



## Module 3

1. Dimensioning Overview
2. Auto Dimensions
3. Manual Dimensions



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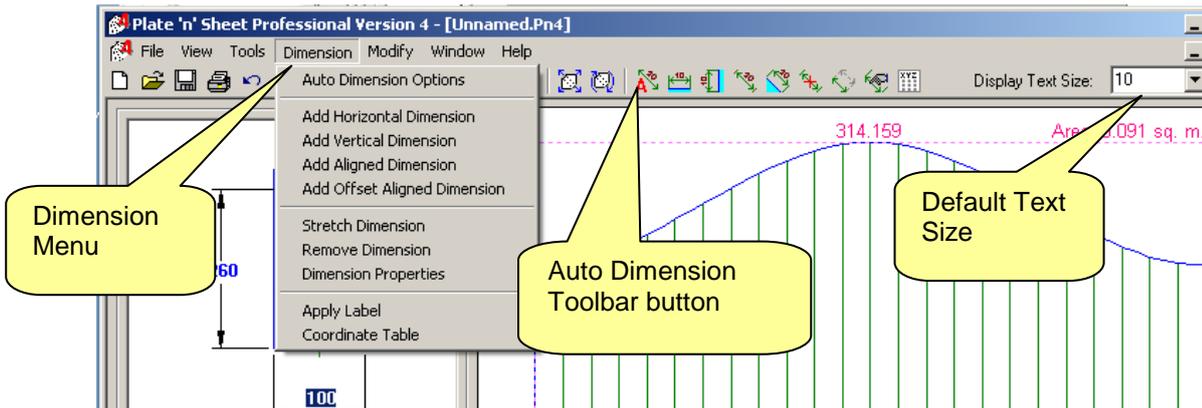
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## Dimensioning

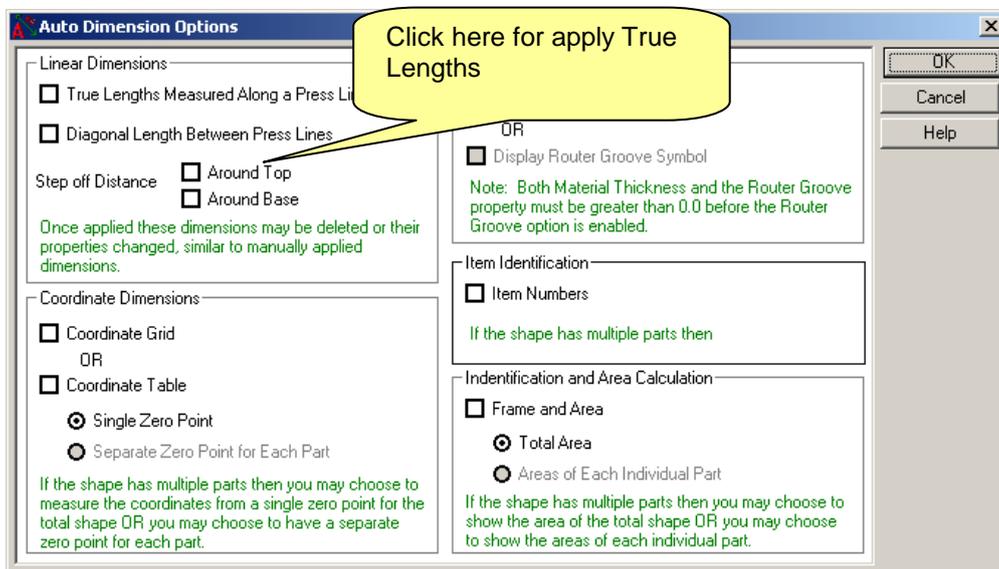
Dimensions may be applied *automatically* or *manually*. The following provides a brief overview of each method.

### Overview of Auto Dimensions

There are several modes of Auto Dimension available. Auto Dimensions may only be applied to pattern View, not to the wire-frame model. The Auto Dimension Options are enabled by selecting from the “Dimension” pull-down menu or by clicking on the tool bar button:

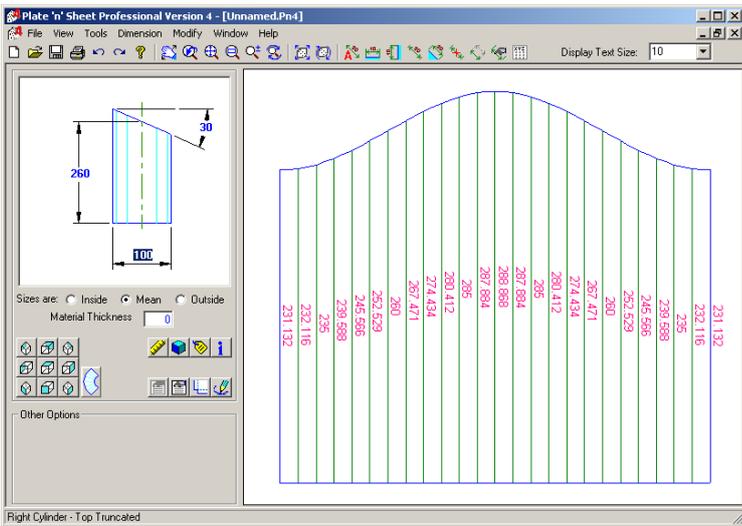


The text size for the dimensions is shown on the drop down list. While text size may be changed after the dimensions have been applied, it is more efficient to set the size before.



The Auto Dimension dialog box shows the various dimensioning modes. Each Auto Dimension mode will be discussed in detail later. For now, just select “**True Lengths**” by clicking on the check box.

The result will be as follows:



The true length of each press line has been placed onto the pattern. Experiment by turning various modes on and off by returning to the Auto Dimension options.

### Overview of Manual Dimensions

A horizontal, vertical or aligned dimension may be applied between any two points. Manual dimensions can be applied to the pattern and also **orthographic** views of the 3D model. That is, the Top View, Front View and Side Views but not isometric or rotated views.



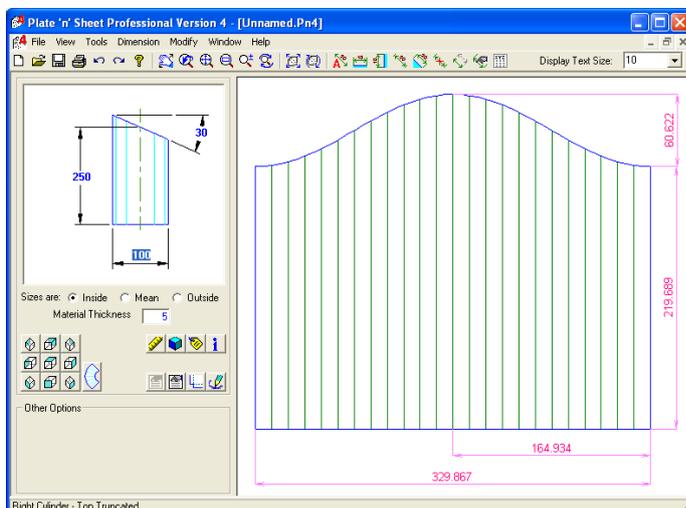
Click this button for a Horizontal Dimension.

To manually apply a horizontal dimension, click on a toolbar button.

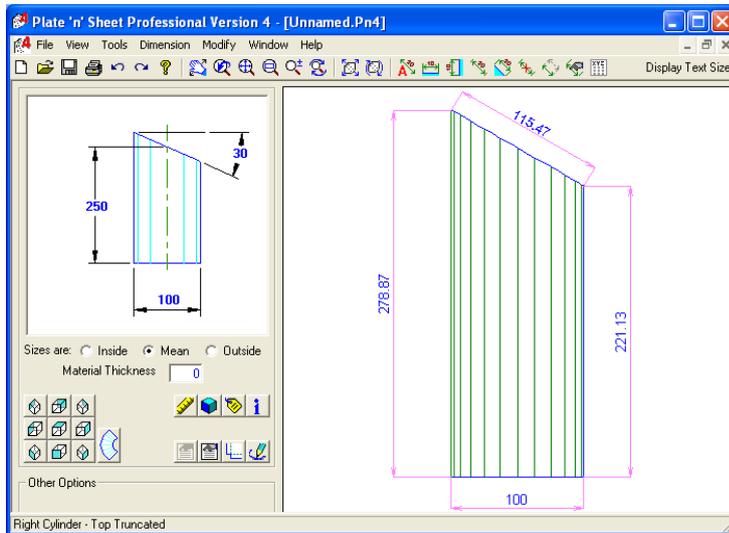
Click a point on the pattern using the left mouse button and hold it down. The dimension will find the closest point. A small circle will appear at each point as the mouse moves over the pattern.

Keep the mouse button down as you drag the mouse to the second point and then release it when the second small circle is over the required point.

Move the mouse until the dimension line is in the correct position and click again.



Horizontal and vertical dimensions applied manually to the pattern.

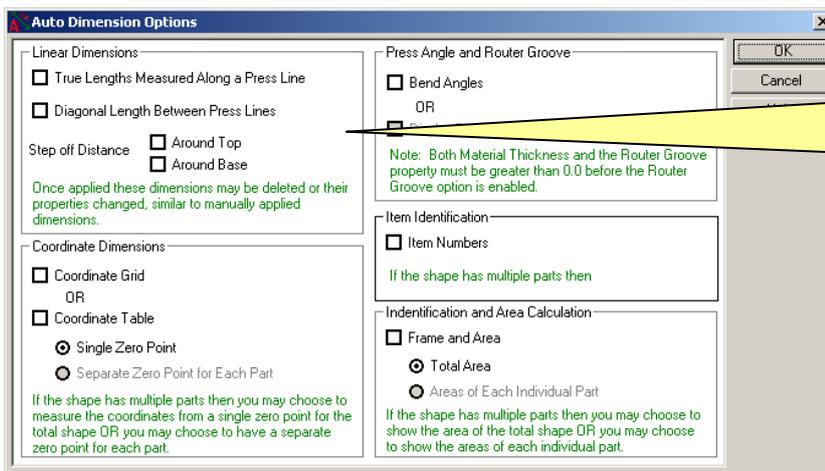


Dimensions applied manually to the wire frame model.

When you change to another view those dimensions will no longer be visible. However, they will be displayed again when you return to the view current when they were created.



## Auto Dimension Modes



Apply "True Length" along each press line, Diagonal dimensions and "Step-Off" distances around the top and base by clicking on the check-boxes.

### True Lengths

This mode displays the "true length" of each press line as an aid to marking the shape onto the material or to check the shape and scale of the pattern.

### Step-Off Dimensions

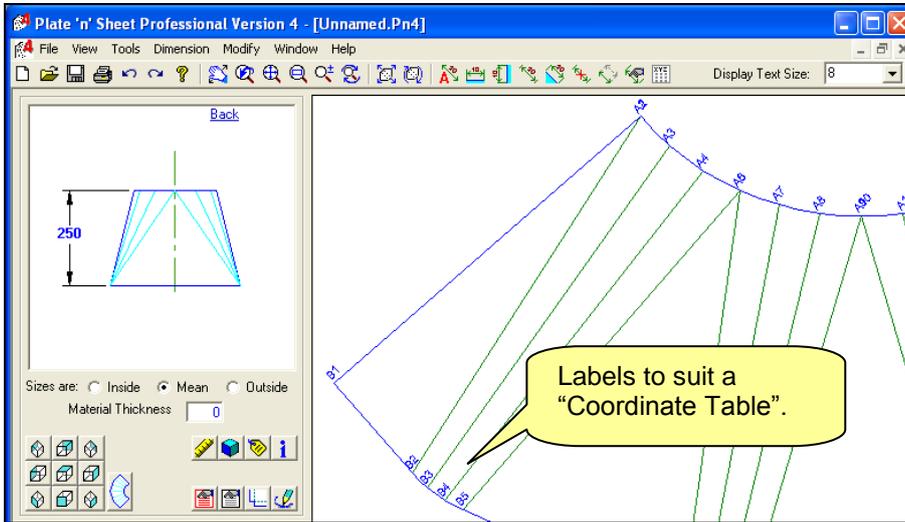
This mode displays the straight-line distance from the end of one press line to the end of the next press line. Step-Off Dimensions may be applied around the base profile or around the top profile.

### Diagonal Dimensions

This mode displays the diagonal distance from the end of one press line to the start of the next press line. This provides a means of triangulating the shape to mark it onto the material or to check the printed shape.

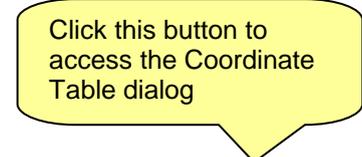


## Coordinate Table

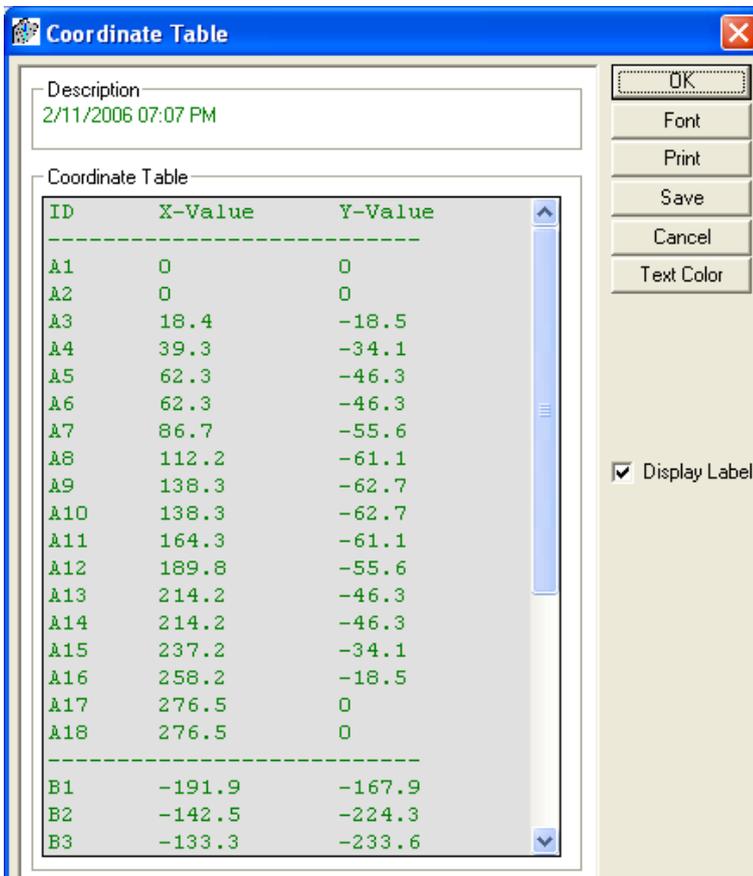


This mode marks the ends of each press line with a code. A table of coordinates of each corresponding point may be viewed by clicking the co-ordinate table button on the tool bar.

Click the Coordinate Table button on the top toolbar.



The Table of Coordinates may be positioned on the drawing by clicking the “ok” button, saved as a text file by using “save” option or printed directly by using “print option”.



If you are using small page sizes such as A4, it is usually better to print the table on a page of its own by clicking the “Print” button, rather than including it on the drawing.

The display of the table on the drawing is controlled by the “Display Label” checkbox. To hide the table, deselect the checkbox.

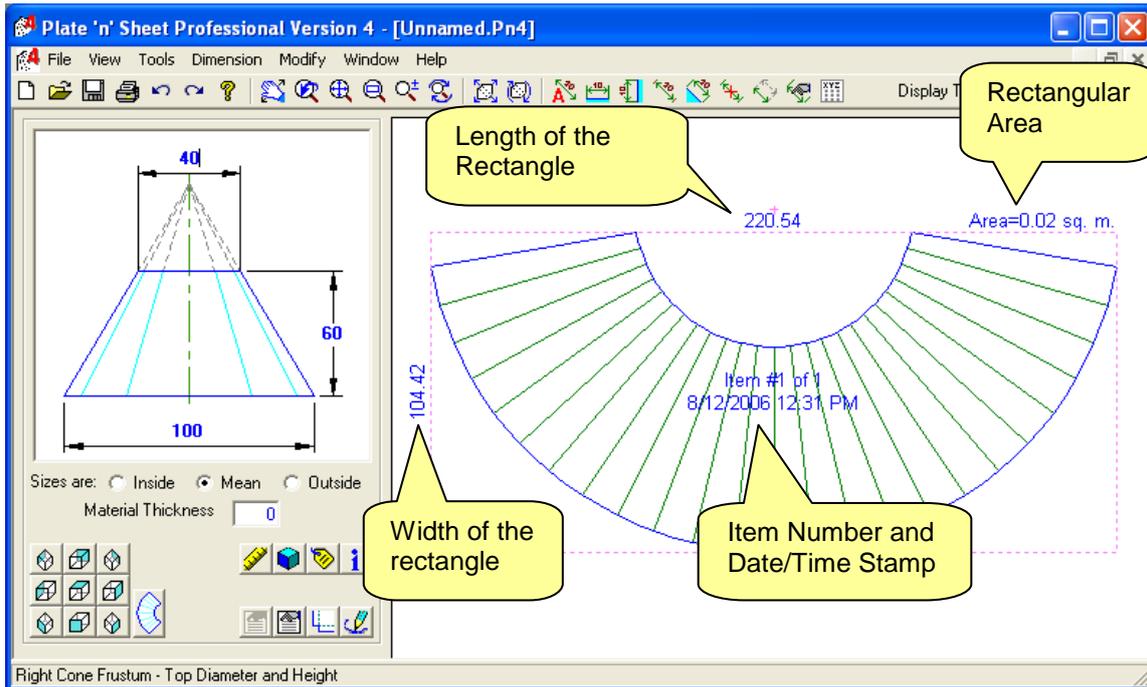
However, if you are sending the result to AutoCAD, displaying it on the drawing is a convenient way of exporting both the pattern and the Coordinate Table.

## Frame and Area

When this mode is selected a rectangle is drawn around the shape with the length, width and area displayed. This is useful for determining the size of the material required to make the shape.

It is also useful for finding the most economical arrangement of the pattern. Sometimes rotating the shape allows the shape to be cut from a smaller piece of material.

If the shape is made up of multiple parts such as an elbow, the Frame and Area may be applied to the entire shape or to the individual parts comprising the shape.



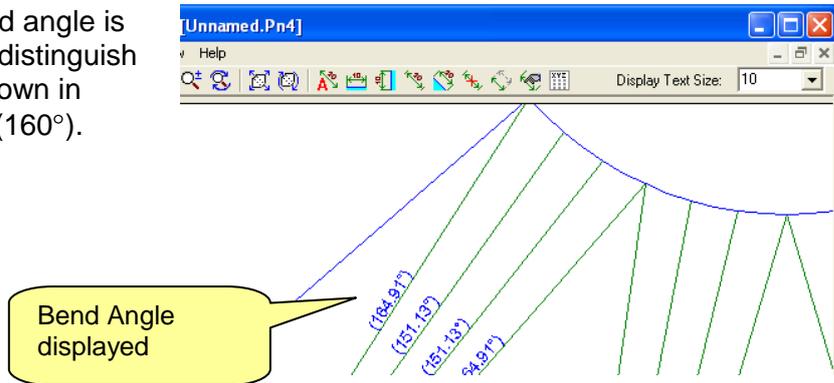
## Item Number

When this mode is selected the number of the item is displayed on the pattern. The date and time that the shape was last updated is also displayed. The item number may not be necessary for single part items but is useful when the pattern is of an item made from several parts, such as a segmented bend.

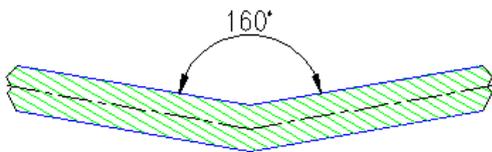
The item number is also displayed on the wire-frame model to help identify the parts.

## Bend Angles

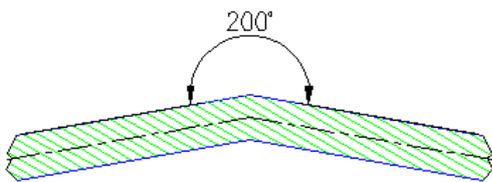
When this mode is selected the bend angle is displayed down each press line. To distinguish it from a true length dimension is shown in brackets and with a degree symbol ( $160^\circ$ ).



The press lines are to be pressed down. For example, the press lines on the transition shown are on the inside surface. The angle displayed is the angle between two adjacent faces measured at right angles to the press line.



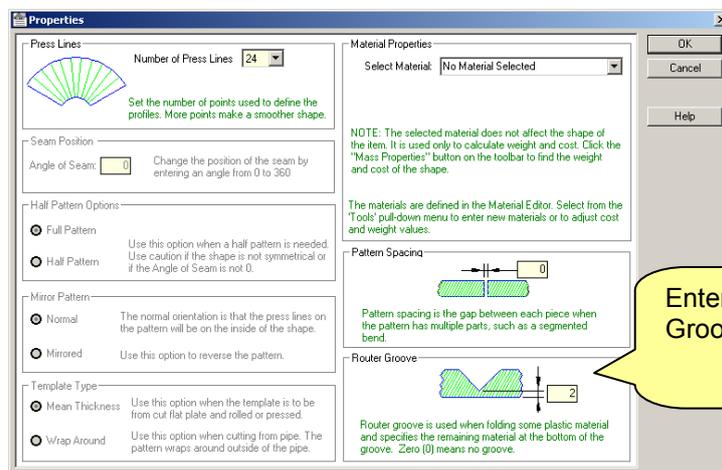
If you are using a press, press down on the press line if the angle is less than  $180^\circ$ .



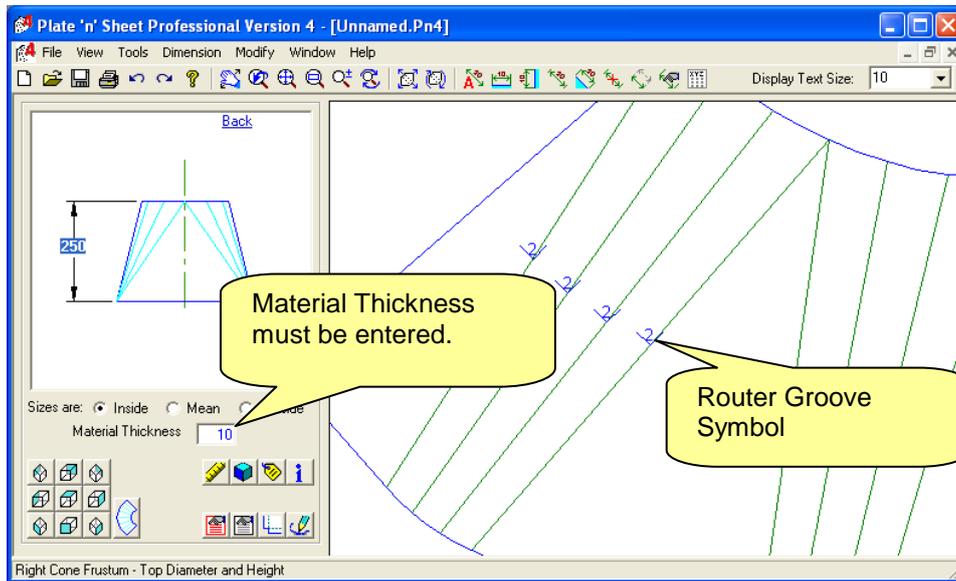
Press up if the angle is greater than  $180^\circ$ .

## Display Router Groove Symbols

Fabricating shapes from plastics may require additional preparation. It is a common practice to cut a groove with a router along the press lines. The bending then takes place at the bottom of the groove.



Before this option is available there must be a value in Material Thickness and in the Router Groove property.



When this mode is selected a symbol is placed on the press line to show the router groove dimension and direction.

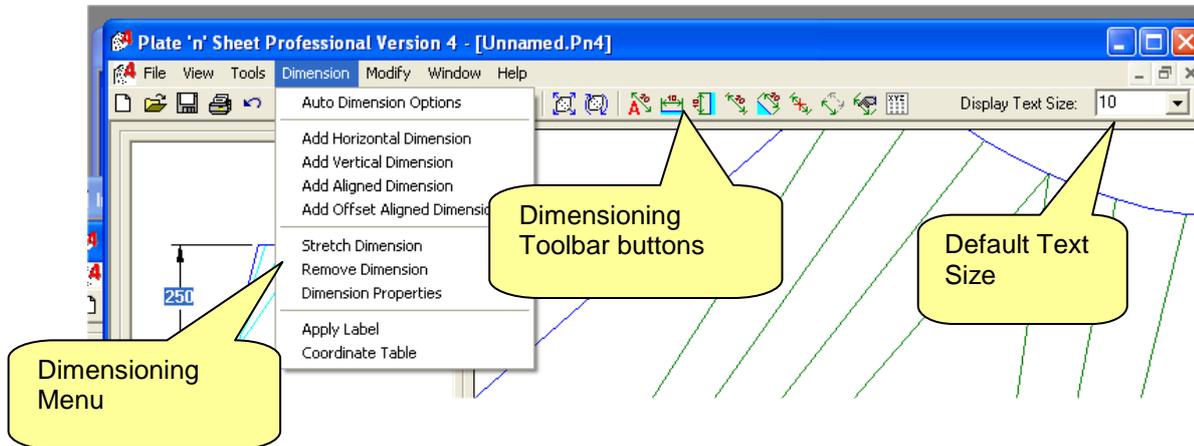


If the symbol points down as shown, the groove is to be cut into the same side as viewed. If the symbol points up, the angle will be greater than 180 and the groove is to be cut on the opposite side.

See the section about Properties for more details on the use of Router Grooves.

## Manual Dimensions

There are several types of Manual Dimension available. Auto Dimensions may be applied to both Pattern View and wire-frame Model View. The Manual Dimensions are applied by selecting from the “Dimension” pull-down menu or by clicking on the toolbar button:

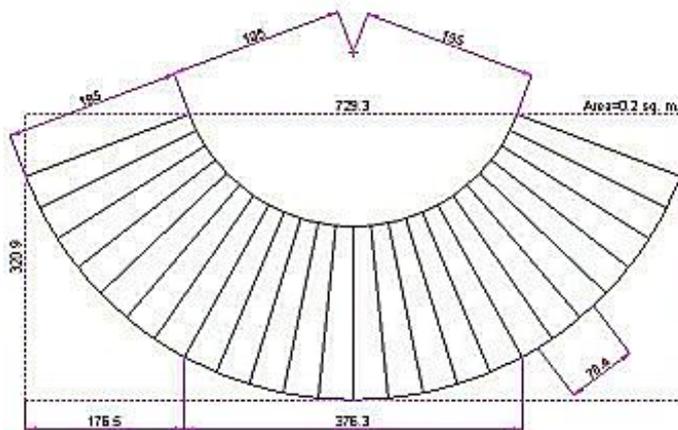


The text size for the dimensions is shown on the drop down list. While text size may be changed after the dimensions have been applied, it is more efficient to set the size before.

## Dimension Points

Dimensions may be attached only to certain points on the shape, as follows:

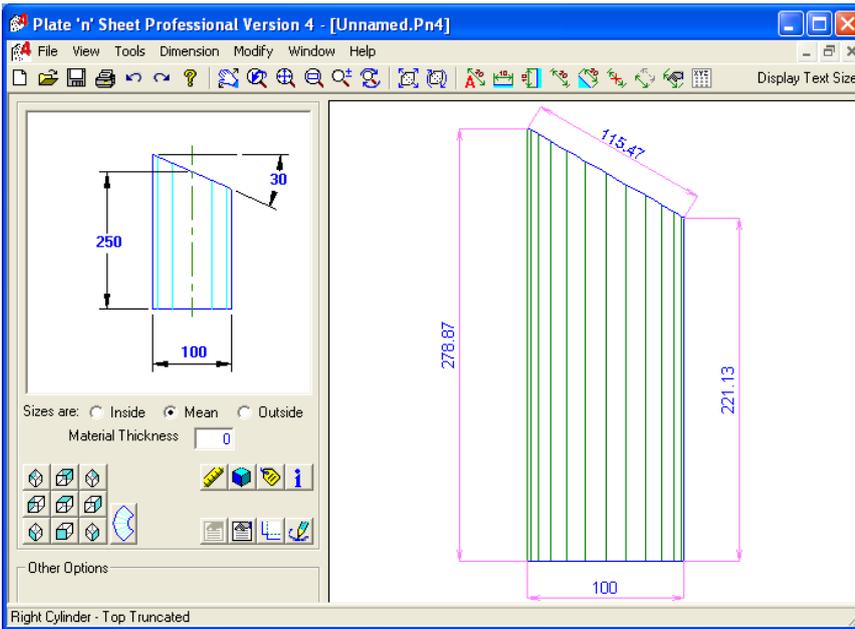
- 1 The endpoints of the press lines.
- 2 The reference point (the little cross that appears at the apex of a cone pattern).
- 3 The midpoint between the endpoints of press lines along the profile.
- 4 The corner points of the rectangular frame when “Frame and Area” is enabled.
- 5 Points around the perimeter of the lap join allowance.
- 6 Points around the perimeter of a hole, such as in a branch.



## Dimensioning in Model View

Manual Dimensions may be applied in Model View in orthographic views only. Orthographic views are the Top View, Front View, Rear View and End Views only. Dimensions cannot be applied in isometric or rotated views.

When you change to another view those dimensions will no longer be visible. However, they will be displayed again when you return to the view that was current when they were created.



Manual dimensions applied to the wire frame model.

## Applying a Manual Dimension

Click a toolbar button or select from the dimension pull-down menu.

Move the mouse over the shape and a small circle will jump to the closest dimensioning point. Click with the left mouse button when the small circle is on the required point. **Keep the mouse button down.**

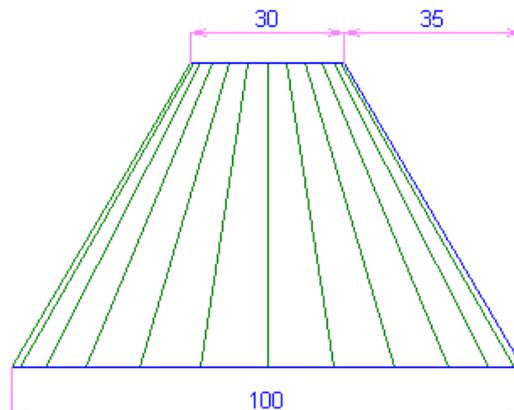
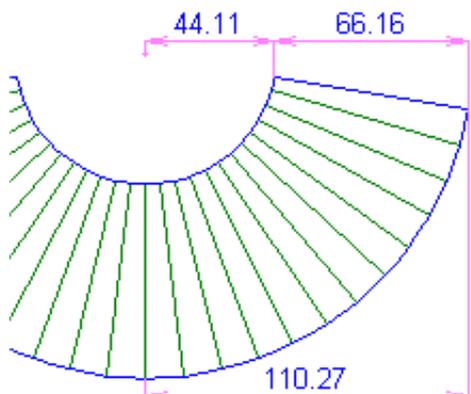
With the left button held down move the mouse over the shape until the small circle is over the second point then release the button.

Move the mouse again and stretch the dimension out until the dimension line is in the required position. Click and release the left mouse button.



## Horizontal Dimensions

A horizontal dimension may be applied between any two points. It displays the **horizontal distance** between the two points even if the points are not aligned horizontally.





### Vertical Dimensions

Similarly, a vertical dimension may be applied between any two points. It displays the **vertical distance** between the two points even if the points are not aligned vertically.



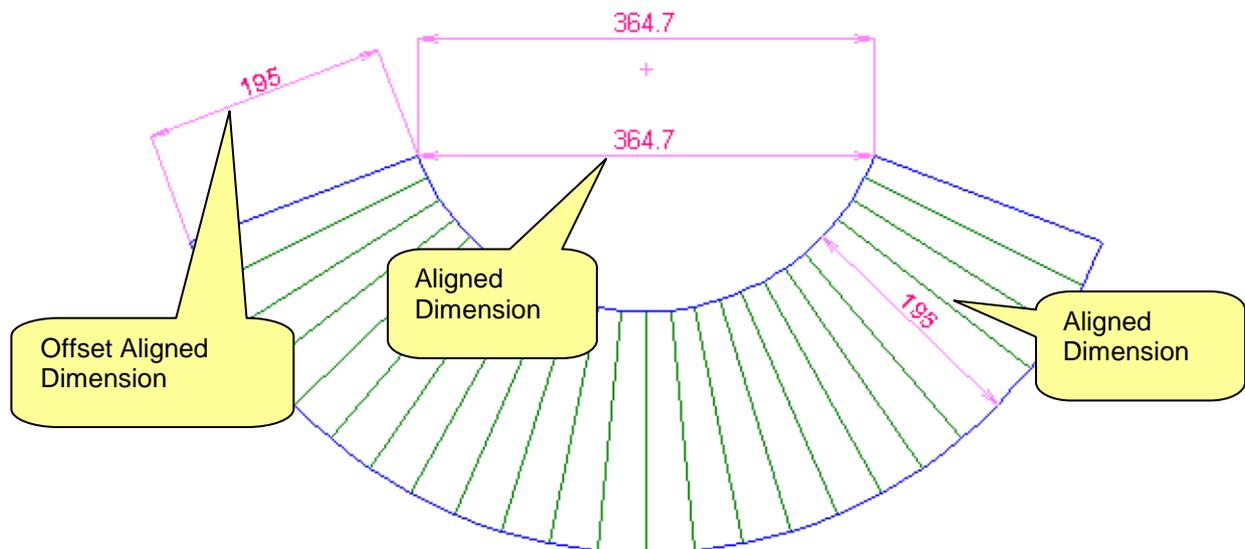
### Aligned Dimensions

An Aligned Dimension may be applied between any two points. It displays the **direct, shortest distance** between the two points. The dimension is drawn as a line connecting the two points. It is similar to the manual dimension in Version 3 of the program.



### Offset Aligned Dimension

An Offset Aligned Dimension may be applied between any two points. It displays the **direct, shortest distance** between the two points and allows the dimension to be offset from the origin points.



### Remove Dimension

To remove a dimension:

Click the tool bar button or select from the “Dimension” pull-down menu.

Move the mouse over the shape on the screen. Each dimension will be highlighted in turn.

Left click with the mouse when the correct dimension is highlighted.



### Stretch Dimension

Stretching allows the dimension line and text to be moved while leaving the origin points attached. To Stretch a dimension:

Click the tool bar button or select from the “Dimension” pull-down menu.

Move the mouse over the shape on the screen. Each dimension will be highlighted in turn.

Hold down the left mouse when the correct dimension is highlighted.

Holding down the left button move the mouse. Drag the dimension line (shown dotted) to the new position and release the mouse button.



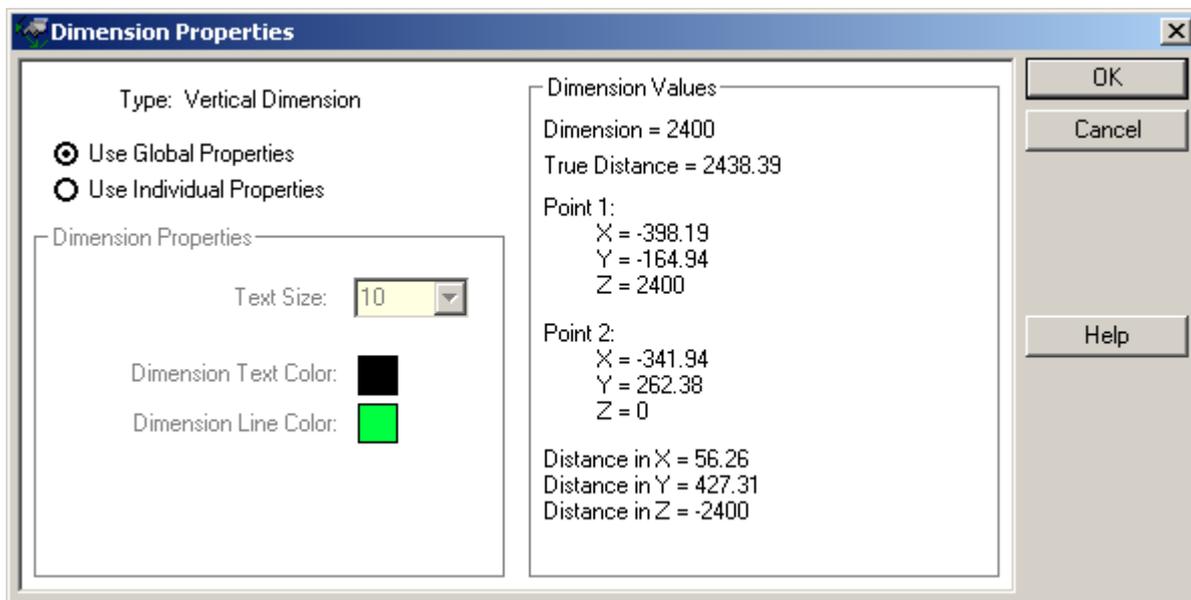
### Edit Dimension Properties

The properties of all or individual dimensions may be modified after they have been attached to the drawing. The properties that may be modified are font size and color and dimension line color.

This can be used to highlight important dimensions using color, to reduce the size of text in congested areas of the drawing and adjust the color to make it more printable.

Click the Edit Dimension Properties button on the Dimension toolbar (or type the Hotkey “**P**” on the keyboard). Move the mouse over the pattern to highlight the required dimension. Click with the left mouse button to select. The dialog box shown below is displayed showing the properties of the selected dimension.

Select Text Size (font size) from the drop down list. Select the text color or dimension line color by clicking on the corresponding color patches and choosing a new color from the color array that is displayed.



**Note 1:** When a dimensions are first applied, the color is determined by the default setting in the “**Global Options**” dialog and the font size is set by the “**Default Text Size**” on the top toolbar. Unmodified dimensions automatically change when either the Global Options or Default Text size values are changed. A dimension with modified properties retains its modified color or text size.

**Note 2:** Dimensions in **Version 4** that have been applied using Auto Dimensioning are the same as dimensions applied manually. This was not the case in **Version 3**. This means that individual Auto Dimensions may be deleted or have its properties modified.

### **Hot Keys**

The dimension operations above can be used more efficiently with the use of certain “**Hotkeys**”. Pressing certain keys on the keyboard can be used rather than clicking a toolbar button or selecting from a menu and some users find this faster.

Hot key	Operation
A	Aligned Dimension
D	Remove Dimension
H	Horizontal Dimension
O	Offset Dimension
P	Edit Dimension Properties
S	Stretch Dimension
V	Vertical Dimension