



## Salt Lake City PTC Email Newsletter November 19, 2001

### Topics:

- PTC Product Focus: **Pro/Report**
- This Week's Technical Tip: **Auto Dimensioning of a Sheetmetal Flat Pattern**
- Upcoming Events and Training Class Schedule

- PTC Product Focus: **Pro/Report**

### **Pro/Report**

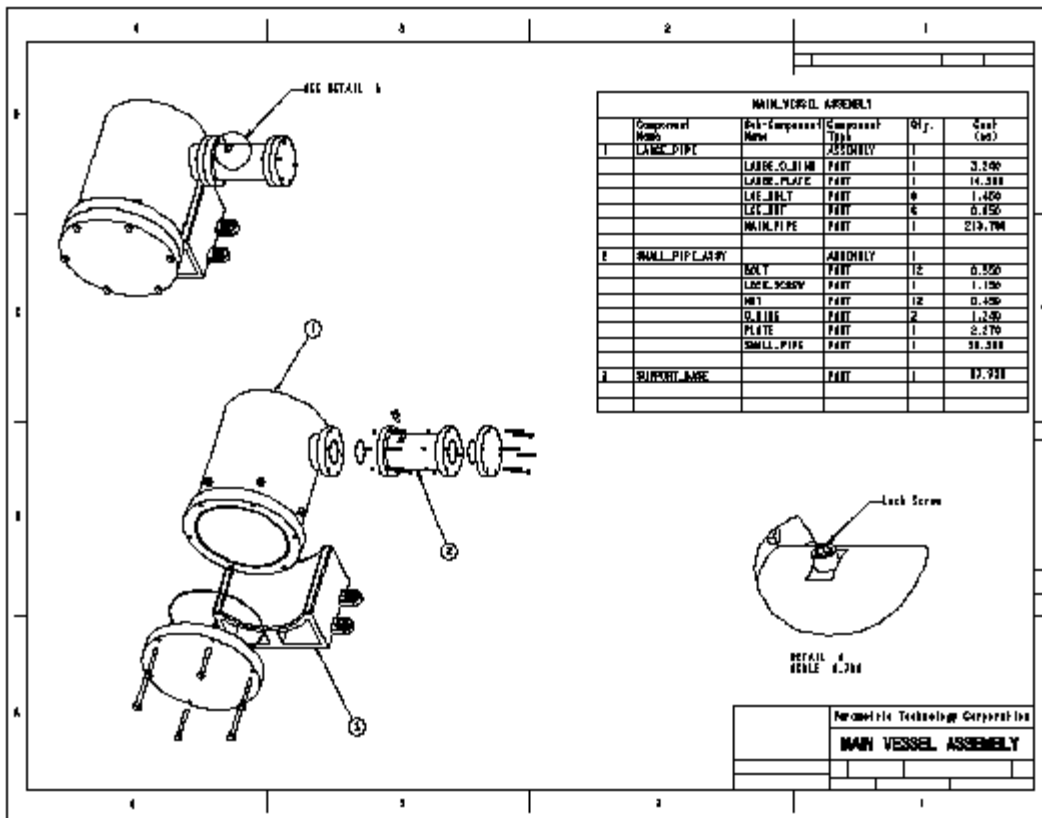
In Drawing mode, you can use Pro/REPORT to create dynamic, customized reports with drawing views and graphics. Pro/REPORT also works with Pro/HARNESS-MFG, Pro/PIPING, Pro/MFG, and Pro/DIAGRAM. By defining repeat regions, you can add model data to a report, which expands and shrinks as the model changes. You can

also use the following techniques to manage the data that appears in a report:

- Add filters to eliminate specific types of data from appearing in reports, drawing tables, or layout tables.
- Search recursive or top-level assembly data for display.
- List duplicate occurrences of model data individually or as a group in a report, drawing table, or layout table.
- Directly link assembly component balloons to a customized BOM and automatically update them when you make assembly modifications.

You can generate several kinds of output using Pro/REPORT such as family tables, associative reports, and graphical wirelists. A common example of a report is a customized Bill of Materials (BOM), such as the one shown in the next figure.

Report with Assembly Views



In Report mode, you can display data in tabular form in reports just as it is in drawing tables. The system takes the data that it reports in the tables directly from a selected model and updates it when you change the model. However, you cannot modify all model dimension values and geometry in Report mode—they are read-only. The system gives report files the file extension .rep. Report Mode Menus Report mode menus are abbreviated versions of the Drawing mode menus. This mode provides all of the associative abilities that Drawing mode offers, except that you cannot modify the model. Refer to the appropriate chapters of this guide for additional information on drawing commands.

## To Create a Report

1. From the Pro/ENGINEER menu bar, choose File New....
2. In the New dialog box, click Report and type a name in the Name box (or accept the default); then click OK.
3. In the Create Report dialog box, specify the report size or retrieve a format:

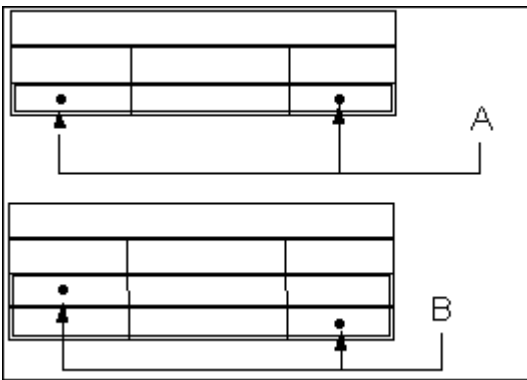
To specify the size, select Set Size and do one of the following:

- Click Portrait in the Orientation box (to make the height larger than the width) and select a standard size from the Standard Size list.
- Click Landscape in the Orientation box (to make the width larger than the height) and select a standard size from the Standard Size list.
- Click Variable in the Orientation box to define both the height and width dimensions. Select Inches or Millimeters and type values in the Width and Height boxes.

To retrieve a format, click Retrieve Format and select a name from the Name list in the Format box. You can also type [?] or click Browse to select a name from the Open dialog box.

4. Click OK. The system displays the report as specified and the REPORT menu appears.
5. Create a drawing table on the drawing sheet by choosing REPORT > Table > Create.
6. Define a repeat region by choosing TABLE > Repeat Region > Add.
7. Select cells to repeat with model information.

## Creating a Repeat Region



- Selecting A yields highlighted (one-row) repeat region
8. Selecting B yields highlighted (two-row) repeat region
  9. Choose Enter Text and type title text in a row or column outside of the repeat region (unless you want that text to repeat with every occurrence of model data).
  10. The repeat region in the table must contain symbol parameter information to be shown in the table. Choose TABLE > Enter Text > Keyboard; or choose ENTER CELL > Report Sym.
  11. Add the assembly model to the report by choosing REPORT > Views and typing the name of the assembly.
  12. The system automatically chooses Add View from the VIEWS menu, and the VIEW TYPE menu appears. To add views of the assembly to the report, add a general view to the sheet. If you do not want views of
  13. the model in the report, choose
  14. VIEW TYPE > Quit and VIEWS > Done/Return. The system still adds the model to the report, but it does not display it on the sheet. You can add drawing views at any time.
  15. If necessary, choose Repaint to display the contents of the table.
  16. Add a drawing format to the report or change it by choosing SHEETS > Format > Add/Replace. You can predefine tables on these drawing formats, save them, and recall them at any time into a report or
  17. drawing.
  18. You may also add the drawing table used in the report by including it in a drawing format that you can retrieve. When you add a drawing format containing a table to a report, drawing, or layout, the table becomes independent of the format. If you decide to replace the format, the system highlights the table, and you can delete it.

# Including Report Parameters in a Repeat Region

Pro/ENGINEER extracts values for parameters from model data and displays them in tables when the parameters are included within a repeat region. These parameters can be user-defined or they can already be defined by Pro/REPORT.

## Pro/REPORT Parameters

PARAMETER NAME	DEFINITION
&asm.mbr.cparams.name	Lists the names of all user-defined parameters in an assembly component. This parameter is defined for the parts making up the connector outside of the cabling environment.
&asm.mbr.cparams.value	Lists the values of all user-defined parameters in an assembly component. This parameter is defined for the parts making up the connector outside of the cabling environment.
&asm.mbr.cparam.User Defined	Lists the specified user-defined parameters used in an assembly component. These parameters are defined for the parts making up the Connector outside of the cabling environment.
&asm.mbr.name	Displays the name of an assembly member. To show tie wraps and markers, the region attributes must be set to Cable Info.
&asm.mbr.param.name	Lists the names of all user-defined parameters in an assembly member.
&asm.mbr.param.value	Lists the values of all user-defined parameters in an assembly member.

<code>&amp;asm.mbr.type</code>	Displays the type (part or assembly) of an assembly member.
<code>&amp;asm.mbr.User Defined</code>	Lists the specified user-defined parameter for the respective assembly components. Note that "&asm.mbr." can be used as a prefix before any User-defined parameter in an assembly member.
<code>&amp;fam.inst.name</code>	Displays the name of a family table instance.
<code>&amp;fam.inst.param</code>	Displays the dimensions of suppressed features in a 2-D repeat region as dashes.  This is not a complete symbol; only suppresses dashed dimensions if the drawing setup file option "dash_supp_dims_on_region" is set to "yes," the default.
<code>&amp;fam.inst.param.id</code>	Displays the ID of a family table parameter if it is a dimension.
<code>&amp;fam.inst.param.name</code>	Displays the name of a family table parameter.
<code>&amp;fam.inst.param.value</code>	Displays the value of a family table parameter for an instance.
<code>&amp;lay.param.name</code>	Lists the names of all user-defined parameters in a layout.
<code>&amp;lay.param.value</code>	Lists the values of all user-defined parameters in a layout.
<code>&amp;mdl.param.name</code>	Lists the names of all user-defined parameters in a model.

<code>&amp;mdl.param.value</code>	Lists the values of all user-defined parameters in a model.
<code>&amp;rpt.index</code>	Displays the number assigned to each record in a repeat region.
<code>&amp;rpt.level</code>	Shows the recursive depth of an item.
<code>&amp;rpt.qty</code>	Displays the quantity of an item.

### Assembly Component Parameters

The following system parameters exist for parameters that you enter into an assembly as feature relations and those that exist at the assembly level only:

- `"&asm.mbr.cparams.name"`
- `"&asm.mbr.-cparams.value"`
- `"asm.mbr.cparam.parametername"`

Note: These parameters do not exist at the part level as `"asm.mbr.parametername"` and other parameters can. In the next example, subassembly "LARGE\_PIPE" is assigned three parameters:

- "price," a number parameter
- "availability," a string parameter
- "in\_house," a "yes\_no" parameter

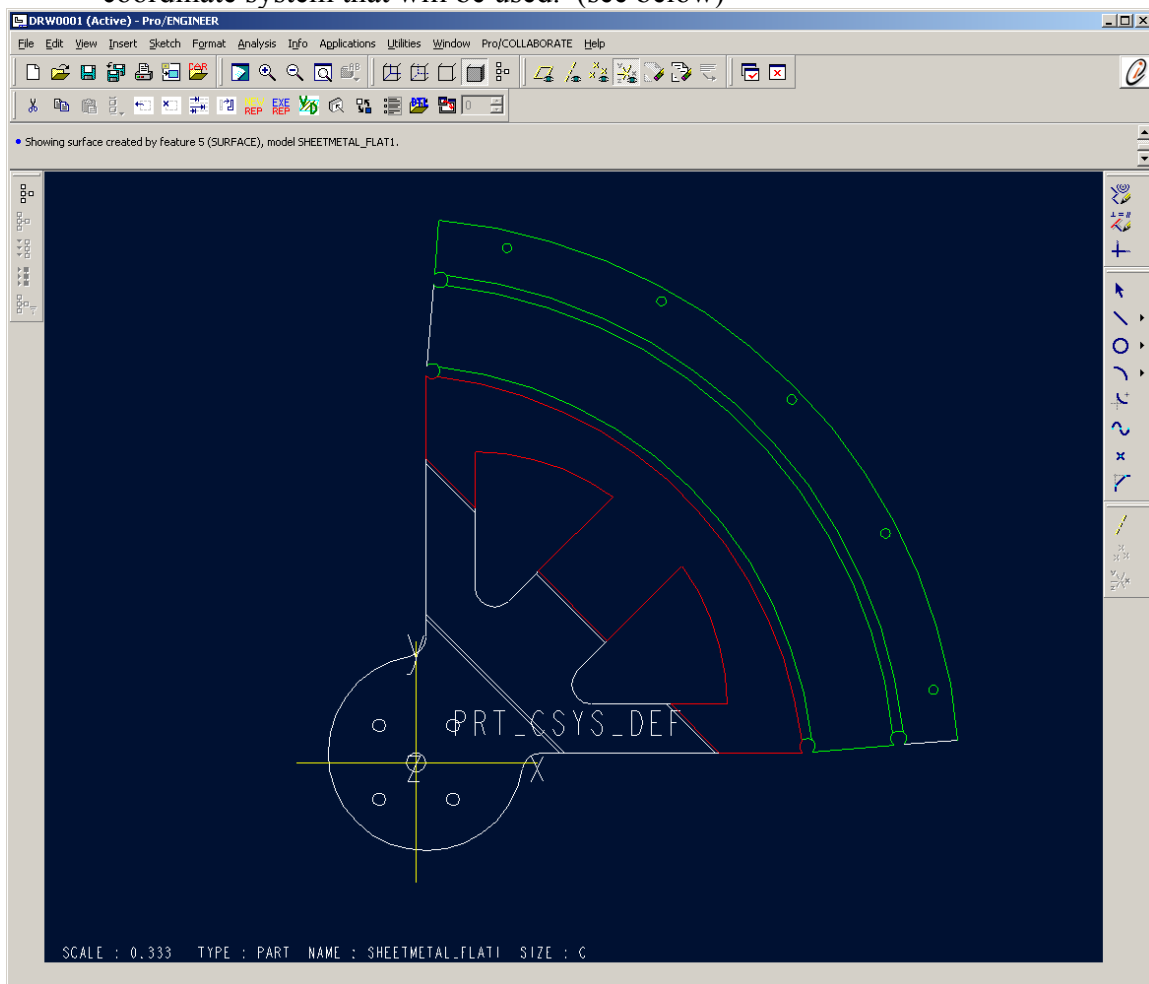
The part "LARGE\_PLATE" is given the "price" parameter as well. In subassembly "SMALL\_PIPE," one occurrence of part "O\_RING" is assigned the "price" parameter also; the parameter exists in assembly "SMALL\_PIPE" and not in part "O\_RING." When the system parses the component parameters for the top-level assembly "Main Vessel," it includes the parameter. Every parameter that is assigned to a component in this example is added to its parent assembly as a feature relation. This figure shows the repeat region template using the component parameters. Repeat Region Template Using the Component Parameters

- *This Week's Technical Tip:* *Auto Dimensioning of a Sheetmetal Flat Pattern*

## *Auto Dimension a Sheetmetal Flat Pattern* *In Drawings*

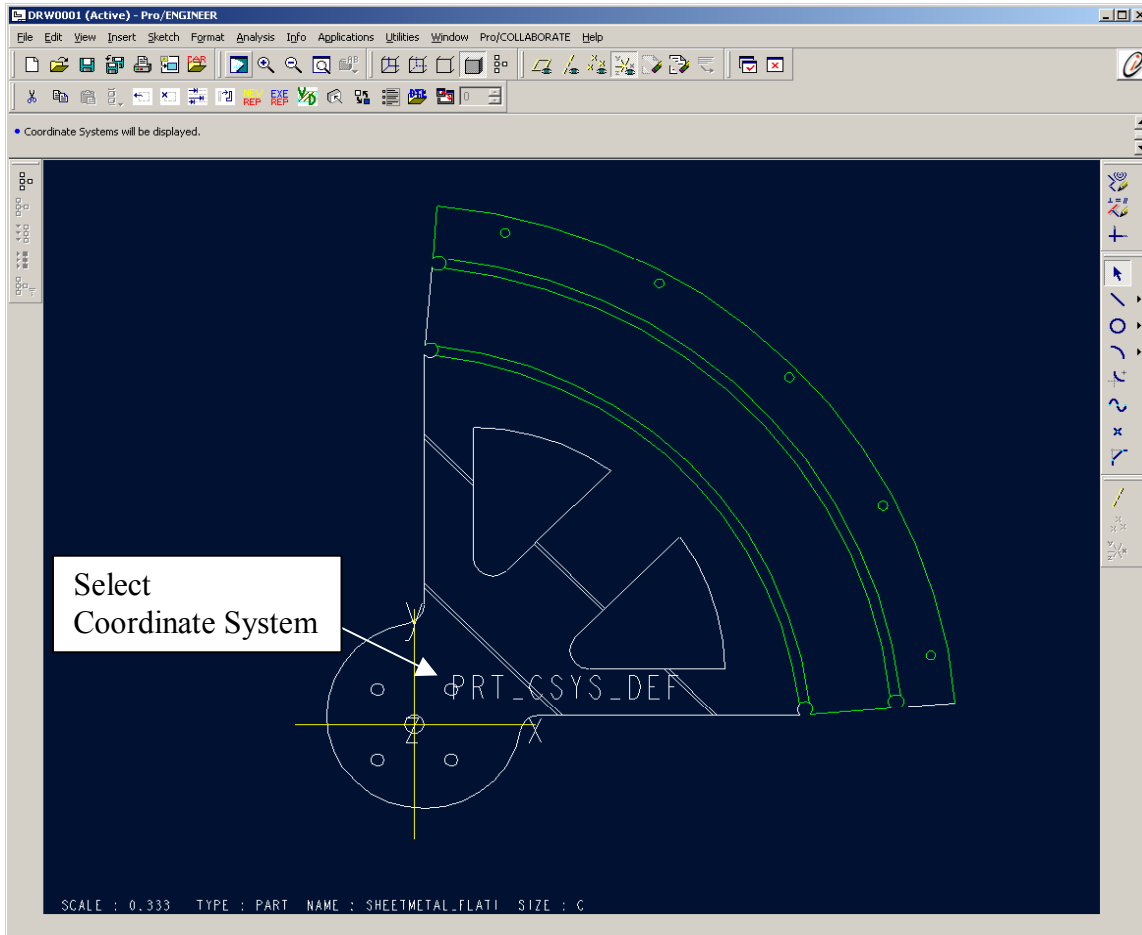
This steps below will explain how to automatically created ordinate dimensions on a view of flat pattern sheetmetal model. The dimensions will be auto-jogged by Pro/ENGINEER.

1. Create a new drawing in ProE and place a view of a flattened Sheetmetal part in the drawing.
2. To create the dimensions automatically, select **Insert, Dimension, Ordinate, and Auto Create**.
3. Next, follow the prompt and select a surface on the part that is parallel with the X, Y, plane of the coordinate system that will be used. (see below)



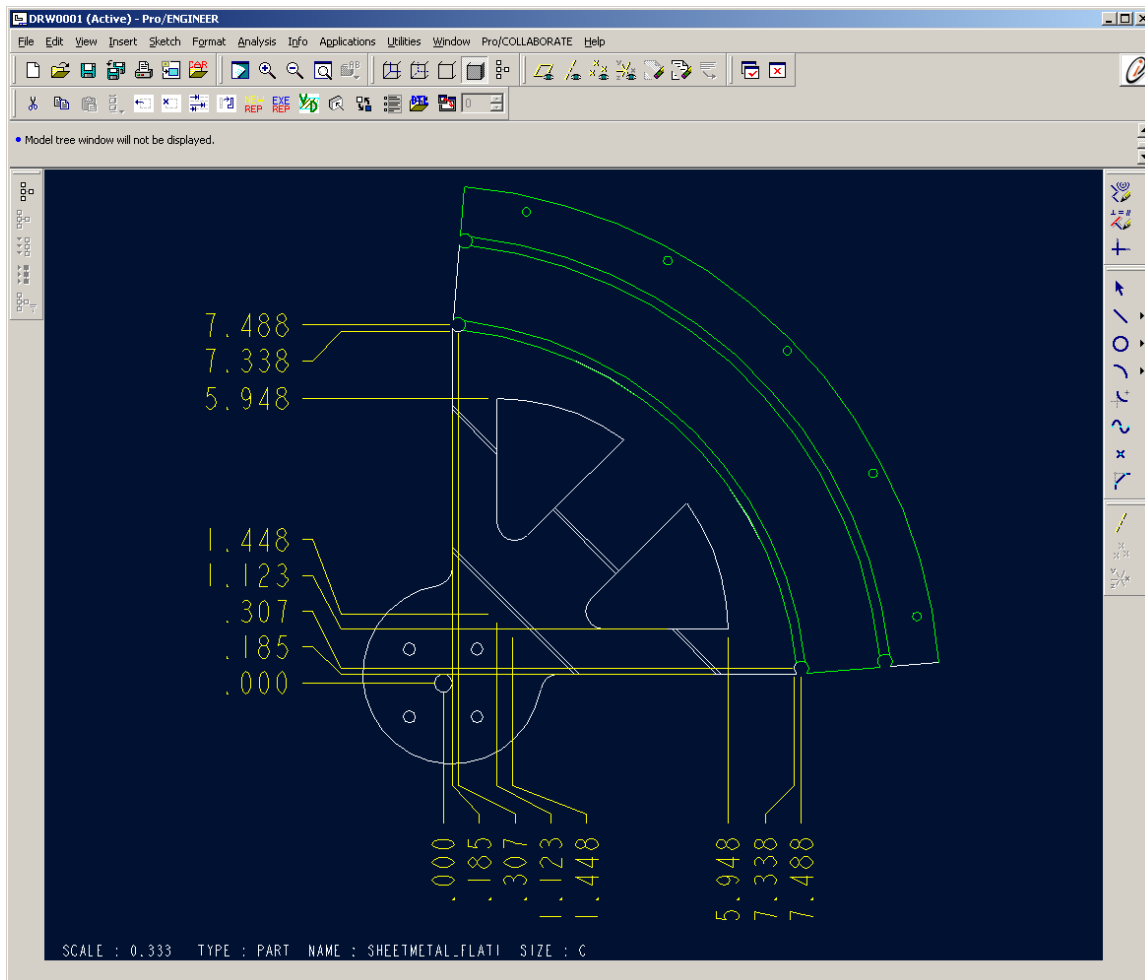
4. Choose **Done Sel.**

5. Next, select the coordinate system to use as the reference for the ordinate dimension baselines. (see below)





Pro/ENGINEER will then automatically add the dimensions to the view.



## Upcoming Events and Training Class Schedules

Date	Class Name	Location
December 3	Introduction to Pro/ENGINEER	Salt Lake City, UT
December 3	Fundamentals of Design	Phoenix, AZ
December 10	Creating Milling Sequences	Salt Lake City, UT
December 10	Fundamentals of Drawing	Phoenix, AZ
December 17	Core Pro/ENGINEER Release 2001 Update	Phoenix, AZ
December 18	Large Assembly Management Training	Phoenix, AZ
December 20	Pro/ENGINEER Advanced Top Down Design	Phoenix, AZ

Register for a PTC training course.

By phone: 1-888-782-3773 or 1-781-370-5103

On-line: [www.ptc.com/services/edserv/index.htm](http://www.ptc.com/services/edserv/index.htm)



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