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PTC Product Focus

Peer-to-Peer Design Conferencing w/ Pro/Engineer Wildfire

Invite whole teams to your Pro/Engineer Wildfire session with the new Peer-to-Peer Design Conferencing functionality.

Now you can instantly connect to other users regardless if they're across the hall... or the across the country. It all starts with a shared workspace. Upload a model and then show your team how you want the model to look, pass control of the session to others, or watch others create geometry – all in real-time. In fact, invite whole teams into interactive design sessions. Don't worry - performance and security are powered by Groove technology.

Performance - Pro/ENGINEER Wildfire uses unique technology from Groove that changes the landscape for performance. First, the encrypted model is downloaded to each participant. From there, only lightweight commands -not mammoth design files -- are exchanged between two or more sessions of Pro/ENGINEER Wildfire, seamlessly. So whether you're leading the design session or following it, you see instantaneous results.

Security - The days of a product's design representing only topology are long gone. In today's model-centric world, designs are valuable intellectual property. Often, they are your competitive advantage. When you put your model into a secure shared session, it's encrypted so it can only be opened by a session of Pro/ENGINEER Wildfire

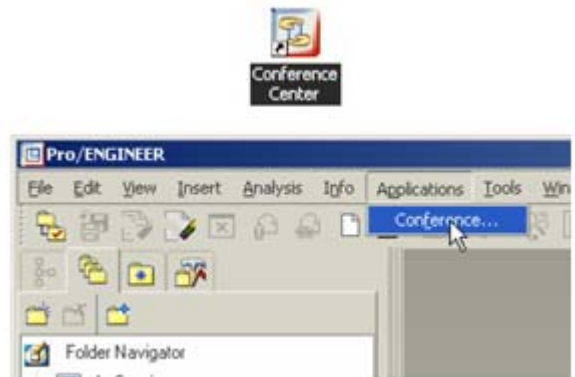


Powered by
groove™ 

that only you created for your partners via the shared space. With Pro/ENGINEER Wildfire, the host controls who can download and save the data, so your intellectual property is secure.

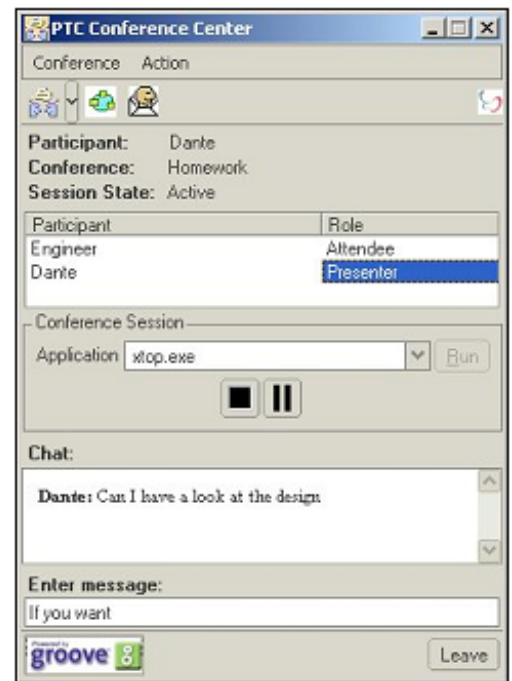
Design Conferencing Functionality Overview:

- Design sessions can be started from outside or inside a Wildfire session.
- Embedded Groove technology provides real-time performance, robust security, and seamless interaction between models and users independent of geographic location.
- Collaborate in real-time with one or many Pro|Engineer Wildfire users.
- Easily pass control between users and perform and Pro|Engineer operations including feature creation, dimension editing, part assembly, etc.
- Instant message session users and conduct online dialogs.
- Easily and securely share files and documents to anyone to the internet.

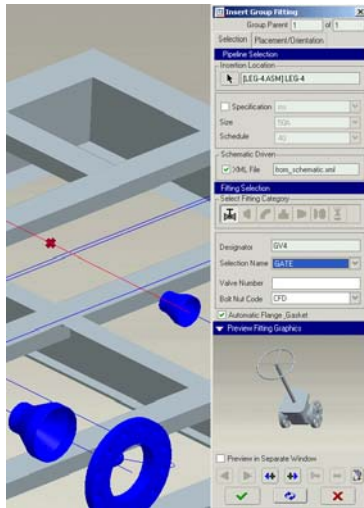


Design Conferencing Uses:

- Instant design collaboration sessions
- Engineering team design reviews
- Online training sessions
- Instant how-to sessions
- PTC Technical Support sessions



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Once the XML file has been imported into Wildfire piping the 3D mechanical designer is supplied with a list of pipelines to be routed around the mechanical design, what components the pipelines are connected to and all the fittings that are to be used. The mechanical designer can now concentrate on finding the optimum paths for pipelines, confident that pipe spec, the connections and the fittings are as intended.

Once each pipeline has been created the mechanical designer can always use the design rule checker within Wildfire to double check the mechanical design conforms to the P&ID, guaranteeing accuracy and minimizing costly mistakes.

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Tips of the Week

Controlling Sketcher Constraints to Capture Design Intent

There are times when a section is sketched, but the design intent was not quite captured by the assumptions that Pro/ENGINEER made. When the design needs to be redefined or finalized, the **Constrain** functionality within Sketcher can make it extremely easy to drive and manipulate the section.

Procedure

1. Starting with a section to be redefined, only horizontal constraints are appearing as *strong*. The section will be redefined entirely by constraints.

Definition of strong – strong dimensions or constraints, in yellow, are those that have either been created by the user, modified, or have been made strong using the right-mouse-button option. Also, strong dimensions or constraints, or sketcher entities, will not be automatically replaced by user-created over-defined sketcher entities. In contrast, weak sketcher entities, in grey, will always be replaced by an over-defined strong sketcher entity.

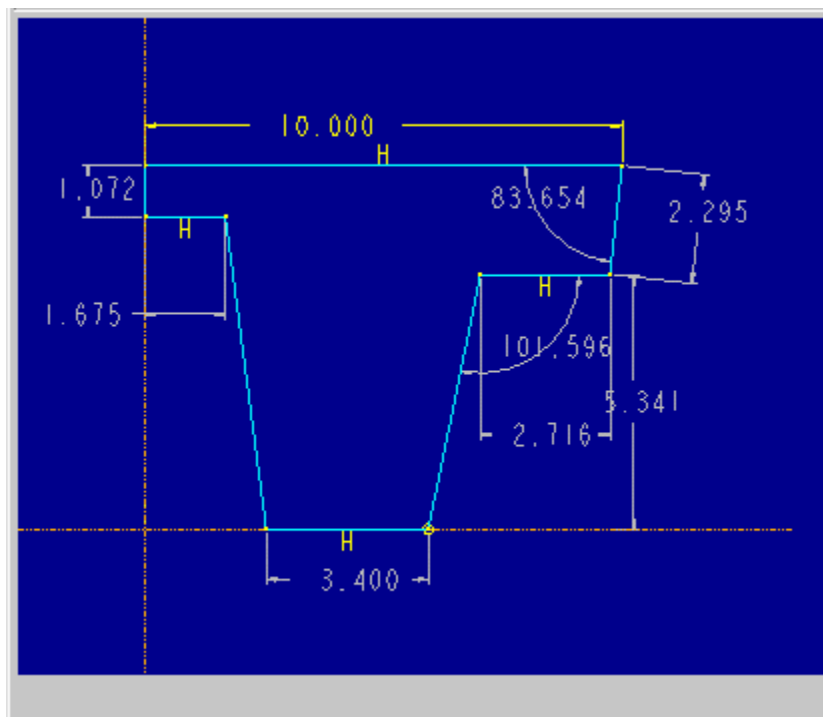


Figure 1

- To begin, the upper right entity will be made vertical using **Sketch, Constrain, Vertical**. A 'V' symbol will be generated and placed next to the entity. Then the two vertical entities will be given **Equal Lengths** from the CONSTRAINTS dialog box. A 'L₁' symbol will be generated on each corresponding entity. Note that in the Figure 2, the 'L₁' is covering the 'V' symbol on the upper right entity.

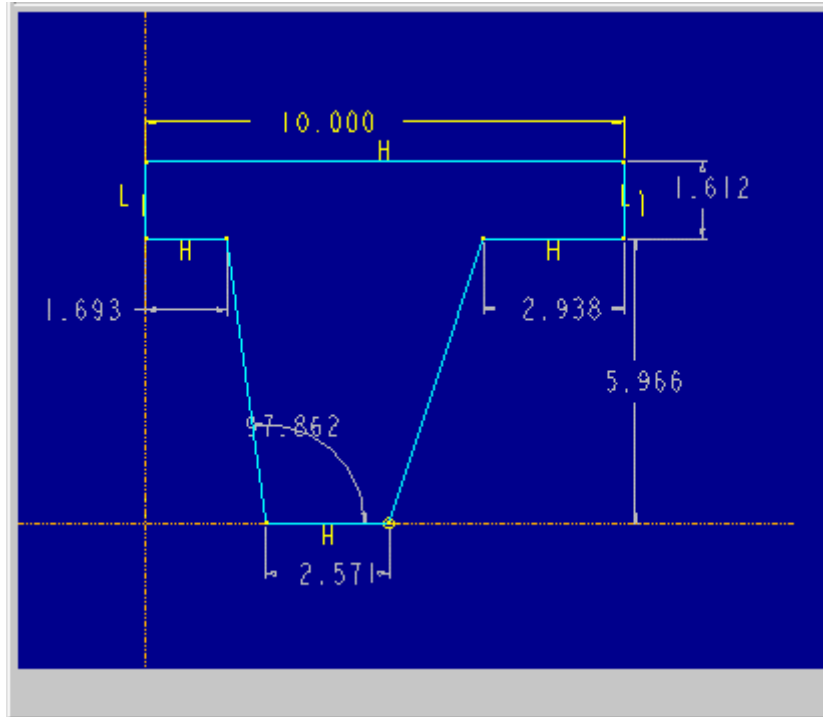


Figure 2

- Next, to create symmetry in the section, a centerline and symmetry constraints will be added. Select **Symmetric** from the CONSTRAINTS dialog, then pick the centerline and then the lower two vertices. Or, by picking the upper two inside vertices, and then selecting **Edit, Convert To, Strong**, this will do the same thing, as they are already appearing as "weak" constraints (gray in color) when the centerline was added. The arrows pointing towards the centerline represent the symmetry constraint. Modifying the 2.571 dimension to 4.000 to strengthen it verifies that the symmetry constraints between the vertices is holding.

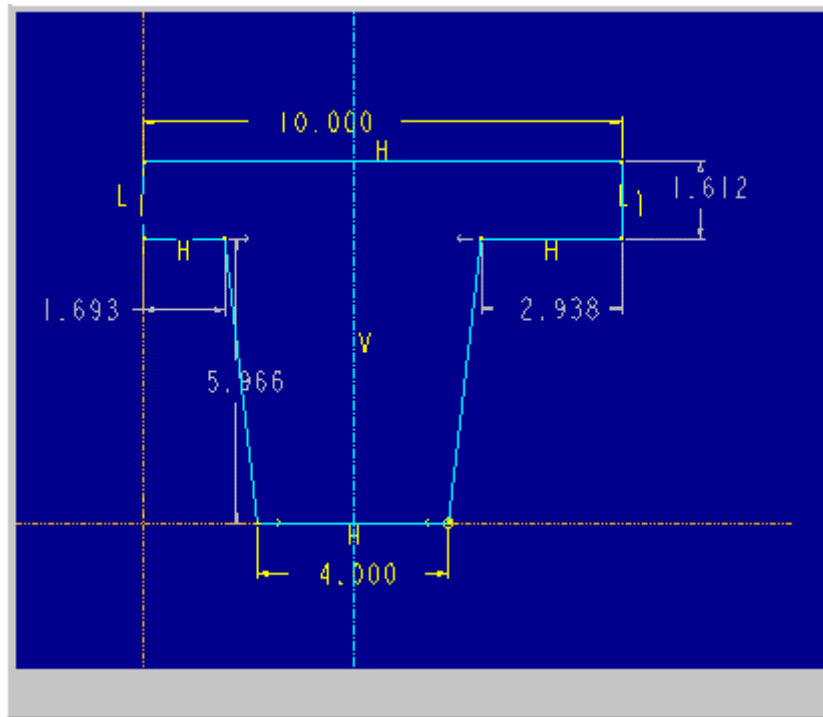


Figure 3

4. To make the two short horizontal entities the same length, select **Equal Lengths** from the CONSTRAINTS dialog and pick on the two entities, as was done for the vertical lines entities in Step 2. A 'L₂' symbol will appear to denote that now these two entities are equal length too, but not the same length as the vertical entities. Modifying the vertical dimension from 1.612 to 1.000 and the one new horizontal dimension to 1.750 verifies the entities holding at the same length.

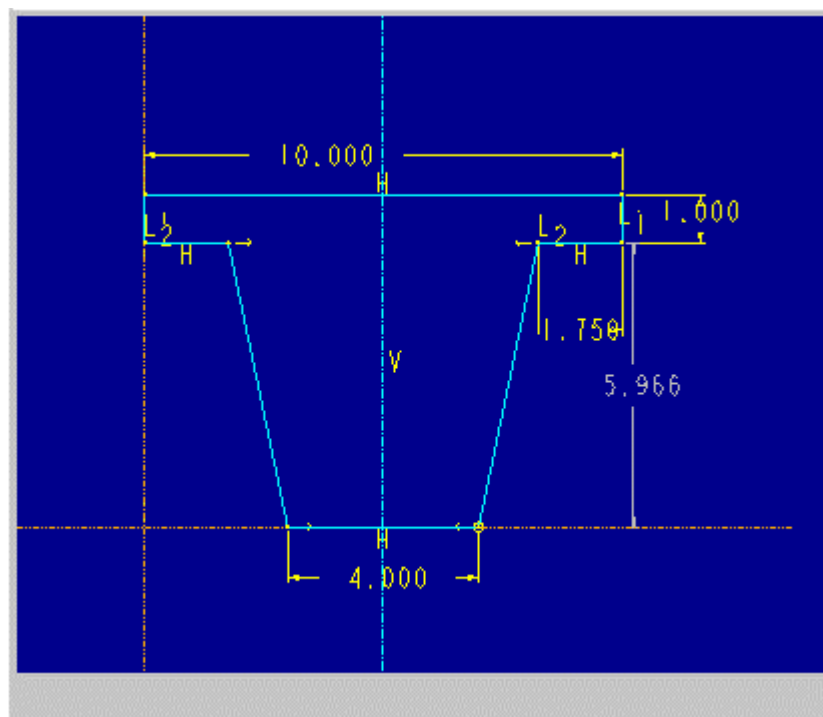


Figure 4

5. If at any point, two conflicting constraints are created on the same set of entities, Pro/ENGINEER will prompt the user to either **Delete** or **Undo** one of the constraints. These conflicting constraints will appear in red. There is also a menu pick for **Explain**, in case the constraint is not apparent.

To finish the section, the 5.966 dimension could be modified to 6.000, strengthened, or left alone, and then **Done** selected from the SKETCHER menu.

In summary, the only item added to the section was the centerline, and then the constraints alone transformed the section in Figure 1 to the one in Figure 4. Remember, if a constraint or dimension is still gray ("weak"), then these can be removed from the section at any time if another constraint is applied, without prompting the user, so it is good practice to **Strengthen** any dimensions or constraints that are required to be in the design intent of the section.

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Tips of the Week

Intelligent Part Numbers

INTELLIGENT PART NUMBERS

Using some simple techniques it is easy to generate intelligent part numbers to describe your Pro/ENGINEER parts. Why would someone want to take advantage of intelligent part numbers? An in-depth description of your products can be obtained simply by looking at the part number. What function it serves. What material it is made of. Dimensional descriptions. Even which customer it is produced for.

Pro/PROGRAM will be used in this article not only to query a user for this needed information but also to handle the underlying logic that a casual user need not know. It is quite simply an interface into the customization or modification of a part or an assembly.

What follows is a simple example of extracting information from a part and using it as a basis for an intelligent part number. The hydraulic cylinder, illustrated in Figure 1, is a good candidate for intelligent part numbers. We will be looking at just one component of this assembly, the cylinder tube, shown in Figure 2.

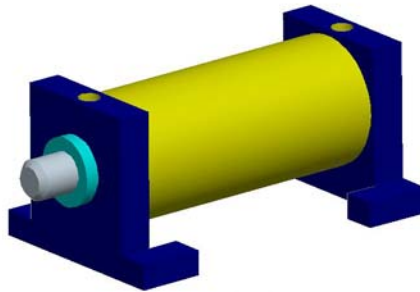


FIGURE 1

Our hydraulic cylinder comes in various sizes and can be used in many different applications. We need a good description of each component so that the correct configuration is obtained. Let's consider the cylinder tube. Important information needed here could be a company standard part designator, material type, dimensional information, or any custom features asked for by the customer.

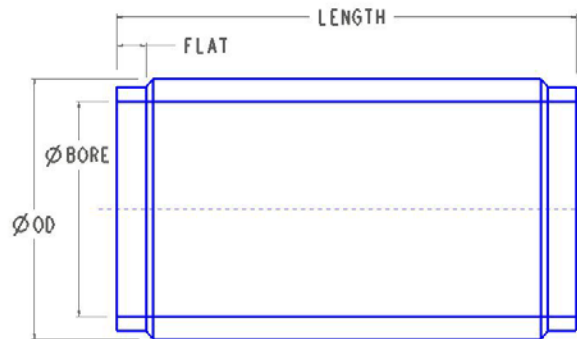


FIGURE 2

For example, a valid part number such as TB100-675-20-AA provides the following information:

TUBE, CHROME STEEL, LENGTH = 6.75 inches, BORE = 2.0 inches, and AA indicates non-standard features added.

This part-numbering scheme adheres to the following convention:

<u>MATERIAL</u>	<u>LENGTH</u>	<u>BORE</u>
100 – CHROME STEEL	200 – 2.0 inches	10 – 1.0 inches
200 – STEEL	513 – 5.125 inches	25 – 2.50 inches
300 – BRASS	1000 – 10.0 inches	48 – 4.75 inches

With the parameters of the cylinder tube named as shown in Figure 2, a Pro/PROGRAM was written to extract the necessary information from the part, query the user for any additional information, and then and then format these values to conform with the desired output. These values (parameters) were then assembled and fed into a user-defined parameter named PART_NUM. The value of this parameter is the intelligent part number we desire. It updates as the cylinder tube changes and can be used in a variety of places including a bill of materials.

The Pro/PROGRAM in this example will do three things:

1. It will ask the user for information. The input section of the program has the following lines added:

```

INPUT
MATERIAL NUMBER
“WHAT TYPE OF MATERIAL? 1 = CHROME STEEL, 2 = STEEL, 3 = BRASS”
CUSTOM YES_NO
“DO YOU WANT A GROOVE? (Y/N)”
END INPUT

```

Here the two parameters (MATERIAL and CUSTOM) are followed by a question in quotes. These questions will be presented to the user upon regeneration of the model. This is one way information can be obtained for use. The other parameters needed and their values are derived from features in the part.

- The program will have relations that obtain other information needed for our intelligent part number. The following relations were added:

```

IF MATERIAL == 1
  FIELD1 = “TB100”
ENDIF
IF MATERIAL == 2
  FIELD1 = “TB200”
ENDIF
IF MATERIAL == 3
  FIELD1 = “TB300”
ENDIF
FIELD2 = ITOS(LENGTH * 100)
FIELD3 = ITOS(BORE * 10)
IF CUSTOM == YES
  FLAT = .5
  FIELD4 = “-00”
ELSE
  FLAT = .25
  FIELD4 = “-00”
ENDIF
PART_NUM = FIELD1+”-“+FIELD2+”-“+FIELD3+FIELD4

```

If CUSTOM is set to yes the flat area on the cylinder tube is extended to .5 inches to make room for the custom groove.

- The Pro/PROGRAM will include or not include certain features by using an “if” statement placed around the feature in the part program. An example follows”

```

IF CUSTOM == YES
  ADD FEATURE
  INTERNAL FEATURE ID 101
  TYPE = CUT
  .
  .
  .
END ADD
END IF

```

In this way if CUSTOM is set to YES the cut feature will be added to the part.
The final result is a cylinder tube with a parameter named PART_NUM having our intelligent part number as its value.

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Announcements

Special Hardware offers for customers updating to Pro/ENGINEER Wildfire

http://www.ptc.com/partners/hardware/current/wildfire_tlo.htm
<http://www.3dlabs.com/PTC/>

PTC Sponsored Events

Click below to see PTC sponsored events:

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Click on the poster to gain access to the Product First Virtual Event resource center!! See the software and pick up a screen-saver or two.



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Please take a few minutes to check out this new web site. We don't think you will be disappointed.

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Upcoming Events & Training Class Schedules

Upcoming, 2003 Your local Pro/Engineer User Groups
<http://www.prouser.org/rugs/>

Upcoming, 2003 Pro/USER International Conference
<http://www.prouser.org/>

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/E 2001 Student Edition!

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

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 products that are available at PTC
- 3) Tips & Techniques using PTC Products

Note: These messages are compiled in the local PTC office and will be distributed via e-mail.

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