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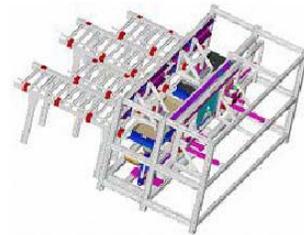
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PTC Product Focus

Expert Framework Extension (EFX)

Product Description: PTC's Expert Framework extension (EFX) was created to aid the user in the fast design of assemblies consisting of structural beams. These beams can consist of both standard and custom cross sections. Typical applications are for steel structures used in oil and gas refineries, offshore oil rigs, large machines, plants, buildings, and cranes, or for structures assembled with aluminum extrusions such as conveyor systems, handling/packing machines, and work cells. Custom beams can also be created for use in window frames, doors, and roofs.



Capabilities: EFX adds functionality to Pro/ENGINEERS assembly environment. The extension has an easy to use UI to aid in the transition from other 2D CAD products. It supports Pro/ENGINEERS top down design process allowing the user to create a concept or skeleton of the framework using curves and then assign the appropriate beam sections from either a standard or customized library. Brackets, endplates, screws and other connector elements can be added by drag and drop simplicity. Beams can also be replaced for other cross sections at any time or modified by mouse dragging or entering in new parameters. Beam and assembly drawings can be created automatically based on the assembly.

Benefits:

- Automates standard Pro/E functions resulting in a productivity gain of up to 10:1 over standard Pro/ENGINEER
- Supports the design process which is commonly followed by structural designers.
- Drawings are created “behind the scenes” for every beam.
- Drawings and BOMs updated automatically as the structure is modified.

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PTC Product Focus

PDMLink : Product Structure Baselines

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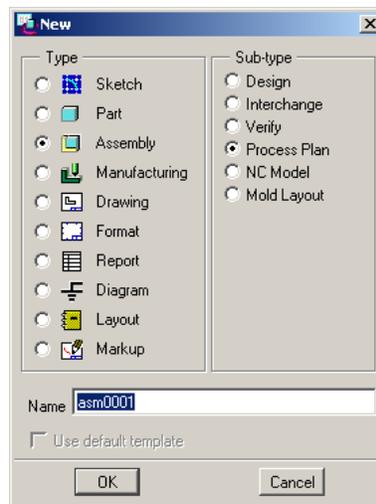
Tips of the Week

Setting Up Inspection Dimensions Using Pro/PROCESS

Summary: This tip describes how to use Pro/PROCESS to set up and document inspection dimensions. The user can combine as many dimensions into one step as desired as well as create as many steps as needed. The inspection dimensions will be created as parameters for each step in the Process assembly and can automatically be reported out using Pro/REPORT.

Step 1: Create the Pro/Process Assembly

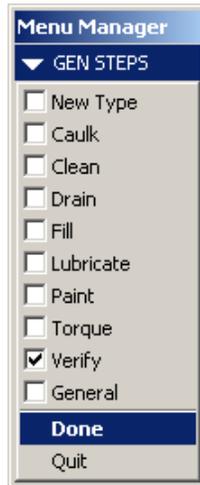
Select *File, New* to bring up the new file dialog box. Select the assembly and process plan radio buttons as shown below.



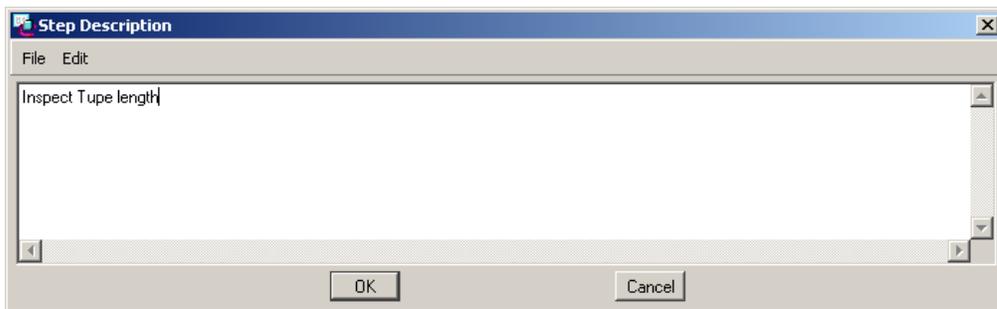
This will create a new assembly file and bring the user into the Pro/PROCESS environment. To begin, the user needs to bring in the part or the assembly to be inspected. Select *Sequence, New Step*, and accept the default of an assembly step by selecting *Done*. Choose *Open* to bring up the file open dialog box and select the model to set up the inspection dimensions. More than one model can be brought in if desired. The assembly steps are designed to bring in an entire assembly model and select components to be assembled at each step. Assembled components are shown in standard display (shaded, wireframe, or hidden line) and unassembled items are shown in phantom lines. If an assembly is selected all components will initially be shown in phantom line. Since the purpose of this process is not to show assembly steps, but rather inspection dimension steps, select all the components to be assembled in this first step. Once all the components have been selected, select *Done* from the menu manager. Additional information can be entered into this step using the optional items in the Step dialog box if desired. Since this step is only to bring in the model for this application, most likely no additional information will be needed. Select *OK* from the step dialog box to finish the step.

Step 2: Create First Inspection Step

Select *New Step* from the Step Sequence menu. Select *General* from the step type menu. This will define the step, bring up the step dialog box, and present the user with a list of the different types of general steps. Although setting the type is not critical, a verify step seems to make the most sense for this application. Select the check box next to *Verify* and select *Done* as shown below.



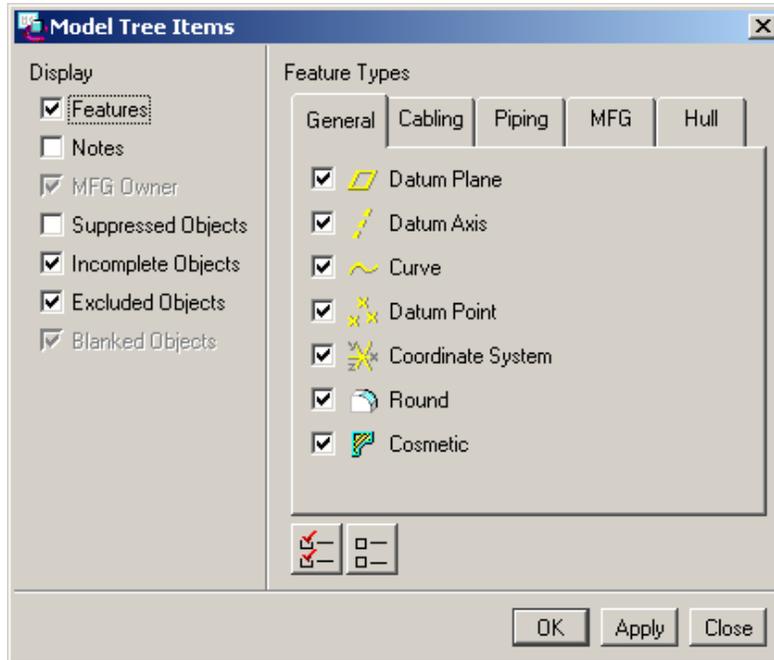
Add a description to the step by double clicking on *Description* in the Step dialog box. This will bring up the dialog box shown below and allow the user to type in a description.



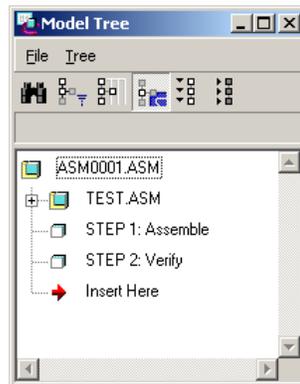
Select and define any other of the optional items shown in the Step dialog box. These items include views, explode states, time estimates, cost estimates, and simplified representations. None of these items need to be set to complete our task. Once finished, select *OK* from the dialog box to finish the step.

Step 3: Tying Inspection Dimension to the Inspection Step

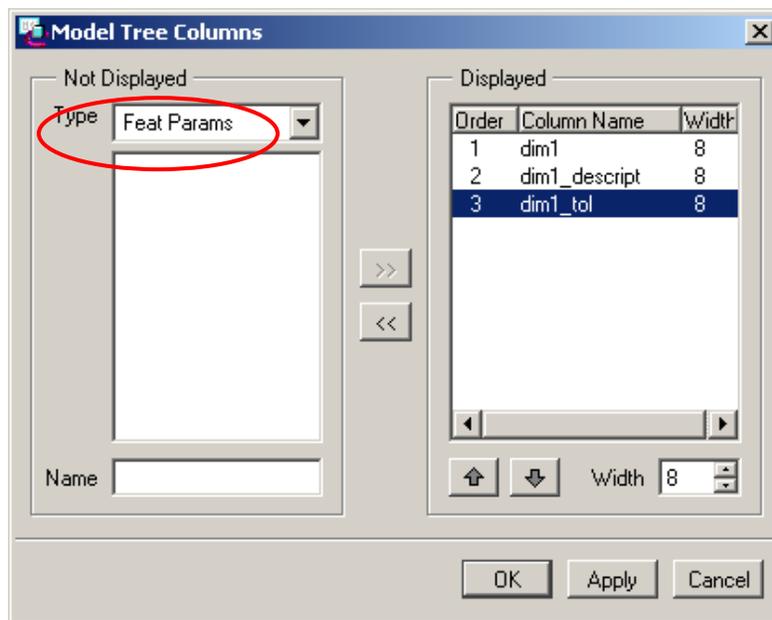
Parameters and relations will be used to create the inspection dimensions. The easiest way to create these parameters is to use the model tree. The parameters need to be tied to the step that was just created. These steps are considered features of the process assembly. Start by adding features to your model tree. Select *View, Model Tree Setup, Item display* from the pull down menus. Select the features check box as shown below.



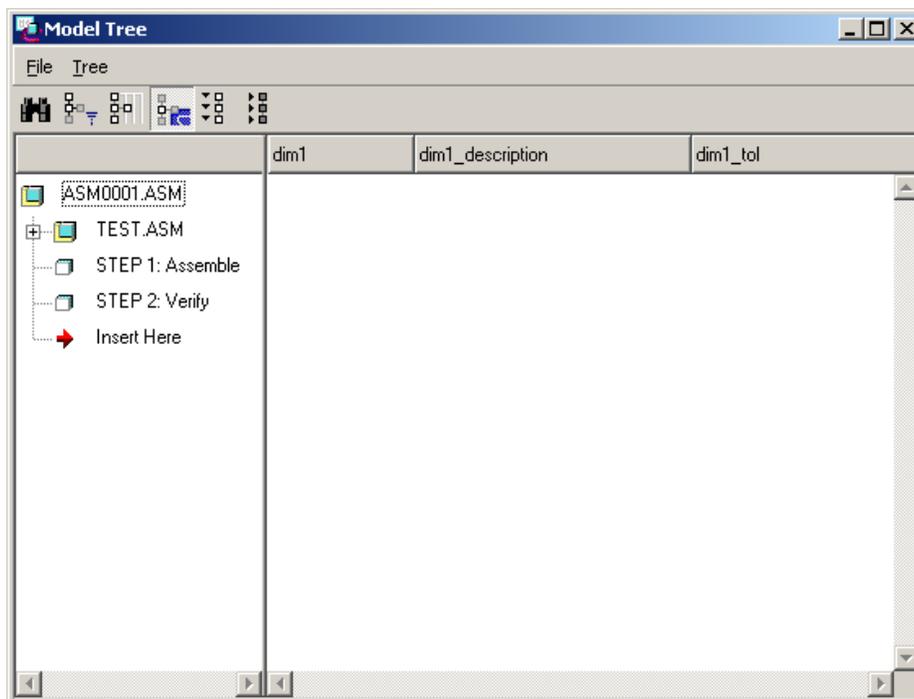
The steps should now be shown in the model tree.



Next add feature parameters to the model tree to create the inspection parameters. The parameters need to be feature level parameters. *Select View, Model Tree Setup, Column Display* from the pull down menus. This will bring up the Model Tree Column Display dialog box. Select Feature Params (Very Important!!) from the type pull down menu. Enter in a name for the parameters. For example, use dim1, dim1_description, dim1_tol. Use the >> to add the parameters to the displayed list. More parameters can be added as needed for multiple dimensions. The dialog box should look like the example below.



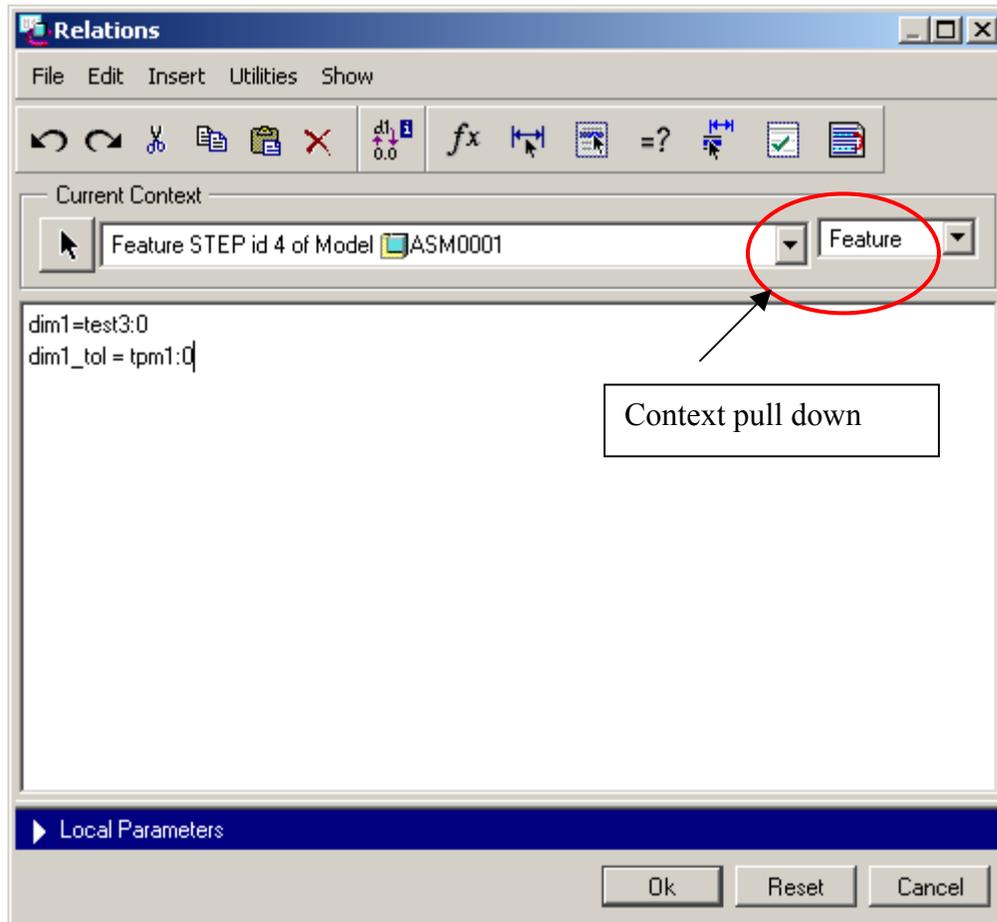
This will add the parameters as columns in the model tree as shown below.



A standard model tree configuration file should be saved off and simply retrieved for future use. This will eliminate all the previous work described in this step

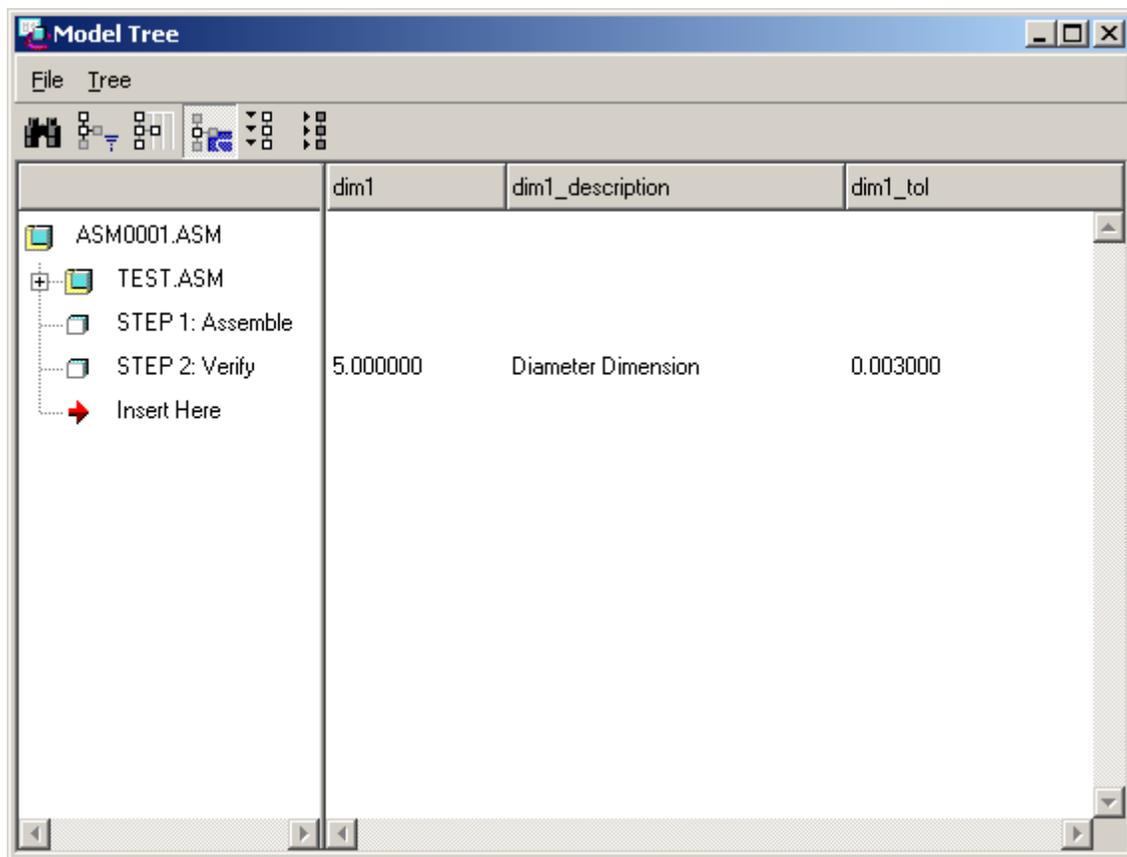
To continue with the process, select in the cells next to the verification step in the model tree to create the parameters. Create dim1 and dim1_tol as real number parameters and dim1_description as a string parameter. Next tie the parameters to a dimension value using a relation. Select *Modify* from the ASM Process menu and select on the model to bring up the desired inspection dimensions. Select *Relations* to bring up the Relations dialog box. **Change the current context to the feature level.** Do this by using the pull down tab in the relations dialog box and selecting feature. Then select the Step number from the model tree. Add relations that

set the previously created feature parameters to the inspection dimensions. Once finished the dialog box should look like the one shown below.



Test3:0 is the parameter name from the model of the dimension that needs to be inspected. Tpm1:0 is the parameter name of the tolerance on that dimension (Be sure to add the :0 at the end of the tolerance parameter in order to tell Pro/E which part to grab that dimension from). If the tolerance can't be seen on the dimension, make sure that tolerance display is turned on in the environment and then change the format of the dimension by selecting *Modify, Dim cosmetics, Format* and select desired tolerance format. Finally regenerate the model to update the values in the model tree.

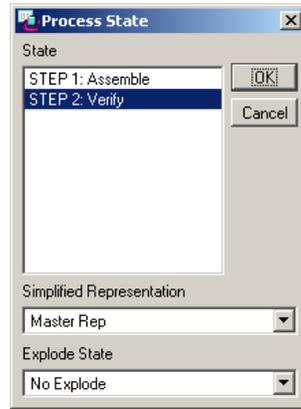
A description can be typed right into the model tree by selecting on the cell. Once finished, the model tree should look like the one shown below.



The above steps can be repeated to create more dimensions in one step or to add more steps with their own dimensions. Non model parameters could also be added by creating an analysis feature to measure the dimension and then writing a relation to the analysis feature.

Step 4: Drawing creation and Table Generation

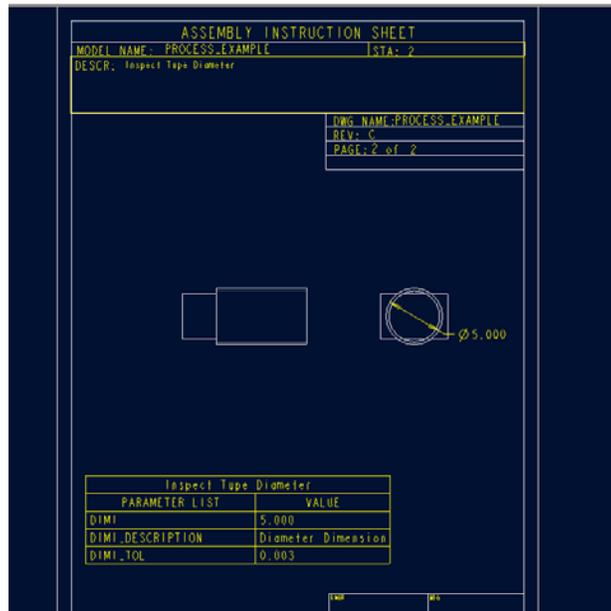
Creating a drawing based off a process assembly is no different than creating a standard drawing. Create a drawing by selecting File, New and selecting drawing from the radio buttons. In the New Drawing dialog box select the process assembly as the model and use a standard format. Pro/E will prompt you for the Process State. Select the first verification step. In this example there is only one.



The format will ask which sheet to use. The first sheet consists of a Pro/REPORT table that sums up all the steps in the process assembly. There are many different report parameters for a process assembly to format these reports. Please refer to Pro/HELP for additional information.

Add a second sheet to the drawing and notice the table in the lower left. This filled out automatically with the parameters tied to the verification step along with their value. The first row of the table reports the description of the step. A view could be added to further show where this dimension is. An example is shown on the next page.

Additional Sheets can be added by using the *Copy Process* pick in the Sheets menu. This option will add another sheet for another step in the process assembly that is formatted the same way as the current sheet.



Conclusion

This is one technique in Pro/E to get the inspection information out. Once a format file and tree.cfg file are set up the process should be fairly fast to generate this information. Text files output from this process could be managed in Intranet to link them to the correct versions of the assembly.

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Tips of the Week

Creating Product Structure Baselines

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PTC

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- 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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