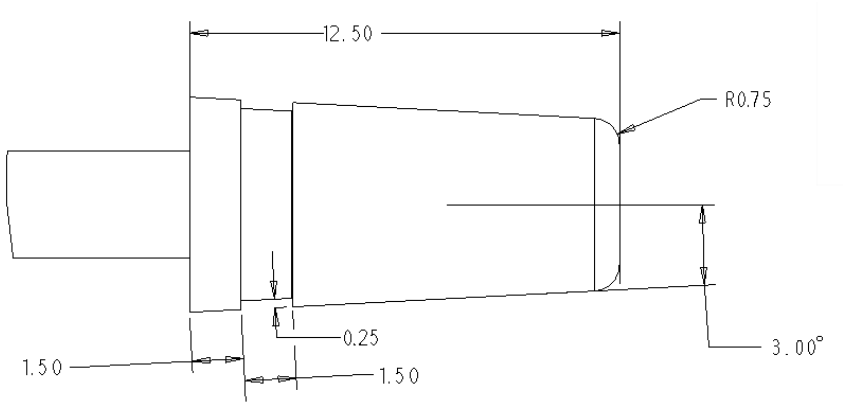
6

Creating Drawings in Pro/ENGINEER



This chapter shows you how to bring the cell phone models and the assembly you've created into the Pro/ENGINEER Drawing mode to create a drawing. A mechanical drawing consists of specifically oriented representations of the object being drawn. Each of these representations is called a *view*. There are several types of views, each drawn and dimensioned to show a certain kind or amount of detail.

In this exercise you'll learn how to add an assembly drawing to a bill of materials using the table function. On subsequent sheets you'll add a few of the more widely used view types to detail some of the parts. The object of these exercises is not to create a full drawing of all eight parts. Instead you are introduced to the basic workflow of the process of creating drawings.

First, review the information on dimensions and views in the next section, then follow the exercises to create the drawing files.

Understanding Dimensions and Associativity

Applying dimensions to drawing views in Pro/ENGINEER uses a different process from that of other programs. The difference is the Pro/ENGINEER associativity factor— instead of using the drafting program to add a dimension to a view where you need it, you choose to selectively show a dimension that has already been passed, along with the view, from the 3D model. This dimension is actively linked to the 3D model. As a result, you can directly edit the 3D model through the dimension in the drawing. When shown in the drawing, these dimensions are called *driving* dimensions, because they can be used to drive the shape of the model through the drawing.

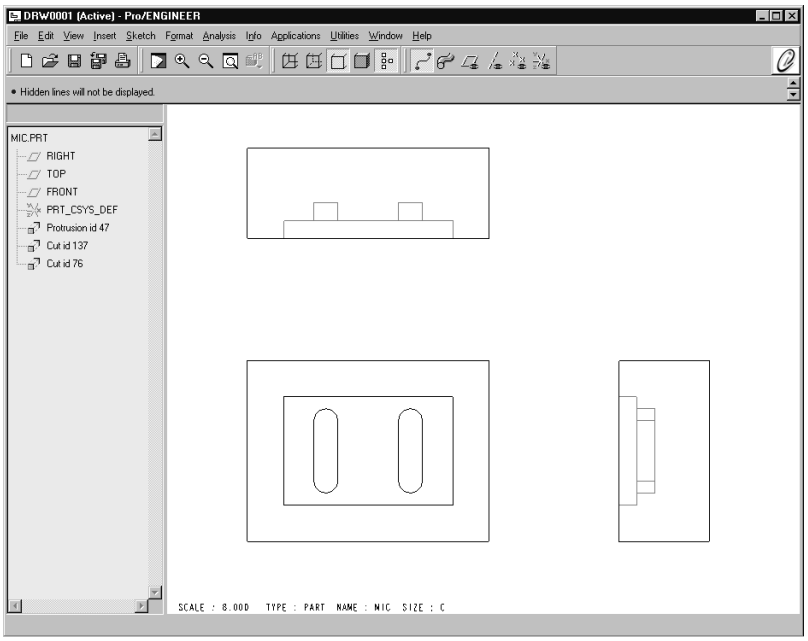
Of course, there will be instances where you need additional dimensions to show the same value for the same object, for example, a view repeated on another drawing sheet. To add these, use the **Dimensions** commands from the **Insert** menu. These inserted dimensions are called added or *driven* dimensions, because their association is only one-way, from the model to the drawing. If dimensions are changed in the model, all edited dimension values and the drawing are updated.

Displaying passed dimensions rather than adding them means your drawing will not be overdimensioned and there won't be any conflicting values on different drawing sheets for the same objects. Once driving dimensions are shown, they can be hidden. They are never permanently deleted.

The nature of driving dimensions has consequences that a drafter must bear in mind when making drawings in Pro/ENGINEER:

- Only one driving dimension for each model dimension may exist in a drawing. A drawing may have several views of the same object, but only one driving dimension for each feature of the model can be displayed. This is to avoid overdimensioning the drawing and to avoid editing a driving dimension on one sheet but not another. You can move a driving dimension from one view to another, for example, from a general view to a detailed view where it is more appropriate. To provide dimensioning for views when driving dimensions are "used up" in other views, you use added driven dimensions.
- It is possible to unintentionally edit the model. If a driving dimension is edited, it turns white as a warning that there is a discrepancy between the drawing and the model. When you regenerate the model, the drawing accepts the new dimension. The link between model and drawing can be broken when configuration options have been set to do this, but this is not recommended.

Drawing view of 3D model with no dimensions showing



Detail Items

Dimensions are the most important detail items passed from the 3D model, but they are not the only items available or necessary to detail a drawing. In the same way you show or hide dimensions, you can show notes, surface symbols, geometric tolerances, datum planes, and axes.

Adding Models vs. Adding Views

Before you can add a view of a model file to a drawing, the model file must be associated with the drawing file. This is called “adding the model,” not to be confused with adding a view. You can associate any number of models with the drawing, but only one model can be actively worked on at a time. The active model is the one ready to have views created for it. The active model name appears in the lower-left corner of the work area.

You add the first model when you create the new file, during the new file setup. Click **File** > **Properties** and then **Drawing Models** > **Dwg Models** > **Add Model** on the Menu Manager to add.

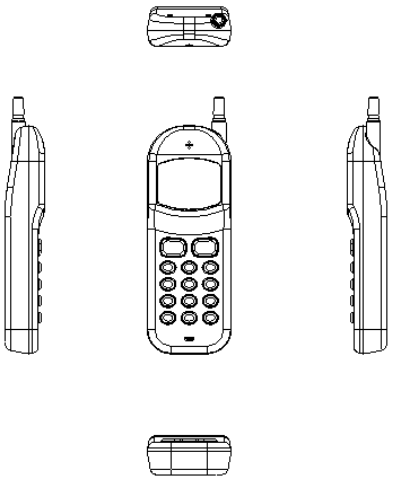
If you are working with an assembly, you can use the shortcut menu to add and activate any of the parts from the Model Tree.

Placing General Views and Projection Views

The first view you place will be a type called a general view. You can regard a general view as a parent view because its orientation and scale can be adjusted, and these properties determine the look of any projected views derived from it.

Projections (projected views) are views derived from the general view, which show alternate faces of the general view. Using projections you can quickly dimension every surface of a 3D object without repeating a dimension. Each projection view exists in either a horizontal or vertical projection channel to the right, left, above, or below the general view. Projection views are automatically aligned with the general view within their projection channels. By default, they can only be moved within the channels they occupy.

General view (center) and four projections



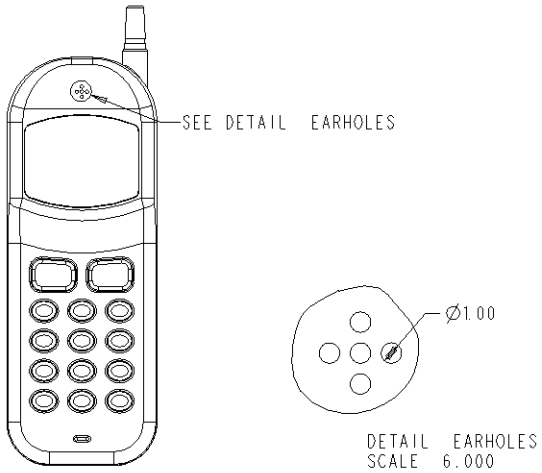
Note

Although alignment with the general view is the usual way to show projection views, you are not limited to this method. Double-click a projection view. The **Drawing Views** dialog box opens. Select **Alignment** and uncheck the **Align this view to other view** checkbox to unalign the selected projection view and place it anywhere on the drawing sheet.

Creating Detailed Views

A *detailed view* shows a small area of another view at a larger scale. The process puts a boundary and note around the detailed area on the parent view and creates the new detailed view to a new scale. You can add driven dimensions, or use the shortcut command **Switch View** to move a driving dimension from the parent view to the detail view.

Detailed view



Scaling Drawings and Views

Pro/ENGINEER automatically determines a scale for a new view based on the sheet size and the size of the model being placed. This scale value appears in the lower-left corner of the screen. To reset the scale, click **Edit** > **Value**, then click the scale readout and edit the value in the prompt line. (A value of 1 means the scale is one-to-one with the actual dimensions, a value of .25 means the drawing is scaled to one-quarter the actual dimensions.)

> **Note**
>
> If necessary, use the `default_draw_scale` configuration option to force a global default scale.

Only two view types may be scaled independently of the default scale setting: the general view and the detailed view. When you rescale a general view, its projections also rescale. Because a detailed view is an enlargement of a small area, you can scale a detailed view independently of its parent view as well as independently of the default scale.

When you place a general view, you choose **Scale** or **No Scale** as part of the view properties setup in the Menu Manager. If the view is placed as **No Scale** (the default) and you later want to change the scale, you must first modify the view type. If your drawing references several parts, you can set the scale independently for each part added to the drawing.

In other words, a default scale is set for each model added to the drawing. The scale readout in the lower-left corner refers to the active model. Any modifications values for scale, relations, and so forth, are applied to the active model.

Using Formats and Templates

Format files contain the collections of lines and text that border a drawing sheet. These serve to divide the sheet into sections and state the company name, design name, and so forth. When you associate a format file with a new drawing file, the format graphics appear on all sheets created in the drawing file. You can also change the format attached to a drawing file after the file has been created and saved.

There is a default format for each standard sheet size, installed in a default format directory. To customize a format, save a default format as a new format file, and then add text in the form of notes or graphics, such as a company logo.

Templates are a more advanced concept in Pro/ENGINEER. Templates contain all the format information and instructions on how to lay out views and projections automatically, as well as how to create tables and bills of materials. Templates are a powerful functionality that can save hours of interactive work on drawings that follow a standard flow. You may use templates in the course of creating drawings eventually, but this exercise will explain the more basic drawing functions.

Creating a New Drawing File

The finished drawing will have an exploded view of the assembly on the first page, with a bill of materials and BOM balloons calling out the parts. However, because these are the more complex elements of the drawing, you'll add them last. First you'll add a view of the antenna part to the drawing. Later in the tutorial you'll add the assembly view and the BOM. Start the new drawing file:

1. Click **File > New** and select Drawing from the New dialog box. Name your drawing ***antenna_DR_INL_PE_1*** and clear the Use default template check box. Click OK. The New Drawing dialog box opens.

2. In the Default Model field, Browse to find your antenna part file.

3. In Specify Template, click **Empty**. Click "Landscape" under orientation.
   In the Size panel, choose **Standard Size,** then **"A"** from the drop down menu.

Watch Video

Create a Table

Now, you will add a table to the bottom right corner of your drawing.

1. Go to **Table > Insert Table**. **Choose Descending > Rightward > By Length > Pick Point.**

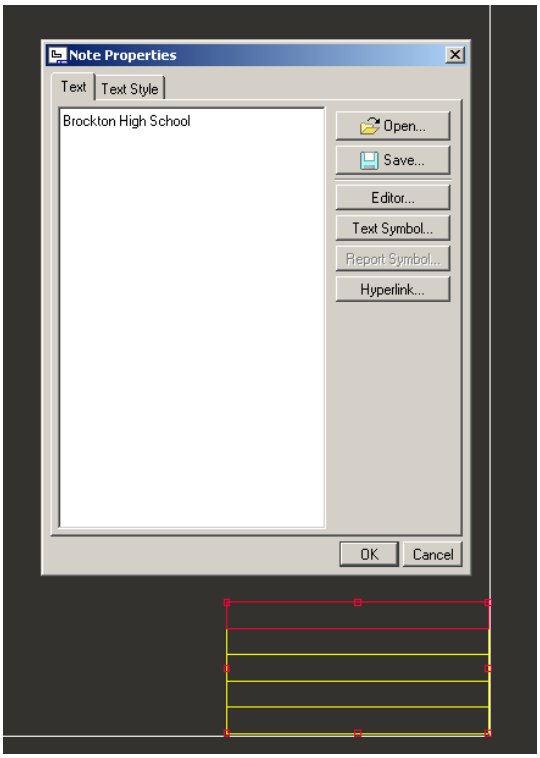
2. Click in the area of the bottom right of the Drawing.

3. You will be prompted to ***"Enter the width of the first column in drawing units (MM) [Quit]"*** in the dialogue just above the drawing area. Enter 60, then click the check mark or hit enter. Since you only need one column, click the check mark or hit enter again. This will prompt ProE to stop adding columns.

4. You will be prompted to ***"Enter the height of the first row in drawing units(MM) [Quit]"*** in the dialogue just above the drawing area. Enter 6, then click the check mark. Since you want five total rows, repeat the procedure four more times, then click the check mark or hit enter again. This will prompt ProE to stop adding rows.

5. You can move the table to the bottom right by clicking on the table and using the arrows to drag it.

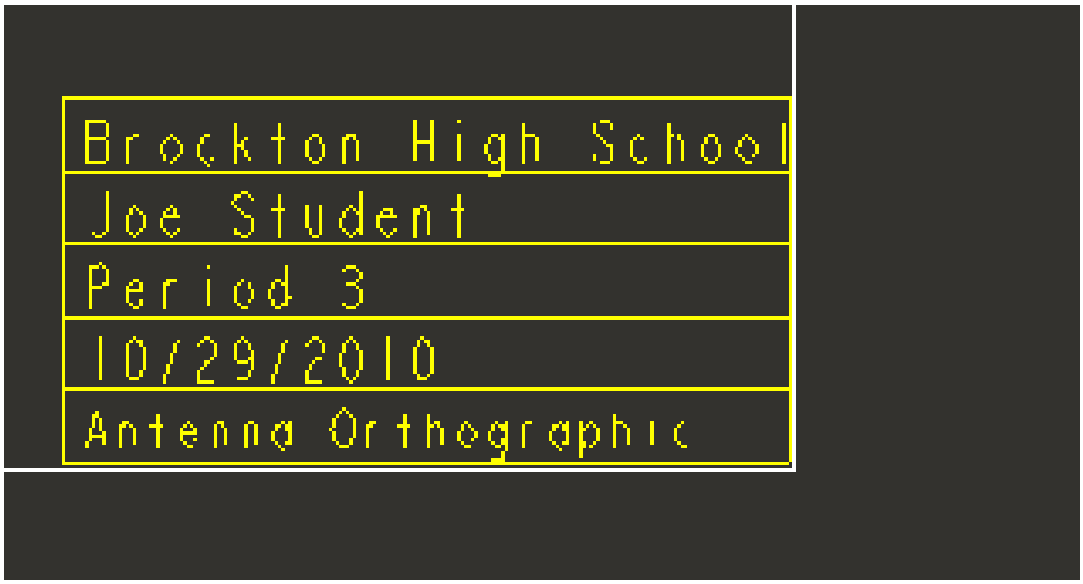
6. Double Click in the top cell of the table. The Note Properties box will pop up. Type in Brockton High School, then click OK.

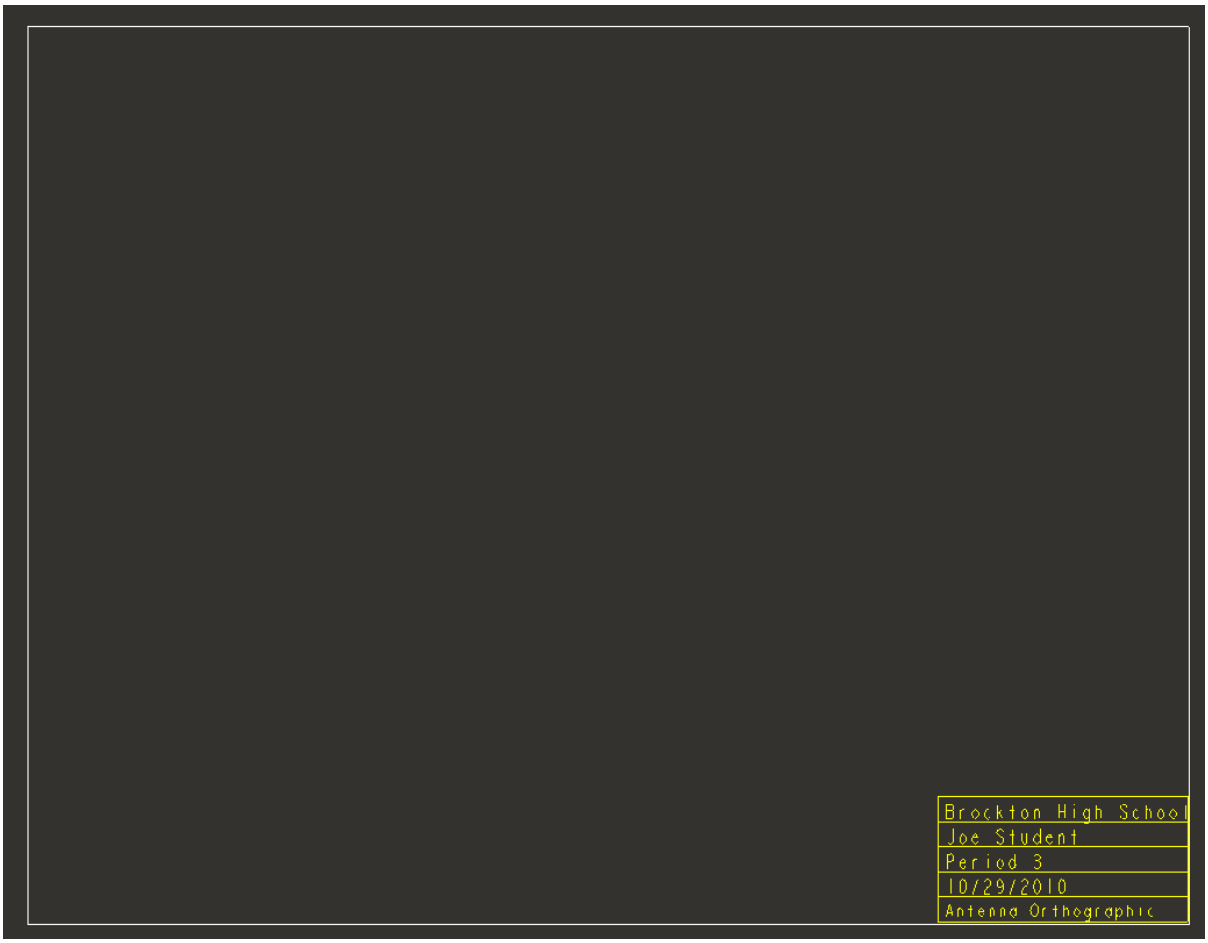


7. Fill in the rest of the table as it is below, with your name and information.

8. You can change the size of the text by clicking on the Text Style tab in the note properties box. Right Click on the table or cell to bring up the Note Properties box.

Watch Video





Now you will add a general view and a projection to the sheet, orient the view, change the view's properties and show dimensions.

1. Click **Insert > Drawing View > General**. You are prompted to select a center point for the drawing view. Click the center-right area of the sheet.

2. The view is placed and the **Drawing View** dialog box opens, showing all the saved 3D orientations. In the dialog box, under **Model View Names**, select **Front** from the list. Click **Apply**.

3. Under **Orientation Method**, select **Angles**. Enter **270** for the angle value and click **Apply**. The view reorients to the horizontal, with the tip pointing to the right. Click **OK** in the dialog box to close it.

4. Under **View Display**, choose **No Hidden** in the Display Style box.

Watch Video

Modify the view

When you placed the view, you accepted the default attributes for the view type. Now modify the view for a custom scale.

1. Right-click the view and click **Properties** from the shortcut menu.

2. Click Scale from the Categories list, and then choose Custom Scale. Enter **2**, and then click **Apply** and **Close**. The view is drawn to the new scale.

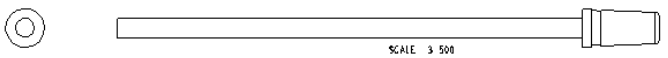
3. Select the view again, right-click and uncheck **Lock View Movement** in the shortcut menu. Use the mouse to move the view to a new position. When the view is placed, check **Lock View Movement**. This setting applies to all of the views on the sheet, not only the selected view.

Now you'll add the projection view to show the shaft and tip outer diameters.

1. Click **Insert > Drawing View > Projection**. Click in the space directly to the left of the general view. The projection view is added as shown in the next figure.

2. Make sure that your display is set to **No Hidden**  to show only the outer diameter and shaft diameter of the tip. When you change display properties, click the Repaint icon to refresh the display. Watch Video

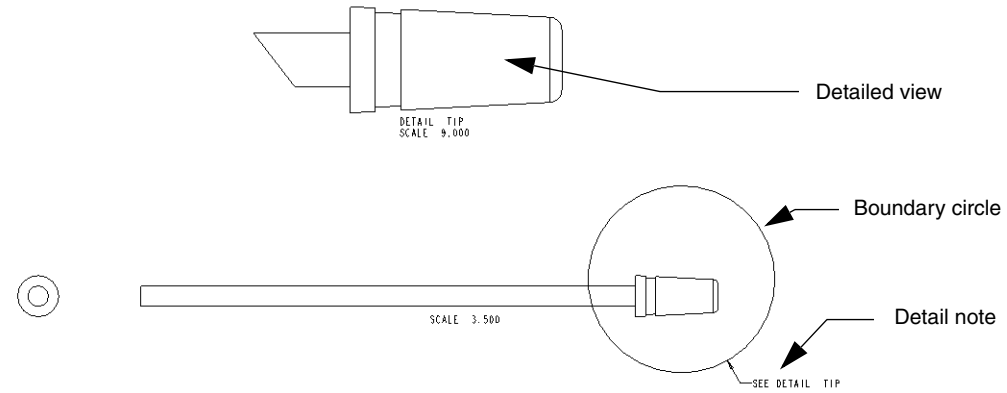
General view and one projection



Add the Detailed View

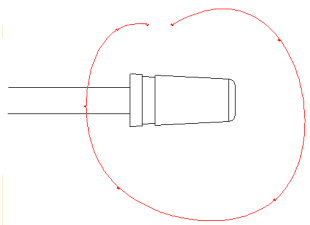
Now you'll add the detailed view. This involves a series of prompts that define the elements shown in the following figure.

Elements of a detailed view



1. Click **Insert > Drawing View > Detailed**. Click along the outline of the tip in the general view. At the spline prompt, use the mouse to click-draw a circle to enclose the detail to be enlarged. Middle-click when the circle is almost complete. A circle is added around the tip.

Sketching the outline for the detailed view



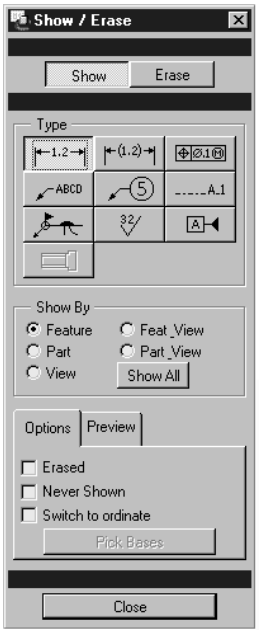
2. Define the note and leader location by clicking anywhere near the circle. The note is added at the selected point (it can be dragged to another position at any time).
3. Click in the upper middle of the sheet, and the detailed view is added. Drag it in any direction to redefine its position.
4. Right-click the detailed view, and then choose **Properties** from the shortcut menu. Choose **Scale** from the **Categories** list and enter a value of 9 for the custom scale, and then click **OK**.

Show Dimensions

There are several ways to show or hide dimensions passed from the 3D model. You can access the commands from the main **Show/Erase** dialog box or from the shortcut menu when you select either a feature or a model in the Model Tree. In this step you'll use the **Show/Erase** dialog box. In subsequent steps you'll use other methods.

1. To start, click **View** > **Show and Erase**. The **Show/Erase** dialog box opens.

Show / Erase dialog box



The **Show/Erase** dialog box controls the display of all information objects that can be passed from the 3D model. The **Show By** area lets you selectively show dimensions feature by feature or view by view, an ability you'll find useful when drawings get complex. The **Options** tab is used to filter showing between **Erased** and **Never Shown**. You may have erased some dimensions on one view with the intention of showing them in another. The **Erased** command lets you show only those you erased.

2. To start showing dimensions, click the Dimensions icon in the **Type** area in the upper-left corner.

3. Click **View** in the **Show By** area.

4. Click the detailed view on the sheet. The passed dimensions are now shown on that view.

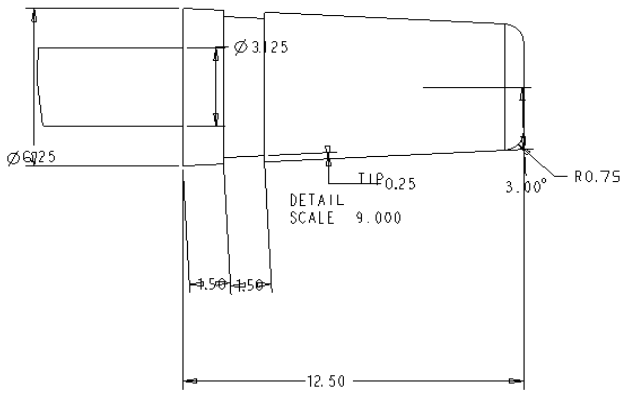
In the **Show/Erase** dialog box, the **Preview** tab activates. You can select certain dimensions to keep or to remove before you close the dialog box. (Remember, these dimensions are never really removed. They are only hidden.) By default the **Sel to Remove** button is active.

Note

The actual position of the dimensions in your file may vary from the illustration, but the same dimensions should be showing.

At this point, if the dimensions don't look as organized on the page as you'd like, don't worry. First you'll work on uncluttering the detail view by moving some of the dimensions to more appropriate views. Then, you'll select and drag the dimensions to their final plotted positions.

Tip detailed view after showing dimensions



5. The two diameter dimensions could be better shown on the end projection view of the shaft. Hold down the Ctrl key and click them both to select them for removal. When they are selected, middle-click to complete the selection. They are both erased. Now, you'll show them on the end projection view.

6. Click the **Options** tab in the **Show/Erase** dialog box. Leave the **Show By** setting on **View**, and select the **Erased** checkbox. Zoom out so you can see the whole sheet. Click the end projection view. The two dimensions are added to the view. Middle-click to complete. Make sure the **Never Shown** and the **Switch to Ordinate** boxes are cleared. Close the **Show/Erase** dialog box.

Return to the detailed view of the tip. As you can see, the length dimension of the shaft would also be better on the general view. Transfer it as follows:

1. Select the dimension, then right-click and choose **Move Item to View** from the shortcut menu.
2. Now click the general view. The dimension is transferred. This is a quick way to move shown dimensions from one view to another.

Insert Added Dimensions

Finally, before you clean up the layout of the dimensions, you will add a reference dimension. This dimension will call out a length that is not passed with the 3D antenna: its overall length, the shaft plus the tip. The reference dimension will have the suffix `REF` to show that it is not a driving dimension.

1. From the main menu click **Insert** > **Reference Dimension** > **New References**. The cursor changes to a pencil shape.
2. Click the line representing the end of the tip, and then click the end of the shaft. (You can zoom and pan without losing your selection.) Middle-click to finish. The dimension is added with the `.REF` suffix.

Clean Up the Dimensions

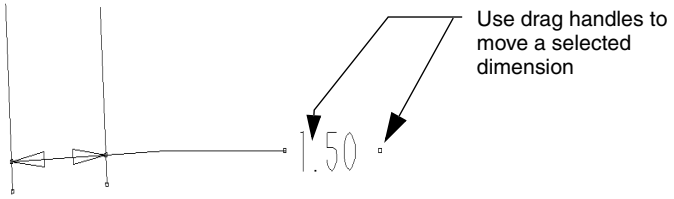
Now you're ready to arrange the dimensions as you want to print them. Review the techniques below, and then try to place each dimension on the tip detail view so it matches the finished illustration. This is not an exhaustive review of all the edits you can make to dimensions, but it does cover the basic concepts.

A selected dimension is highlighted and surrounded by drag handles that you use to drag it from one spot to another. You must move the cursor over each handle to see the direction of movement possible:

- A four-sided cross cursor means you can drag the dimension in any direction.
- A two-way arrow shows that you can drag on one plane.

When you select a dimension and drag it into position, the leader and witness lines follow with it. You will probably have to zoom in to properly pick a drag direction.

Dimension drag handles



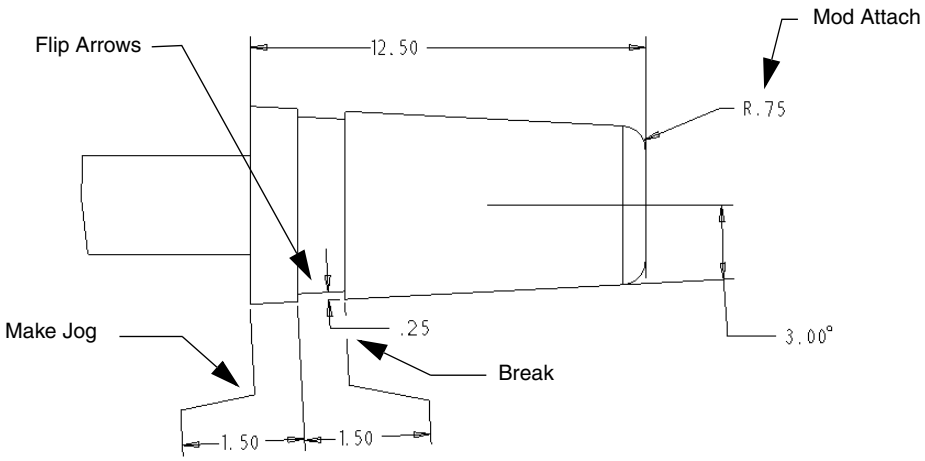
Edit Witness Lines and Arrows

You can edit witness lines the same way you edit dimensions: by selecting them and using drag handles located at either end. You can lengthen or shorten dimension lines or skew them away from their default angles. You can also use the shortcut menu to create breaks in them where they must cross other lines or leaders, and to create jogs in the witness lines to widen the dimension area.

To edit arrows, select the dimension and use the shortcut menu commands. You can change the style of the arrowheads or reverse their position on the witness line.

The next figure shows a solution for the tip dimension. Use the right mouse button shortcut commands to make the changes indicated.

One solution for detailing the tip view



Create a Second Sheet

Now you'll create a second sheet, and see how to place an exploded view of the cell phone assembly. Follow the guidelines below to create an exploded assembly view on a new sheet, create a bill of materials, and show the BOM balloons.

Create an Exploded Assembly View

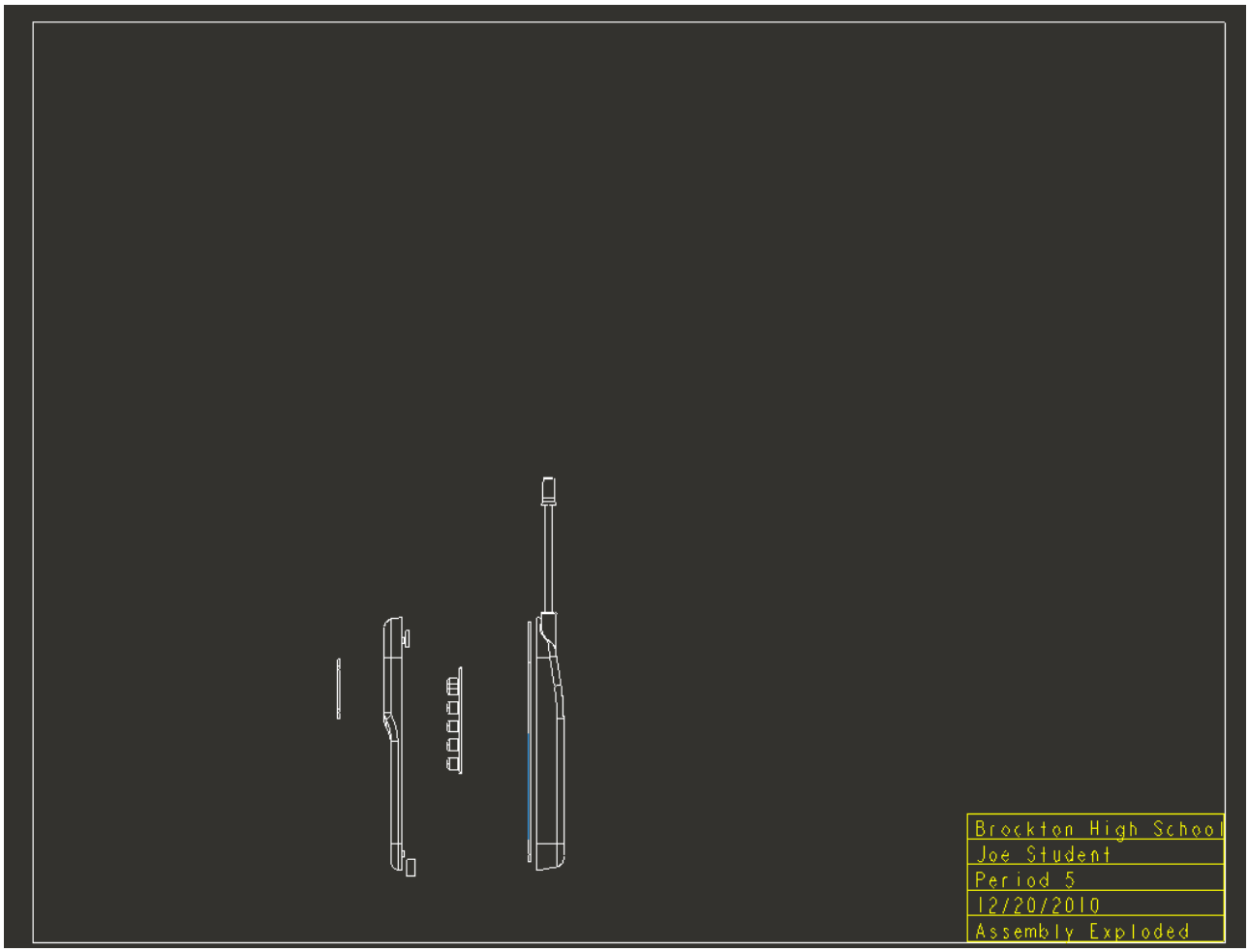
Click **Insert** > **Sheet**. A new sheet is added, and listed in the lower right corner as 2 of 2. Click **Edit > Move Sheet.** In the dialog box, select **Insert at Beginning** and click **OK**. The new sheet is now `Sheet One`. Now you'll add the assembly file to the drawing models.

1. Right-click the new sheet and choose **Properties** from the shortcut menu. The Menu Manager opens. Click **Drawing Models** > **Add Model**. Use the browser to choose the assembly file.

2. Right-click and select **Insert > General View**.

3. Choose **DEFAULT ALL** in the **Select Presentation** dialog box. Click the sheet, and then **OK** to place the view.

4, Set up a table like you did for the first sheet.



Sheet with the Left View inserted

Create a Bill of Materials

In this sequence you'll see how to add a very basic bill of materials to the drawing. A bill of materials is the most common use of a report table in Pro/ENGINEER. The report table "reads" parameters you enter as text into the cells and automatically adds cells in additional rows to accommodate information returned from the design database.

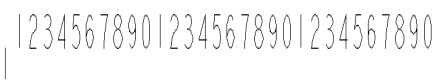
Setting up a report table can be a complex process, but once defined you can save and reuse report tables on other drawings. This exercise demonstrates as simply as possible how report tables work in Pro/ENGINEER.

Create the Table

A table is a series of cells of a given height and width. You can enter "dumb" text into the cells, for example as a column heading. The first step in creating the BOM is to define the table.

1. Click **Table > Insert > Table**. In the **Menu Manager,** select **Descending** > **Rightward** > **By Num Chars** > **Pick Point**. Click at a point where the upper-left corner would be on the sheet. A scale of numbers appears horizontally from the clicked point.

2. Click around the number 5 or 6. The width of the first column is defined, and the scale moves to the right.



3. Click the scale again to define another column width about twice as long. When you have defined this column, click the middle mouse button. The horizontal scale is replaced with a descending horizontal scale. Now you'll define the number of rows.

4. You'll need two rows to start, one for the column headings and one for the parameters. Click twice down the scale to create the two rows and middle-click to complete the cells. The table should look like this.

Note

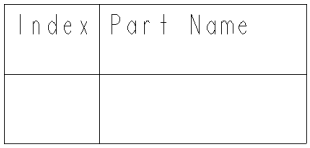
You can always resize cells after you create them but in this case, as you practice, it may be easier to delete the table and start again.

Enter Text Headings

Now you'll enter the headings for each column.

1. Double click the upper-left cell. The Note Properties dialog box opens. Use it to type `Index` in the text area and click **OK**.

2. In the same way, insert the heading `Part Name` in the next column. The table should look like this:



You created some table cells, and entered simple text into two of them. To adjust the cell size, right-click the cell and choose **Height and Width** from the shortcut menu.

Now you will set up the remaining cells to expand with information. To do this, you first designate the empty cells as repeat regions, or cells that will expand automatically to create the finished table. You then "build" the parameters you want to enter by selecting text strings from a hierarchical menu.

Define a Repeat Region

1. Click **Table** > **Repeat Region.** The Menu Manager opens.
2. Click **Add**. You are prompted to select the corners of the region.
3. Click the lower-left cell, then the lower-right cell. Both cells are highlighted. Click **OK**, then **Done** on the Menu Manager.

Now you are ready to enter the parameters for the assembly index number and matching part name.

Add BOM Parameters

1. Double-click the left side cell of the repeat region. Because it is now a repeat region, the **Report Symbol** menu opens by default. These are the top-level prefixes, denoting the modules for which reports may be generated. This menu lets you assemble a report parameter string by "punching down" through eligible parameters.
2. Click **rpt**..., then **index**. The cell now contains the heading `rpt.index`. Now double-click the lower-right cell and then **asm. > mbr. > name** on the **Report Symbol** menu. The assembly member name parameter you have just created is entered into the cell.
3. Click **Table** > **Repeat Region** > **Update Tables > Done**. The table is expanded to show the information as defined by the parameters.

Show BOM Balloons

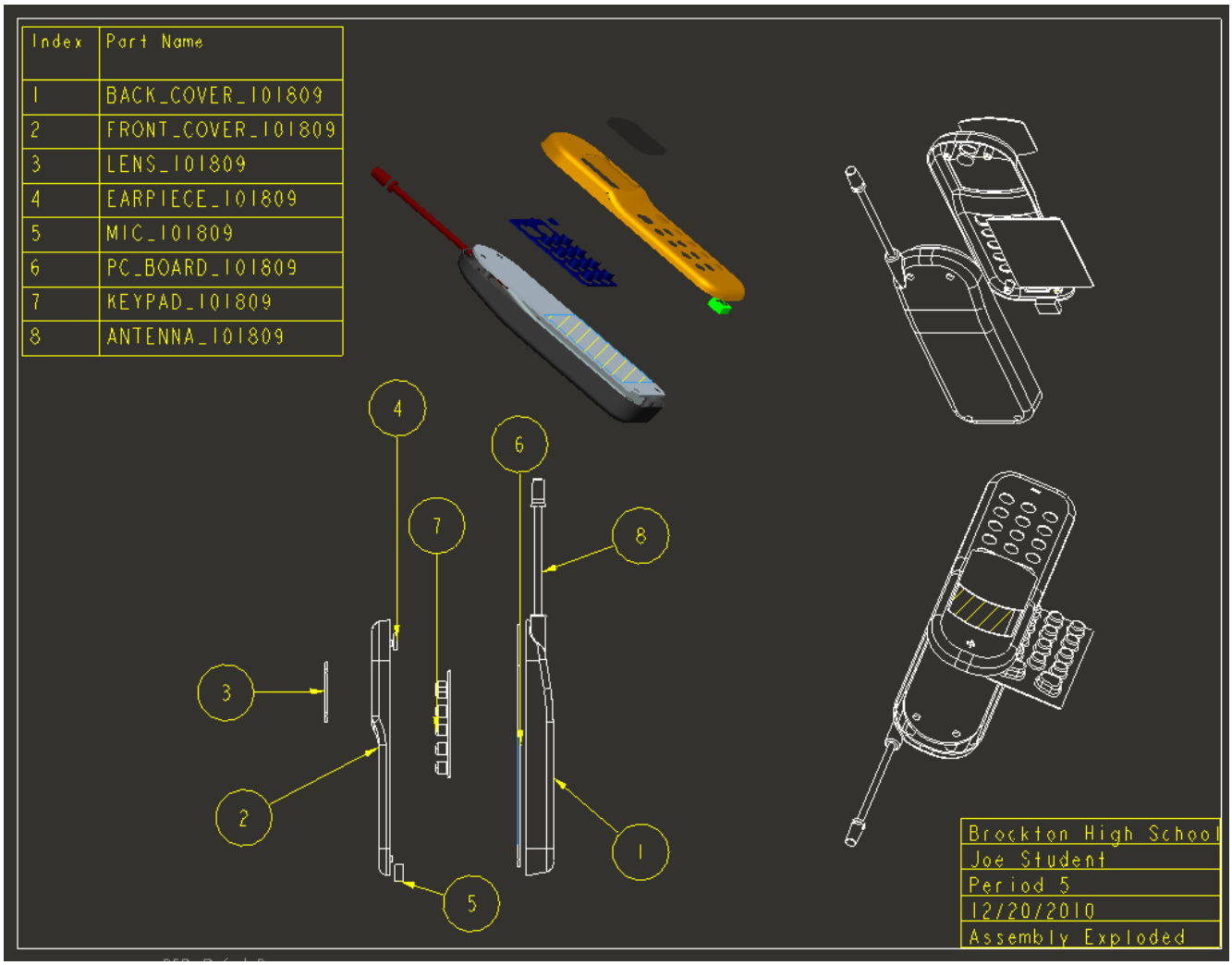
BOM balloons are the finishing touch to the first sheet. First you'll assign an existing repeat region as a BOM balloon region. Then all you need to do is choose to show the balloons, and arrange them as you want them.

1. Click **Table** > **BOM Balloons**. In the Menu Manager, choose **Set Region**. You are prompted to select a region in the table. Because the table has only one region, click anywhere in the table to select the region. Once the region has been selected, the rest of the Menu Manager items are available.

2. Now click **Create Balloon** on the Menu Manager under BOM BALLOONS. You are prompted to select the view in which to show them. Click the **Left** view, and then **Done** on the Menu Manager. The BOM balloons are added to the view.

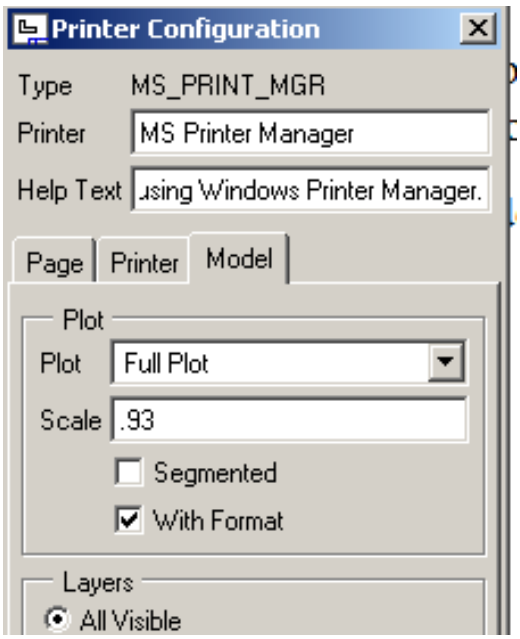
3. You can right-click the view and choose **Cleanup BOM Balloons** from the shortcut menu to modify the positions of all balloons. To fine-tune the layout, select and drag each balloon separately to a new location, or right-click a specific balloon and choose **Edit Attachment** to select a new attachment edge on the object.

Exploded view with BOM balloons on next page

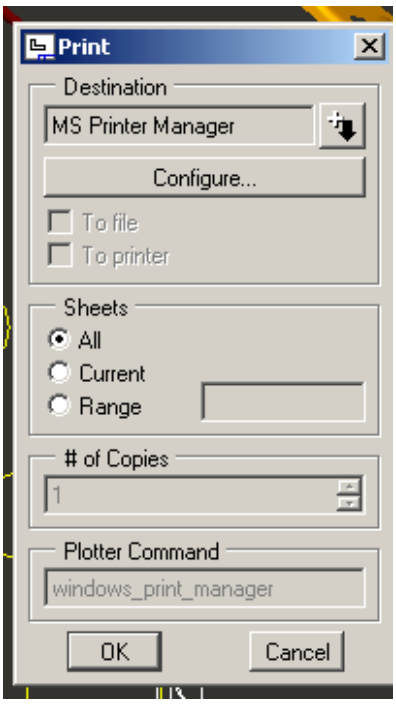


Finished Sheet with BOM Balloons.

1. Go to **File > Print > Configure > Model.**
2. In the **Plot** box, choose **Full Plot**. In the **Scale** box, type in **.93**.



3. In the sheets range, choose **All.** Click **OK**.



4. Choose the **HP 4200 LaserJet Printer**. Click OK and Print. Staple your drawings and hand them in.