



## Minutes from February 8, 2007 User Meeting

The meeting was held at Front Range Community Collage in Westminster Colorado, The meeting started at 3 PM and there were 30 people in attendance.

Kevin Van Liere of 3Dawn Consulting LLC made a presentation of 3Dawns EfficiencyWorks product and demonstrated some of its functionality to aid users in productivity in all areas of design and drafting. The functions presented were:

Starter Templates Wizard is a tool allowing someone setting up new part, assembly or drawing templates to quickly set these up without having to go thru the slower learning curve of setting up each template feature individually.

Set Properties Wizard This tool walks the user through basic questions about the active document. What is the part description? What is the part number? Depending on what type of document is active determines what questions are asked.

Materials Selection Wizard is a tool to aid the user with an interface to quickly identify a material and finish which meets their needs and can be linked to a preferred company materials list.

Get Next Part Number Wizard will select the next available part number from an Excel database.

Export Wizard will export files in formats such as IGES, STEP and Pdf allowing a quick method of generating files for purchasing or vendors.

Kevin also passed out prizes to people selected from a drawing for the people that filled a survey of their feed back on the products shown in his presentation.

During a break for snacks Gerald Davis presented pictures from SolidWorks World 07 which he and Paul Larsen had just returned from that day.

The remainder of the meeting was for “Open Mouse” for attendees to present or discuss favorite tips and tricks. Several tips were presented but the most helpful was voted to be for a method to copy custom properties from one part or assemble to another.

The tip is:

- Go to the custom properties of the file you want to copy from
- Left click and highlight the number (left column) of property or properties you want to copy and do a CTRL C.
- Open the destination file, go to the custom properties and left click on the number (left column) of the first blank row
- Enter CTRL V and the first item will appear
- Place the cursor in the Value Column and hit tab if importing multiple properties.

DPM Associated LLC provided a \$50 prize for the winning tip.

Thank to Kevin Van Liere for the presentation and for all presenter of the tips and tricks and of course to DPM Associates for the prize.

Steve Ray  
COSUG Communication Director



## Minutes from April, 2007 User Meeting

The meeting was held at Front Range Community Collage in Westminster Colorado, The meeting started at 3 PM and there were 36 people in attendance.

Due to a last minute change of plans we only had one presentation for this meeting. Kurtis Anliker a Territory Technical Manager for SolidWorks made a very interesting presentation on the varied uses of the powerful surfacing commands within SolidWorks. Surfacing within SolidWorks has made the transition between the Industrial designer and the Mechanical engineer much smoother. The strength of the capabilities is not just limited to surfaces models but also very useful with solid models. Kurtis explained how many of the commands can be very useful when working with imported solids where there is not feature history to work from. The ability to use commands such as fill, delete face, replace face can allow you to modify existing faces without having access to the commands that created the features.

Being able to use 3D sketches to create surfaces is very useful when defining many of the creative shapes required for products today. When working with the 3D sketches the designer only has to grab and drag points to be able to re define shapes and that pictures can be imported into the sketch and the points moved to match the shape of the object in the picture. Kurtis demonstrated how 2D sketches can be created on the fly within 3D sketches

Kurtis also pointed out that SolidWorks Office Premium offers the capability of being able to bring in Point Cloud and Mesh files to be able to convert to either surface or solid models.

It was announced that since the Denver SolidWorks User Group Network Technical Summit will be held on August 28<sup>th</sup>. The regular August User Group meeting which would have been on August 9<sup>th</sup> will be combined with the Tech Summit.

During the mid session break attendees were treated to snacks of sandwiches and refreshments and an opportunity to mix and mingle with each other.

A very big thank you goes to Kurtis Anliker for the very interesting presentation and to SolidWorks for the assortment of shirts and copies of “SolidWorks for Dummies” for the attendees to take with them. Also we would like to thank all the attendees for their support.

The next scheduled meeting is June 7<sup>th</sup>. A meeting notice will be sent out announcing the location but also visit [www.cosug.com](http://www.cosug.com) for meeting details.

Steve Ray  
COSUG Communication Director  
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## **Minutes from June 8, 2007 User Meeting**

The meeting was held at Front Range Community College in Westminster Colorado, The meeting started at 3 PM and there were 25 people in attendance.

We were honored to have Mr. Vic Leventhal a Group Executive with Dassault Systems meet with us and present the history and future of SolidWorks. Vic was the first CEO for SolidWorks and was very involved with making the company what it is today. We heard many stories of what the founding members went through to get the product out into the marketplace and that the company has remained true to its founding principles of working to make its VARS, users and investors successful. To keep everyone thinking Vic rewarded people for correct answers for questions concerning the history of the company total payout, of his own money, was \$80. For the future Vic presented an overview of SolidWorks 2008 that no one outside of SolidWorks has seen, he had just received it the day before. Some of the high points presented were a improved user interface, the ability move & rotate PhotoWorks images, and 2 new packages Drive Works & Flow Express, for Premium users a tolerance analysis package. Form what I heard we have lots of good things to look forward to.

After a brief break for snacks and refreshments a question and answer session concerning issues people were having. Fred Zobel from MCAD was present to field many of the questions.

After the meeting Mr. Paul Mott of Peak Solutions demonstrated a hand scanner system from Z CORP for capturing existing parts into a database.

Just as a reminder the August User group meeting will not be at the usual time but we are combining with the SWUGN Technical Summit which is August 28<sup>th</sup>. The following link will provide full information on the summit.

[http://www.swugn.org/pages/summit/SWUGN\\_Technical\\_Summit\\_Denver1.html](http://www.swugn.org/pages/summit/SWUGN_Technical_Summit_Denver1.html)

Steve Ray

COSUG Communication Director



## Colorado SolidWorks User Group

Voted 2006 SolidWorks User Group of the Year

### Minutes from the October 4, 2007 User Group Meeting

Our meeting was held at the WarrenTech Career & Technical High School in Lakewood, Colorado and started at 3:00PM. There were 43 people in attendance. Dan Riffell and Rick Chin were the guest speakers.

Gerald opened the meeting by inviting everyone to participate in the CSWA exam. This is a 3 hour proficiency examination that includes a series of questions and a modeling challenge. The test can be taken from any workstation. To participate at no charge, you need to have attended a recent SolidWorks user group meeting. Send your name, company name and email address to [president@cosug.com](mailto:president@cosug.com). You'll receive an access code and web link to the test.

Gerald also recommended attending SolidWorks World 2008 in San Diego. Visit [http://www.solidworks.com/pages/swworld08/fees\\_policies/fees\\_policies.html](http://www.solidworks.com/pages/swworld08/fees_policies/fees_policies.html) for more information.

Dan J. Riffell's live SolidWorks modeling demonstration emphasized efficiency. To paraphrase Dan's recommendation, don't clutter your workspace with menu toolbars. Instead, customize the toolbars so you only see the functions you commonly use. More importantly, take advantage of keyboard shortcuts. Customize the keyboard shortcuts so they are easy for you to remember.

Dan used a "cam plate" as a sample project. While drawing his master sketch for the base extrude, Dan did not take his hand off of the mouse. He drew lines and arcs by either tapping a keyboard shortcut (the "A" key) or moving the mouse pointer back over the last end point to toggle the line mode.

His initial sketch was just a free form concept. He didn't fuss with dimensions or scaled shape. The main goal was to quickly get the idea roughed out. Once that was completed, he then started adding dimensions and relations to fully constrain the sketch. His sketch included some solid lines in the interior of the closed profile. This alarmed some of the old timers in the audience. Dan would soon reveal his purpose.

Dan took the time to position the dimensions to eliminate the tangled bird's nest. He also pointed out the changes in the mouse pointer to indicate mouse button options for locking the direction of dimension orientation. He recommended that we organize our sketch dimensions so they resemble a finished mechanical drawing; include the tolerance on critical dimensions. His reasoning would become evident later when he generated a mechanical drawing and imported the model dimensions and annotation views.

With the base sketch completed, Dan created the base extrude. He pointed out that future mating in assemblies can take advantage of the planes in the parts. So, take advantage of mid-plane extrude if that will result in a useful plane relationship in the model.



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As he added extrudes to the model, he demonstrated using a shared sketch as well as a derived sketch and pointed out the benefits of each technique. Closing the sketch to the model was a quick trick that got a strong reaction from the audience.

To demonstrate a more efficient method of creating the model, Dan set up a second configuration and suppressed the several extrudes that he had just created. He then used his master sketch and used the contour select option to select regions of his sketch for extruding. This was the reason he had used solid lines in the interior of the sketch. With just a few clicks he reproduced the same model that had taken extra sketches and several extrude / mirror operations. Contour Select can be a powerful (efficient) tool.

With the extruding completed, Dan used the Hole Wizard to add holes and counter bores to the cam plate. He demonstrated customizing the Hole Wizard to include his “favorite” hole sizes.

Dan then moved on to adding fillets, which he pointed out should generally be the last series of features added to a model. There are times when that rule does not apply. He demonstrated that by filleting a surface which resulted in fillets around the perimeter of holes. By sliding the fillet step up the Feature Tree, the fillets were removed from the holes, which is what he wanted.

With the feature modeling completed, Dan added material to the model (copper in this case). He also demonstrated the application of PhotoWorks materials and recommended doing that early in the modeling process. If you wait until the final assembly is completed, it is time consuming to return to each part and update the material properties. Adding the materials while the model’s design intent is fresh in your mind is a better work flow.

Dan used the multi-view tool to split the screen into 4 quadrants. He did this so he could quickly do a test rendering in PhotoWorks. With a few adjustments to lighting, he was satisfied with the result and used the mouse to drag the view borders to return to a single view. This little trick emphasized the capability to set up the views to be of maximum value while modeling.

Looking ahead to the mechanical drafting, this cam plate model did not lend itself well to standard orthogonal projections. Dan decided to set up a cross sectional view of the part. He used a 3D sketch to create reference features for a plane. Dan used the cross-section icon and the new plane to set up a cross sectional view. He saved the view as a custom view and then exited the cross section mode.

He added some annotations to surfaces of the model (datum reference and surface finish call outs). He also used File>Properties to add custom properties to the file (material, part number, date drawn, etc.)

Using his favorite drawing template, Dan then created a mechanical drawing of the cam plate. He used the view pallet to drop appropriate view onto the drawing, including his cross sectional view. He pointed out that the title block had updated with the file properties that he had set up in



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the part model. He then inserted the dimensions from the model. The result was a finished mechanical drawing with just a few clicks.

Dan closed with a quick demonstration of smart components. He created a “training assembly” to add bolts to the cam plate. Once the cam plate “learned” about its smart features (the bolts), he was able to drop the cam plate into new assemblies and have the bolts automatically add themselves and be properly mated.

All of that modeling, in spite of the pauses taken to describe what he was doing, took less than 40 minutes.

We then took a short break to enjoy the sandwiches and beverages provided by WarrenTech.

When we resumed, Vic Vandamme addressed the meeting. Vic is the instructor of CAD technology. His students are eager to serve as interns at companies that use SolidWorks and Vic encouraged us to contact him. The interns could be very useful temporary labor. If you’re interested, send an email to [vvandamm@jeffco.k12.co.us](mailto:vvandamm@jeffco.k12.co.us)

Rick Chin then took the stage and presented a method for organizing the design process. His speech “Designing with the Absurdly Ideal” gave us insight into how he manages development projects. eDrawings is an example of the result of applying this methodology. The full PowerPoint presentation is available for download at <http://www.cosug.com/Downloads.html> so I won’t go into too much detail here.

The essence is a 4 step process.

- 1) Work on problems that matter (to the customer)
- 2) Describe absurdly ideal scenarios (don’t worry about the how, just describe the perfect)
- 3) Generate lots of ideas (take your time and explore – use the ideal as a guide)
- 4) Set criteria for evaluation of the ideas

PTM (problems that matter) is an acronym that anchors the thought process. If the customer feels strongly about the problem, it is likely that they will feel strongly about the solution – *Wow! why didn’t I have this before?* If your product generates that kind of reaction, it will sell itself.

The *absurdly ideal* exercise serves to keep the development team working in the same direction and at the same time avoids unnecessary constraint. If you’re working on an airplane, you might imagine an absurd solution that takes no time to get from one location to another, uses practically no fuel, is super reliable, and absolutely safe if anything goes wrong. With that thought, the next step is to come up with practical ideas that approach the ideal.

Rick suggested that ideas are plentiful, but they may take time to emerge. As you gaze at a starry sky, the longer you look the more stars you see; it takes time for your eyes to adjust to the environment. It may take time for your mind to capture the landscape of solutions to the



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problem you're working on. At this stage, just patiently capture the ideas, don't evaluate them. You want quantity, not necessarily quality.

When it comes time to evaluate the ideas, Rick offered a priority filter.

- 1) Does it solve a PTM?
- 2) Is it highly effective?
- 3) Realistic resources, technology, and schedule?
- 4) Is the effectiveness obvious?
- 5) Is it easy to adopt?
- 6) Is it an unexpected solution?

With diminishing priority from 1 to 6, these criteria help you to sort out the ideas and focus on successful product solutions. With the core ideas identified, the race is on to get to market!

The meeting adjourned at 5:00PM.

The COSUG Board would like to enthusiastically thank Dan Riffell and Rick Chin for making their presentations. The hospitality of WarrenTech is sincerely appreciated. We send our thanks to everyone who attended.

Gerald Davis,  
President



## Minutes for the December 6, 2007 User Group Meeting

The meeting was held at the Warren Tech Career & Technical High School in Lakewood, Colorado. The meeting started at 3PM and there were 27 people in attendance and the featured guests were students at Warren Tech.

Gerald Davis, COSUG President, opened the meeting by welcoming all and announcing the results of the election for officers of the COSUG Board of Directors. Steve Ellenberg and Terry Ziemek (founding officers) had declined to run for reelection due to other time constraints so there were 2 open positions to be filled. The Board would like to thank Steve and Terry for their hard work in helping getting the group to where it is today. The election results show that all incumbent officers were reelected and Kevin Van Liere is now the Presentations Director and Kelly Frazier is the Promotions Director. We welcome Kevin and Kelly and look forward to working with them in the future. Gerald also reminded all the users of SolidWorks World 2008 coming up January 20 thru 23<sup>rd</sup> in San Diego and pointed out the many benefits to those that can attend.

The focus of the meeting was on the Warren Tech students and their work with SolidWorks. Vic Vandamme, the instructor of CAD Technology at Warren Tech, introduced some of his students and explained that in their curriculum the students start out working with AutoCAD and after they have completed all the requirements of that section they are given the chance to move into SolidWorks. He admitted that this provides an incentive for the students to complete the AutoCAD section.

As group projects, the students had built parts from existing drawings, and mated the parts into assemblies. They presented an animation of a steam motor driven flywheel assembly and a model of a radio controlled car. The level of detail they had built into their designs was very impressive. There were several comments from the audience to point out that while the high level of details for such things as tire treads and screw threads is very impressive that in the industrial world much of this detail is left off of working models to help with regeneration speed and memory usage.

A sampling of the question the students had for the user group were concerning:

*Aligning features in assemblies.* Since the piece parts for the motorized flywheel assembly were all built from individual drawings some of the holes were not exactly aligned. Gerald Davis explained the principle of using "in context design" in the assembly to insure features are aligned and stay relative to each other as changes are made.

*How to bring design parametric dimensions into the detail drawings.* Dan Riffell created a simple part and walked through the steps to utilize the dimensions in the drawing.

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*How best to mate 2 spherical feature in an assembly* (for a spherical rod end in a CV joint). Kevin Van Liere suggested creating sketch points and lines in the piece parts and then mate the components using the sketches.

*Whether to use lofted or revolved features to created such items as tapered pins.* Gerald Davis pointed out that the revolved feature typically with help produce smaller file sizes and can make it easier to make design changes.

Other student projects that were presented were concepts for a water filtration system (that could be used in situations such as disaster relief) and model of a guitar body.

I was very impressed with the amount of work the students had put into building their models and how eager they were to learn better modeling techniques.

Snacks were provided and the giveaway items provided by SWUGN were snapped up quickly. We would like to express our appreciation to Warren Tech for hosting the meeting, the time of the students to present their projects and of course the users attending for proving their experience to answer the questions.

The next meeting will be February 7, 2008 at Warren Tech's founders Room.

The meeting adjourned at 5:30 PM.

Steve Ray  
Communications Director

# Colorado SolidWorks User Group



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## **December 6, 2007 COSUG Board of Directors Meeting**

### Attendees:

Gerald Davis  
Steve Ray  
Cathy Ross  
Paul Larsen  
Kelly Frazier  
Kevin Van Liere

A board meeting was held following the December User Group meeting at Warren Tech.

### Business discussed:

The incumbent board members welcomed newly elected Kevin Van Liere and Kelly Frazier to the Board. Steve Ellenberg and Terry Ziemek had declined to run for re-election. The Board thanks them for their service.

Discussed ways to better survey the user group. Kelly recommended passing out surveys at the meetings and have attendees drop them off on their way out of the meeting rather than doing an online survey in the days following the meetings. It was decided to try this for the next meeting in February. Kelly will generate a survey for review.

There was a discussion that last year we tried to plan the agenda for the years 6 meetings at the beginning of the year and that this led to several of the "planned" presentations did not materialize due to many of the presenters not being able to attend for various reasons. It was suggested and agreed on that we will just look forward 2 or 3 meetings at a time for presentations.

The next meeting will be February 7, 2008 at Warren Tech's Founders Room. The agenda will be a overview of SolidWorks 2008 and TBD technical session to be set up by Kevin and Kelly.

The April meeting will be held in Fort Collins. Cathy will help locate a facility and local speakers to present.

The goal for the meetings this next year is to provide at least 1 technical presentation at each meeting and to work to recruit members to present their story of "how they utilize SolidWorks".

The meeting adjourned at 6 PM.