



NEWSUG Meeting Minutes
March 1, 2005, 5:00 p.m.
Fox Valley Technical College, Appleton, Wisconsin

While this report generally covers the meeting events, they have been arranged into a logical sequence and refined with the purpose of making them helpful without necessarily precisely representing the facts as they happened.

25 people attended this meeting.

Click on these links for easy navigation.

[SolidWorks Tips](#)

[Weldments](#)

[Large Assemblies](#)

[Next Meeting](#)

Announcements – Bob Braun

Member Survey

The members took a survey to help us select the topics for future meetings and refine the NEWSUG organization. Here are the topics, with number of positive responses.

- Sheet metal – 8
- Detailing and drafting techniques – 5
- Best modeling practice – 5
- Molding and castings – 4
- Equations and other methods to automate design and imbed intelligence – 4
- Configurations – 3
- Macros and customization – 3
- FEA – 2
- Rapid prototyping – 2
- SolidWorks peripheral hardware – 2
- PDM software – 2
- Design tables – 2
- Weldments – 1
- Surfacing – 1
- Lofting – 1
- CAM software – 1
- Certified SolidWorks Professional test and certification – 1
- Migration from Autocad – 1
- Large assemblies – 1
- File management – 0
- Libraries – 0
- Industrial design – 0
- Hardware requirements – 0
- Benchmarking performance – 0
- Other third party software – 0
- SW version updates announcements – 0

For additional topics, one person mentioned COSMOSMotion

These suggestions for change were made:

- Let users come up and give tips that have saved them work.
- “More board members or users to share the load.”

In response to the question about making the meeting longer or shorter, one person said “longer”, one person said “shorter” and six people indicated that the length should not change.

Through the survey and discussions immediately around the survey, four members volunteered to present topics. A VAR also volunteered to let their technical people present.

The NEWSUG Board will give strong consideration to the recommendations that came out of the survey.

[Return to the top](#)

SolidWorks Tips – Moshe Saraf

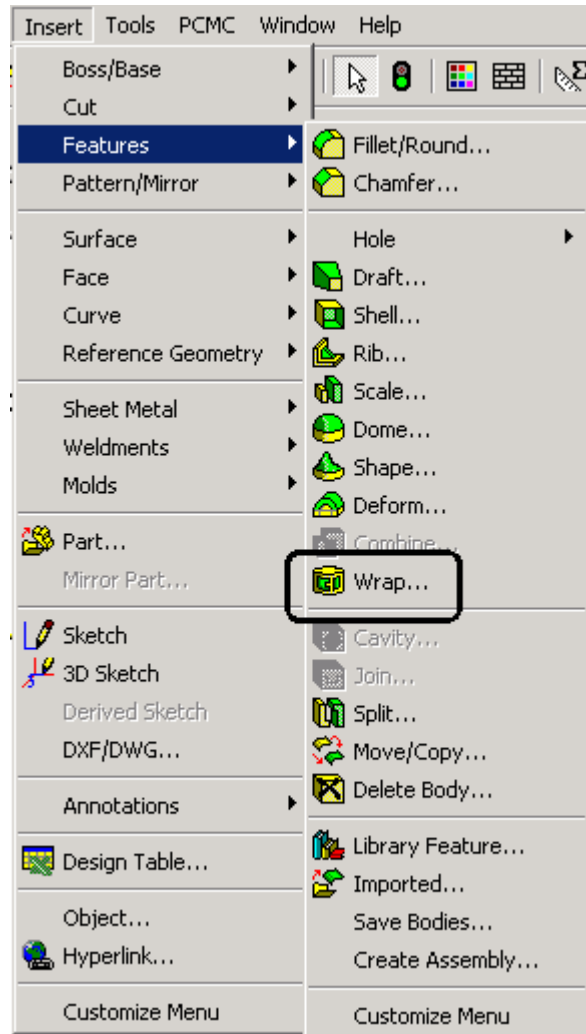
How to split an irregular face

This tip is helpful to split an irregular, or regular, face.

1. Create a continuous irregular face.
2. Create a closed sketch on a plane that will project on to the curve in step one.
3. Follow these command steps:



- a. Use the *Insert|Features|Wrap* menu selection sequence.



- b. On the dialog box, select *Scribe*. In the other blank boxes, select the surface and the sketch you made in step 2.
- c. Select the OK check mark.

Note these requirements:

- The irregular face must be one continuous face. It may be a spline but it cannot be a series of tangent lines and arcs.
- The sketch must fully wrap on to the surface. If it wraps off the edge, it will fail.

If you use this feature regularly, you can drag the icon to a menu that you usually use. It is not a part of one of the existing standard toolbars.

[Click here](#) for a sample part that demonstrates both successful and unsuccessful attempts to split an irregular curve.

Rotating around an edge or vertex

In order to rotate a part or assembly around an edge or vertex, select the *Rotate View* icon from the tool bar. With the icon for a cursor, wake the edge or vortex that you want to rotate the model around and click to select. Then, press the left button and drag.



Note that this works reliably with the *Rotate View* icon. Other methods to rotate the part, like using the middle mouse button, may or may not work, depending on your software configuration and version.

Auto size planes


When you have a model with a plane that is too large or small for the model or you cannot easily find it, right click on the plane in the tree and select *Autosize*. The plane will expand or contract to match the model size.

[Return to top](#)

Weldments – William Davis, from Impact Engineering

William Davis presented the portion of his SolidWorks World 2005 presentation that covered weldments. Link to the presentation by clicking [here](#).

In a simplified summary of the presentation, SolidWorks weldments permit the user to efficiently create welded assemblies. Here are the general steps.

1. Open a part file.
2. Make it a weldment by clicking on the weldment icon. 
3. Create a sketch that outlines the shape of the first set of structural members. Here are some sketch limitations:
 - a. All sketch lines must be connected.
 - b. Only two sketch lines can meet at one point.




4. Select the structural member icon to add structural members that follow the sketch.

With this dialog box open, select the structural shape that you want to use, select the relationship between the shape sketch and the sketch created in step 3 and select how the corners between structural members are made. Structural shapes can also be rotated.

5. Create additional sketches and fill in the structural members, like was done in steps 3 and 4.



6. End caps can be added by selecting the end cap icon and selecting the ends you want capped. This feature lets you control the end cap thickness, corner chamfer size and end cap offset from the edge of the structural member, typically a tube.

7. Gussets can be added nearly as easily as end caps by clicking .

8. Nonstandard parts can be added by extruding them.

Other features that were presented were:

- Non-straight sketches can be used to create bent structural elements. These can be trimmed and stretched to fit the mating parts.
- Weldments automatically create derived configurations for the as welded and machined parts.
- The welded components can easily be grouped into drawing cut lists.
- Weld beads can be shown.
- You can create your own structural sketches. When you do this, be sure to add an abundance of sketch points so that you have many options for locating them.
- When you use a structural sketch in a weldment, you are using only a copy of the original. As a result, within limitations you can substantially modify the sketch in your weldment to get the design you need without changing the original sketch and other weldments that use it.
- Weldment functionality can be used for parallel thought processes that have nothing to do with weldments, like extrusion assemblies.

[Return to top](#)

Large Assemblies – Bob Braun

This presentation was a summary of two presentations given at SolidWorks World 2005. The nature of this presentation is that it is lists of things to check. Not all items are valid for all users. You can see the presentation by checking [here](#).

Next Meeting

The next NEWSUG meeting is scheduled for May 31, 2005.