



NEWSUG Meeting Minutes
February 7, 2006, 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.

28 people attended this meeting.

Click on these links for easy navigation.

[NEWSUG Survey](#)

[Efficient Slots](#)

[SolidWorks Simulator](#)

[Flexible Assemblies](#)

[SolidWorks Animator](#)

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Announcements – Bob Braun

We voted to accept a revised NEWSUG charter. It can be viewed on our web site. Revisions primarily were designed to bring the charter in line with how NEWSUG operates and to give us more flexibility on the number of board members.

A SolidWorks blog can be found at www.solidworkscommunity.blogspot.com.

We explored interest in starting a sister NEWSUG group in Green Bay.

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NEWSUG Survey

We surveyed took a member survey to see how NEWSUG can best meet member needs.

The survey was circulated to the members before the meeting with one response. The person who responded to the electronic survey was also at the meeting. 23 total surveys were handed in.

We asked a range of questions about the meeting format, like time and location. The vast majority of the members did not respond or provided positive comments about the format. Here are the change suggestions:

- About 13 percent of our responders would like to see a meeting in Green Bay. . No one volunteered to be a part of the leadership team.
- About 17 percent of our responders would like to see us start a little later. There was strong correlation between those who wanted to see a Green Bay meeting and those who wanted to start a little later.

We asked the members to rank first through fifth their preferences for future presentation topics. The results were weighted based on how strongly they were wanted with the below results. Dates of recent presentations on these topics are included in braces.

High Popularity

- Most efficient practices for assemblies
- Sheet metal [September 6, 2005]
- Equations, linked features [November 15, 2005]
- Configurations/Design tables [November 15, 2005]
- Lofting and sweeps
- Surfacing and shells
- SolidWorks tips and ticks

Medium Popularity

- Motion Simulation (such as Simulator and Animator) [February 7, 2006]
- Weldments [March 1, 2005]
- Advanced mates
- Large assemblies [March 1, 2005]
- Macros, API [September 1, 2004]
- Assembly component patterns
- Interference detection
- COSMOSExpress (FEA)
- Molding and castings
- Best drawing practice [May 24, 2005]
- Best modeling practice
- CAD libraries, creation and management
- SW add-on software such as Animator, PhotoWorks, CosmosFloWorks

Low or No Interest

- Imbedded notes and documents
- SW Partner software such as CAMWorks, SheetmetalWorks, CircuitWorks
- 2D to 3D: Techniques for importing into SolidWorks Sketches
- Creating parts from within an assembly, Design-in-place
- SolidWorks Explorer
- Top-down vs. bottom-up design
- CAM software
- Fixing bad mates
- Rapid prototyping [December 7, 2004]
- Certified SolidWorks Professional test and certification

The NEWSUG Board of Directors plans to use these survey results to choose topics for future presentations.

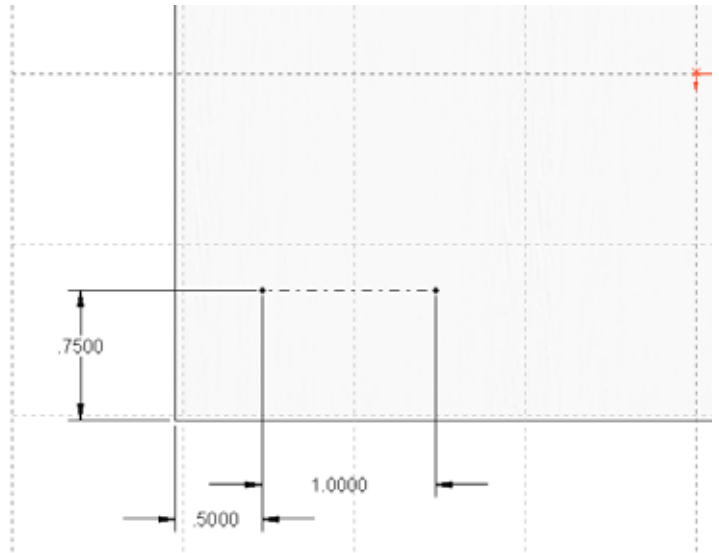
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SolidWorks Tip, Efficient Slots – Dan Sheber, KC

An efficient way to make slots is to sketch them with the offset command and extrude them. Here is the procedure:

1. On your part, create a sketch of a center line that will represent the centerline of your slot. In the illustration, a slot 1 inch long has been illustrated with a center .75 inches from the edge and .5 inches from the side.

Note that you can create multiple slots simultaneously.



2. Select the center line for the slot. If you do not select the center line for at least one slot, this will not work. If you have multiple slots that you want to create, you need only select one at this time but you can also select more than one.

3. Click on the offset icon, which looks like this:
 - a. Tools
 - b. Sketch Tools
 - c. Offset Entities

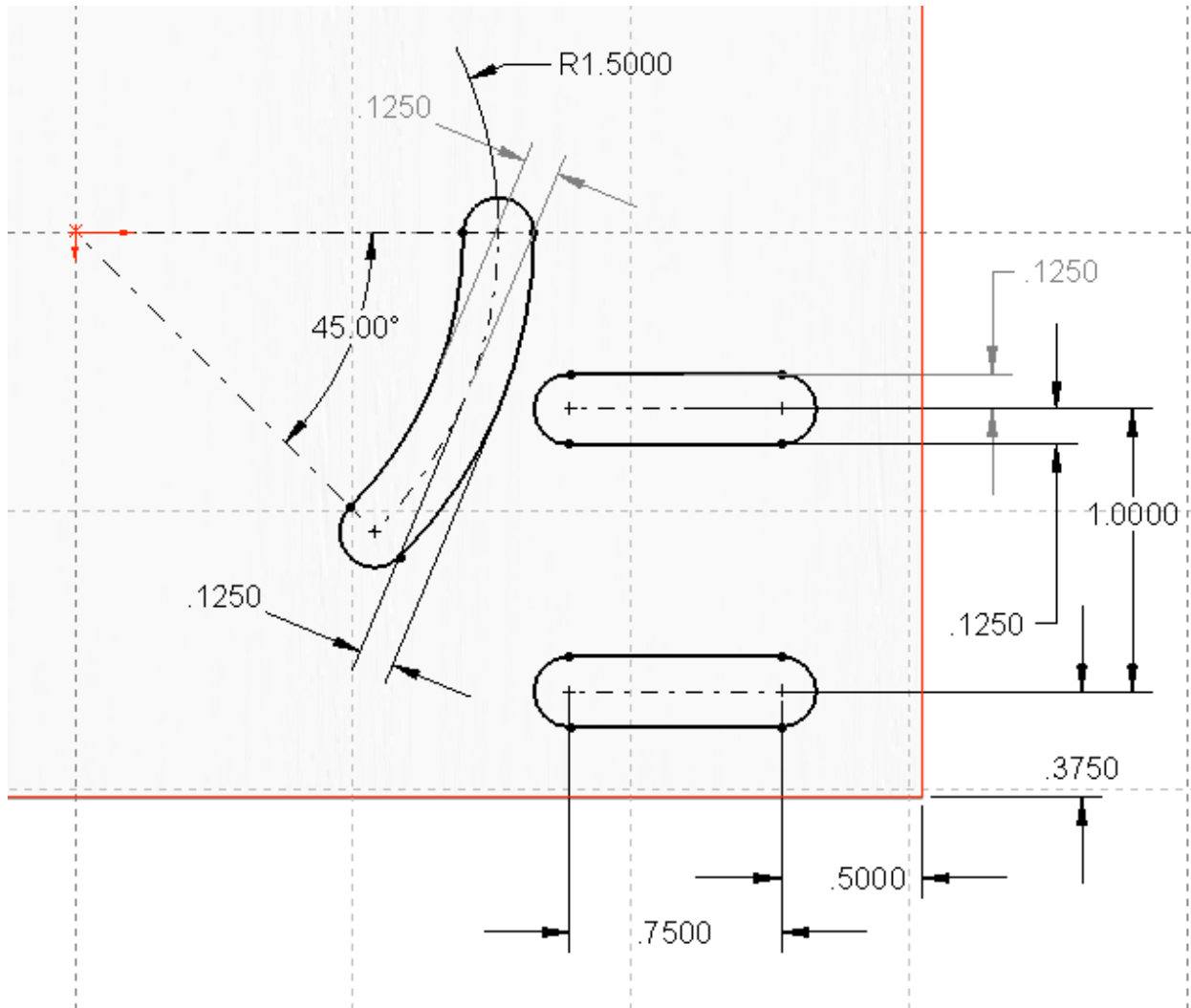


4. In the dialog box, set your radial offset (half slot width) in the dialog box. Be sure *Bi-directional* and *Cap ends* are checked. The capped end type should be *Arcs*.

If you have multiple slots that you want to make and you have not previously selected them, select them now.

5. Select the OK check box.
6. Do an extruded cut to finish the slot.

Note that you can create arc shaped slots as easily as straight slots. Also, if you have slots that are a combination of arcs and straight segments, they can be created in the same way provided that you have the *Select chain* box checked in the *Offset Entities* dialog box. In the below illustration, a pair of slots were created in one operation and an arc shape slot was created in another.



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SolidWorks Simulator – Carol Beard, Peer Technical

Note:

To keep parts visible when in motion:









To save resources, SW2005 makes parts cube up. To show parts in motion, you must turn this off. Tools/Options/Performance/drag the “Level of Detail” slider to off.

First, create any assembly with partially constrained parts that are free to move in some direction.

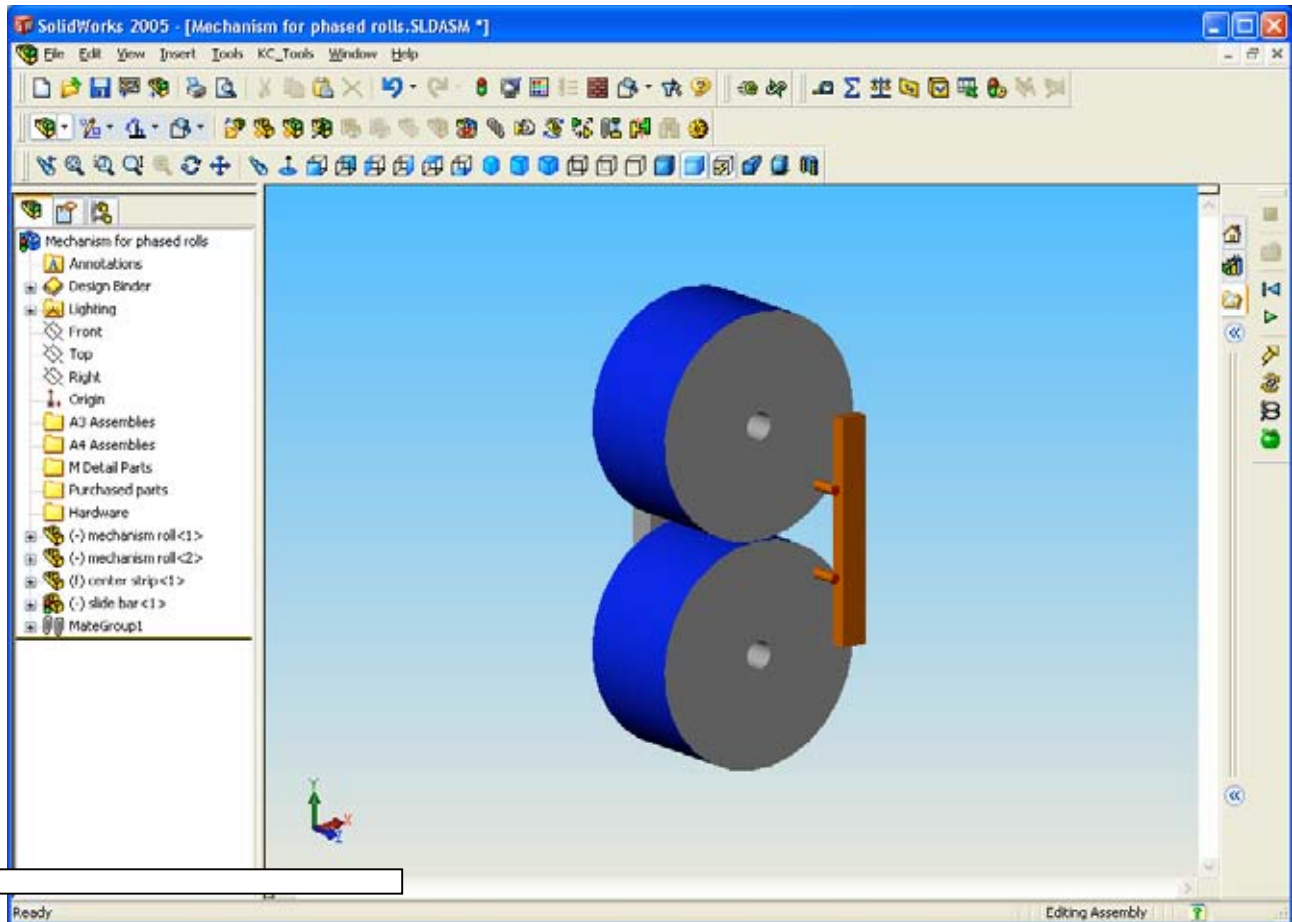
Right-click on the toolbar to open SIMULATION toolbar. (Or click on the gear to the far right of the *mating clip*.)

Simulation Toolbar

The **Simulation** toolbar contains the tools for using **Physical Simulation**.

	<u>Stop Record or Playback</u>
	<u>Calculate Simulation</u>
	<u>Reset Components</u>
	<u>Replay Simulation</u>
	<u>Linear Motor</u>
	<u>Rotary Motor</u>
	<u>Linear Spring</u>
	<u>Gravity</u>

In the following example, only the “center strip” maintaining a relationship between the two rolls is fixed. The rolls are free to spin. The vertical bar is parallel to the right plane, and coincident to the surfaces of the pins on the rolls.

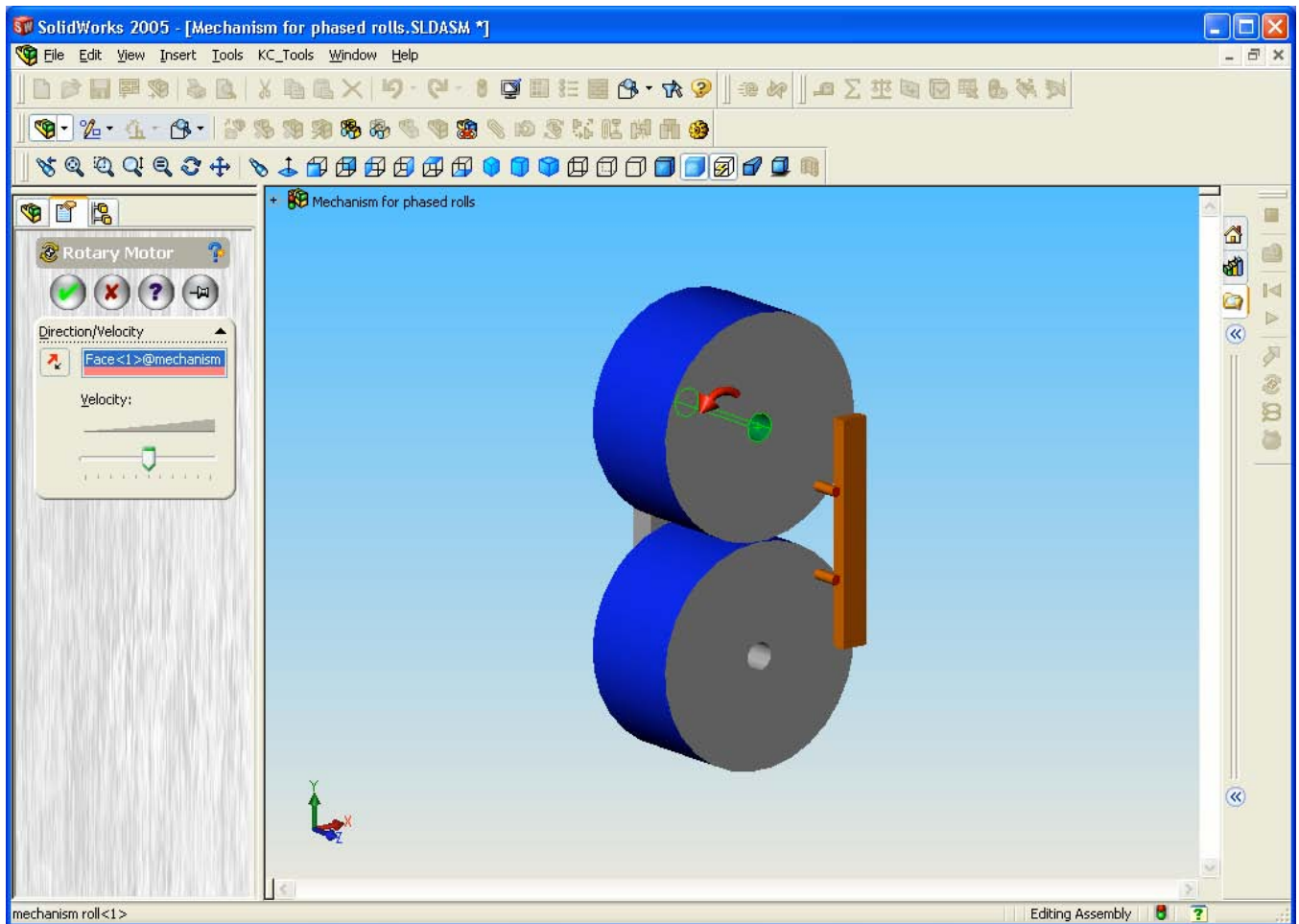


Click the gear icon to the right of the Mate clip to open the Simulator toolbar. Select Rotary Motor.



The box for Direction/Velocity indicates that you need to select the axis for rotation.

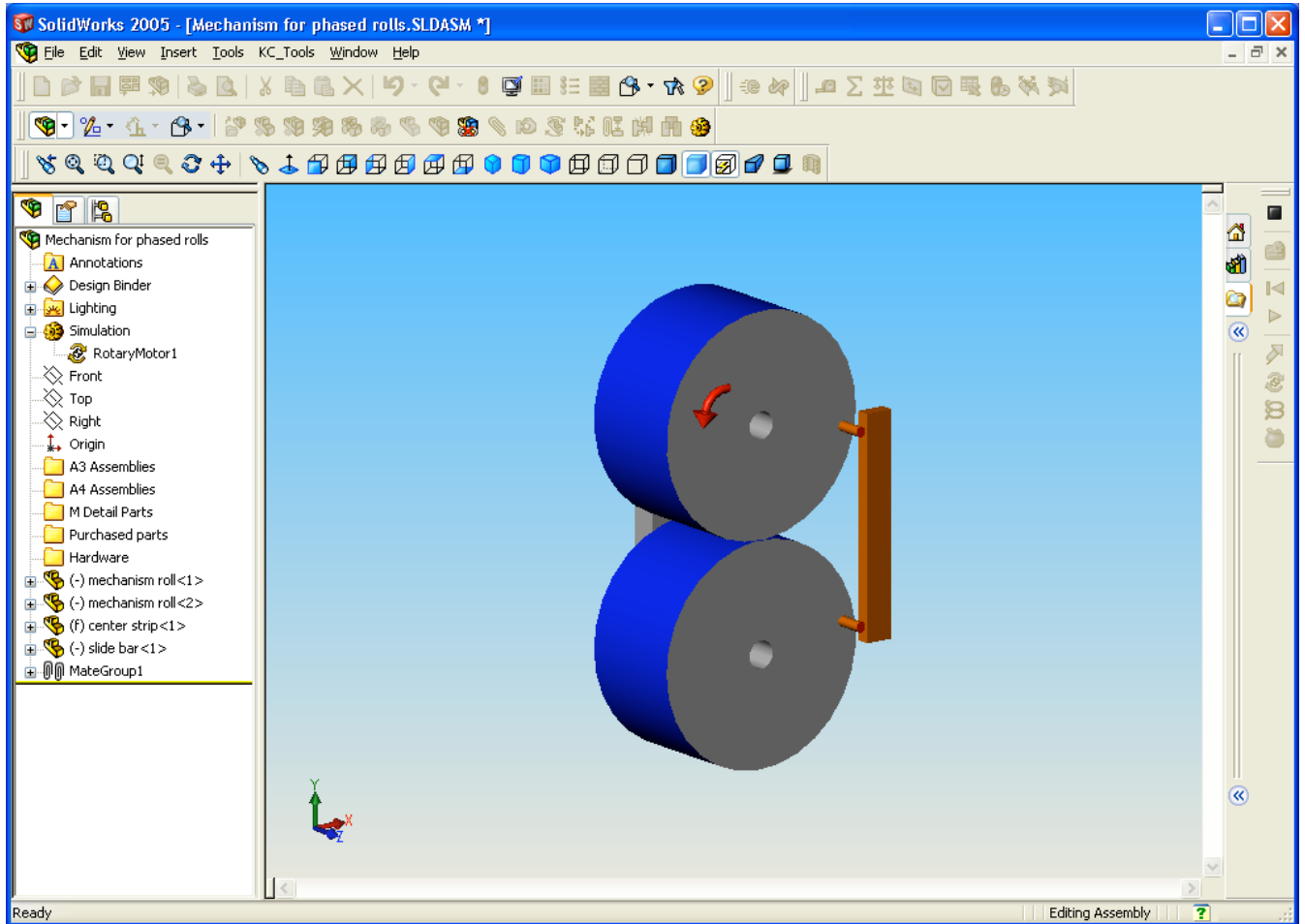
Select the core diameter of a roll.



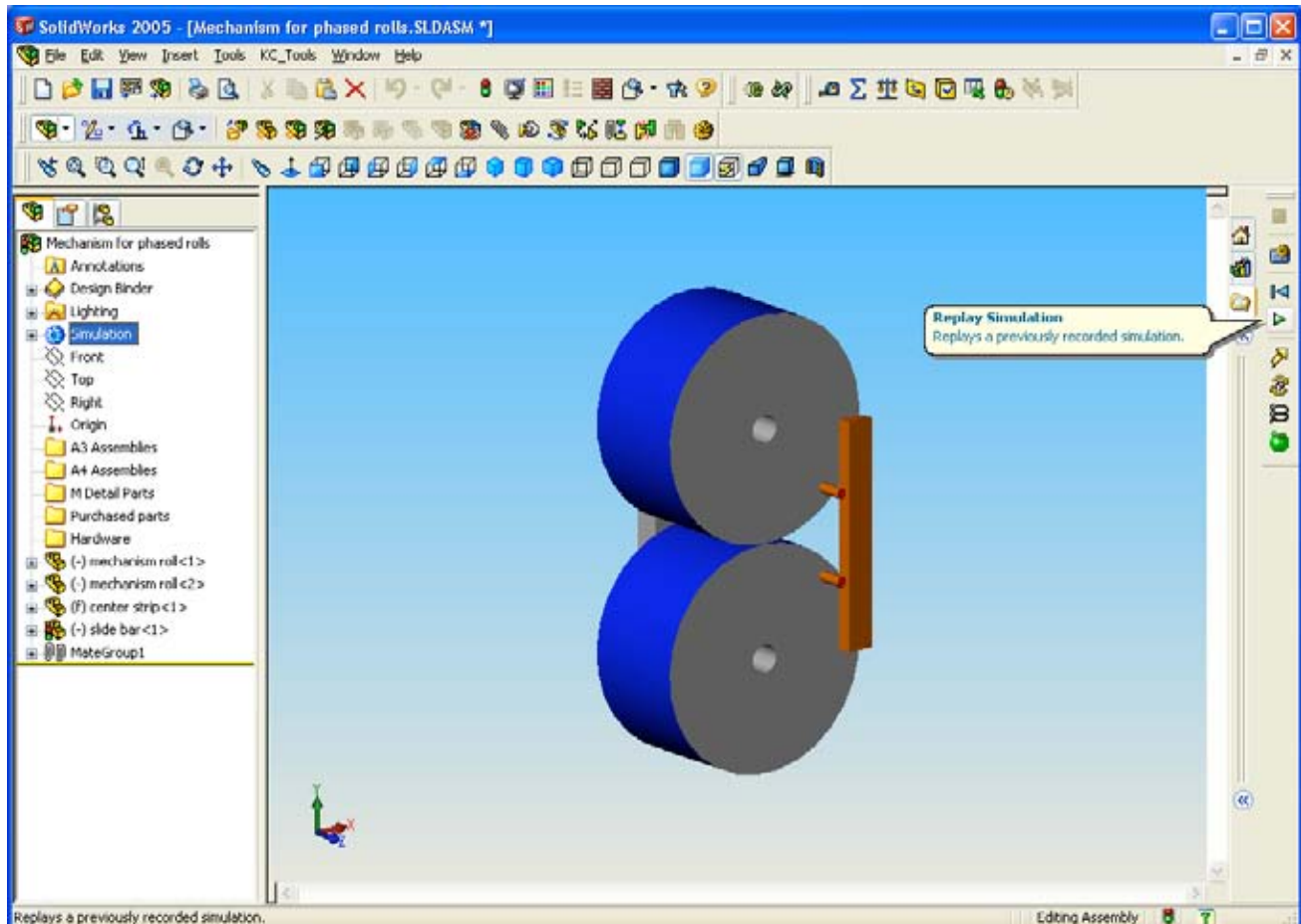
To record, click Calculate Simulation



And record:



Now you can replay the simulation anytime.



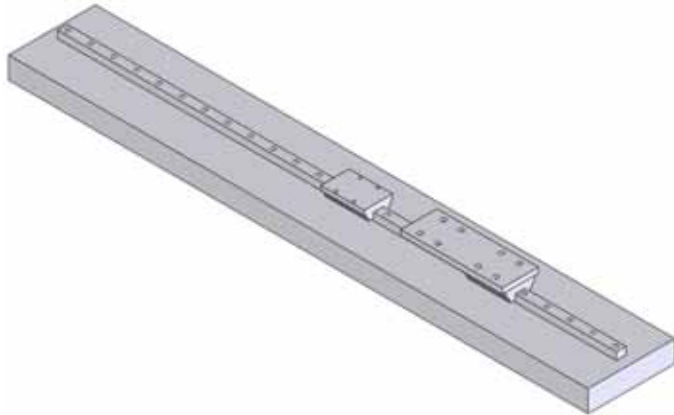
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Flexible Assemblies – Bob Braun, C3

The general idea of a flexible assembly is that you fix degrees of freedom for an assembly at a higher level of assembly than they are created at. This was illustrated with a linear bearing system.



The linear bearing cars were mated to be centered over the rail and to the assembled height but their position along the rail were not fixed.



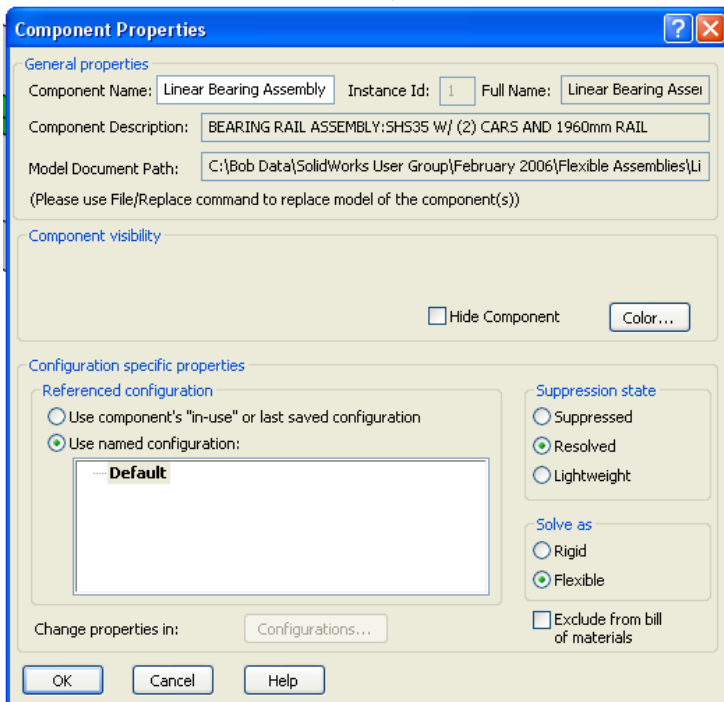
The linear bearing assembly was put into a higher level of assembly with a base and a load block. The linear rail was mated to the base and the attachment was fixed to one of the bearing cars. With the default settings, the bearing cars are fixed in the position they happened to be in when the linear bearing assembly was last touched.

In the feature tree, right click on the linear bearing assembly and select *Properties* from the drop down list. In the lower right corner, select the option button under

Solve as for Flexible. When you select the *OK* command button, the linear bearing cars will be

free to move. They can be mated to the load block and the load block and bearing cars can be positioned by an outside actuator, like a pneumatic cylinder.

Here are some things to note about this exercise:



- In the *Component Properties* dialog box, you have the option to select *Exclude from bill of materials*. In most cases, you do not want to do this here. This totally excludes the linear bearing from the next level of assembly. Usually, you want to do is go back to the linear bearing assembly and under the *Configuration Properties* select the box *Don't show child*

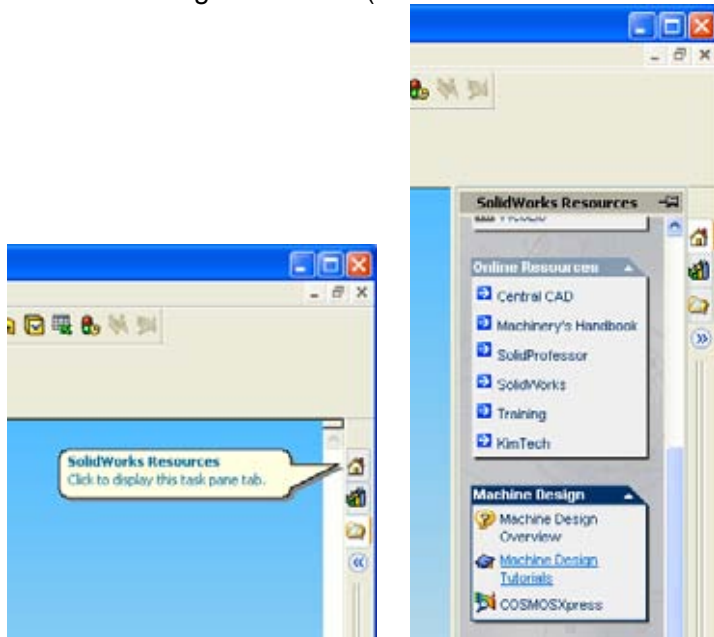
components in BOM when used as sub-assembly. This will let you carry forward the linear bearing as a single part number in this top level assembly and not separate items for the linear rail and the bearing cars.

- Flexible assemblies do not come free. If you have a large assembly with many components, the added computation time it takes to resolve flexible assemblies may be more than you can wait for or your computer will support.
- Flexible assemblies solve many design problems where you need to illustrate how things go together or how they work. They can bring our models a step closer to simulating reality.

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SolidWorks Animator – Matt Luedke, Graphic Systems

Matt provided an in-depth and elaborate demonstration that can not be duplicated here. To access a great step-by-step tutorial on Animator, click on “SolidWorks Resources”, then “Machine Design Tutorials” (often labeled “On-Line Tutorials”).



Online Tutorial

Show Back Forward Home Print

Tutorials for Machine Designers

These tutorials discuss some basic SolidWorks concepts and terminology. Read the [Conventions](#) information.

If you are new to the SolidWorks software, complete the 30 Minute Lesson first. Lessons 1-3 then cover basic part creation, assembly operations, and drawings. All other tutorials can be completed in any order.

30 Minute Lesson

Lesson 1 - Parts	Import/Export
Lesson 2 - Assemblies	Multibody Parts
Lesson 3 - Drawings	Pattern Features
AutoCAD and SolidWorks	PDMWorks
3D Sketching	PhotoWorks
Advanced Design	Revolves and Sweeps
Advanced Drawings	Sheet Metal
Assembly Mates	SolidWorks Animator
COSMOSXpress	SolidWorks API
Design Tables	SolidWorks Utilities
eDrawings	Toolbox
FeatureWorks	Weldments
Fillets	

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Animate an assembly by displaying motion, changes in viewpoint, and display mode, as well as exploding/collapsing the component.

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Next Meeting

The next meeting will be May 2, 2006 at FVTC in room F108. The main presentation topics will be Lofting and Sweeps and extensive sessions on tips and tricks.