

Accucadd dimensions are stored using special Dimension Entity records. Previous versions (e.g. RoboCAD 20) used regular lines and text. Some of the Dimension Entity properties are worth knowing about when drawing with Accucadd, and this section describes them.

**Dimensions** When a dimension is displayed, on the computer screen or on the printer, it has several components:

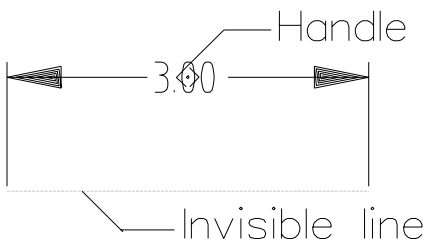
Lines and arcs, which are used for the dimension lines and leader lines.

Text, forming the dimension and tolerance labels.

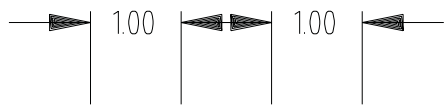
Terminators and other markers. These are actually text, in a special dimensioning font (see Dimension: Terminators).

In RoboCAD 20 and earlier systems, the Entities making up a dimension were simply added to the drawing's data as if they had been drawn 'by hand'. The Dimension functions made it *much* easier to draw dimensions correctly, but that was all they did.

**Dimension Entities** The Accucadd Family programs have a new type of Entity, which stores the points selected in drawing the dimension, the Dimension: Set Up settings, and other information. As the dimension is a single Entity, it can be stretched, edited or deleted in one operation. When a dimension is altered, its label is changed to reflect the dimension's new size (associative dimensioning).



In some ways Dimension Entities behave like invisible lines, arcs or circles, in the same position as the corresponding 'Entity' that they dimension. They can be selected in the same way as those Entities — which usually means that they are selected at the same time — or by the Handle of their text label. Selecting the line, arc or circle can part-select the Dimension Entity, if you are using an appropriate Group icon; selecting the text handle always selects the entire Dimension.



If dimensions are part-edited, a new dimension is produced for each resulting part, using the same dimension style. The examples shown here are for lines, but the same principles apply to all types of distance and angle dimensioning provided by Accucadd.

Other kinds of Entity can be used to define parts of Dimensions, but a Dimension can't be used to define parts of another Entity, even another Dimension.

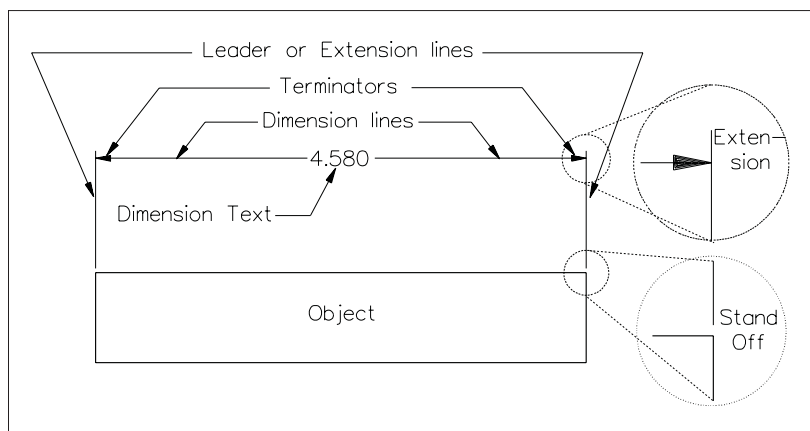
**Details:** The Handle of a Dimension's text label is always found at the center of the block of text (like the Middle Justification style of Draw: Text). If a tolerance label is used, it shares the same handle.

If an Insert containing Dimensions is Exploded (see Edit: Explode), the Dimensions' labels are updated.

As a Dimension Entity stores all the details of the Dimension Style used to produce it, changing the Style settings will not alter any dimensions that have already been drawn. (Dimension: Edit Dimension is provided for you to use to change pre-existing dimensions).

Advanced: Storing dimensions as single Entities makes it much easier for Accucadd add-on programs (such as RoboDATA) to recognize and/or ignore dimensions, when they read RDF files.

A dimension Entity consists of a header describing the dimension's style, followed by the Entities that make up the current version of the dimension: lines and/or arcs, plus text Entities. This technique makes it easy for third-party programs to read and work with dimensions, if required. It also makes Dimension Entities rather memory-hungry, but this is worthwhile, given the abilities it adds to Accucadd.



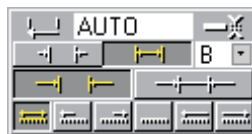
Dimensioning terminology

This function controls the attributes and arrangement of dimensioning lines, markers and text.

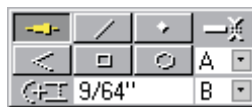
The Dimension: Set Up palette is displayed. This palette contains controls for all of the attributes of dimension lines, markers, dimension text and leader lines. The function of each icon will be explained in detail in the rest of this section. The palette is divided into five regions:



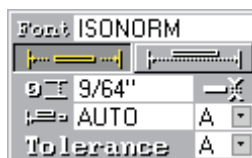
Leaders or extension lines: Controls the length of the leader (distance of the dimension line from the object), the distance from the object to the leader, the distance the leader extends beyond the dimension line, the line weight used to draw the leader line, and which leader lines are drawn.



Dimension Lines: Controls the placing of dimension lines within or outside the leaders, whether the dimension line is continued or broken, the line weight used to draw the dimension line, and which dimension lines are drawn.



Terminators or Markers: Controls the style of the markers placed at the ends of dimension lines, and the line weight used to draw them, and controls the size and line weight of the marker used for the centers of arcs and circles.



Text: Controls font, position, size, offset, and line weight used for the dimension text, and the line weight used for any tolerance text



Style: Controls whether the entire dimension set is placed on the currently active layer (A) or whether all dimensions are “forced” onto a single layer, and allows you to save or retrieve dimension set-ups as named files.

Making use of all the controls on the Dimension: Set Up palette can be fairly complex, but is usually unnecessary. The control settings can be saved as Styles, and Accucadd provides a number of pre-defined Styles to conform to ISO, ANSI, DIN and BS308 standards. The “Accucadd” style provided is simply an example “house style”, which you can alter.

For most drawing work, you should be able to make use of one of the standard styles. These are loaded in the same way as other Styles in other Accucadd functions. Details are given below under Dimension: Set Up: Style.

This region of the palette controls leader or extension lines.



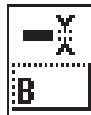
**LEAD LINES:** These are the lines on either side of the dimension text. Select this icon's value window to alter the lead line length. When you do this an input window is displayed. Type in a new value for the length of the lead lines or press **Enter** to 'free' the lead lines, (this lets you determine the length of the lead lines with the cursor).



**STAND OFF:** This is the distance between the start of the dimension point and the start of the lead line. The default setting is 1mm. To set the stand off distance, select the icon's window, type a value in the input window and press **Enter**. To return to the 'free' setting press **Enter** without keying in a value. Note that STAND OFF and LEAD LINES cannot both be set to 'free' at the same time.



**EXTENSION:** This is the distance between the terminator and the end of the extension line. The default setting indicated by 'AUTO' in the value window is 1.333 times the text height. Select this icon's window to specify a new extension distance; the text height will be added to it, so you are entering the 'margin' above the text. An input window is displayed into which you can type a value followed by **Enter**. To return to the 'AUTO' setting just press **Enter**.



**WEIGHT:** This icon displays the line weight used for leader lines. Select the value window to change it; a window is produced from which you can select the number of the new linewidth.



**BOTH:** Select this icon if you want both leader lines to be drawn.



**LEFT:** Select this icon if you only want the left hand leader line to be drawn.



**RIGHT:** Select this icon if you only want the right hand leader line to be drawn.



**NONE:** Select this icon if you don't want any leader lines to be drawn.

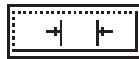
**Details:** The distances you can set up with these icons are all measured in 'paper units' — they aren't affected by the drawing scale. Unlike some 'paper units' distances, they aren't altered when you change the drawing scale.

This region of the palette controls dimension lines.

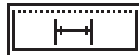


**HALF LINE:** This function allows you to alter the length of the dimension lines between the leader lines and the text. For a dimension line that is broken by text this is the length of each half of the line up to the edge of the text box. By default Accucadd draws dimension lines up to the edge of the text box.

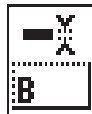
You can change the length of the dimension lines by selecting this icon's window and typing a new value in the input window. Accucadd will ignore values which are greater than half the dimension distance. For example, a line 20mm long which is given a half line length of, say, 50mm (too long!) will result in two dimension lines being drawn around the text at 10mm each. The length is measured in 'paper units' — it isn't affected by the drawing scale. Unlike some 'paper units' distances, this length isn't altered when you change the drawing scale.



**INVERT:** Select this icon if you want all dimension lines to be drawn outside of the lead lines. You can break dimension lines between the lead lines or draw a continuous line between the lead lines by selecting **BROKEN** or **CONTINUOUS** (below) in conjunction with this icon.



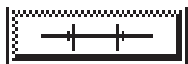
**FIT:** When this icon is selected, dimension lines will always be drawn inside the lead lines, except when the text and dimension lines are too big to fit between the lead lines. In this case the dimension lines will automatically be drawn outside of the lead lines. If text planted between the dimension lines is wider than the space between the dimension lines then the terminators will be inverted (drawn outside the lead lines).



**WEIGHT:** This icon displays the line weight used for dimension lines. Select the value window to change it; a window is produced from which you can select the number of the new lineweight.



**BROKEN:** Select this icon to break dimension lines between the leader lines. This is the default selection. This icon works in conjunction with **INVERT**.



**CONTINUOUS:** Select this icon to draw a continuous line through the space between lead lines. This icon works in conjunction with **INVERT** and **TEXT ABOVE LINE** (see Dimension: Set Up: Text). It only applies if you have selected **INVERT**.



**BOTH:** This is the default selection; it draws both dimension lines.



**LEFT:** Select this icon if you only want to draw the left hand dimension line.



**RIGHT:** Select this icon if you only want to draw the right hand dimension line.



**NONE:** Select this icon if you want no dimension lines to be drawn.



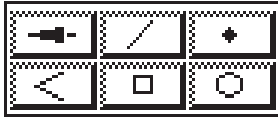
**FULL LEFT:** Select this icon to draw a full-length dimension line with only one terminator, at the left-hand end.



**FULL RIGHT:** Select this icon to draw a full-length dimension line with only one terminator, at the right-hand end.

**Notes:** The FULL LEFT and FULL RIGHT options are often useful for CNC dimensioning.

This region of the palette controls markers — dimension line arrowheads and circle center markers.

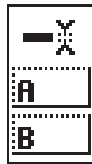


**TERMINATORS:** These icons allow you to select dimension line terminators. Position the cursor over the desired terminator type and click the button. The size of the terminators is scaled to the dimension text height, in accordance with the drawing standards supported by Accucadd. It is possible to alter the size of the terminators individually if they do not suit your in-house standards; the process is described under Dimension: Terminators.



**CIRCLE MARK SIZE:** This icon allows you to change the size of the circle center marker; select the current value and type a new size in the input window. The size is measured in `paper units` and isn't affected by the drawing scale. Unlike some `paper units` distances, this size isn't altered when you change the drawing scale.

Although you can enter larger values, the maximum size that will be drawn is about 20mm or 0.8 inches



**WEIGHT:** This icon shows the line weights used for terminators and circle center marks. The upper value window shows the weight for terminators, the lower, for circle centers. To change a line weight, select the value window and then select a new line weight from the list, as for the line weights of leader lines (see above).

This region of the palette controls dimension and tolerance text.



**FONT:** Select this icon's window to change the font in which the dimension text is drawn on the screen. When you do this a window is displayed containing all of the text fonts available in Accucadd. To select one of these fonts, position the cursor over the desired font and click the button. You can scroll up and down the font window to view all of the fonts that are available.



**TEXT ON LINE:** Select this icon if you want text to be placed in the middle of the dimension line. Note that the dimension line will always break around the text in this mode. This is the default selection.



**TEXT ABOVE LINE:** Select this icon to place text above the dimension line. Use this icon in combination with TEXT STAND OFF.

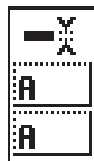


**TEXT HEIGHT:** The text height to be used for dimensioning. This is the 'standard' height — you can use other heights in dimensioning functions, but this is the default. Select the current value to enter a new value.

The size of dimension terminators (see Dimension: Set Up: Markers) and the height of the text used to display tolerances are calculated from the dimension text height. It is measured in 'paper units' and isn't affected by the drawing scale. Unlike some 'paper units' distances, the text height isn't altered when you change the drawing scale.



**TEXT STAND OFF:** This is the distance between the baseline of the text and the dimension line. It can be positive or negative. Select this icon to change the distance between the dimension line and the text. A positive value will place the text above the dimension line. A negative value will place the text below the dimension line. For vertical lines, read left for top and right for bottom.



**WEIGHT:** This icon shows the line weights used for dimension and tolerance text. The upper value window shows the weight for dimension text, the lower for tolerance text. To change a line weight, select the value window and then select a line weight from the list, as for the line weights of leader lines (see above).

This region of the palette controls the **Style and Layering** of dimensions drawn with the **DIMENSION** menu functions.

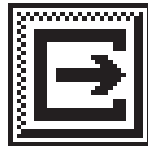


**LAYER:** Select this icon's window to change to another layer. This lets you place all of your dimensions on a separate layer conveniently, without changing layers. A window is displayed at the bottom of the screen which prompts you to enter a layer number. The default selection, ("A"), places all dimensions on the current layer.

This setting does not apply to "dimensions" created using the Draw functions with line weights A and B, only to Entities created by the Dimension menu functions.



**STYLE:** Select this window to save or load a Accucadd Dimension Style. A window is displayed which gives you the option to Save or to Load a Style. Select one of these options, then select a Style name from the catalog. An Accucadd Dimension Style saves all of the settings on the Dimension: Set Up palette.



**EXIT:** Select this icon to leave Dimension: Set Up.

**Details:** When you select Save or Load from the window the Dimension Style Catalog is displayed. This contains pre-defined styles for the four main dimension standards, DIN, ISO, ANSI, and BS308 plus the default dimension setup called Accucadd.

If you select, say, ANSI from the window all of the dimension line, dimension text, and lead line attributes associated with ANSI dimensioning will be activated in the Dimension Set Up palette. All you have to do is plant the dimension on the drawing to get perfect ANSI dimensioning.

To save a combination of settings as a Dimension Style activate any combinations of icons you wish on the Dimension: Set Up palette and type in values for text height, text stand off etc. Then select the Style icon and select Save in the window that is displayed. The Style catalog will be displayed along with a window at the bottom of the screen prompting you to type in a name for the new Style that you have created. Do this, then press **Enter**.

This function allows you to edit the style and position of Distance dimension Entities. The endpoints of the dimensions are not altered, so this function does not alter the dimension's length. It does not affect other types of dimension (radius, diameter and angle).

Edit Distance presents sub-menu for you to select **Position** or **Style** editing:

**Position** The Whole Group palette, described under Edit: Group, and the Dimension: Distance palettes appear. Change the settings on the Dimension: Distance palettes to reflect the type of dimension you want. Note that you can turn absolute into horizontal, vertical or CNC dimensions, or make any other changes available on the palettes.

Use the Whole Group palette to select the Distance dimensions to be altered, and then select **OK**. The dimensions are altered to correspond to the new settings: depending on the type of dimension they become and the Path Dimension Offset setting, you may have to position each one, using the cursor.

You can change the settings on the Distance palettes as you position each Dimension. Each setting will affect the Dimension you're currently changing, and those following it.

**Edit Style** The Whole Group and Dimension: Set Up palettes appear; select the (new) settings you want to use on the Set Up palette (or load a new Dimension Style).

Use the Whole Group palette to select the Distance dimensions to be altered, and then select **OK**. The dimensions are altered to correspond to the new settings.

**Details:** Edit Style and Edit Distance each allow you to change the settings on their palettes; you may need to use both for some changes. If you wish to change a drawing from one dimension standard to another (e.g., ISO to ANSI), the ALL icon on the Whole Group palette is very useful.

If you use Path Dimension Offset to set a group of dimensions to the same dimension offset, each one will be offset in its original direction.

The selected dimensions are re-created during this process. A useful side effect is that, since they are drawn over again, you can change units. To change an inch dimensions to millimeters, change the current units to millimeters, use Edit Style, Change All, and the dimensions will be redrawn in millimeters.

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# Distance

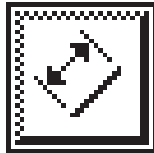
# Dimension

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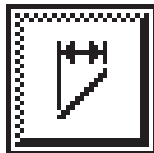
This function dimensions the distance between two points. Several sets of points can be dimensioned by using Path selection.

The Distance, Distance Selection, Tolerance, Dimension text height, Prefix & Suffix and Dimension Display palettes are all displayed:

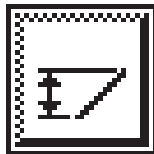
The Distance palette contains the following icons:



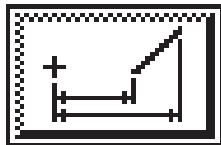
**ABSOLUTE:** Select this icon to dimension the absolute distance between two points.



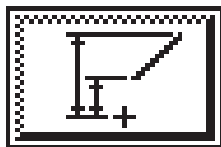
**HORIZONTAL:** Select this icon to dimension the horizontal distance between two points.



**VERTICAL:** Select this icon to dimension the vertical distance between two points.



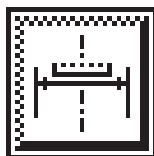
**X-CNC:** Select this icon to dimension distances in X from the current Datum point. This is the newest Handle in the drawing, or the center of the paper if there are no handles.



**Y-CNC:** Select this icon to dimension distances in Y from the current Datum point. The CNC dimensioning options are so called because of their usefulness in preparing drawings for CNC work; their effects depend on the settings of the Distance Selection palette (see below).



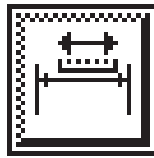
**PATH DIMENSION OFFSET:** When the PATH icons of the Distance Selection palette are used (see below), this value can be used to set the distance between the dimension line and the line being dimensioned (Absolute dimensions) or the offset of successive dimension lines (Horizontal, Vertical or CNC dimensions — see below). When it is set to FREE (click on the value, then press **ENTER**), each dimension must be positioned with the cursor. Note that this value can be set to zero: type **0** and press **ENTER**.



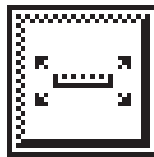
**CENTRAL:** Select this icon to position the dimension text centrally between the two ends of the line.

# Dimension

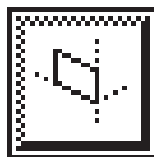
# Distance



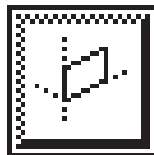
**ON LINE:** Select this icon to position dimension text anywhere as long as it 'rests on' the dimension line. If you use the PATH icons from the Distance Selection palette — see below —, you must position each dimension's text block individually.



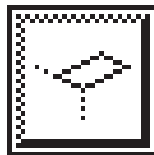
**FREE:** Select this icon to position dimension text anywhere on the drawing. Again, if you use the PATH icons from the Distance Selection palette — see below —, you must position each dimension's text block individually.



**ISO LEFT:** Select this (and the following ISO icons) for dimensioning isometric drawings. If you use this in conjunction with the HORIZONTAL or VERTICAL icons, the dimension is corrected for the isometric view. For ISO LEFT, dimension text is slanted at  $-30^\circ$  and rotated by  $30^\circ$ . It fits onto the left face of an isometric cube.

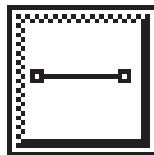


**ISO RIGHT:** Select this icon for dimension text slanted at  $30^\circ$  and rotated by  $330^\circ$ , which fits onto the right face of an isometric cube.

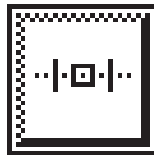


**ISO PLAN:** Select this icon for dimension text slanted at  $-30^\circ$  and rotated by  $330^\circ$ . It fits onto the top face of an isometric cube.

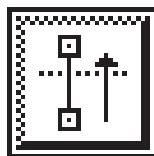
The Distance Selection palette is used to control the selection of endpoints for dimensions. It contains the following icons:



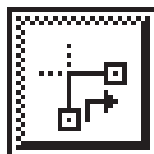
**LINE:** Enter the endpoints by positioning the cursor twice and clicking, as if you were drawing a line using DRAW: ELEMENTS: LINE. If you are using a CNC dimension setting, the first point is already fixed, at the datum.



**WHOLE ITEM:** Select a Line Entity, using the Select Item cursor, as if you were using the WHOLE ITEM icon from the GROUP Palette (see EDIT: GROUP). For most Distance settings, the endpoints of the line form the endpoints of the dimension; with the CNC settings, the first endpoint is at the datum and the second is at the endpoint of the line that is closest to the point where you selected the line.



**WHOLE ELEMENT PATH:** This icon is used to select a path, in the same way as the corresponding icon on the GROUP palette. *All* of the lines on the path are dimensioned individually; arcs can form part of the path, but are not dimensioned. For most Distance settings, the endpoints of each line on a path form the endpoints of the line's dimension.



**PART ELEMENT PATH:** This operates in the same way as the previous icon, except that parts of Entities can form segments of the path. Again, lines are dimensioned, and arcs and parts of circles forming segments of the path are ignored.

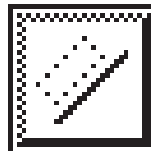
## PATH dimensioning effects

With a CNC setting, two dimensions are generated for each line: they have their first endpoint at the datum, and their second endpoints at opposite ends of the line. With Horizontal, Vertical or CNC dimensions, if two dimensions have the same position and the same value, one is suppressed.

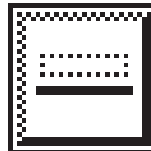
## Using PATH DIMENSION OFFSET

If the PATH DIMENSION OFFSET value is set, you must use the cursor to position the first dimension generated by the Path, and then the others are placed automatically. For Absolute dimensions, PATH DIMENSION OFFSET gives their offset from the line they dimension; Horizontal, Vertical and CNC dimensions are placed at intervals equal to the PATH DIMENSION OFFSET distance.

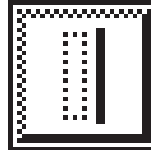
The Tolerance palette contains the following icons:



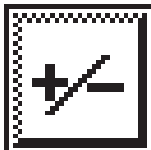
**PARALLEL TEXT:** Select this icon to display dimension text parallel to the distance dimensioned. This isn't actually part of tolerancing, but it is a convenient place for the icon.



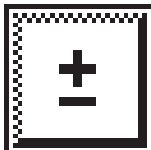
**HORIZONTAL TEXT:** Select this icon to display text horizontally.



**VERTICAL TEXT:** Select this icon to display text vertically.



**PLUS AND MINUS:** Select this icon to enter separate positive and negative tolerances. The TOLERANCE icon must be active — see below.



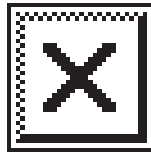
**PLUS-MINUS:** Select this icon to enter a single positive/negative tolerance. Again, the TOLERANCE icon must be active.



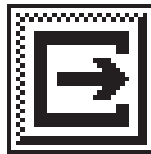
**MAX-MIN:** Select this icon to enter absolute minimum and maximum dimensions. TOLERANCE must be active.



**TOLERANCE:** Select this icon to enable tolerance entry.



**CANCEL:** Select this icon to cancel this dimension and select a new distance to dimension.



**EXIT:** Select this icon to stop dimensioning and remove the palette.



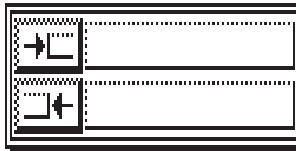
**DIMENSION STYLE:** Select the window containing the current Dimension Style name to reset the text height to the style selected in DIMENSION: SET UP.



**TEXT HEIGHT:** Select this icon to set the dimensioning text height. Selecting the DIMENSION STYLE icon returns this to the value set in DIMENSION: SET UP.



**DIMENSION SCALE:** Drawings can be dimensioned using a different scale: the dimensioning scale. Select the DIMENSION SCALE icon to use the dimensioning scale; click on the scale displayed to enter a different scale, as per SET UP: SCALE. DIMENSION SCALE only affects the *values* of dimensions: the drawing isn't altered, and the dimension text height remains in 'paper' units.



The Prefix & Suffix palette contains two windows into which you can enter up to 12 characters. The two strings of characters can be used as a prefix or suffix to the dimension text generated by Accucadd.



**PREFIX:** Select this icon to apply the Prefix string to the dimension.



**SUFFIX:** Select this icon to apply the Suffix string to the dimension.



The Dimension Display palette varies in appearance according to the tolerancing mode selected. This is its form with no tolerancing in use.

**Operation:**

Select the type of dimension required, the text height and orientation and the tolerancing required. All of these can be changed after selecting a line, if desired. Select the Distance Selection setting required, and then follow the instructions next to its icon (above).

**PATH DIMENSION OFFSET**

If you have set this value, defining the position of the dimension completely, it is drawn immediately. However, you will usually position

dimensions with the cursor:

**Positioning with the cursor** When you have selected the endpoints of a dimension, Accucadd confirms your selection by indicating where the dimension will be displayed. A rectangular box represents the position of the dimension text.

**Overriding dimension text** The dimension text is shown in the dimension display palette. You can change it by positioning the cursor over the text, clicking the button and typing the new text, terminating it by pressing **Enter**. This doesn't alter the true length of the dimension; just the dimension text to be added to the drawing. The value entered will be converted to the current precision. To enter a number or value without converting it to the current precision, type a quote-mark (") before it. If you override the dimension's value, it won't be updated when you stretch or edit the dimension.

Position the dimension lines with the cursor, then click the button draw the dimension. If the text position is CENTRAL, then it's drawn along with the rest of the dimension. Otherwise, position the text box and click the button again to draw the text.

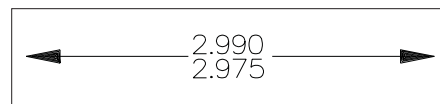
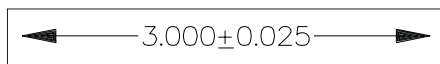
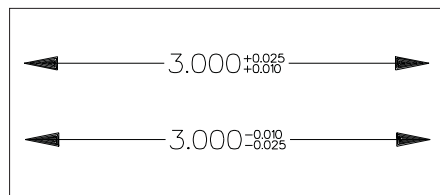
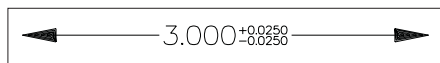
**Details:** Note that this function can dimension the distance between any two points, allowing easy overall dimensions, or measurement of distances.

Dimensions are reported according to the current precision and zero-display settings. To change these, use Set Up: Precision.

Dimension: Entities describes the special Entities which are used to record Dimensions.

**Tolerancing:**

A palette will also be shown displaying the value of the dimension and the values of its tolerances. This palette varies in appearance, depending on the tolerance mode selected.



**PLUS AND MINUS (+ / -):** This allows separate positive and negative tolerances. Enter both in the boxes on the tolerance palette.

You can make both values positive, by prefixing the 'negative' value with a plus sign ('+'). This is usually used for shaft diameters: you can also make both values negative, by prefixing the 'positive' value with a minus sign ('-').

**PLUS-MINUS (±):** This allows a single value, which should be entered on the tolerance palette.

**MAX-MIN:** Enter the actual maximum and minimum dimensions on the tolerance palette.

**Tolerance details:** The size of the small text used in tolerances can be controlled, as described under Dimension: Terminators: Tolerance.

Accucadd stores tolerance values as an amount to be added onto, and an amount to be subtracted from, each dimension. In PLUS-MINUS mode, both are set to the same value. In PLUS AND MINUS mode, they have distinct values. MAX-MIN mode is the same as PLUS AND MINUS, except that the tolerance values are applied to the dimension. All this means that you can change between tolerance modes without having to re-enter tolerance quantities.

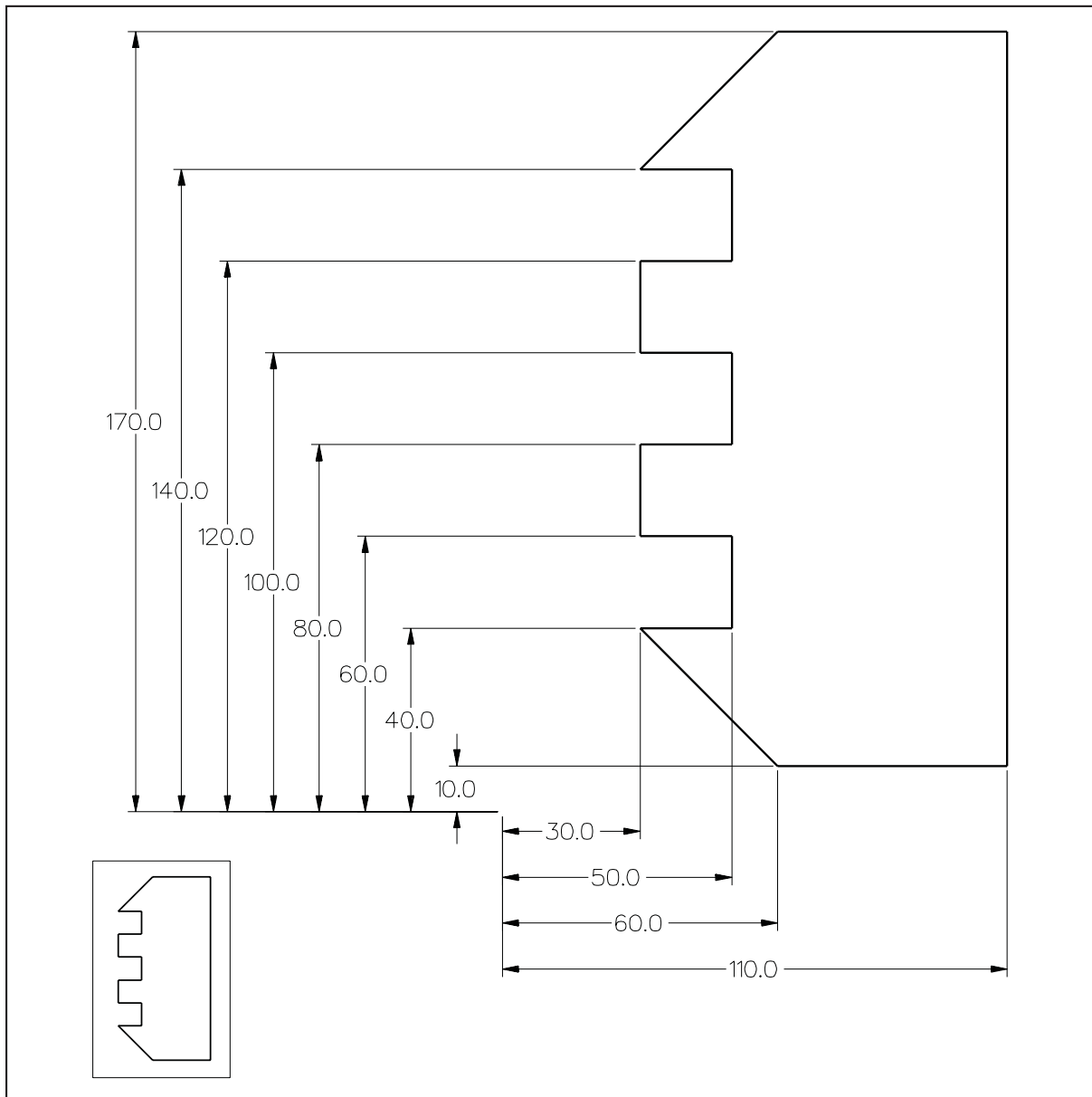
The tolerance values are retained, so that setting a tolerance of, for example,  $\pm 0.05$  will result in that value being used for all dimensions until it is changed. Changing to MAX-MIN would result in a distance of 1.000 being displayed as (0.950, 1.050), and a distance of 7.55 being displayed as (7.50, 7.60). A little experimentation is advised to find the tolerance mode and values correct for your work, which can be saved as part of the Dimension Style.

**Please note that Accucadd does not attempt to check if tolerances are sensible.**

**Dimension Styles** All text, dimension line and leader line attributes can be altered using Dimension: Set Up. This provides a flexible way of defining and using Accucadd dimension styles (this includes the pre-defined standards).

**Example drawing** The drawing below was dimensioned using Whole Element Path and a Path Dimension Offset of 10mm with X-CNC and Y-CNC. The inset shows the object without any dimensions.

To draw each set of dimensions takes just four mouse clicks: beginning of path, orientation, end of path, and first dimension position. To dimension the entire drawing, as shown, required four clicks for the X set and four clicks for the Y set—that's just eight mouse clicks to add twelve dimensions, all neatly positioned.



This function allows you to edit the style and position of Distance dimension Entities. The endpoints of the dimensions are not altered, so this function does not actually alter the dimension's length.

EDIT DISTANCE presents a window, for you to select style or position editing:

**Edit Position** The WHOLE GROUP palette, described under EDIT: GROUP, and the DIMENSION: DISTANCE palettes are presented. Change the settings on the DIMENSION: DISTANCE palettes to reflect the type of dimension you want. Note that you can turn absolute into horizontal, vertical or CNC dimensions, or make any other changes available on the palettes.

Use the WHOLE GROUP palette to select the Distance dimensions to be altered, and then select **OK**. The dimensions are altered to correspond to the new settings: depending on the type of dimension they become and the PATH DIMENSION OFFSET setting, you may have to position each one, using the cursor.

You can change the settings on the DISTANCE palettes as you position each Dimension. Each setting will affect the Dimension you're currently changing, and those following it.

**Edit Style** The WHOLE GROUP and DIMENSION: SET UP palettes are displayed; select the settings you want to use on the SET UP palette (or load a new Dimension Style).

Use the WHOLE GROUP palette to select the Distance dimensions to be altered, and then select **OK**. The dimensions are altered to correspond to the new settings.

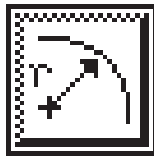
**Details:** Edit Style and Edit Distance each allow you to change the settings on their palettes; you may need to use both for some changes. If you wish to change a drawing from one dimension standard to another (e.g., ISO to ANSI), the ALL icon on the WHOLE GROUP palette is very useful.

If you use PATH DIMENSION OFFSET to set a group of dimensions to the same dimension offset, each one will be offset in its original direction.

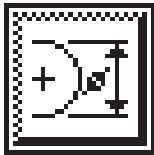
These functions are used to dimension diameters and radii.

Radius and Diameter are two separate functions in the Dimension menu. However the palettes for both functions are exactly the same except that the default settings on each palette are different. When you select Radius from the menu the palette appears with the single line radius icon active. When you select Diameter from the menu the palette is displayed with the single line diameter icon active.

The Tolerance, Dimension text height, Prefix & Suffix and Dimension Display palettes are also displayed. They are described under Dimension: Distance and operate in the same manner for Radius and Diameter.



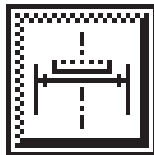
**SINGLE RADIUS:** Select this icon to draw the radius of a circle or an arc with a single line dimension line. Select the item anywhere on the perimeter to display the dimension lines. Click again to plant the dimension. The text and dimension line can be altered using Dimension: Set Up.



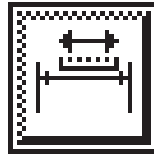
**DOUBLE DIAMETER:** Select this icon to draw the diameter of a circle with double dimension lines. Select the item anywhere on the perimeter to display the dimension lines. Click again to display the orientation of the cursor and then click again to plant it. The dimension lines, lead lines and text can be altered in Dimension: Set Up.



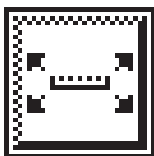
**SINGLE DIAMETER:** Select this icon to draw the diameter of a circle or arc on a single line. Select the item anywhere on the perimeter and move away to display the dimension line. Click again to plant it. The dimension line and text attributes can be altered using Dimension: Set Up.



**CENTRAL:** Select this icon to position the dimension text centrally between the two ends of the line.



**ON LINE:** Select this icon to position dimension text anywhere as long as it rests on the dimension line.



**FREE:** Select this icon to position dimension text anywhere on the drawing.

**Operation:** Select the circle or arc to be dimensioned by positioning the cursor over its circumference and clicking the button.

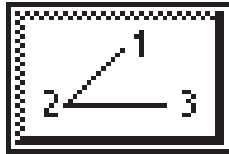
In PARALLEL TEXT mode (selected from the Tolerance palette), a radius or diameter is displayed with a text box parallel to it. Move the cursor to position the diameter and click the button to draw it. Repeat this process to draw the dimension text.

If the circle is “small” (its diameter is less than three times the width of an arrow head) the diameter is extended so that the arrow heads point in rather than out. Position the cursor and click the button once to draw the dimension lines and again to draw the text.

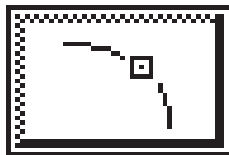
In HORIZONTAL TEXT mode, a radius or diameter is displayed, extended to join the handle at the lower left corner of the dimension text box. Move the cursor to place the diameter at the required position and to extend or shorten the leader line. Click the button to draw the two lines and the text simultaneously.

This function adds angle dimensions to the drawing.

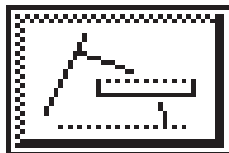
The Angle Dimension palette is displayed, along with the Tolerance, Dimension text height, Prefix & Suffix and Dimension Display palettes (all described under Dimension: Distance)



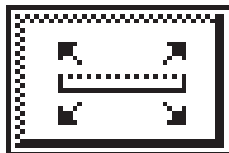
**THREE POINT ANGLE:** Select this icon to dimension an angle defined by three points in the drawing. Position the cursor and click the button to place three points as shown by the icon. Position the dimension and click the button to draw it. If FREE TEXT is selected, position the text and click the button to draw it.



**ARC ANGLE:** Select this icon to dimension the angle of an arc in the drawing. Select the arc anywhere on its perimeter, position the cursor to display the dimension and click to draw the dimension. If FREE TEXT is selected, position the text and click the button to draw it.



**FIXED TEXT:** Select this icon to fix the dimension text in the center of the dimension arc.



**FREE TEXT:** Select this icon to position the dimension text on the drawing with the cursor.

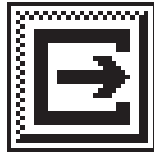
Tolerances and text height are used in the same manner as for the other dimensioning functions.

The attributes of the dimension arc, lead lines and text can be altered using Dimension: Set Up.

Note that when you are using Three Point Angle the angle will be calculated and displayed in a clockwise direction from the first point selected. In every case either the major or minor angle may be dimensioned simply by moving the cursor from "inside" to "outside" the angle.

This function enables you to draw arrow heads or other terminators on selected Entities, for instance to indicate a composite dimension line that you have added to the drawing by hand. The terminators created are the same as those currently selected under Dimension: Set Up. The ability to find intersections automatically, and draw “arrows” there, is included.

The Dimension text height palette is shown to allow you to change the text height (which controls the arrow head size). This operates as described under Dimension: Distance.



A palette with one icon is shown; select **EXIT** when you wish to leave Arrow.

The Select Item cursor is displayed. Select an Entity from the drawing. Accucadd finds the intersection or endpoint of the chosen Entity which is nearest to the cursor and draws a terminator at this point, with the terminator pointing away from the cursor.

Details: Circles must have at least one intersection point in order for the system to calculate the position of the terminator.

If the terminators are to be treated as part of the drawing's dimensioning (see View: Speed Filter), draw them in line weight A, and place them in the same layer as you are using for the dimensions created with Dimension functions.

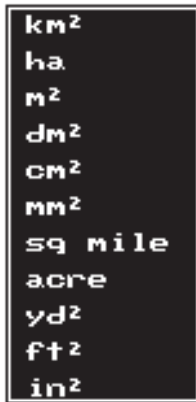
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# Area (and Perimeter)

# Dimension

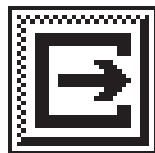
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This function reports the Area and Perimeter of a region on the drawing and lets you place the Area and Perimeter values on the drawing as dimension text.



When you select Area, the Area Units Selection window is displayed. This presents the available units in which the Area and Perimeter can be reported. Select the units you wish to use.

Next you will be prompted to enter a resolution value. This controls the accuracy of the Area calculations. Enter a value between 1 and 0.01 — its significance is explained below.



An Exit palette appears and the Hatch cursor is active. Click **Exit** to leave the Area function.

As described under Draw: Hatch, place the seed point for the area to be calculated and then a box surrounding it. Accucadd finds the borders of the area, and then hatches it. If this takes a long time Accucadd will show you a progress counter.

Several of the Dimension Text palettes appear: Height, Prefix and Suffix. These can all be used as described under Dimension: Distance.

The Prefix & Suffix palettes are pre-loaded with the “Area=” and “units of measure” text: change these if you wish. Move the Text cursor (containing the Area value) and position it where you want the area value to appear in your drawing. Click to draw the text. The Area value is also shown on the Status Bar.

To draw the Perimeter text, position the cursor and click the button. Again, you can use the Prefix & Suffix palette to alter the text supplied by Accucadd if you wish.

**Details:** The Area calculations are based on the area which is covered by the Hatch pattern. This Hatch pattern is made up of vertical strips. Accucadd calculates the length of each strip and then multiplies the sum of the lengths by their common width to give a value in square units.

The hatching shown on the screen isn't part of the drawing. It gives you an indication of the region that was measured and will disappear the next time you redraw, regenerate, zoom, pan or shrink.

You can edit the Area and Perimeter text using Edit: Text.

**Accuracy:** The Perimeter value calculated by Accucadd is as accurate as double precision floating point permits. This is generally to 16 decimal digits. The result of the calculation will be displayed rounded to the number of decimal places set under Set Up: Precision—a maximum of six. The accuracy of the Area value depends on the resolution value entered. The area accuracy is never perfect; varying the resolution enables you to increase accuracy at the expense of longer calculation time. An accuracy of one part in one million is typical with judicious use of the resolution value for strictly geometric boundaries (lines, arcs and circles).

Accucadd—and every other CAD system—approximates curves with line segments, and it is this segmented boundary rather than the “mathematical” one which will be used.

**Resolution:** The resolution value you enter gives the width of the strips used to calculate the area, in the units you selected for reporting of the area and perimeter.

For example, select **m<sup>2</sup>** as the units and then type **1** for the resolution value. The strips will be one meter wide (in the drawing’s scaled coordinates) and the AREA value will be fairly approximate.

If you were to use a resolution of **0.01**, the strips would be 0.01m (one centimeter) wide, and the AREA value would be much more accurate. It would still be reported in square meters, and it would take much longer to calculate.

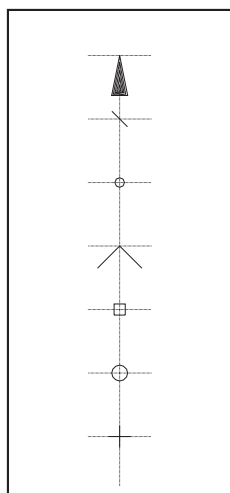
This section describes the dimension terminators used by the Accucadd automatic dimensioning functions and the techniques used for customizing them. The text used for displaying dimension tolerances can also be customized, and this is described below, under “Tolerances” You don’t need to know anything about this section to draw with Accucadd: it only applies if you wish to use advanced customization of dimensioning.

Dimension terminators are single text characters, drawn from the special-purpose DIMTERM font. The text records that hold them are created automatically by the DIMENSION functions.

The size of the dimension terminators can be customized by altering the special file **ac\_dterm.siz**, stored in the Accucadd System Directory. On a network or machine-sharing installation, the file is located via the Search Path and should be placed in the GLOBAL directory to act as a proper “in-house standard”

Two “size” files are supplied with Accucadd: **ac\_dterm.big** and **ac\_dterm.sml**. The first one (.big) contains data conforming to the national and international dimensioning standards supported by the program. Many people, however, find that this produces arrowheads that are larger than they would like to see. The second (.sml) file contains data that, while differing from these standards, produces more esthetically pleasing arrows. Accucadd uses (only) **ac\_dterm.siz**. Accordingly, if you want “big” arrows, delete **ac\_dterm.siz** and copy **ac\_dterm.big** to **ac\_dterm.siz**. If you want “small” arrows, copy **ac\_dterm.sml** to **ac\_dterm.siz**. You must then re-start Accucadd for the changes to take effect.

Altering the installed file allows for further in-house variations and “fine tuning”; the only part of **ac\_dterm.siz** that normally needs to be altered for this purpose is the third column, “Size”.



Code	Adjst	Size	Max	Min	Position	Draw/Through
192	1.0	1.8	20.0	1.0	center	draw
193	1.0	0.7	20.0	1.0	middle	draw
194	1.0	0.3	1.5	1.5	middle	draw
195	2.0	1.0	20.0	1.0	center	draw
196	1.0	0.5	10.0	1.0	middle	skip
197	1.0	0.7	2.5	2.5	middle	skip
198	1.0	1.0	20.0	1.0	middle	draw

`ac_dterm.siz` There are seven dimension markers described in this file; the six dimension terminators shown on the Dimension: Set Up palette and the marker used for the centers of circles. See Dimension: Set Up: Markers.

**ac\_dterm.siz** is an ASCII text file, suitable for editing with the Accucadd Text Editor (see Draw: Text). Its normal contents are shown above.

There is one line of information for each marker; the columns are separated by `|' characters and spaces and tabs are used for readability. The information in the columns is:

Code	The ASCII character code for the marker. This is the code of the relevant character in the DIMTERM font, which doesn't have the same characters as a standard font. You won't need to alter this unless you use RoboFONT to create new characters in DIMTERM.
Adjustment	This is a scaling factor, used to adjust the size of marker characters so that they fill the full size of the character grid. A value of 2.0 (for example, line 4 in <b>ac_dterm.siz</b> ) doubles the size of the character drawn on the grid. You won't need to use this unless you design your own characters with RoboFONT; adjust the size of terminators using the next field.
Size	This is a scaling factor for the size of the marker. It is multiplied by the Accucadd Dimension Text Height to give the size of the marker. The size of each marker can therefore be controlled independently. To double the size of the marker, double this number, to halve it, halve the number. The most common change is to shrink the solid arrowhead (character 192); the usual Size factor adopted is 1.35.
Maximum	The maximum size of the marker, in millimeters, after the Adjustment and Size factors have been applied.
Minimum	The minimum size of the marker, in millimeters, after the Adjustment and Size factors have been applied. Note that Maximum and Minimum can be used to create fixed-size markers, such as the dot and circle (characters 194 and 197).
Position	The position and orientation of the marker. You will not need to alter this unless you are using RoboFONT to design new markers. Markers that point in a direction (such as arrowheads) are drawn on the character design grid pointing upwards, with their vertical axis on the center-line of the grid. Such markers should have "center" in this column. Markers that are symmetrical (such as circles) are drawn in the center of the character design grid, and should have "middle" in this column.
Draw-through	In this column, "draw" draws the dimension line through the terminator to its end; "skip" stops the line short at the terminator.
Details:	The characters in the DIMTERM font can be altered using RoboFONT, available through your supplier. This product enables you to alter any

of the Accucadd fonts and create new ones. Note that if you change the characters used for dimension markers, the images on the icons of the Dimension: Set Up palette won't change accordingly.

**Tolerances** When dimensions are drawn using the Plus And Minus icon ( + / -)—see Dimension: Distance—the separate plus and minus tolerances are written out in smaller text than the main dimension value, with a size scaled to it.

By default, each tolerance text block is  $\frac{1}{16}$  of the dimensioning text height. This ratio can be altered by *creating* a special file, **ac\_tolht.siz**. (There is no such file in the initial installation). This file should contain one number, written out as a “real”, or floating point number. This is the ratio of the tolerance text height to the dimensioning text height.

For example, the default tolerance text height would be re-created by a **ac\_tolht.siz** file containing:

0.4375            ( $\frac{1}{16}$  as a decimal)

The Accucadd Text Editor (see Draw: Text) is suitable for creating and altering the file; you will have to exit from Accucadd, and re-start it to observe the effect of each change. The default size matches the dimensioning standards that Accucadd supports: most organizations that change it make the tolerance text larger. The useful range of values is about 0.3 to 0.8.

**Details:** Note that the value in **ac\_tolht.siz** only affects dimensions as they are drawn. Changes will not affect existing dimensions.

**ac\_tolht.siz** should be placed in the System Directory. On networked or machine-sharing copies of Accucadd, it is found using the Search Path, and should be placed in the Global directory if it is to affect all drawings done on the network.