save the current part before you reload or all changes will be lost.

I made an excel page of all the standard hot keys that SolidWorks comes with. Often we forget about these and they are helpful if you remember them. Also if you create your own hot keys, you can look and see what is used already so you don't duplicate it. Unfortunately, SolidWorks will not warn you against creating multiple commands for one hot key. I suggest if you are trying to start using the standard hot keys that you create a cheat sheet to hand up on or near you monitor. You may want to customize the hot keys list (make a copy of it first) by getting rid of the ones you know and adding your own before you print it out. You can shrink it when you print it, either by changing the font size or printing it at less than 100%, if you need to. See attached spreadsheet of SolidWorks default hot keys.

Solid Solutions magazine is doing a series of articles that will total 100 "tips to increase productivity". So far it was publish 10 at a time in the May/June and July/August 2000 issues. This is another place you can look for tips.

Greg Johnson – LA Darling Store FixturesTopic: "3D Meeting/NetMeeting"

Break-out Session "Union Terrace Social"

Free beverages and munchies served on the Terrace. Perfect evening on the Terrace!

Special Thanks



Gary Shruck, Carie Locher, Jeff Wong, Jeff Setzer, Jim Boda, Kent Ziemer, Mike Alex, Christine Shorey, Ron Pulvermacher, Greg Johnson and Steve Krueger for giving their time as presenters.

Paul Oliphant for reserving the room and the perfect weather for us.

Graphics Systems Corp for use of the computer and projector.

Stacey Dunn at SolidWorks Corp. for paying for our food and beverages.

Special thanks to George Skupniewicz for running the last 8 meetings.

Please forward these minutes to others who work at your company or are interested in joining MASWUG. Please send any secretarial business (i.e. potential new member e-mail addresses for the database, comments about the meeting, the meeting minutes, etc) to the new co-secretary Ron Pulvermacher: info@matrixdesign.net

Submitted by old MASWUG secretary: Jeff Wong