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new!

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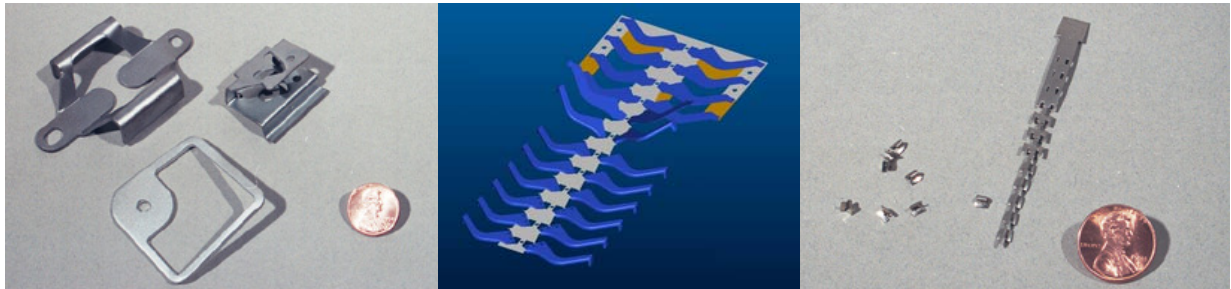
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PTC Product Focus

Progressive Die Extension 2.2

Tooling development for progressive dies is best approached with a purpose-built solution. With Pro/ENGINEER Progressive Die software, easy-to-use wizards guide the user through automatic strip layout definition, cut stamp creation, and placement/modification of die components. Documentation, clearance cuts, and drilled holes are created automatically, enabling die designers to avoid manual, error-prone tasks.

Examples of parts made from a Progressive die:



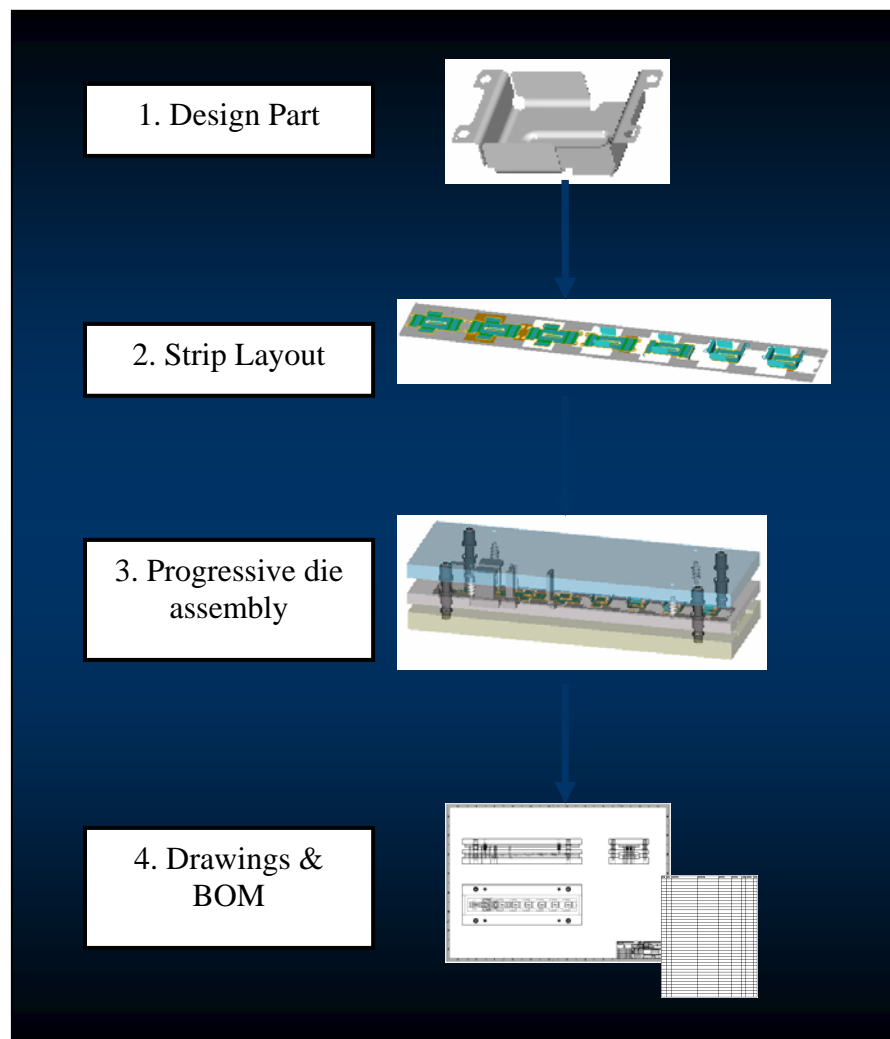
30,000 Lb. Progressive Die
1,000-Ton Try Out

Why Pro/ENGINEER Progressive Die Extension (PDX)

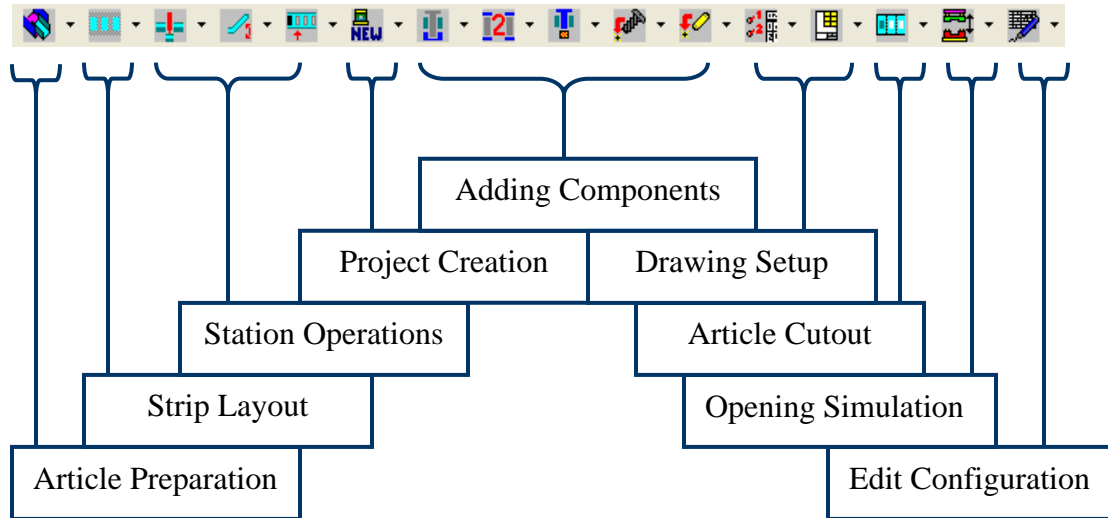
In the past tools for this type of specialty application were limited to 2D systems. Considering the complexity of progressive die sets there is a severe limitation with this method. For example there is no capability for interference checking which will cause accuracy/tolerance issues. Also there is no associativity with the model thus updates are very difficult.

Pro/ENGINEER (without PDX) lacks the tools necessary for certain specialized functions required to create a full Progressive Die Set. These tools include the ability to create strip layouts, project definition, die set definition, and smart libraries of components that will automatically create the necessary clearances in the die set plates.

PDX has the capability to automate this process which specialized ‘wizards’ that guide you through the process. The steps required to create a die set are as follows.



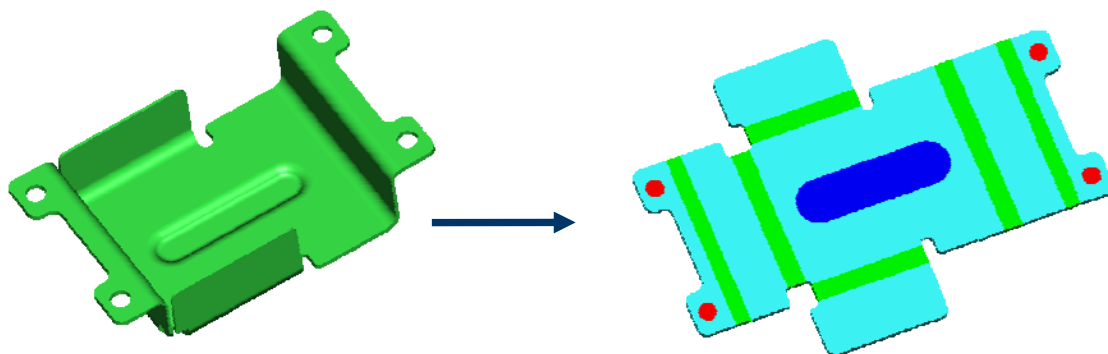
The PDX UI follows the above logical flow and leads the user through the steps.



At this stage we will go through the steps in more detail and review the advantages in the methods used with PDX.

Step1: Sheetmetal Part Preparation (Article Preparation)

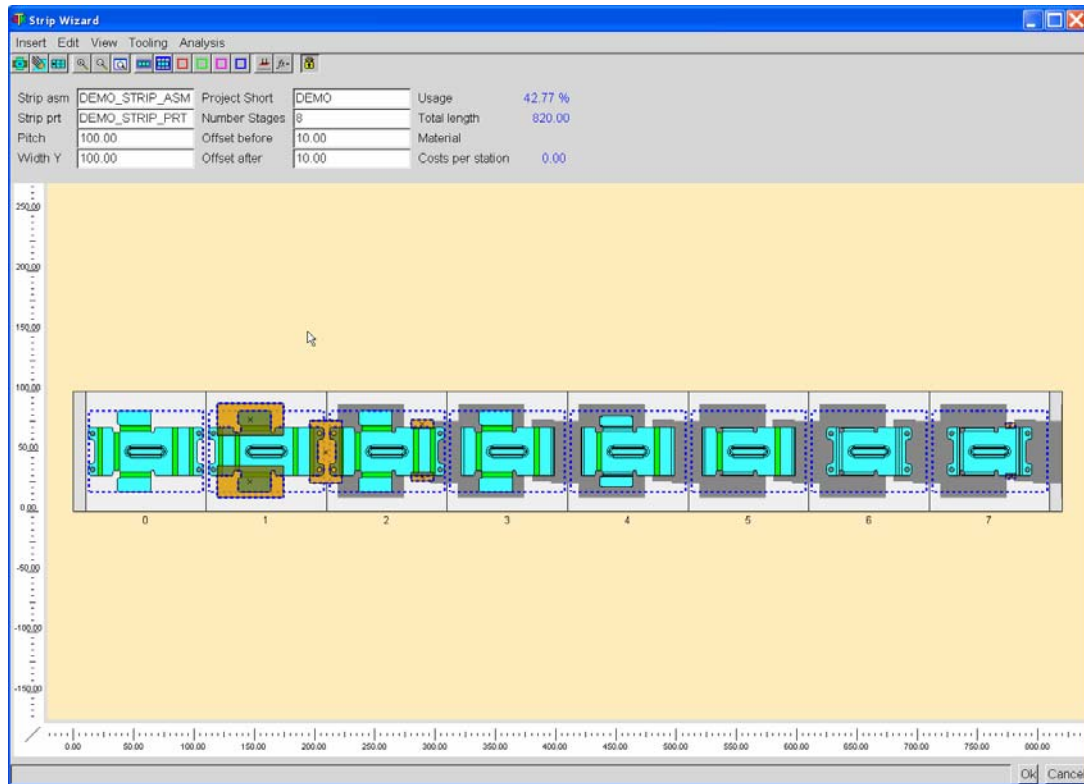
The first thing you need is a Sheetmetal part. It does not necessarily need to be created in Pro/ENGINEER. It can also be imported as well. As long the part can be converted to a Sheetmetal part and can be developed or flattened. This will be called the article.



At this point PDX will create an intelligent copy of the original. It will automatically un-bend the part and identifies the manufacturing geometry like forms and cuts. It color codes the geometry to easily identify all the operations required to create this part. Green represents bends, blue represents forms and red represents cuts or piercings. All of these are used for the strip layout step. By directly referencing the model the prepared article part is completely associative.

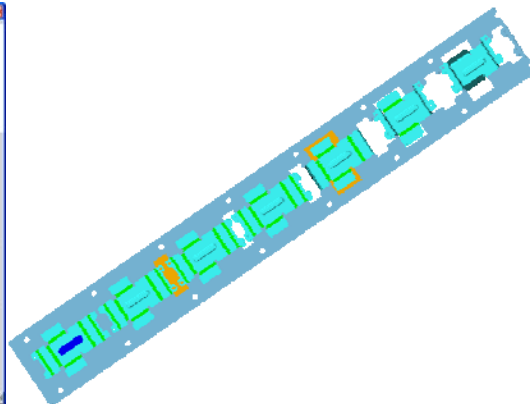
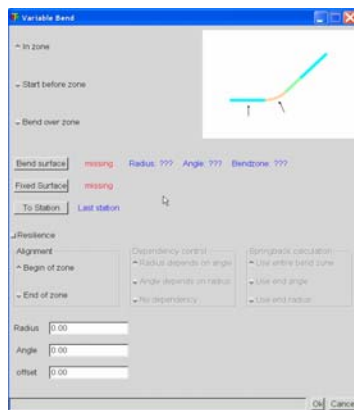
Step 2: Automated Strip Layout

Using the strip wizard the flattened article is positioned within the strip. The strip size and number of stages can be entered as well. Once positioned the Strip layout wizard calculates the strip usage percentage, material costs and design space.



Also within the wizard, stamping references can be created as seen in orange in the image above. Once done with the strip layout the progression can be setup. Just choose the geometry identified in step one (green, red and blue areas) with the respective operations (i.e. bending station).

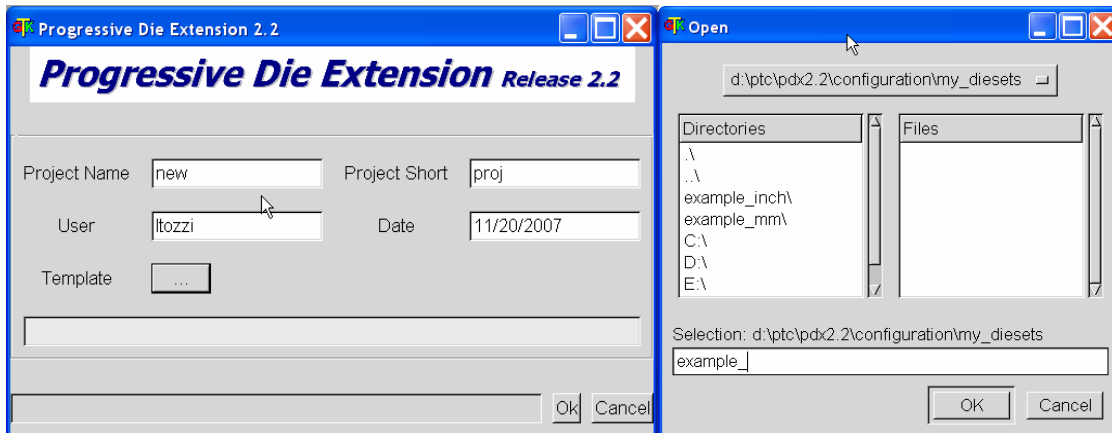
Variable bending can also be used. This operation has the option of entering the radius and angle according to the bend factors. This operation is often used when you want to pre-bend a value of 45 degrees before fully bending to 90 degrees in the next station.



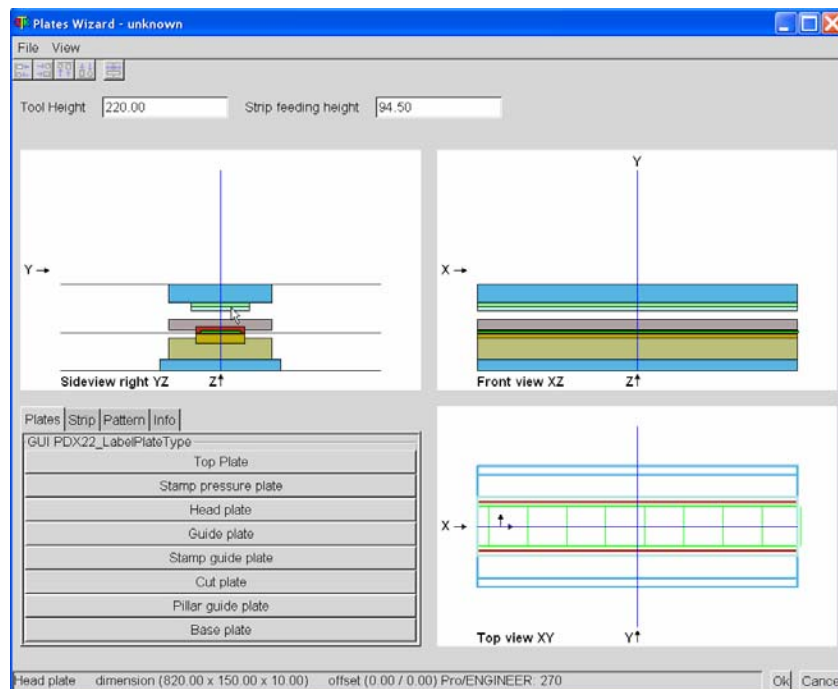
The Strip is automatically created and as a bonus you can set it up to create a drawing of the strip with a pre-defined template.

Step 3: Project Definition and Die Set Creation

This is where we create a project. Essentially this is where we name the assembly and how the components are renamed. The Project Short value will become the prefix to all the component names of the die set. Templates can be used to maintain company standards. The template can include manufacturing files, drawing templates and any other component that you may want to include.



Now we use the Die Set Wizard to define the size and position of the main components. This easy-to-use wizard contains all the necessary dimensions. Just double click the plate and enter the values.

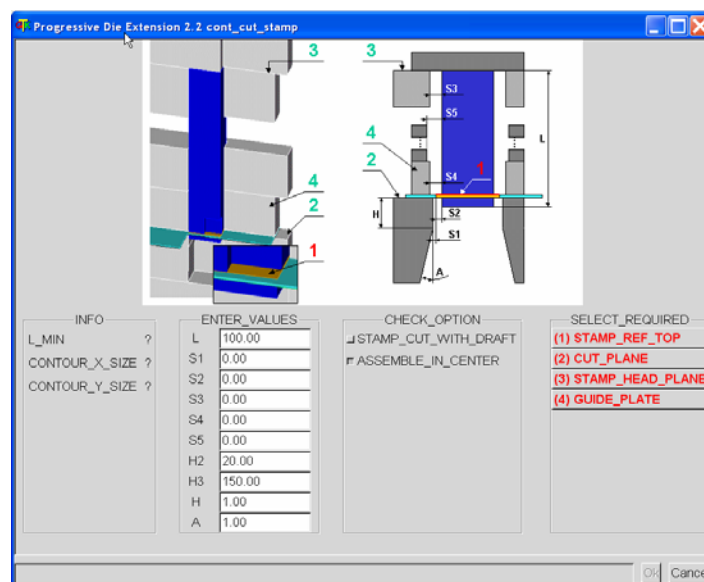


There is also provision to set up the location of guides, springs, spacers, top screws, guide screws, base screws, top pins, guide pins and base pins. This will create the axis pattern that can be used as a reference to position the actual components.

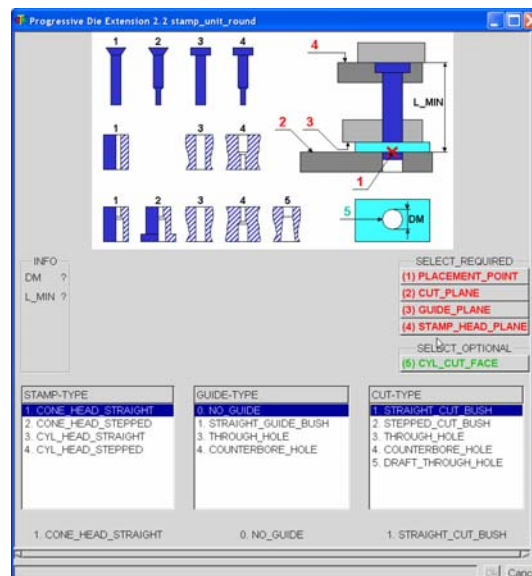
Now that the plates have been assembled it is time to create the tooling. There are a couple of techniques that can be used.

First the stamps can be directly referenced (in orange) to create the stamp with the proper shape. The stamp takes on the contour of the stamp reference created in the strip layout. The properties of the stamp are defined via a library of predefined shapes. The round piercing operations are also defined in this way except that we chose from a library.

Example of Contour stamp:

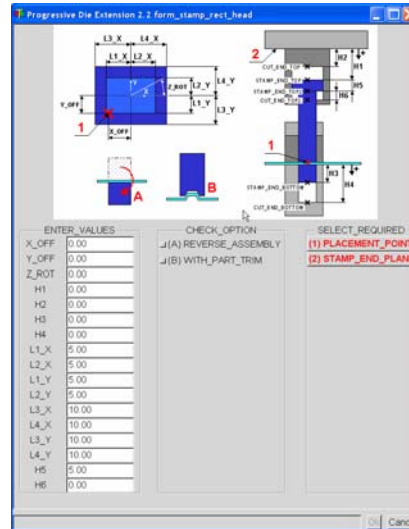


Example of a round stamp:



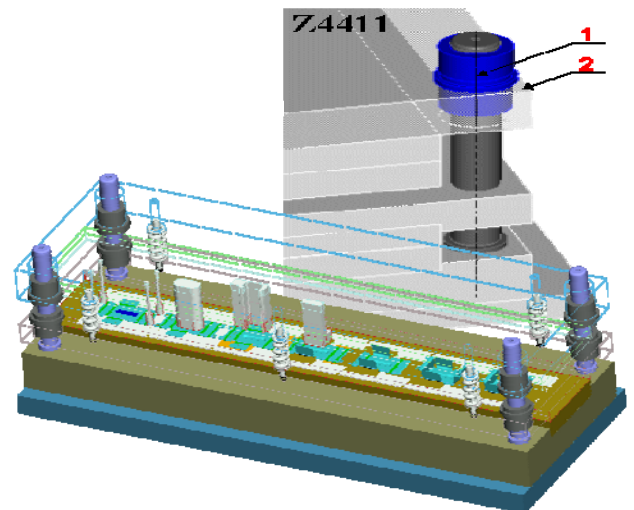
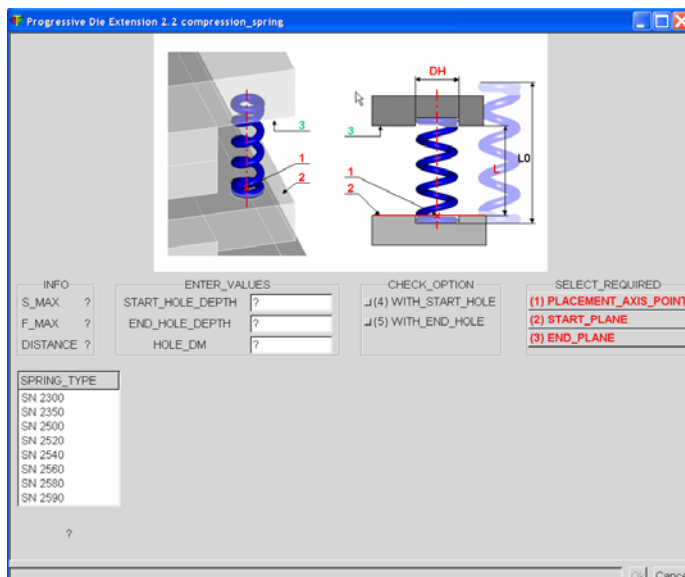
In this case we chose from the library of available round stamps out-of-the-box. These include **Misumi & Strack** standards.

Example of a form tool dialog box:

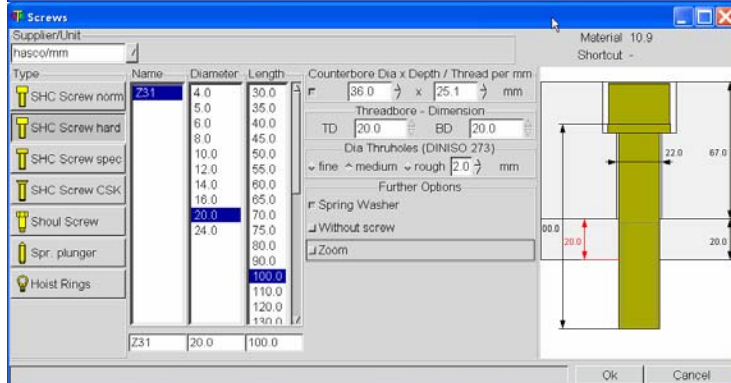
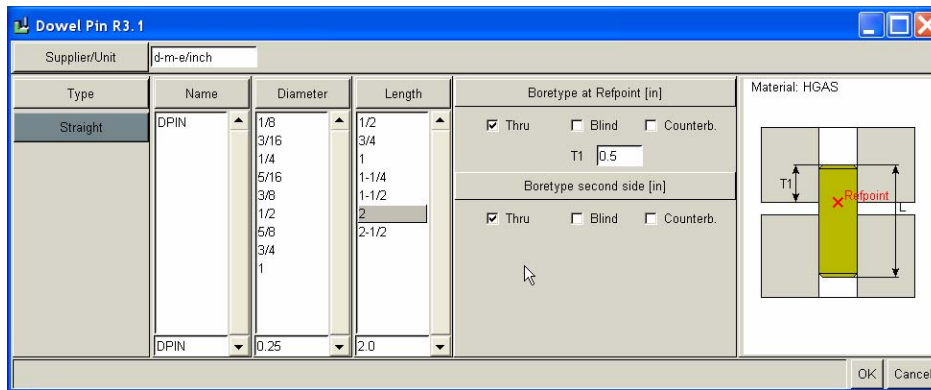


Other components are also placed in this way. The interesting part to this is that the library and the associated UI for placement can be customized to suit any company standard. Pictures can be included to show the correct picks as well as dialog boxes for entering values. This flexibility allows for a custom library to be built and effectively all components will become drag and drop. The other benefit to this is that the clearance cuts thru all the plates are automatically created.

Example of Spring Placement and Guide Placement:

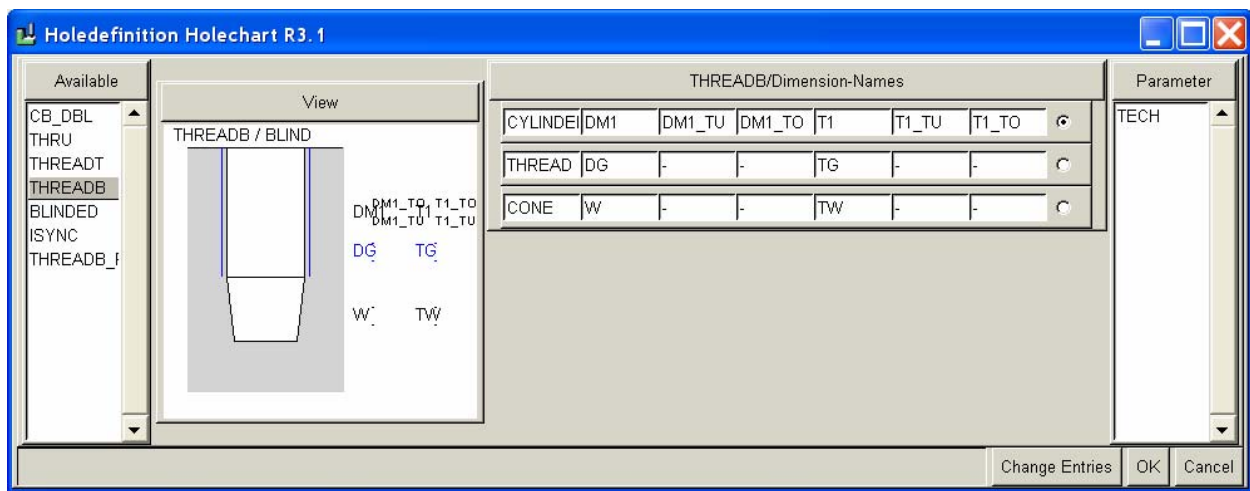


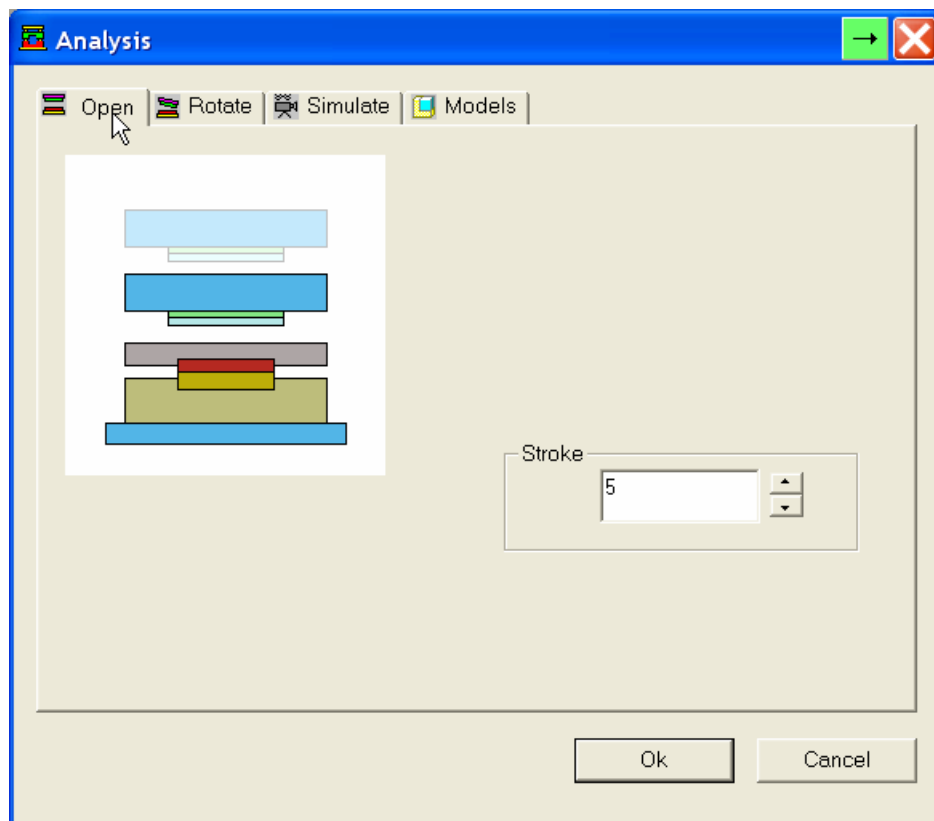
Since the die set was created by a template that included the drawings of all the plates the drawings are updated as well. There is also the ability to add screws and dowels from the library.



Step 4: Drawings, Bill Of Materials & Holecharts

PDX has the added capability to define hole charts. It will automatically recognize all the different kinds of bores in the plate and automatically place a table on the drawing. Below is the configuration options for how the bores are labeled and how the table will look. It can also output NC specific bore information.





Once done the entire project can be saved and re-used as a template for the next progressive die. The un-paralleled productivity gains vs. traditional methods make this tool extremely easy to justify. It keeps company standards in check and produces high quality deliverables at reduced costs. Since the whole assembly is in Pro/ENGINEER any change made to the designed part will regenerate thru the

PDX 5.0 Sneak Peak

Within the next couple of months PDX 5.0 will be released. Here is a quick overview of the improved functionality.

General Enhancements:

- Native Windows Look and Feel
- Support for Windows 64-bit

Strip Layout:

- Support for proxy stamp reference parts to make references more robust
- Improve calculation time for stamp reference part interference checks

Plates Wizard:

- Creation of structured die sets
- Improved handling techniques

Analysis Enhancements:

- Calculate line chain center

Hole Chart Enhancements:

- Modify hole chart content on the fly.
- Simplifies definition of bore tables.
- Setup of header and footer in the table.
- Control over page breaks

Bill of Material support

- Pre-define standard parameters for all components.
- Modify existing bill of material information.

Change Over functionality

- Create next version based on existing Die Sets. Rename parts automatically based on naming convention or by specifying the name.

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PTC Product Focus

Windchill R9 Part Classification and Reuse

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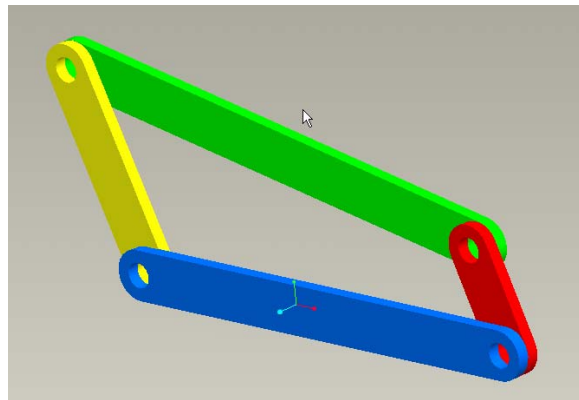
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Tips of the Month

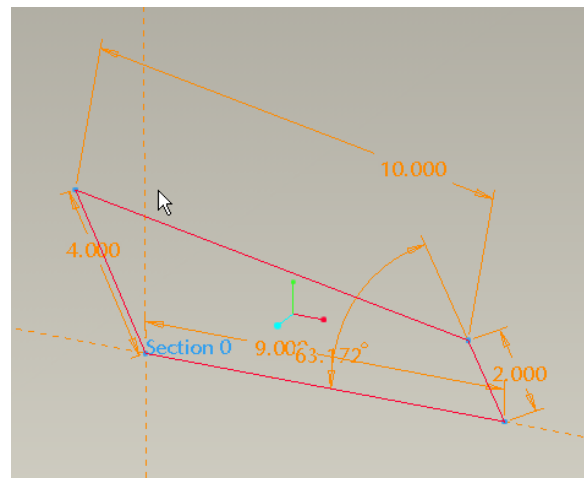
Motion Skeleton Example Using 4-bar Mechanisms

Following traditional top-down design techniques for assembly with moving components has been difficult in the past. Usually some variability is built in the skeleton to simulate motion. Unfortunately this does not allow the usage of mechanism/dynamics to study the motion of the assembly. With the introduction of motion skeletons in Pro/ENGINEER Wildfire 3.0 this issue has been resolved. This short example will give you an idea on how to get started with Motion Skeletons.

The goal is to build the following with full mechanism constraints, based off a skeleton that will define the length of all the links. At any given point the skeleton can be changed to adjust the lengths but the mechanism needs to be maintained.

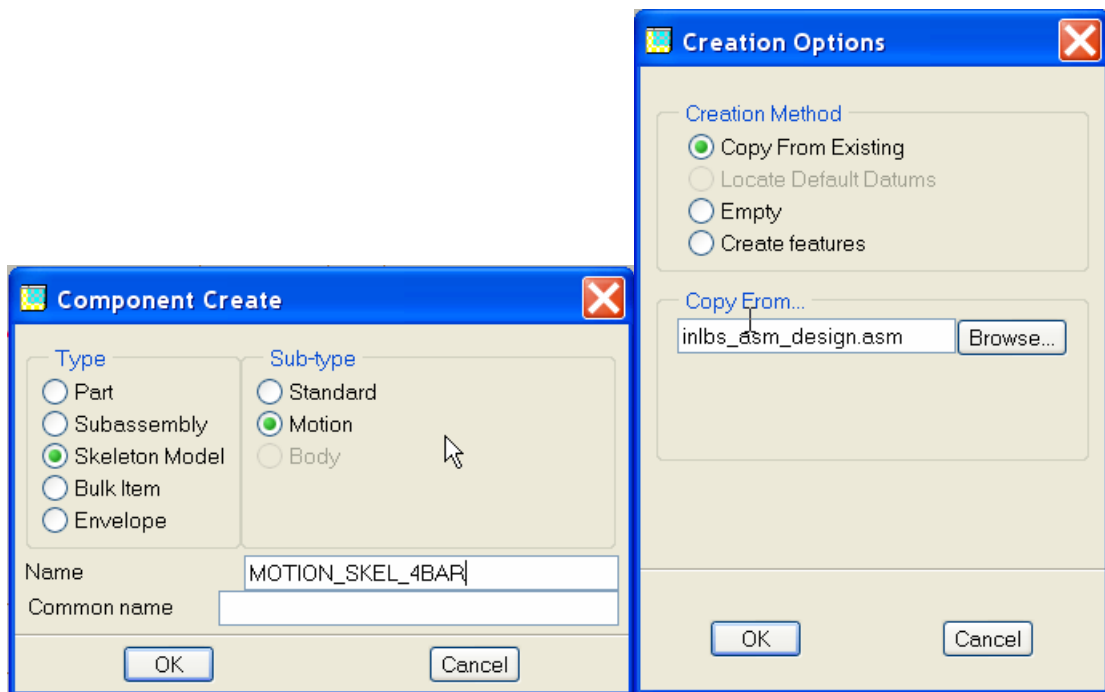


The skeleton for something like this would look like the following. Basically it is a simple sketch that defines the lengths of all the links. The angle in this case was needed to properly constrain the sketch but it is meaningless as far as the mechanism is concerned. Although it can be used to define the start position of the mechanism

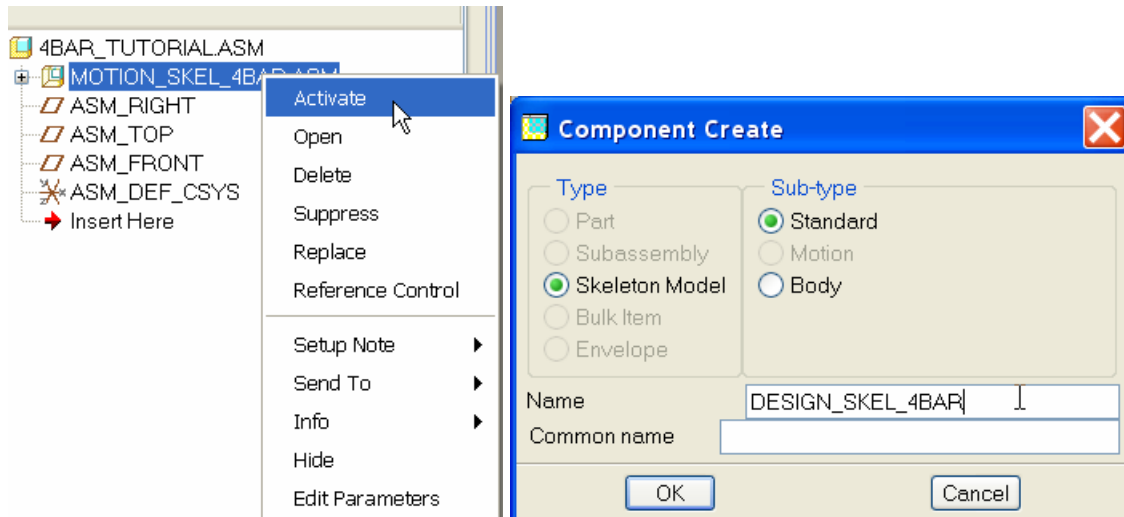


4-BAR Mechanism Tutorial using a Motion Skeleton

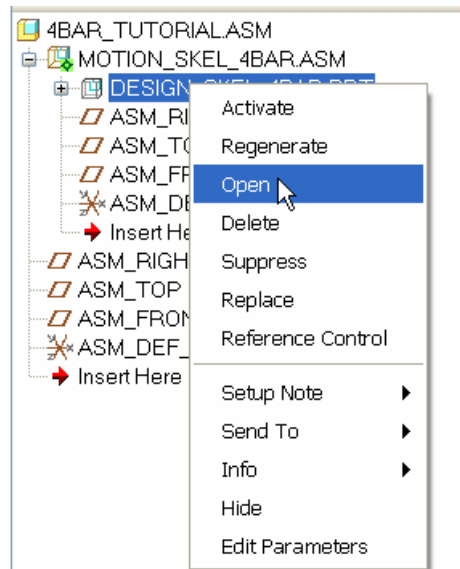
1. Create a new empty assembly.
2. Create a new component and select the **Skeleton Model Type** and choose **Motion** for **Sub-Type**. Choose a standard start assembly to **Copy From**.



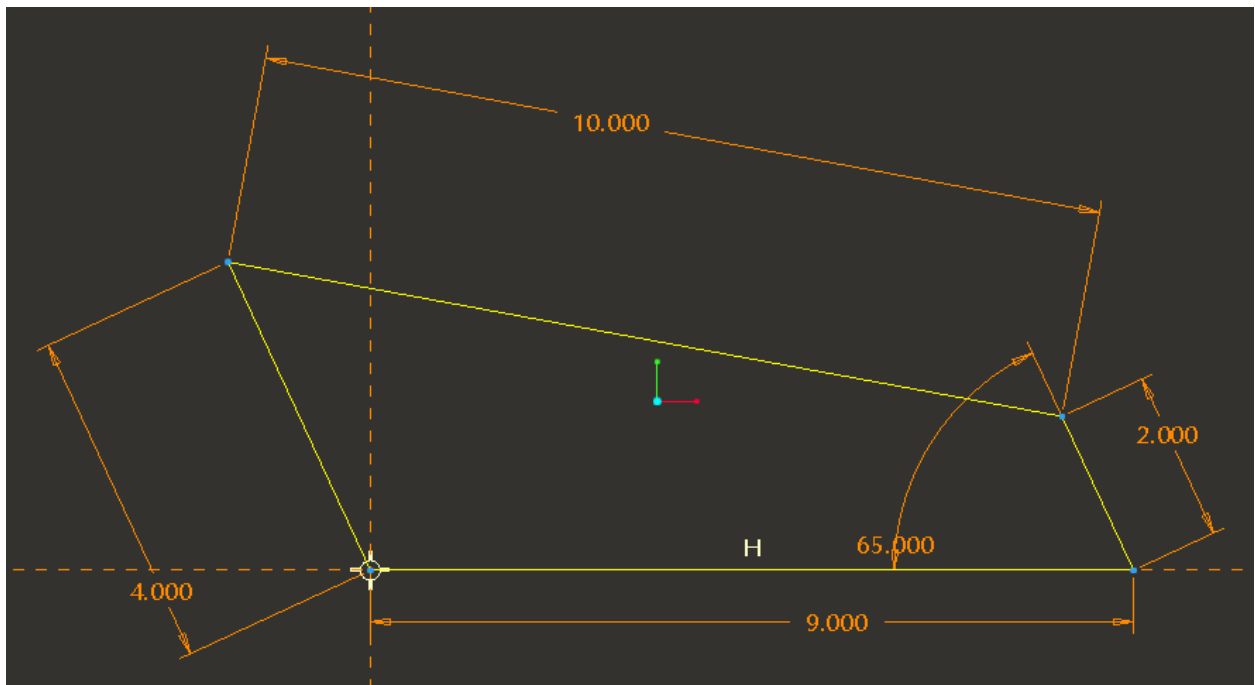
- Now we need to create the base skeleton as shown above. **Activate** the motion skeleton as shown below and create a new component. In this case choose **Standard** as this will be our reference for the bodies. Choose a standard start part as a template for the new skeleton.



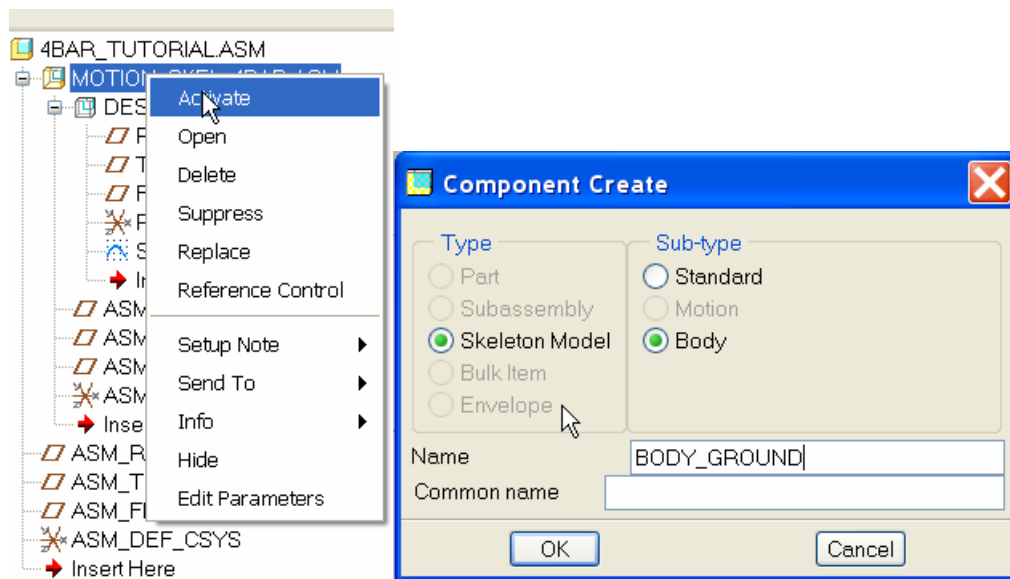
- The model tree will look like the following. **Open** the just created skeleton as shown.



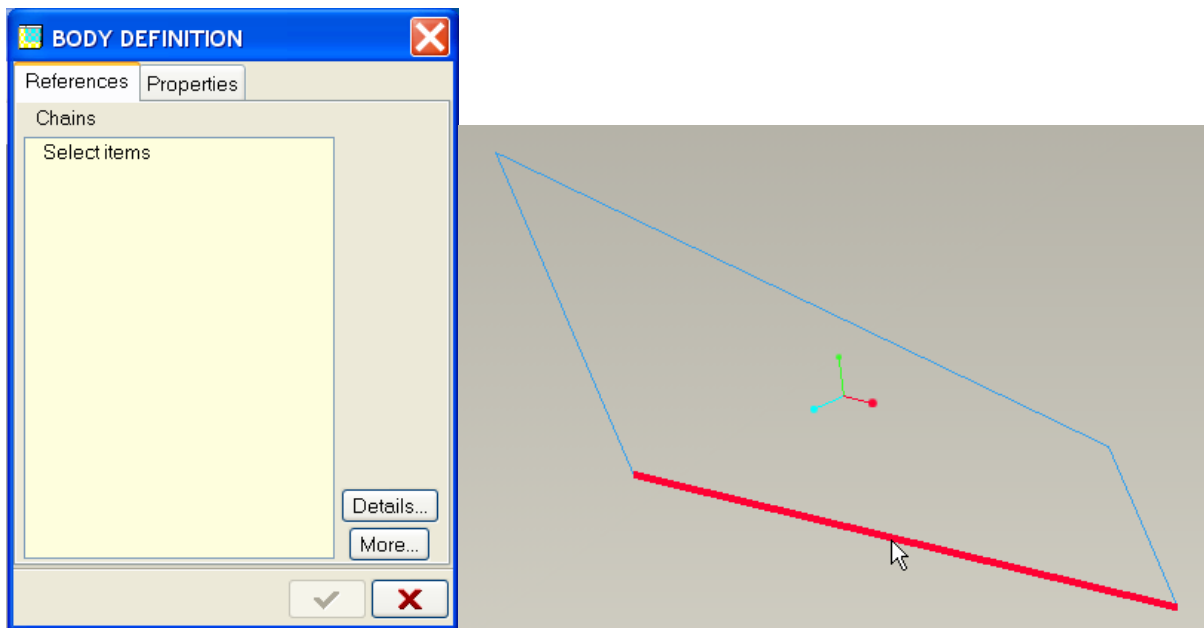
5. Create a sketch like shown below.



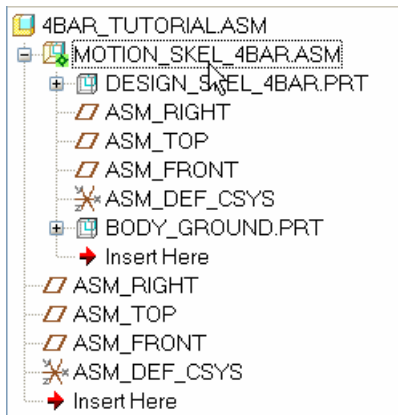
6. Return to the main assembly and **Activate** the **MOTION_SKEL_4BAR** skeleton. Create a new component and select **Body** as the **Sub-type**. Name it **BODY_GROUND**. This represents the fixed link at the bottom. The above image shows it as the blue link.



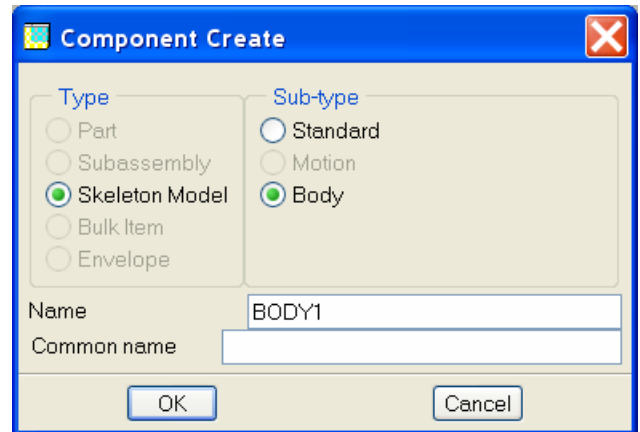
7. This brings up the following dialog box. Select the bottom curve as the Chain reference. This will automatically create a copy of this curve into the BODY_GROUND part. The first component created is always designated as ground.



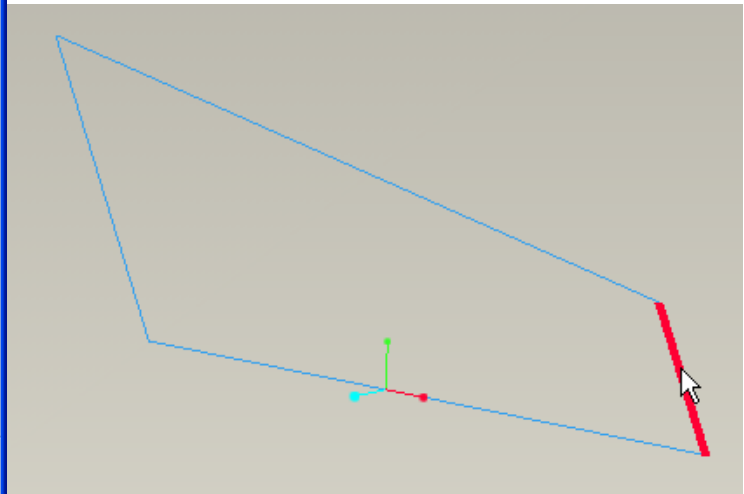
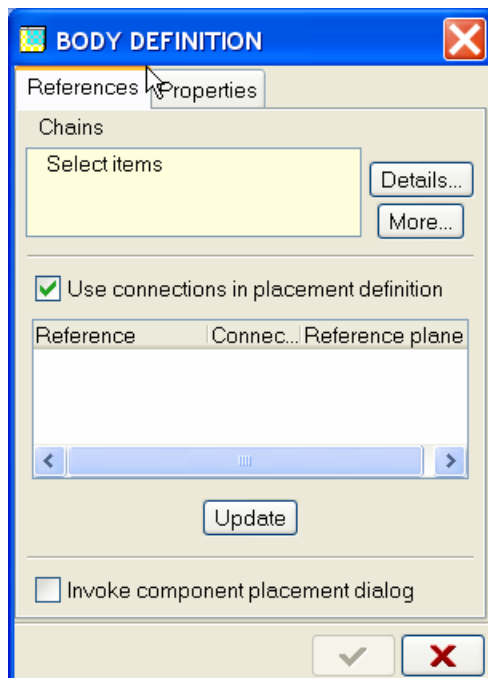
8. The model tree should look like this at this point.



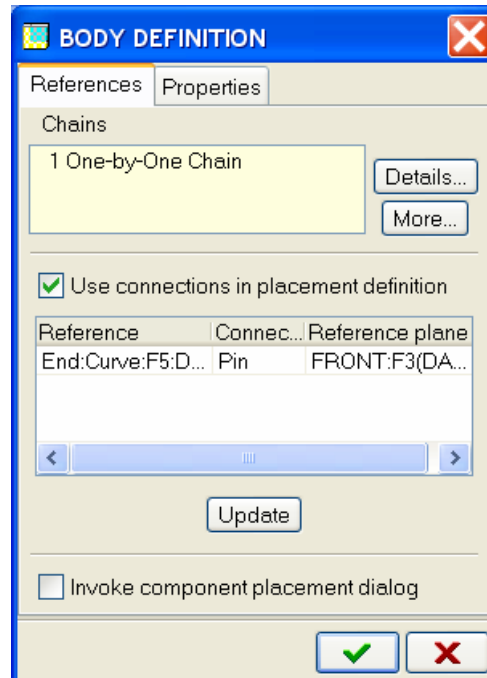
9. With the **MOTION_SKEL_4BAR** **Activated** create a new **Component>Body**. Name it **BODY1**. Ensure that you use a **Start Part** selected in the next dialog box.



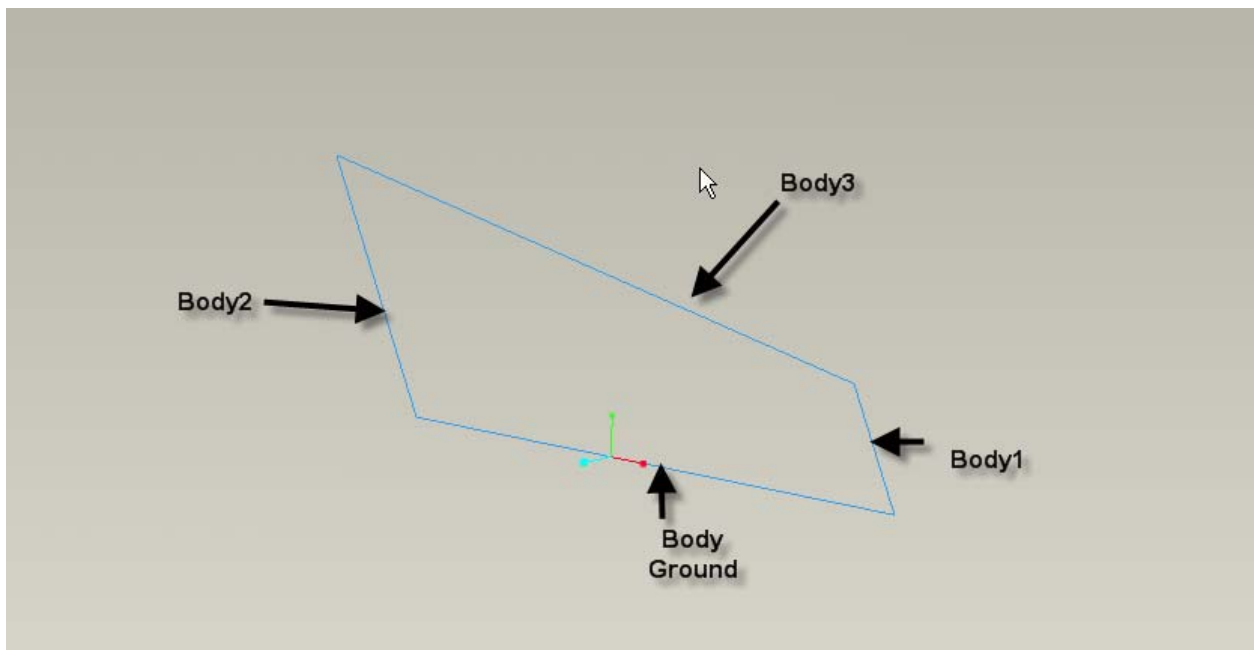
10. Now we reached the point where we define our first moving body. The following dialog box asks to select a chain reference. This will become the skeleton for the first link.



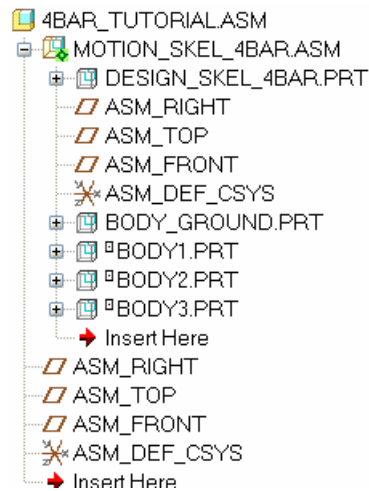
11. Once the curve is selected click on the **Update** button. Notice that it automatically recognized a Pin joint between the ends of the 2 curves. Sometimes this assumption is not the desired connection type. In this case select the **Invoke component placement dialog** radio box. This will bring up the standard assembly dialog dashboard and you can choose any type of constraint at that point.



12. Repeat steps 9 to 11 for the other 2 bodies. Name the bodies BODY2 & BODY3 respectively.

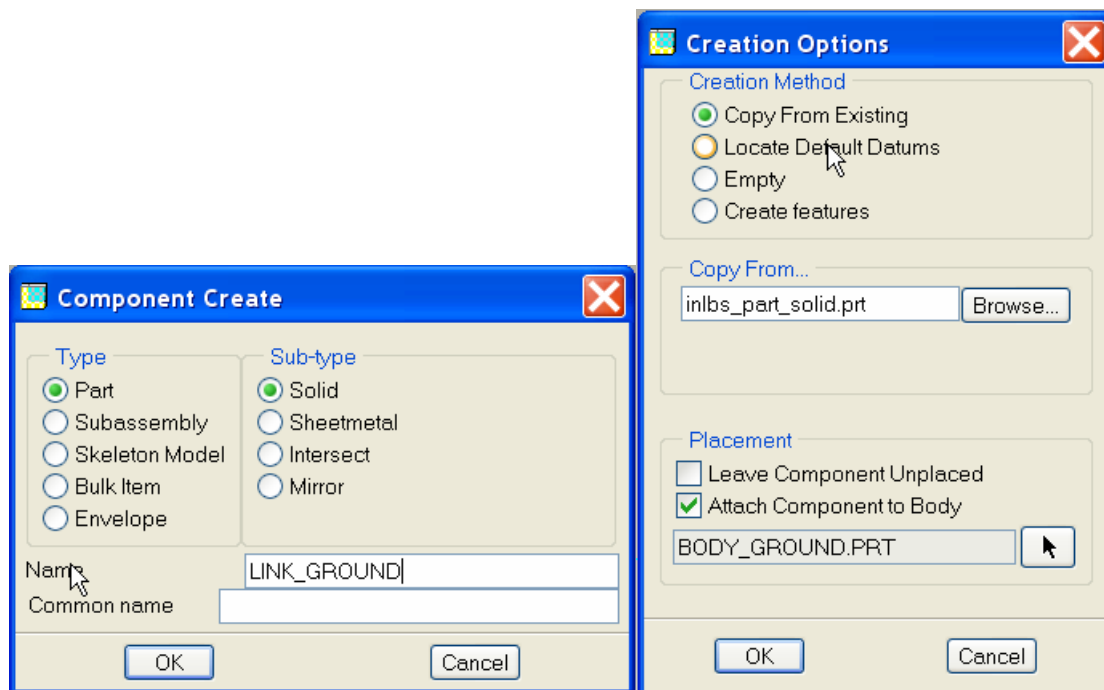


13. The model tree should look like this at this point. Now we are ready to create the parts.

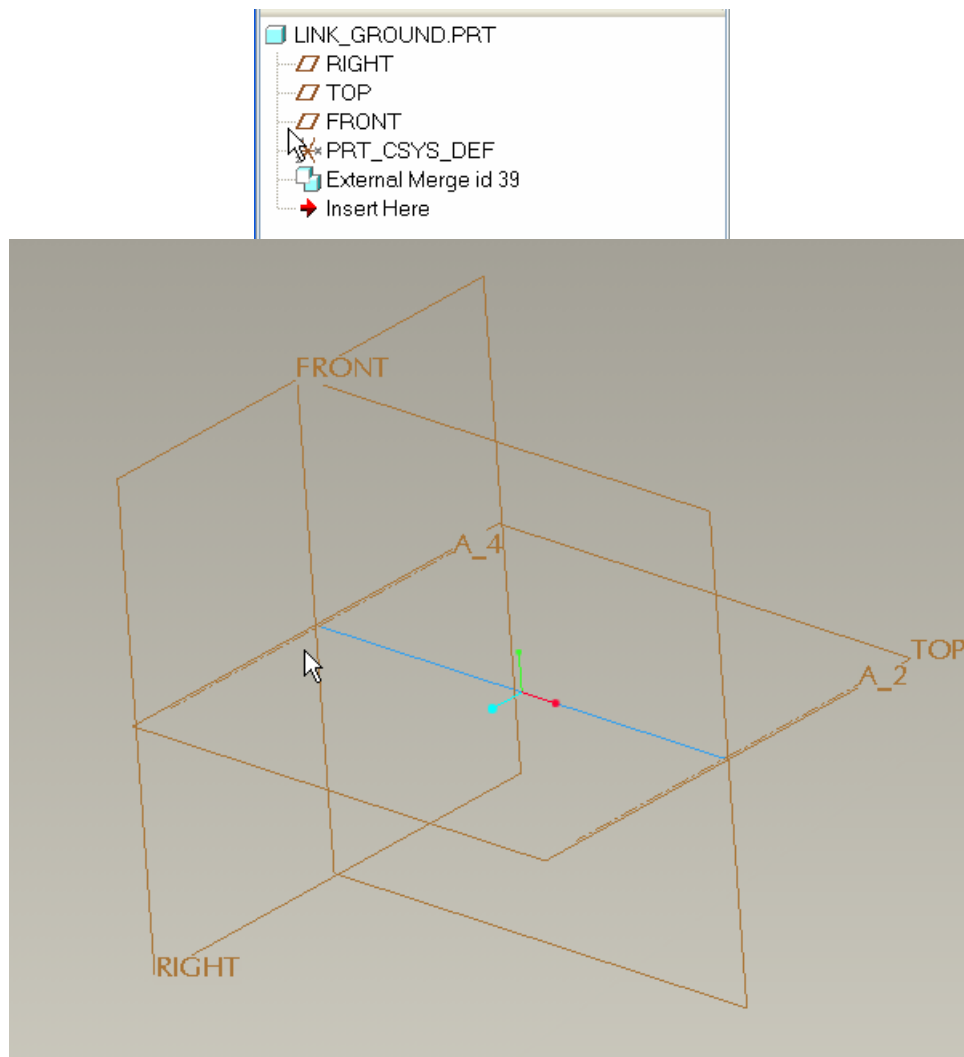


14. Activate the top level assemble (tip: Ctrl-A is a quick short cut for this)

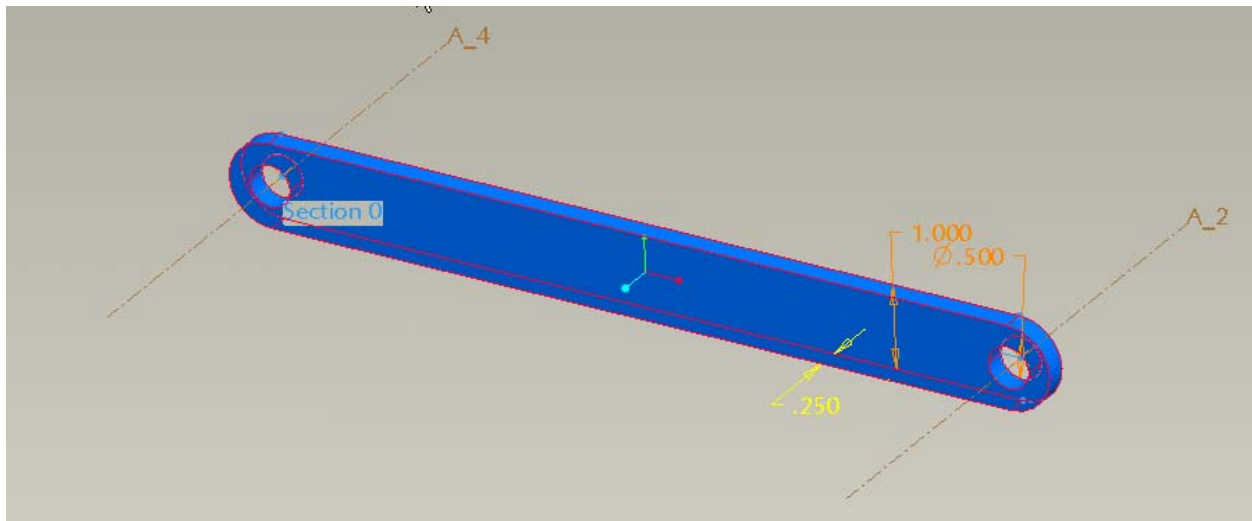
15. Create a new component. Now we are creating a **Solid** Part. Notice that there is a new selection at the bottom called **Attach Component to Body**. Select this option and choose the **BODY_GROUND** part.




16. Open this new part. You will notice that an External merge is added to the model. This is an automatic copy geometry from the **BODY_GROUND.prt**. Notice it contains the curve and the 2 axes that were also automatically created for the pin joints. This geometry can be used as references to create feature such Protrusions.

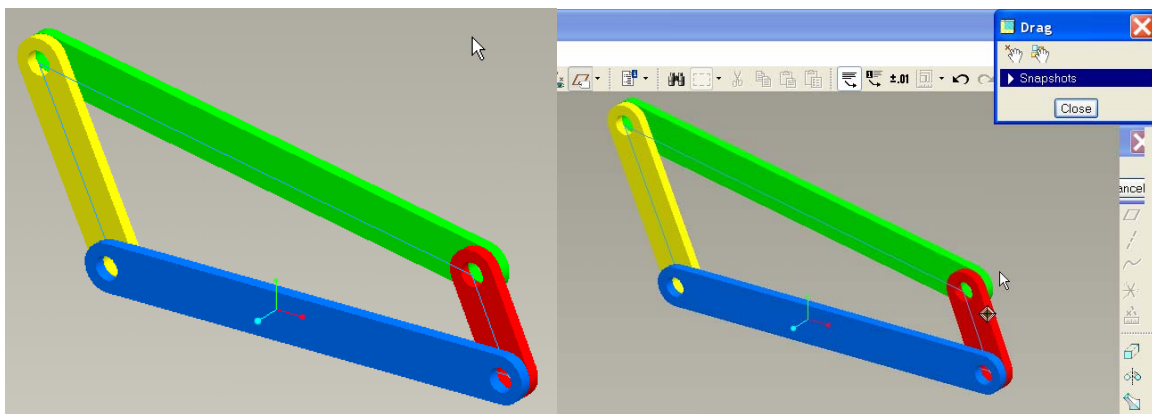


17. Create the solid link as shown, using the curve and axes as sketching references.

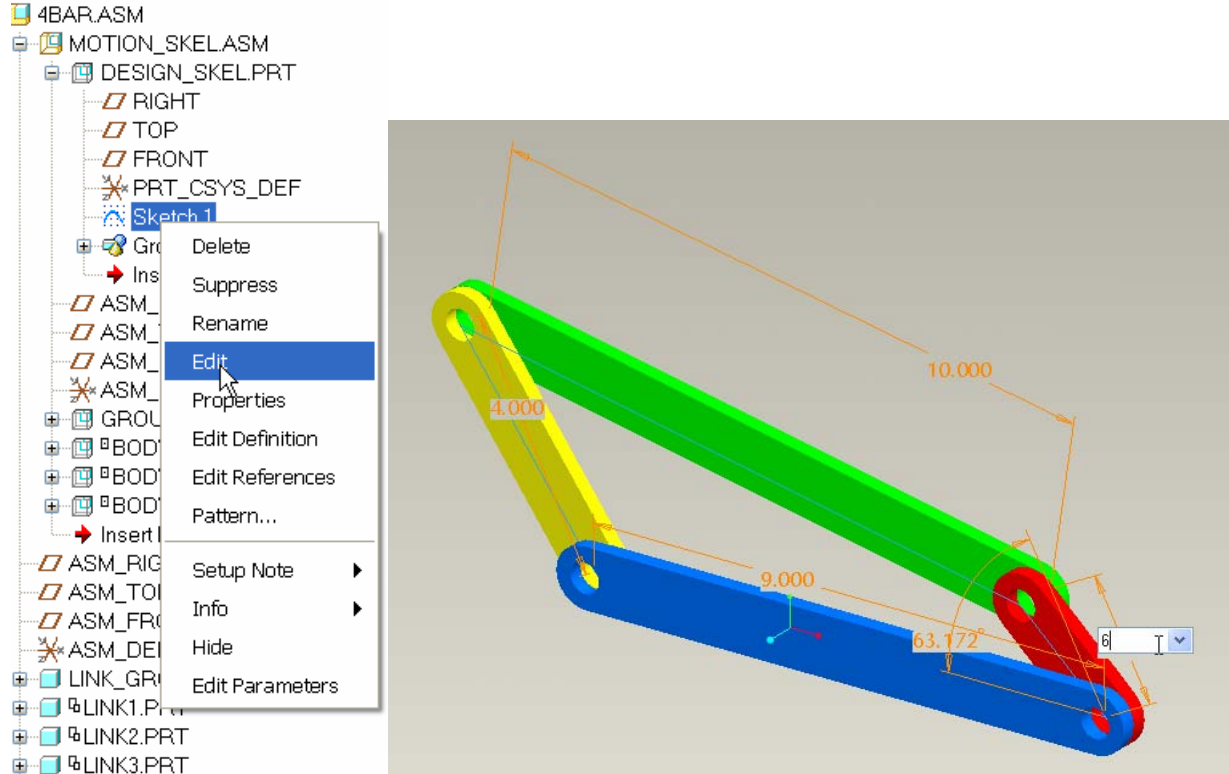


18. Repeat steps 15 to 17 for the other links.

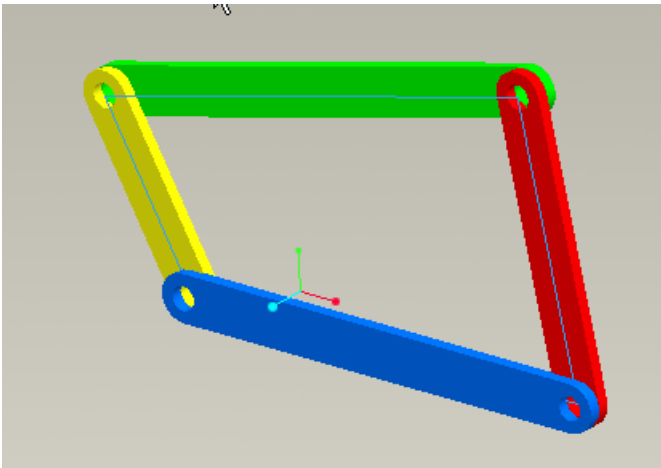
19. The resulting assembly should look like this. Experiment with the motion using the **Drag** command .



20. Now you can experiment by changing the dimensions of the skeleton. Edit the sketch located in the DESIGN_SKEL.prt. Change the 2" dimension to a 6" and regenerate.



21. The assembly regenerates and the mechanism is preserved.



This simple example demonstrates how Pro/ENGINEER Wildfire 3.0 automates the creation of mechanism connections using motion-skeletons and Top-Down design techniques.

This concludes the tutorial.

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Tips of the Month

Edit Structure (now with less java!)

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Announcements

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E-PROFILES IS HERE!!

We have been eagerly anticipating the debut of the new electronic version of Profiles Magazine and now it is here! This new web site will supplement the print edition of the magazine and will

provide new useful features not feasible with paper media. e-Profiles will provide you with 24x7, worldwide access to key information previously available exclusively in the print version. "Tips & Tricks," a popular feature pioneered by Pro/USER, has also moved to the web and will be expanded as the site matures.

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

<http://profilesmagazine.com/>

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Upcoming Events & Training Class Schedules

Upcoming, 2007 Your local Pro/Engineer User Groups
<http://www.ptcuser.org/rugs/>

June 1 – 4, 2008 Long Beach, CA 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.

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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

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

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