



**.NEWSUG Meeting Minutes
November 13, 2007, 5:00 p.m.
Fox Valley Technical College, Appleton, Wisconsin**

While this report generally covers the meeting events, those events have been arranged into a logical sequence and refined with the purpose of making them helpful rather than precisely representing the demonstrations as they happened.

26 people attended this meeting.

Click on these links for easy navigation:

[Beam and Bearing Calculators](#)
[Equations and Linked Values](#)
[Molding and Casting Design](#)
[Next meeting](#)

Announcements – Bob Braun

We invite more members to join the Board. It is an excellent experience.

More presenters are needed. You don't have to be a professional speaker. If you know about subjects many of us seldom encounter, like animation, you can present a short demo that would be appreciated.

Survey Results:

- No significant desire expressed to change meeting format (day of week, start time, length, location).
- Preferred topics
 - Large Assemblies
 - CAD Libraries Creation & Manage
 - Sheetmetal
 - Weldments
 - Advanced Mates
 - Configurations & Design Tables
 - Macros/API
 - Most Efficient Assembly Practices
 - Moldings and Castings
 - Piping, Plumbing & Hoses
 - Best Modelling Practice
 - Design in Place
 - Motion Simulation
 - Setting up for Performance
 - Sharing Extrusions & Tube Shapes
 - Working with Downloaded Models

- We agreed that we should not be worried about subject repetition since the software is always changing, and several approaches may exist.

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Beam and Bearing Calculators – Bob Braun, Valley Mechanical Solutions, LLC

Toolbox Toolbar:

Both Beam Calculator and Bearing calculator (and other helpful utilities) require that the SolidWorks Toolbox be activated. SolidWorks Toolbox has several engineering tools including:

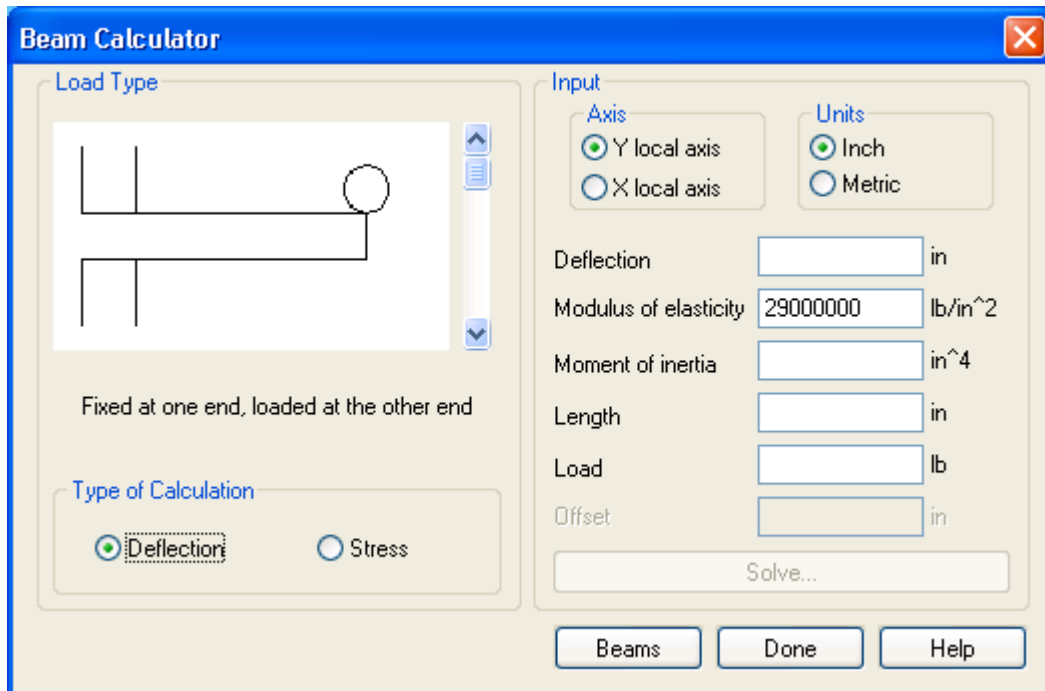
- Beam calculator for determining the stress and deflection of a beam
- Bearing calculator for determining the load capacity and life of a bearing
- Grooves to add standard O ring and snap ring grooves to a cylindrical part
- Structural steel cross-sections to add as a sketch to a part and to find section properties

To activate SolidWorks Toolbox:

1. Click Tools, Add-Ins.
2. Select SolidWorks Toolbox from the list of installed, compatible software products.

Beam Calculator:

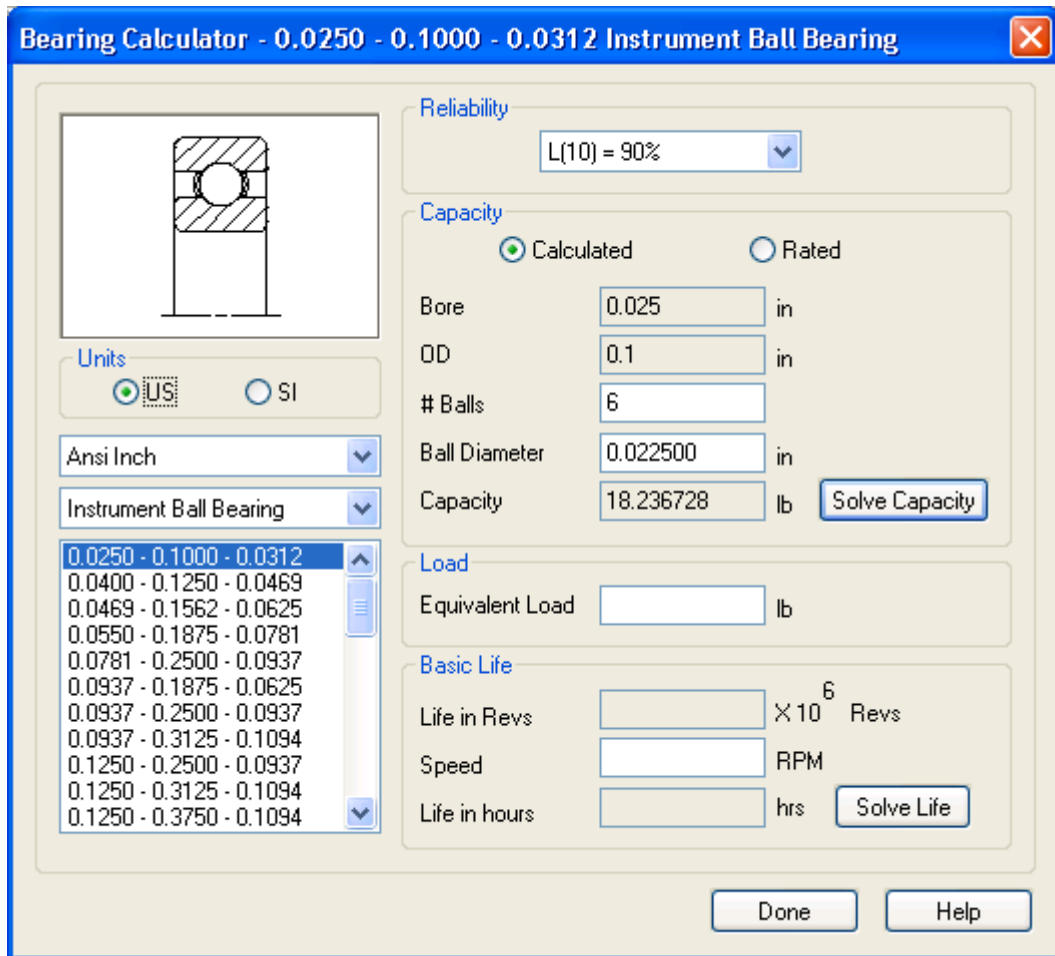
- Start by selecting
 - Toolbox | Beam Calculator . . .
 - Icon from Toolbox palette
- Select the load type from the six illustrated in the scroll down box
- Select either of the calculation types
 - Deflection
 - Stress



- Select Input
 - Y local axis (most common)
 - X local axis
- Select units
- Enter the beam modulus of elasticity, moment of inertia and length.
 - Enter beam values by hand
 - Use the “Beams” button to select standard beams
- Enter either load or deflection while leaving the other one blank.
- Click on the “Solve” button

Bearing Calculator:

- Start by selecting
 - Toolbox | Bearing Calculator
 - Icon in Toolbox palette

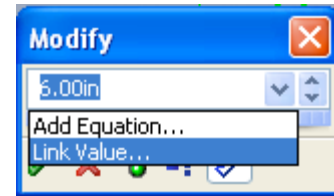


- Select units
 - US
 - SI
- Select bearing type and series
- Enter Reliability from drop down
- Select specific bearing
 - From drop down list
 - Enter bearing properties in text boxes
- Enter equivalent load – up to user to determine equivalent load
- Enter speed (rpm)
- “Solve Life” to get bearing life expressed in both hours and millions of revolutions

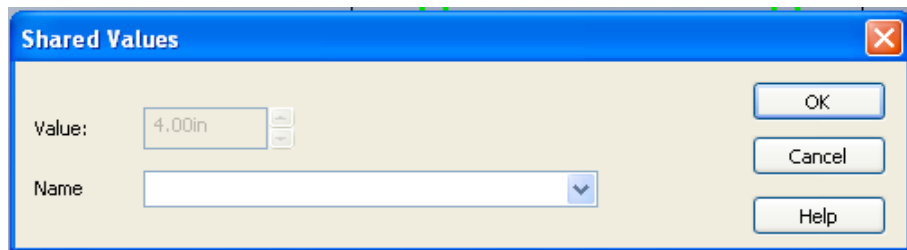
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Equations and Linked Values – Bob Braun, Valley Mechanical Solutions, LLC

- Create **Linked values** when:
 - Entering initial dimension value
 - Editing value



- Select from value drop down
 - If you want to match an existing name, select from drop down
 - If you want to create a new name, select from name drop down.



- If a feature does not have a traditional dimension, like an extruded height or feature array, double click on the feature to show the dimension.
- Linked values will match. Any linked dimension will drive all others.

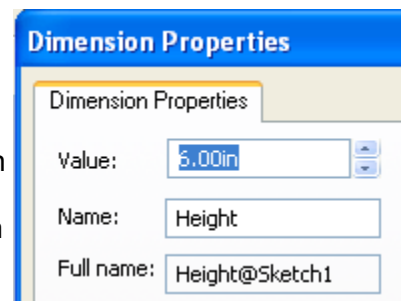


If a dimension is linked, it appears in the graphics area with an infinity sign next to it.

Equations do many of the same things as linked values but are more powerful and not as convenient.

Before using equations (and design tables) it is a good idea to name your dimensions, sketches and features. This will make your work much easier to use because it will help you to know what the formulas mean and what the dimensions are controlling.

- Name dimensions by right clicking on them and selecting “Properties” from the drop down. You can get the same dialog box by selecting “More Properties” near the bottom of the dimension properties box. Edit the name to the unique descriptive name that you want.
- Dimensions that are not part of a sketch, like an extrusion depth or the count on a rectangular feature array can be given functional names. Double click on the feature in the Feature Manager to display the related dimensions. Right click on the dimension and select “Properties” from



the drop down menu.

- Edit sketch names and feature names from the default by either doing a slow double click on them or by selecting them and pushing the F2 key.

While SolidWorks is flexible about naming format, these names can easily be used by other applications like Excel or a database that is not so accommodating. These guidelines will keep you out of most trouble in this area:

- Use only letters, numbers, dashes and underscore characters. Do not use all numbers and avoid starting a name with a number.
- Avoid spaces between words. Use a combination of upper case and lower case letters with an upper case letter at the start of each word or put an underscore between words.
- Avoid punctuation, especially commas, slashes and quotation marks. They are often used as field delimiters in databases.

Start making equations with these steps

- Click on Equations icon
 - Use to establish key variables
- Click on Dimension drop down
 - Use to drive dimension values
- Equation units will be default units plus selected angular units
- Use programming language format
 - Assigned variable name equals assignment equation
 - Enter dimension names by
 - Writing dimension names in quotes
 - Click on dimensions
 - Use available functions Help | Equations | Functions
- Manage equation activities by making them active or not with configurations

With Equations and Linked Values, we can:

- Create Variables of product dimensions to drive machine component size and array count – like the number of bearings in a bowed roll
- Hold size ratio for key dimensions – like roll diameter

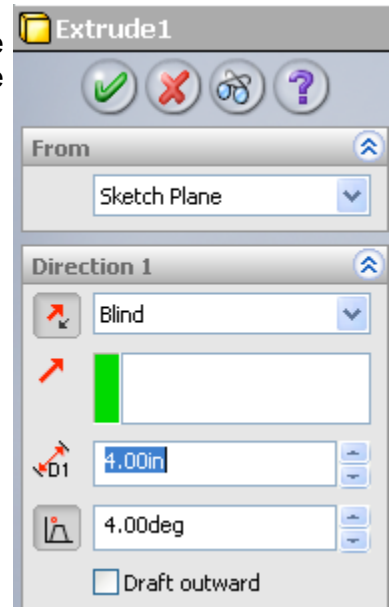
As an alternate to SolidWorks equations, we can also do calculations in the spreadsheet used as a design table. In the area more than one blank column to the right of the working area and more than one blank row below the last entry any calculation work can be done without disrupting the design table. Since many people are comfortable with Excel calculations, this can be an easier way to work with calculations than SolidWorks Equations.

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Molding and Casting Design, John VanEngen, CATI

The challenges of molding and casting design focus on insuring that we have adequate draft and fillets. John demonstrated some SolidWorks tools that many users are not familiar with to do these tasks efficiently. John also demonstrated MoldflowXpress, which is not available after SW 2007. He also showed us how to make a part as a negative of another part, which has broader application than just molding and casting.

Features like the extrude command usually have an option to include draft in the extruded feature. In the illustration of the extrude dialog box, you can see the option in the bottom to add draft by clicking on the command button. You specify the draft angle. You also have the option to reverse the draft direction by selecting the “Draft outward” check box.



DraftXpert

The DraftXpert can:

- Create multiple drafts
- Do draft analysis
- Edit drafts
- Automatically invoke the **FeatureXpert** to resolve draft features that do not initially go into the model easily



Start the DraftXpert by clicking on the Draft icon in the features toolbar or click on Insert | Features | Draft. Click on the DraftXpert tab at the top of the property manager. Select the Add tab.

To draft features, follow these steps:

1. Click in the neutral plane box and select the plane that you want the draft to come from.
2. Select the features that you want to draft.
3. Select “Apply”

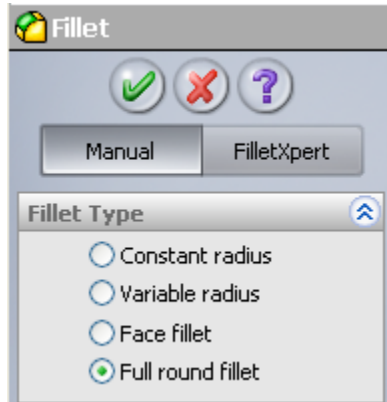
To do a draft analysis, select the “Auto paint” check box. This will color code the surfaces to indicate how well they meet the draft requirements selected at the top. When you move your cursor over a surface, text will pop up to indicate the draft in that area.

You can add draft and check draft repeatedly with new draft angles and neutral axis.

Go to the Change tab to change the draft that you have already entered.

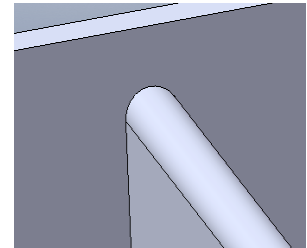
Full Round Fillet

When you have a rib or other feature that needs a fillet at the top, use a *Full Round Fillet* in one step.



After you model the square rib top, select the fillet tool from the tool bar.

In the property manager, select *Full round fillet*. SolidWorks is now looking for three surfaces to join with a tangent fillet to all three and consuming the middle surface. Select the three surfaces that you want the fillet to be tangent to. Select the OK check box and you will have a complete fillet.



FilletXpert

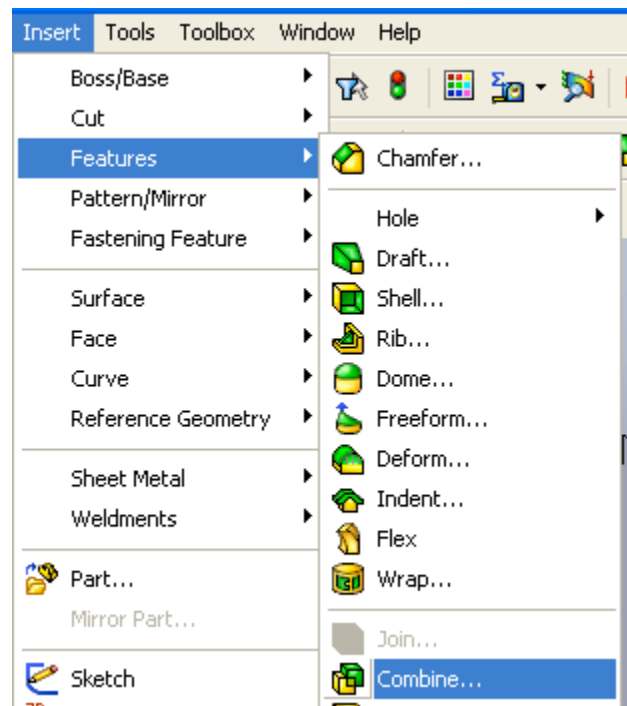
Access the FilletXpert with the fillet tool. Click on the FilletXpert at the top of the properties manager.

The FilletXpert lets you create multiple fillets without selecting the fillet command repeatedly and it automatically reorders fillets. When you select the *Change* tab, you can efficiently edit the fillets that you have already made.

Mold Making

While mold making includes many details that are outside the scope of the presentation, John showed us a basic strategy to make the cavity. This can be done with in any application where you want to model the negative of a part and create the positive part from it.

1. Model the part negative, for example the molded part.
2. Model the mold feature so that it includes the molded part in the same part model. When you extrude the profile, be sure that the *Merge result* box is not checked.
3. Under the *Insert* menu, select *Features* and then *Combine*.
4. In the properties manager, select *Subtract*.
5. For the *Main Body*, select the mold or the part that you want remaining. The most reliable way to do this is to select in the feature manager.
6. For the *Bodies to Subtract*, select the molded part or part that is the negative of what you want.



7. Select *Show Preview* to see if you are getting what you want before you select the *OK* check mark. If you have created a multi-body part, you will have a choice about what bodies to keep, including all of them.

Next Meeting:

The next meeting will be February 5, 2008 at FVTC.
The presentation topics will be: advanced mates, CAD Libraries,
and setting up SolidWorks at your company.

