PTC Technical Specialists E-Newsletter

Date: August 2009

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B) PTC's Legacy Data Migration (LDM) Management

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B) Keeping Your Customers Up-to-date is Only A Click Away

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PTC Product Focus

Sketcher Enhancements in Wildfire 5.0

There are several new, exciting enhancements in Wildfire 5 sketching. Here is a detailed look at those improvements.

Sketcher Constraints Improvements

Sketcher constraints and workflows are more flexible. There are shortcut menus, object-action workflow, and a consolidated user interface. A new constraint type, equal dimension, is introduced.

Benefits and Description

You can use either of these workflows for creating constraints:

- Object-action—Select the entities, right-click, and select available constraints from the shortcut menu. Available constraints depend on the entities selected.
- Action-object—Select the constraints first and then select the entity.

While sketching and dragging sketched entities, new key sequences and mouse clicks speed up constraint creation. During sketching, use consecutive right-clicks to toggle through locking, disabling, or enabling the constraint (Figure 1)
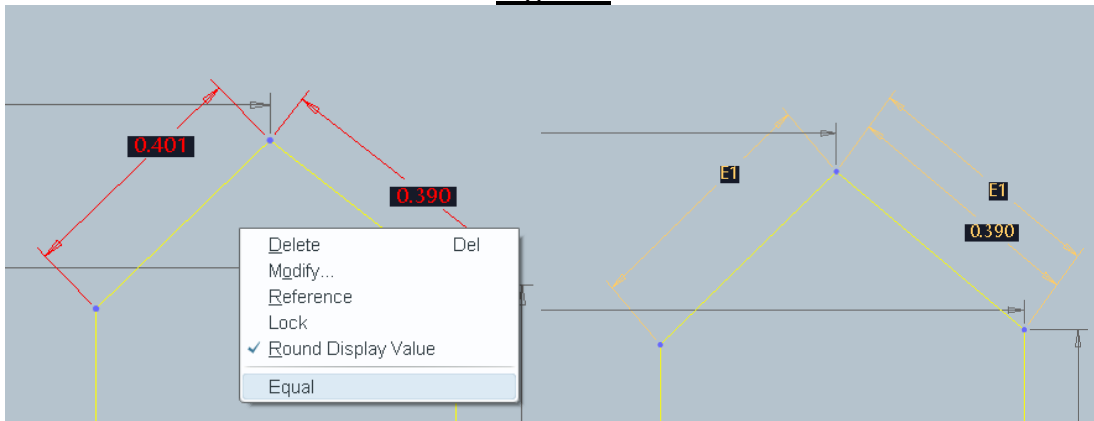
Figure 1
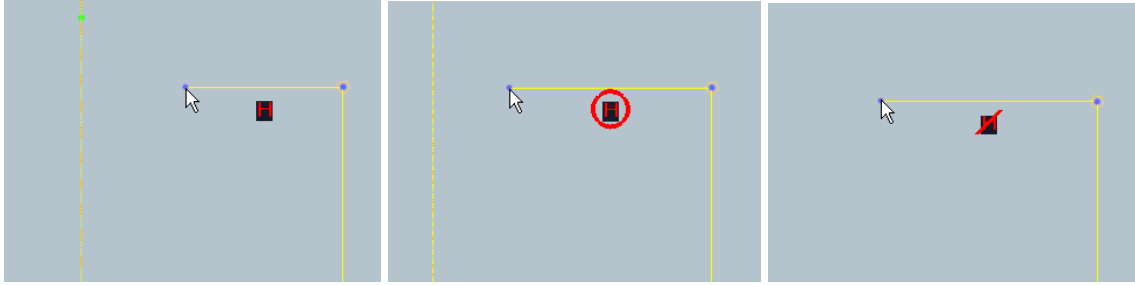
Press and hold down SHIFT to disable all constraints. During dragging, press and hold down SHIFT to enable the offering and accept constraints.

You can apply the new constraint type, equal dimension, to any dimensions of the same type. Applied in the same way as equal length or diameters, an E constraint (E for equal dimension) appears (figure 2).

Figure 2

Sketcher Creation Tools Enhancements

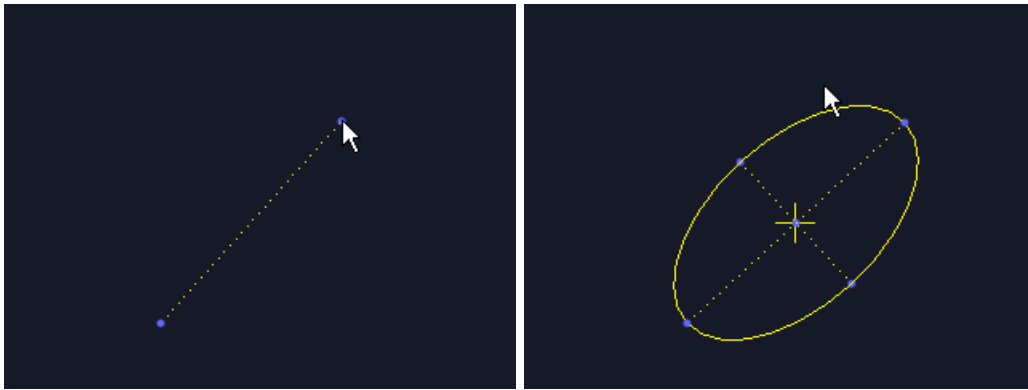You can create new entity types in Sketcher, providing flexibility and speed in feature creation.

Benefits and Description

You can create the following Sketcher entity types:

• Slanted ellipse created by either establishing the major axis endpoints or by defining the center and one end of the major axis. (Figure 3)
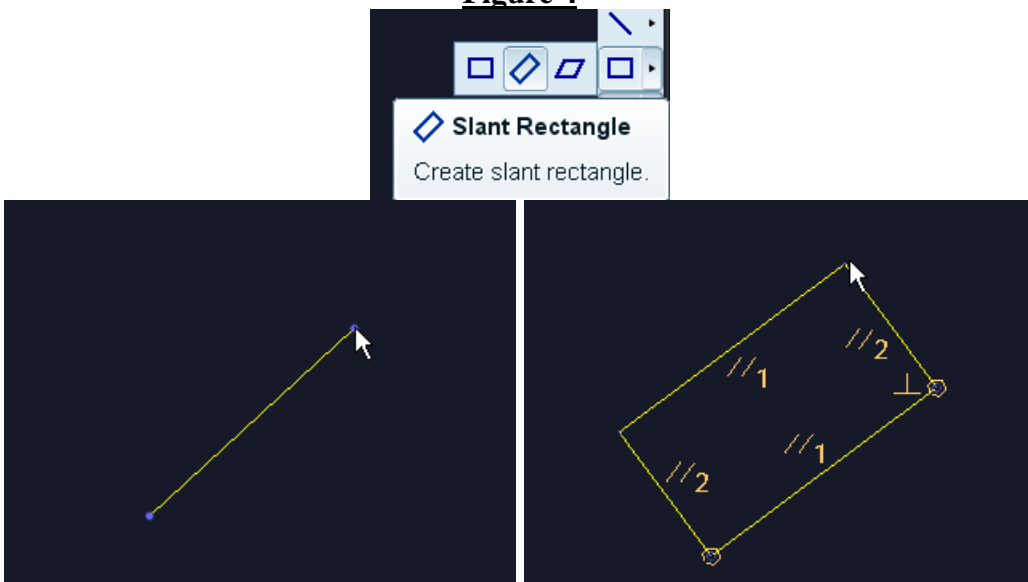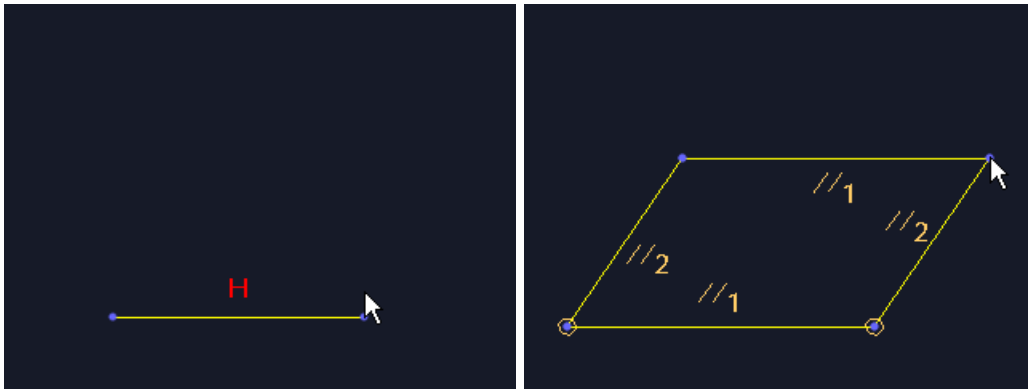
Figure 3

• Slanted rectangle (Figure 4)
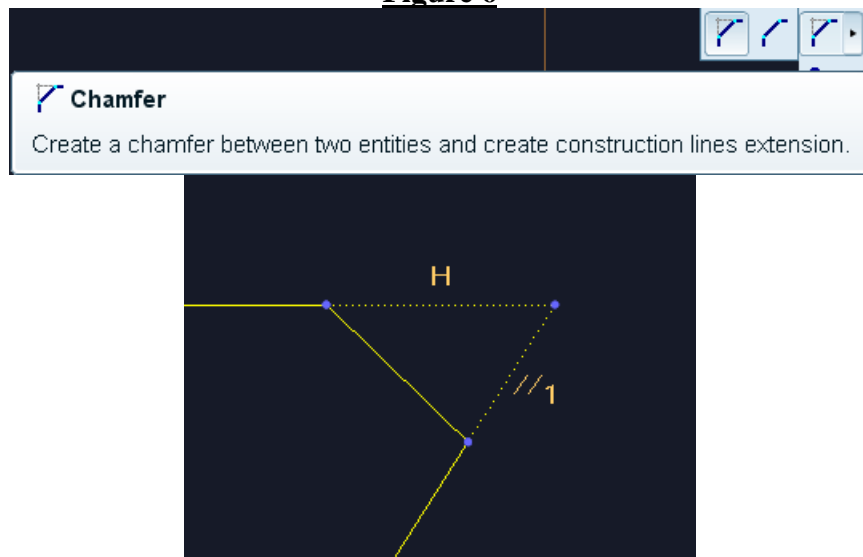
Figure 4

• Parallelogram (Figure 5)

Figure 5

• Chamfer with (Figure 6) or without intersecting construction lines (Figure 7)

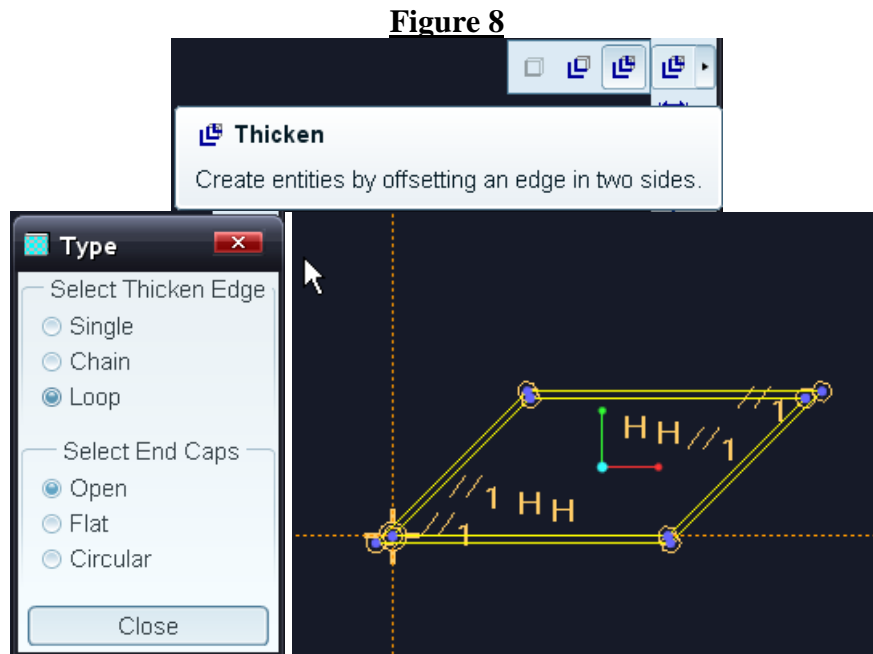**Figure 6**

Chamfer

Create a chamfer between two entities and create construction lines extension.

Figure 7

Chamfer Trim

Create a chamfer between two entities.
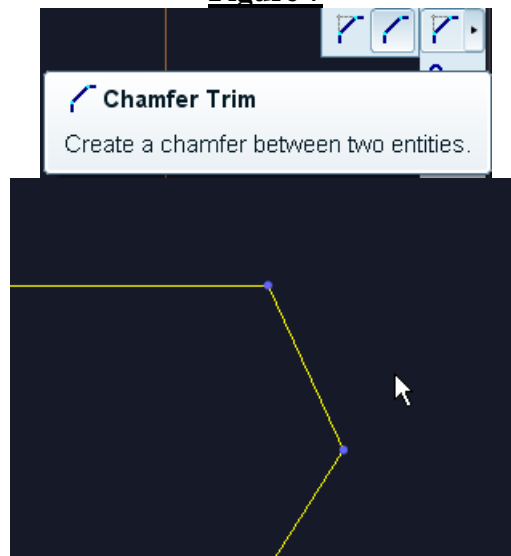
In addition to using or offsetting an existing model edge in a sketch, you can thicken an edge resulting in two parallel entities a stated distance apart and a stated distance from the reference. (Figure 8)

Figure 8

Use commands from shortcut menus (Right-Mouse-Button) to quickly access sketcher options and references. (Figure 9)

Figure 9
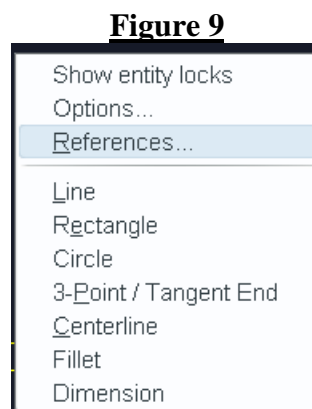
Sketcher Dimensioning Improvements

Several new dimension enhancements in Sketcher improve usability and flexibility.

Benefits and Description

With enhanced Sketcher dimensioning you can:

•   Create a total included angle about an axis of revolution

When dimensioning a section with a centerline and a non-parallel line, you can assign an angle dimension or a total included angle dimension, depending on the design intent. The following is an example of a line with an angle dimension applied:
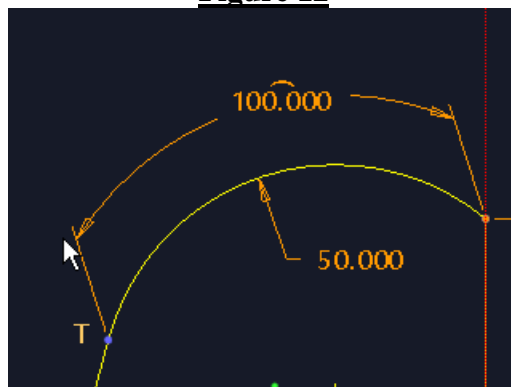
Figure 10
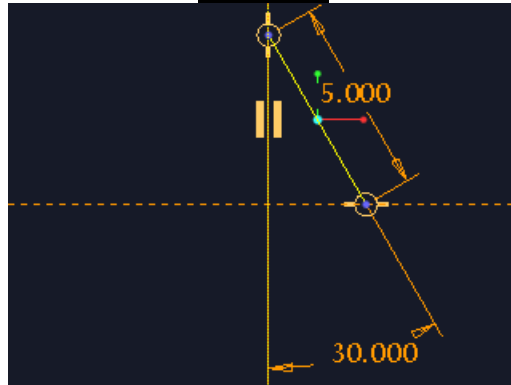
The following is an example of the line with a total included angle dimension applied:

Figure 11
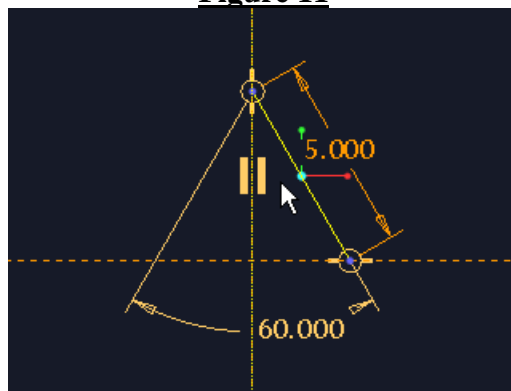
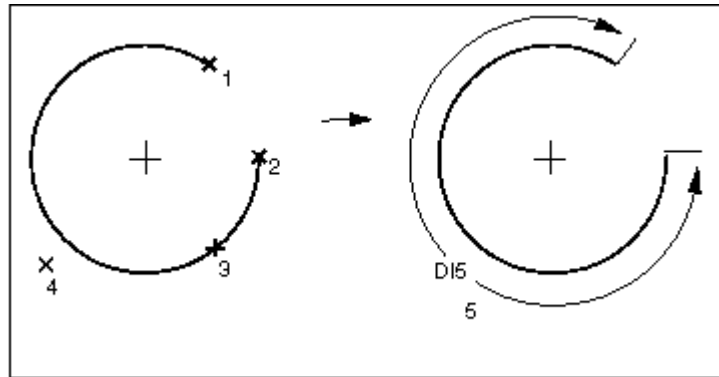•   Create an arc length dimension, with appropriate symbols

Figure 12

Example: Creating Angular Arc Dimensions (from Help files)

1. Click 1 – on endpoint
2. Click 2 – on endpoint
3. Click 3 – on arc
4. Places dimension
5. Resulting dimension

• Toggle between a diameter, radius, and linear diameter dimension

Figure 13
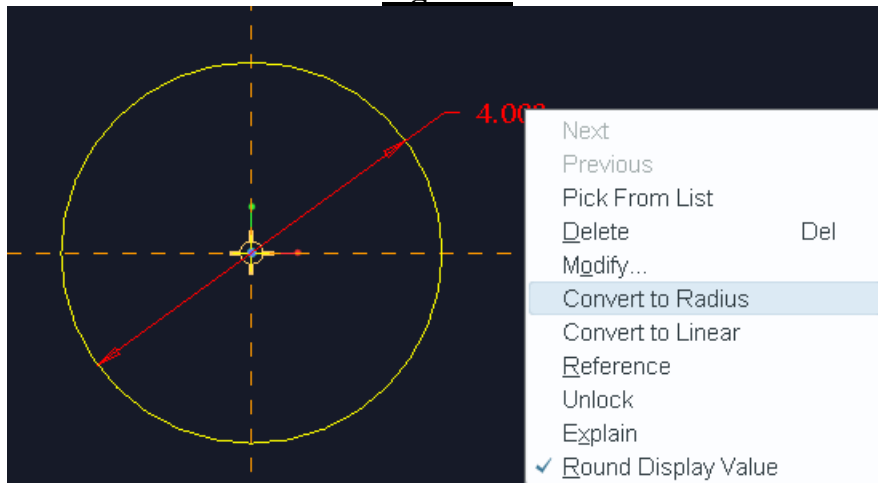
• Create a linear dimension tangent to two arcs in a specified direction.

Figure 14

In this example the 3.00 dimension is tangent to both circles and parallel to the centerline

- Toggle between a reference and driving dimension

Figure 15

- Create a new perimeter dimension for a loop or chain

Figure 16

• Create any type of dimension (Normal, Perimeter, Reference, or Baseline) by clicking the dimension icon on the toolbar
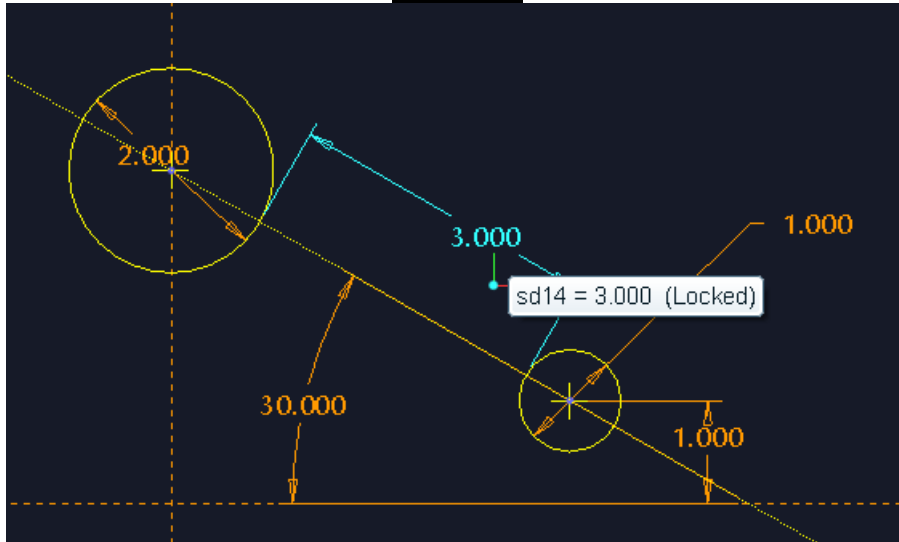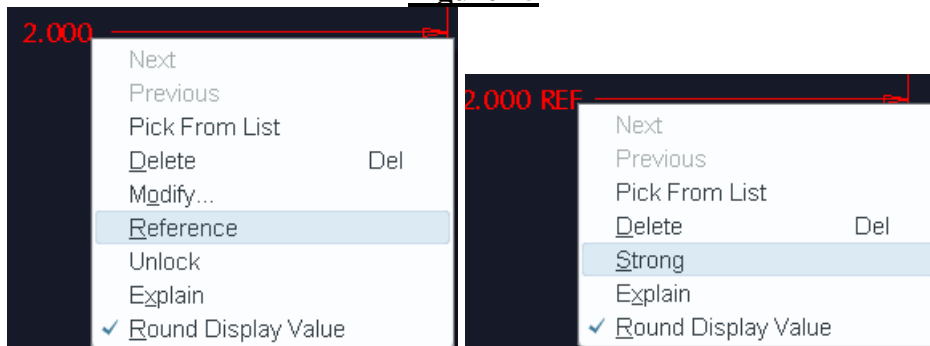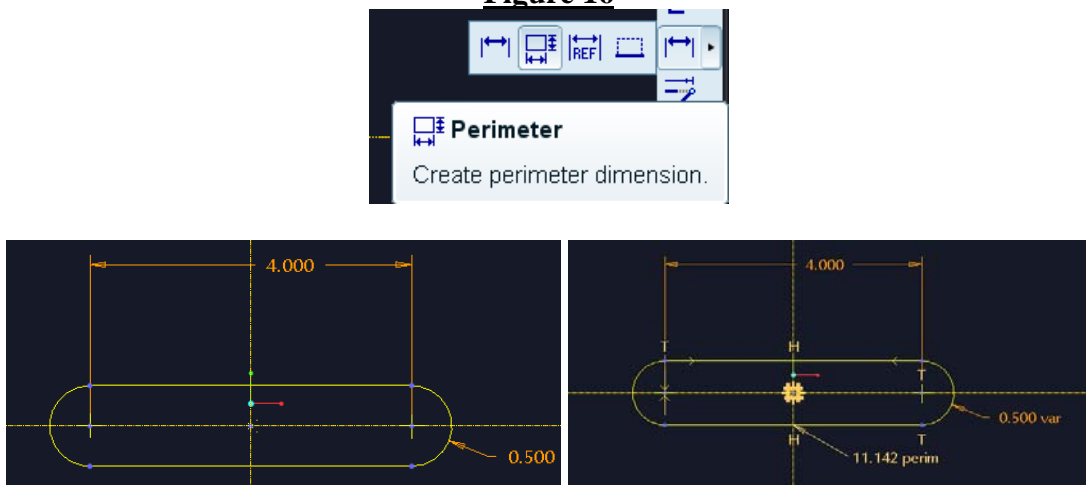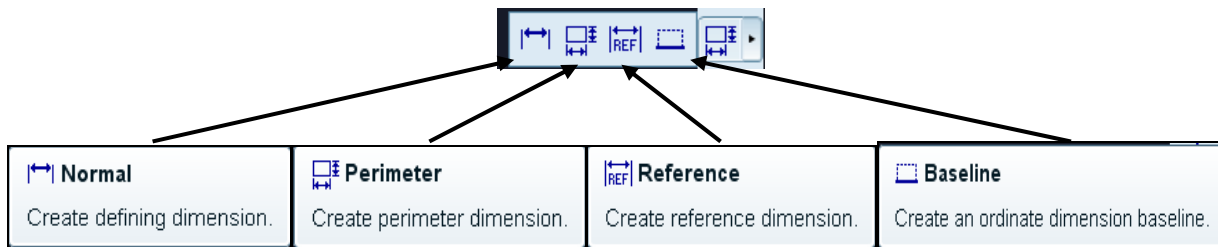

## Sketcher Points and Coordinate Systems Improvements

Sketcher points, centerlines, and coordinate systems are consolidated to simplify their application, and make them more intuitive and flexible.

Benefits and Description

Construction points are used only within the sketch. Depending on how the sketch is used, geometry points can produce a datum point or axes in the resulting model. Geometry points can create axes during extrusion; therefore, axis points are eliminated. Geometry points are saved with the section, improving reuse of common designs, such as the palette.
There is a single workflow for centerline creation and you can toggle between geometry and construction. Use construction centerlines only within the sketch. Depending on how the sketch is used, geometry centerlines can produce datum axes in the model. For consistency in revolved features, you can use only geometry centerlines as axes of revolution.

Figure 17
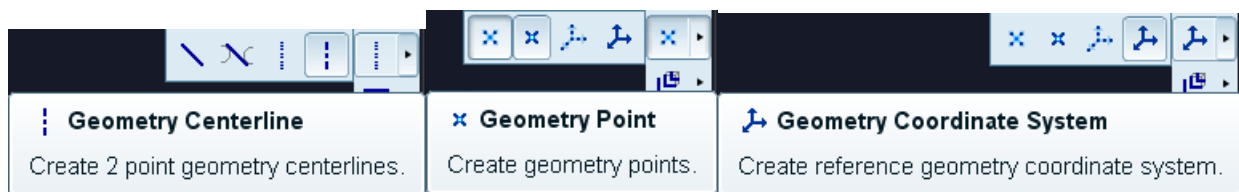
A new type of coordinate system (CSYS) is introduced, and the existing horizontal-vertical (HV) CSYS inside sketches is updated. You can create a new rotatable geometry CSYS (rotatable around the z-axis only; the x- and y-axes remain in the sketch plane) within a sketch, resulting in a true CSYS within the resulting model. The existing HV CSYS, available only within in a sketch is updated so it defaults to HV, but can be rotated.

Figure 18
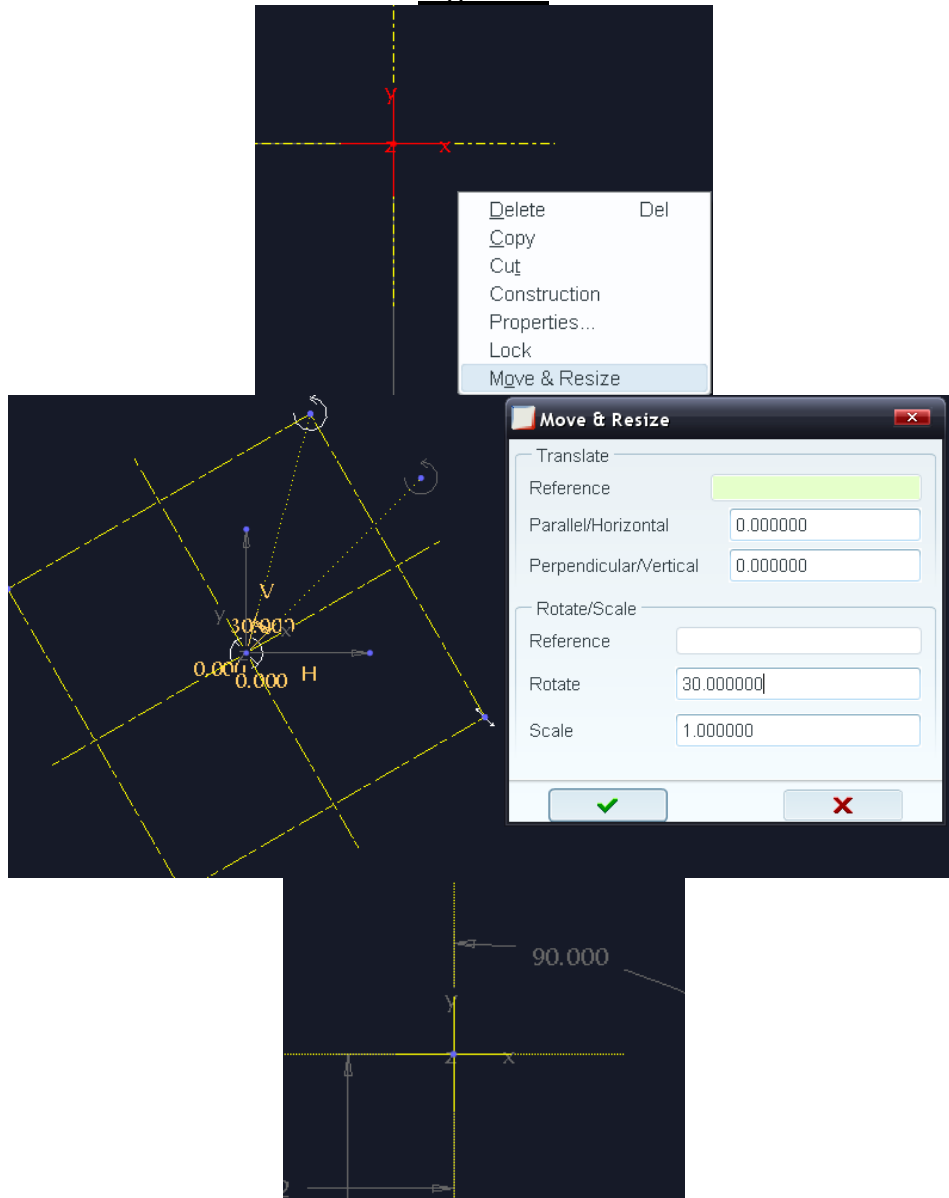
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PTC's Legacy Data Migration (LDM) Management

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Tips of the Month

How to "Replace With Any" Component

In Pro/ENGINEER Wildfire 4.0 it is now possible to replace a component with any unrelated part or subassembly. Pro/ENGINEER provides the tools to map the references required from both objects to ensure that there is no feature failure due to missing references.

Here is a quick example on how this is done. The yellow connector needs to be replaced with another connector with a 90deg angle so that the pipe will have an optimal shape.

Figure 1
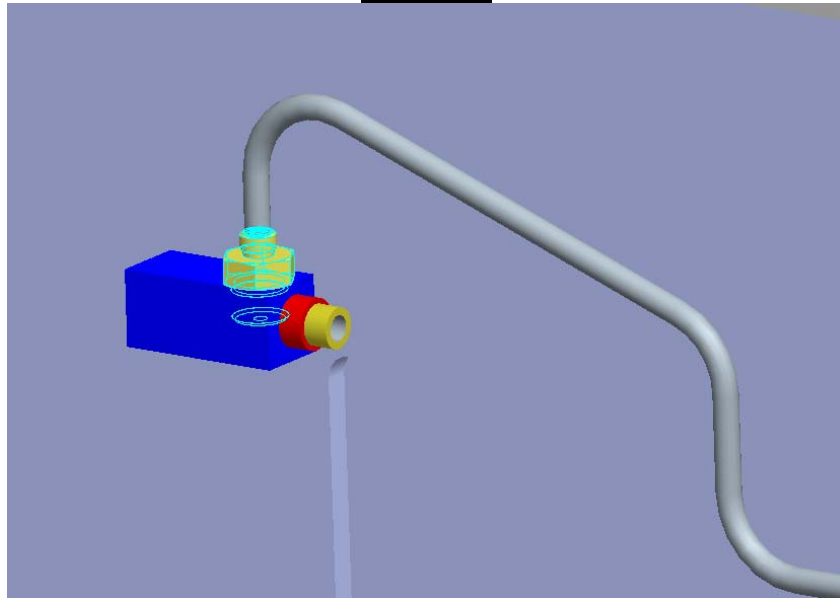
Figure 2
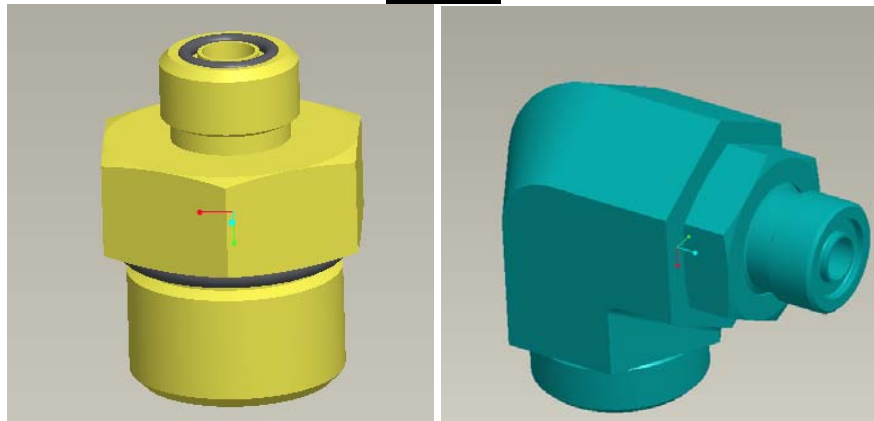
Notice that the connector is actually an assembly that also contains 2 rubber gaskets. When we replace the yellow component in the assembly there are a couple of references that need to be paired so that there is no feature failure due to missing references. In this case there are placement constraints in the main assembly, the pipe connection, and the placement of the gaskets.

Step 1: Select the component and Right-Mouse-Button to select Replace.
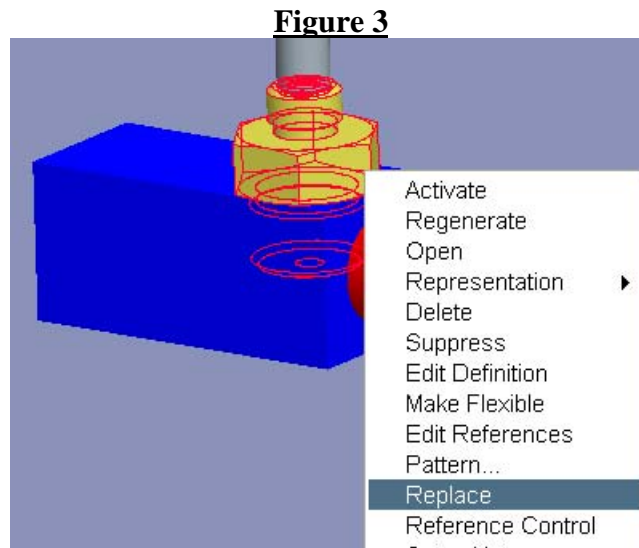
Figure 3

Step 2: Select the **Unrelated Component** option [1]. Select the component to replace the current component [2]. Select the Edit Ref Table to start the **Reference Pairing Table**.
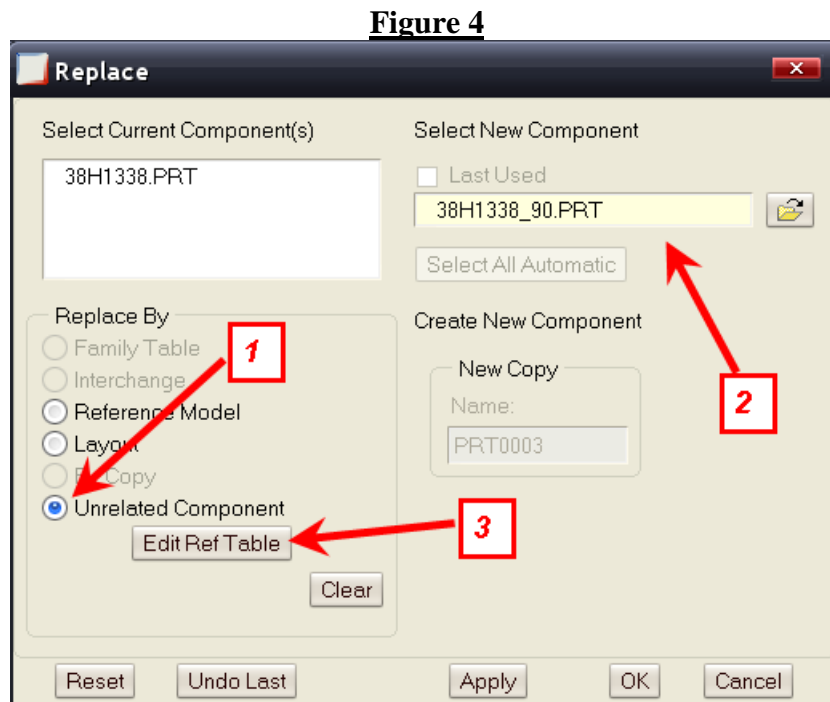
Figure 4

Step 3: This is the interface you are presented with (figure 5). It allows you to pair all references. You can automatically find pairs of references between the outgoing component and the replacement component. There are several pairing rules:

- Same Name— Automatically pairs objects with the same name and type.

- Component Interfaces—Searches for the interfaces with the same names and then examines each definition. If the same reference types are used in each interface then these can be mapped automatically.
- Same History—Searches the replacement model for any external references to the original model. If found, these references are automatically paired.
- Same Parameters— Automatically pairs parameters with the same name and type.

After autotagging, autoselection, or manual selection, a pairing table is created. You can store the pairing table and then use it for replacing the original component or for replacing the same two components in some other assembly. Your storage method may be the current assembly or you may create a new interchange assembly. Both methods have advantages and disadvantages. Storing a pairing table as a separate interchange assembly:

- Advantage: You can easily find all interchange assemblies in which this component is located.
- Advantage: All possible candidates are offered for replacement.
- Disadvantage: It modifies the models (unwanted with library parts).

Storing a pairing table in the context of the current assembly:

- Advantage: Models used for replacement are not modified.
- Disadvantage: During subsequent replacements on other assemblies, you must search and find the appropriate assemblies that define tags for outgoing and incoming components. This may be time-consuming.

Figure 5
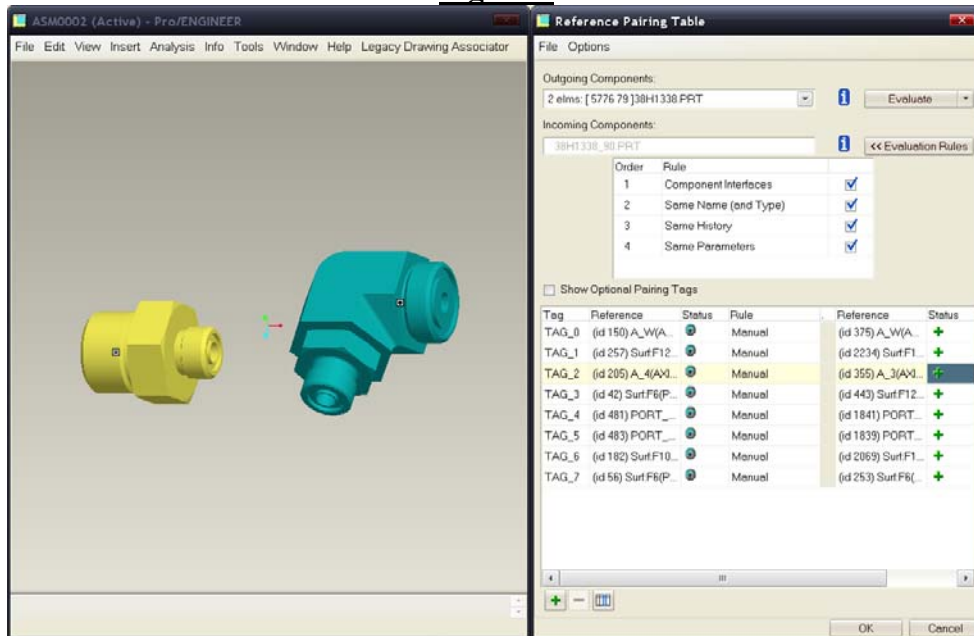
In this example the pairing table was stored so the all references are tagged. We will delete one reference to show how to manually map references.

Step 4: Manually mapping references. In this case the reference for Tag_6 did not get auto tagged. Select Tag_6

Figure 6

| TAG_5 | (id 483) PORT_... | | Manual | | (id 1839) PORT... | ✚ |
| TAG_6 | (id 182) Surf:F10... | | Manual | | | ✚ |

The reference will be highlighted and a green handle will appear.

Figure 7
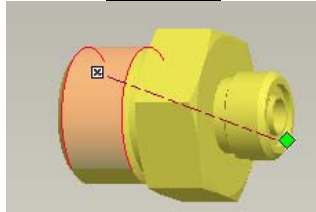
Simply drag the handle to the new reference.

Figure 8

Step 5: Select **OK** in the reference paring table and **OK** in the Replace dialog box. The result is shown below.

Figure 9

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Tips of the Month

Keeping Your Customers Up-to-date is Only A Click Away

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Announcements

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Customer Care Zone

PTC Tips & Techniques Webcasts: Work Smarter. Not Harder.

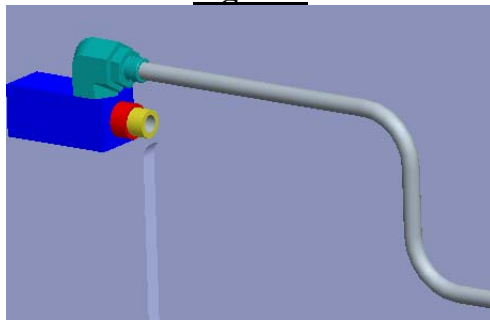Click below to see regularly scheduled Tips & Techniques technical Webcasts that are designed to provide you with the most popular time-saving tricks that Pro/ENGINEER users of all skill levels will find useful. Get more out of your maintenance dollars!

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- http://www.hp.com/go/ptcworkstation

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http://profilesmagazine.com/

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Upcoming Events & Training Class Schedules

Upcoming, 2009   Your local Pro/Engineer User Groups
                          http://www.ptcuser.org/rugs/

June 7 – 10, 2009   Orlando, FL USA
                          PTC/USER World Event
                          http://www.ptcuser.org/

Events

Our seminars and conferences seek to provide you with relevant information regarding product development trends in your industry as well as innovative software learning experiences. Think of them as a constructive day off where you can share experiences and swap ideas with your peers.

If you can't manage to get away, we'll bring it to you. Check back often for regularly scheduled live webcast events.

You're Invited to Attend…

Please visit the PTC Education Services website for the latest training information including course descriptions, schedules, locations, and pricing.

- Attend a course at any PTC Center and receive a free copy of Pro/ENGINEER Wildfire Student Edition!

    http://www.ptc.com/services/edserv/index.htm

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Sessions are performed just like a traditional ILT (including interactive exercises where you and the instructor can work on lab exercises together) and feature some of our most popular ILT courses. These sessions cover the exact same material as the traditional ILT in-center courses. Also look for some of our most frequently requested mini-topics delivered in the same format that are only an hour - two hours in duration.

If you have any questions about these sessions or would like to see getting other courses, not on this list, on the schedule please feel free to contact me for more details. They are a great way to bring training to you without you having to worry about location or being out from work for long stretches.

You can register for these sessions just as you would for any normal ILT class either by:

1. calling order admin at http://www.ptc.com/services/edserv/training/registra.htm or

2. you can go to PTC University directly at http://www.ptc.com/learning and submit a registration request directly. All you have to do is search the catalog by typing in "virtual" in the search field and you will see a listing.

PTC

Note: This PTC E-Newsletter will continue to be used for the following:

1) Inform you on events related to PTC products (user groups, conferences, training schedules, etc.)

2) Educate you on solutions that are available at PTC

3) Tips & Techniques using PTC Products

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