## COORDINATE INPUT METHODS (NON DYNAMIC MODE)

Absolute Cartesian Syntax:              **3,4**
Relative Cartesian Syntax:              **@4,2**
Absolute Polar Syntax:                  **9<75**
Relative Polar Syntax:                  **@5<60**
Spherical coordinate input Syntax:      **7<35<53**
(3 dimensional applications)
Cylindrical coordinate input Syntax:    **60<35,30**
(3 dimensional applications)

## A CHRONOLOGICAL HISTORY OF RELATIVE DRAFTING TECHNIQUES

## DISCUSSION – LASTPOINT (@)

The use, and manipulation of AutoCAD's *"lastpoint"* system variable to establish a **relative** working point, or to make use of a known reference point *during command execution* is the main focus for this section. When a new drawing is started AutoCAD establishes the *"lastpoint"* system variable to 0,0,0.  As you work on  the drawing AutoCAD continuously updates this variable.

## THE ID COMMAND

To establish a relative point (lastpoint) several methods can be employed.
Use of the **ID command** was the primary technique for establishing a relative point.
Simply entering the **ID** command, then selecting a point in the drawing caused
AutoCAD to report the selected point's X,Y, & Z coordinates, on its status line.
Because AutoCAD calculated this point and it was the most recently selected point,
the "lastpoint" system variable was automatically updated. In the next command
sequence (line, circle…) using the @ syntax will tell AutoCAD to calculated the
following coordinates relative to the "lastpoint" system variable just set with the ID
command rather than the cartesian coordinate system.

- Another method typically used (how it use to be done) to establish a *"lastpoint"* is
  the **^C (cancel method)**.  Using this method the CAD Operator begins a new
  command (line, circle…) and responds to the "From point:" prompt by selecting
  the point for which the "lastpoint" is to be established.  Once selected the Operator
  "cancels" the command (control+c, ESC). Using this method causes AutoCAD to

update the system variable "lastpoint" to the point selected.  Using the @ syntax at this point will instruct AutoCAD to calculate the following coordinates with respect to the "lastpoint" system variable.

- The *least recommended* method (and unfortunately *still commonly found* in the industry today) is the **geometric construction** method of establishing relativity. This method refers to the use of additional geometry (such as lines or circles) being generated in a drawing to form a point of reference. The most common technique found in use today is the method of generating *offsets* equal to the distance an entity is to be created "from". The point of reference created is then used as a point to work from (thereby establishing a relative point).

## USING THE "FROM" OSNAP

The **From** object snap differs from the other types of object snaps because it establishes a temporary reference point as a basis for specifying subsequent points. The **From** object snap is normally used in combination with other object snaps and relative coordinates, and most notable its use with the new **direct distance** and **tracking** methods of coordinate input.

For example, at the "start point" prompt when drawing a polyline relative to an existing line's midpoint, enter **from** then **mid**, select an existing line's midpoint, then enter @2,3 to locate the start point two units to the right and three units up from the midpoint of the selected line. The user can specify an absolute coordinate for use as a base point; however, specifying an absolute coordinate for the offset essentially cancels the **From** object snap, and locates the point at the specified coordinates.

Command:  **pline**
From point:  **from**
Base point:  **mid**
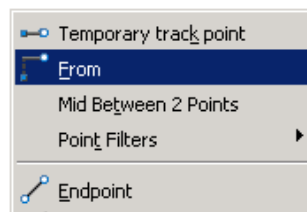of: *Select the line*
of <Offset>: Enter **@2,3**

**Figure 1 – From Osnap**

# DIRECT DISTANCE INPUT

Direct Distance coordinate input appeared in AutoCAD's R13c4a release.  (Imported from the LT version!). This method of coordinate inputs allows the user to specify the polar angle for input (or modifications) by positioning the pointing device (mouse, digitizer...) in the desired angle's direction.

The user then enters a distance and a return key.  This input method can be used during entity creation or *advanced editing*.  When used in conjunction with *Orthomode, From* and *Tracking* Osnap features the user can save a good deal of time by eliminating relative input coordinates manually.

**Note:**  If you toggle the *Snapang* system variable setting to 45 or 30 degrees when drafting in angles of 45 and 30 degrees (isometric), the angle increments are easily accessed when Ortho is set on.

## POINT FILTERS

AutoCAD's Point Filter features provide a means for borrowing and building coordinate pairs from existing geometry. They provide an ideal way to produce accurately aligned drawing features without the use of additional or reference constructions.

Point filters are transparent by design and are intended to be used during both drawing and editing commands.  Once mastered, point filters are a natural add-on to AutoCAD's autoediting and geometric calculator functions.

Every point in AutoCAD has a set of XYZ coordinates associated with it.  This can be seen when using the ID command to identify the location of any point in a drawing.

Command: **ID↵**
Point: *select any point↵*

**X = 2.50      Y = 1.75      Z = 0.0**

The "borrowing" aspect of **point filters** is the process of pulling out the X, Y, or Z value from any point in a drawing, holding it in abeyance, then inputting (or borrowing again) the remaining coordinates to complete a coordinate entry. At a minimum AutoCAD needs a X and a Y coordinate pair, as such whenever a coordinate(s) is missing during the building of a coordinate pair using point filters AutoCAD prompts for it.
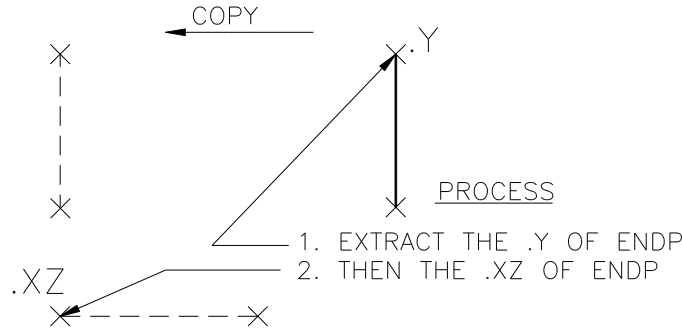
**Figure 2 – Point Filters with Copy Commands**

In Figure 3-2, the CAD Operator wants to copy an existing vertical line to a point directly over the left endpoint of the horizontal line shown, while maintaining the Y coordinate.

**STEPS:**
Command: **CP**↵
Select objects: *select vertical line*↵
<Base point or displacement>/Multiple: **END**↵
of  *select endpoint of vertical line*↵
Second point of displacement: **.Y**↵
of  **@**↵
(need XZ): **.XZ**↵
of  **END**↵
of  **select left side of endpoint of horizontal line**↵

**Note**:  This example illustrates a method of using AutoCAD's *point filters* and may not be the most appropriate command for the task shown.  In the case of figure 3-2, the Offset command's "Through" option would have accomplished the same task with less input and time**.  Can you do it using the offset command's "through" option?**

Figure 3-3 depicts the same methodology for using Point Filters to edit entities during an editing session.  The CAD Operator wants to move the horizontal line from its current position to a position aligned with the center of the existing circle.
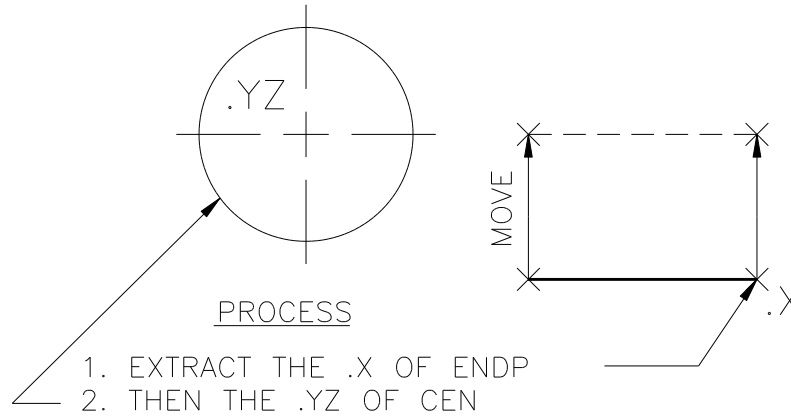
**Figure 3 – Point Filters with the Move Command**

STEPS:

Command: **M↵**

Select objects: *select horizontal line ↵*

<Base point or displacement>/Multiple: **END↵**

of  *select endpoint of horizontal line ↵*

Second point of displacement: **.X↵**

of  **@↵**

(need YZ): **.YZ↵**

of  **CEN↵**

of  *select CEN for the circle ↵*

The relative symbol (@) is commonly used when using Point Filters.  It allows the user to filter a coordinate (X,Y, or Z) transparently from the cursors current drawing position.  This allows the user to work accurately with relative coordinates during command line execution. Many times command line execution can simplify and lengthy command interface process.

Figure 3-4 illustrates the development of a right side view of a circular plate without the need for *additional constructions* or the need to ID existing geometry.
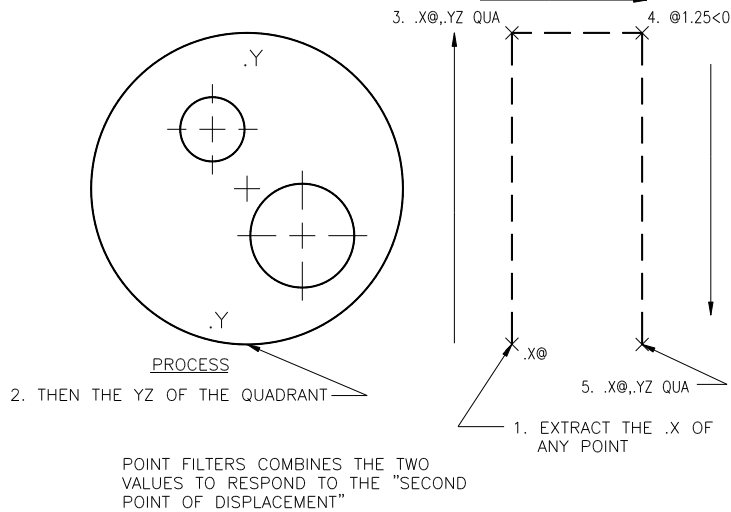
3. .X@,.YZ QUA        4. @1.25<0

.Y

.Y

PROCESS
2. THEN THE YZ OF THE QUADRANT

.X@

5. .X@,.YZ QUA

1. EXTRACT THE .X OF
   ANY POINT

POINT FILTERS COMBINES THE TWO
VALUES TO RESPOND TO THE "SECOND
POINT OF DISPLACEMENT"

**Figure 4 – Construction Side Views with Point Filters**

STEPS:
Command: **L↵**
From point: **.X↵**
of (need YZ): **.YZ↵**
of **QUA↵**
of  *select lower quadrant of existing circle↵*
To point: **.X↵**
of **@↵**
(need YZ): **.YZ↵**
of **QUA↵**
of  *select upper quadrant of circle↵*
To point: **@.75<0↵**
To point: **.X↵**
of **@↵**
(need YZ): **.YZ↵**
of **QUA↵**
of  *select lower quadrant of circle↵*
To point: **C↵**

As can be seen from the preceding examples, **Point Filters** provide valuable access to existing geometry's coordinate input pairs.  They eliminate the need to create additional constructions and simplify the process of drawing accurately.  Positioning new geometry with respect to existing geometry is also greatly enhanced. Now it's time for the **BIG GUNS!**

# THE "TRACKING" OSNAP (THE KING OF BEERS!)

### Tracking

In addition to Object Snaps and point filters, you can use AutoCAD's new **tracking** feature to tentatively locate points visually relative to other points in your drawing. Tracking combines both X and Y point filters and allows you to use them on the fly.  Tracking is a true Object Snap member and resides on the Osnap cursor menu.

You can use tracking whenever AutoCAD prompts you for a point. Tracking points are tentative until confirmed with a <enter> key. When you start tracking and then specify a point, AutoCAD turns *Ortho* on and constrains the next point selection to a path that extends vertically or horizontally from the first point.  (note: to change the orthogonal path, return to the last tracking point and move the cursor in the desired vertical or horizontal direction).  You may **track** as many points as you need, however, you must enter an *enter key* before tracking will terminate.

Tracking is invoked as an Osnap feature at the command line using **TK**. *This feature should be used with the Direct Distance feature when possible*.  See the following discussions for using AutoCAD "From" Osnap in conjunction with Tracking and Direct distance.

**Tip:** The direction you move after selecting the first tracking point determines whether its this point's X or Y value that is retained for constructing the new point. The second tracking point provides the second value of the new coordinate.  If the direction selected is not the direction of displacement desired, simply place the cursor adjacent to the first point and begin moving in the other direction.  Remember only orthographical directions (no diagonal) are allowed when using Tracking (TK) features.

Command: **pl**
From point: **tk**
First tracking point: **mid**
Next point  (Press ENTER to end tracking): **2 <enter>**
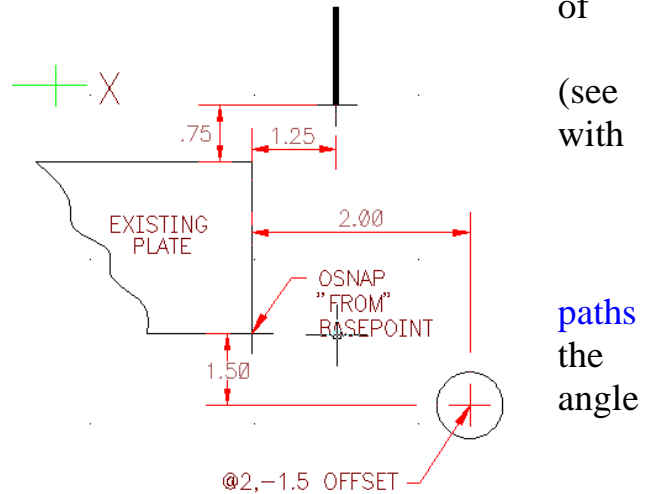Next point  (Press ENTER to end tracking): **3 <enter>**
Next point  (Press ENTER to end tracking):
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **<enter next point>**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **<enter>**

# COMBINING DIRECT DISTANCE & TRACKING

Create the figure below on your Workstation.  Locate the points for the circle's center point and the EndPoint of the line's beginning point (at "X") by using From and Tracking methods to establish the relative points.  Then use a typical relative coordinate entry method for the beginning of the line with the From osnap, and the direct distance coordinate input method Coordinate Input section) in conjunction the Tracking osnap to place the circle.

(see with

# AUTOTRACKING

As you move your cursor, alignment and tooltips are displayed when you move cursor near polar angles. The default measurement is 90 degrees. Use the alignment path and tooltip to draw your object.

paths the angle

**Figure 1 - Placing relative**

You can use polar tracking with Intersection and Apparent Intersection object snaps to find the point where a polar alignment path intersects with another object.

Ortho mode restricts the cursor to horizontal or vertical (orthogonal) axes. Because you cannot have Ortho mode and polar tracking turned on at the same time, AutoCAD turns polar tracking off when you turn on Ortho mode. If you turn polar tracking back on, AutoCAD turns Ortho mode off.

# POLAR TRACKING TAB CONTROLS AUTOTRACK SETTINGS

1. **Polar Tracking On:** Turns polar tracking on and off. You can also turn polar tracking on or off by pressing F10 or by using the AUTOSNAP system variable.

2. **Polar Angle Settings:** Sets the angles used with polar tracking.

3. **Increment Angle:** Sets the polar increment angle used to display polar tracking alignment paths. You can enter any angle, or select a common angle of 90, 45, 30, 22.5, 18, 15, 10, and 5 degrees from the list. This setting is also controlled by the POLARANG system variable.

4. **Additional Angles:** Makes any additional angles in the list available for polar tracking. The Additional Angles check box is also controlled by the POLARMODE system variable, and the list of additional angles is also controlled by the POLARADDANG system variable.

5. **New:** Adds up to 10 additional polar tracking alignment angles.

6. **Delete:** Deletes selected additional angles.

## OBJECT SNAP TRACKING SETTINGS

1. **Track Orthogonally Only:** Displays only orthogonal (horizontal/vertical) object snap tracking paths for acquired object snap points when object snap tracking (OTRACK) is on. This setting is also controlled by the POLARMODE system variable.

2. **Track Using All Polar Angle Settings:** Permits the cursor to track along any polar angle tracking path for acquired Osnap points when Object Snap Tracking is on while specifying points. This setting is also controlled by the POLARMODE system variable.

**NOTE:** Clicking Polar and Otrack on the active status line also turns polar tracking and object snap tracking on and off.

## POLAR ANGLE MEASUREMENT AREA

Sets the basis by which polar tracking alignment angles are measured. Options are:

1. **Absolute:** Bases polar tracking angles on the current user coordinate system (UCS).

2. **Relative to Last Segment:** Bases polar tracking angles on the last object you created.

As shown AutoCAD's users have a variety of drafting tools to enhance "relative" coordinate entry. Ortho mode, SNAP, Direct Distance, Point Filters, TK (tracking Osnap), Object Tracking (OTRACK), Polar Tracking, coordinate input methods round out the set of tools. Now for even **BIGGER GUNS!**

# DYNAMIC INPUT

The first thing you'll notice when you open AutoCAD 2006 and start to draw is that there's a sort of command line at the mouse cursor. This is called the **Dynamic Input (DI)** tool tip. As mentioned above Dynamic Input has three parts: 1) The prompt; 2) A down arrow: press the down arrow on your keyboard to see the current command's options, and 3) a text box where you can enter values, such as the length of the line, or as coordinate. Dynamic Input is controlled by AutoCAD's new System Variable, **DYNMODE**.

DYNMODE turns dynamic Input features on and off. The initial value is 3. When all features are on, the context governs what is displayed.

When DYNMODE is set to a negative value, the Dynamic Input features are not visible, but the setting is stored. Press the DYN button in the status bar to set DYNMODE to the corresponding positive value.

0 = all off
1 = pointer input on only
2 = dimensional input on only
3 = All on

When **DYNMODE** is set to anything but 0, you can turn off all features **temporarily** by holding down the temporary override key, **F12**.



**Figure 5 - Dynamic Input Components**
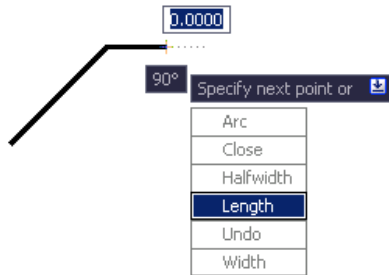
There are five different fields you can choose to display for grip editing:

Resulting Dimension, Length Change, Absolute Angle, Angle Change and Arc Radius (available by selecting Settings under Dimension Input).

*Tip: Set the system variable TOOLTIPMERGE to 1 to display both the indicated object snap and the command prompts.*

**Figure 6 - Down Arrow to Access Options**

**Figure 7 - Arrow Up to Show Last XY Coordinates**
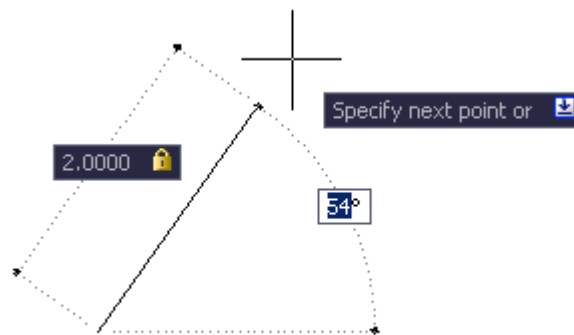
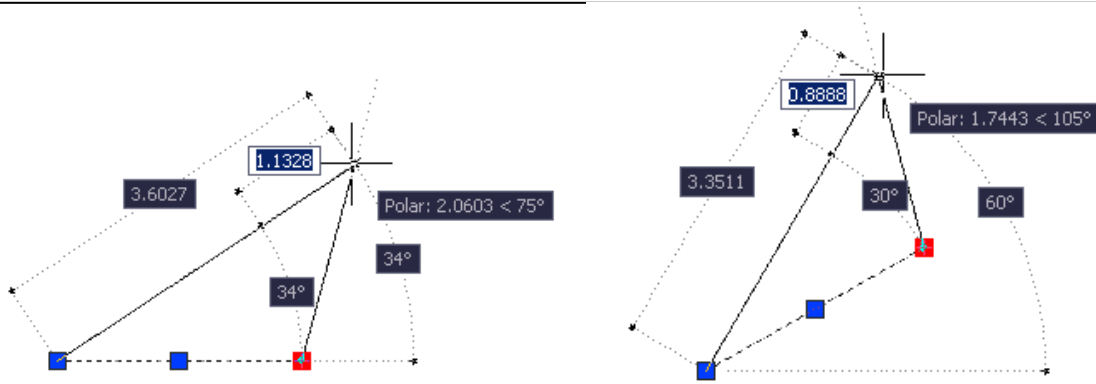**Figure 8 - Constrain Dimensional Input with the Tab Key**
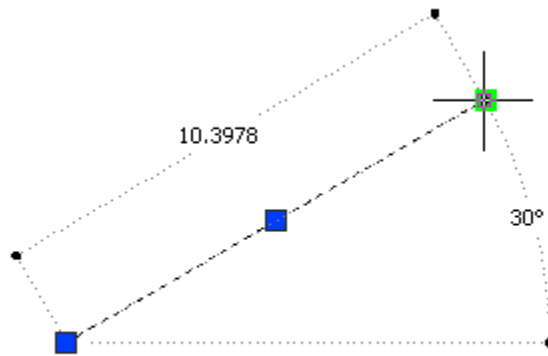
**Figure 9 – Additional Dimensional Input Fields**
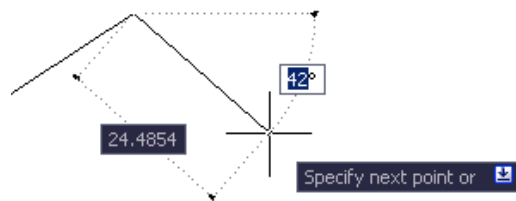
**Figure 10 - Hover Status Readout**

**Figure 11 - Use Tab Key to Dimensional Input Focus**

Specify next point or ⏬ @ 1.3310 < 0°

**Figure 12 - Using the @ Symbol**

Specify next point or ⏬ @ 2 🔒 0.0000

**Figure 13 - Using the @ Symbol then a Comma**

Specify next point or ⏬ @ 2 🔒 < 0°
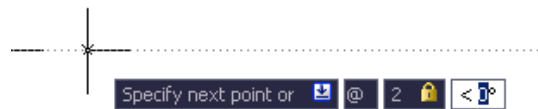
**Figure 14 - Using the Polar (<) Symbol**

- You activate Dynamic Mode using DYN (Dynamic Input) on the active status line, enable Pointer Input using DYNMODE and show the command prompt in the Tool Tip using DYNPROMPT.

- Settings can be set on the command line or in the Drafting Settings dialog box. The Tab key is used to toggle between values, and the Down arrow key is used to access and select the options for a command.

- You may want to experiment with closing the Command Line window altogether (remember you can toggle its display by pressing Ctrl+9), although this will probably give you the jitters from a long habit of depending on that trusty command line. Also, if you don't like DI, you can turn it off by clicking the new DYN button on the status bar. Use can still use the F2 key to toggle the command line history window! Or you can undock the command window and use Auto-hide to roll open or roll up the window.

- You can customize the tool tip quite a bit to specify when it appears, what it looks like, and how it behaves. For example, if you type coordinates in the tool tip, by default they act like relative coordinates. You can change this, but don't. **You may never have to type the @ symbol again!**

- When all three components are active and you select a command, all coordinate, and dimension values as well as command prompts, are displayed near the cursor, and the values get dynamically updated as the cursor moves.

Tip: Set the brand new TOOLTIPMERGE system variable to 1 to merge the OSNAP and DI tool tips and simplify the display.

## DIMENSIONAL INPUT & GRIPS

The behavior has changed when you use grips to stretch vertices when dimensional input is turned on (DYNMODE = 2 or 3). Entering a single direct distance value differs from previous releases because you are editing the segments' total length or angle value, not the vertex position. For best results, turn off dimensional input by setting DYNMODE to 0 or 1, by pressing F12, or by clicking the Dynamic Input button (DYN) on the status bar.

- There are five different fields you can choose to display for grip tool tip editing, they are: Resulting Dimension, Length Change, Absolute Angle, Angle

Change and Arc Radius (available by right-click -> Settings, on the DYN button, then under Dimension Input Settings area.

- When Dimension Input is on, coordinate entry fields are tied to dimensions on the geometry being created or edited. The values in the tool tip changes as you move the cursor. When you use grips to stretch objects or when you create new objects, the dimensional input pointer displays only acute angles. That is, all angles are displayed as 180 degrees or less. Angle specifications rely on the direction of the cursor movement to determine the positive angle direction.

- When you hover over a grip to edit an object, the tool tip displays the original dimensions. As you move the grip, length and angle values are updated dynamically.

## DYNAMIC INPUT GUIDELINES

- You can now enter coordinate values in a tool tip instead of on the command line. The *X,Y* coordinate values you enter for most commands are by default interpreted as relative polar coordinates--**not absolute coordinates**, as in previous releases of the product.

- To enter relative coordinates, you usually do not need to enter the at sign (@); you only need to enter the relative offset values. To indicate absolute coordinates, use the pound sign (#) prefix. For example, to move an object to the drawing origin, enter #0,0 at the second point prompt.

- The **DYNPICOORDS** system variable controls whether pointer input uses relative or absolute format for coordinates. You can use symbol prefixes to temporarily override these settings:

- Remember, to enter absolute coordinates when relative coordinates are displayed in the tool tip enter #, to enter relative coordinates when absolute coordinates are displayed, enter @. And as always, to enter absolute world coordinate system (WCS) coordinates, enter * (an asterisk).

- When dimensional input is turned on (DYNMODE = 2 or 3), the program switches to pointer input when you enter a comma (,) an angle bracket (<) or when you select multiple grip points.

- When you enter a dimensional value and press the TAB key, the field displays a lock icon, and the cursor is constrained by the value that you entered.

- You may get unexpected results if you enter values in a Dynamic Input tool tip when the cursor is off the drawing area and the tool tip is not visible.

- Dynamic Input is not supported for perspective views.

## POINTER INPUT GUIDELINES

When Pointer Input is on, the cursor displays coordinate values as you move it in the drawing area. To enter coordinates, type the value and press the TAB key to switch the focus to the next tool tip, then type the next coordinate value. When you specify the point, the first coordinate is an absolute coordinate. The format for the next point is relative polar coordinates.

- For dimensional input (**DYNMODE=3**): When you select multiple objects that share a common grip and then click the grip, the TAB key cycles through all the dimensions for each selected object

- For pointer input (**DYNMODE=1**): If you enter a value during a command, use the left angle bracket (<) or comma (,) instead of the TAB key to display an additional input field.

## CONSTRAINING DIMENSIONAL INPUT

1. You can constrain any length or angle value by using the TAB key to validate your input. In other words the number 4 followed by the TAB key would constrain the length of an object on an alignment path to 4" no matter what the angle of travel was.

**Example:** A line is to be drawn 2" long at 54 degrees. A command sequence of 2 TAB key will strike a 2" line and constrain that length while it is moved into the 54-degree position.
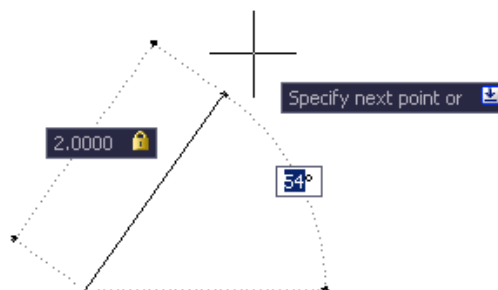
## PASTE VALUES INTO A DYNAMIC PROMPT TOOLTIP

- To use PASTECLIP to paste values from the Windows Clipboard into a dynamic prompt tooltip.

Note: type a letter in the tool tip and then backspace to delete the letter before you paste the entry Otherwise, the entry is pasted into the drawing as text.

## FINAL BIG GUN! TEMPORARY OVERRIDE KEYS (!!!)

- You can temporarily turn on or off drawing aids such as object snaps, tracking, and Ortho, by holding down a key or combination of keys.

- Holding the **SHIFT key** down while pressing the mouse "middle-button" will activate the **3DORBIT** command in "constrained mode" and you can drag the mouse to orbit the drawing without issuing any commands!

- Hold down the **CTRL + ALT key** without being in a command executes the **PRESSPULL** command automatically!

- Holding the **SHIFT Key** down while in 3Dorbit toggles the orbiter between free and constrained orbits.

- Holding down the **CTRL Key** prevents tool palettes from docking

- Holding down the **F12 Key** Toggles Dynamic-input mode (**DYN mode**)

- Hold the **SHIFT Key** down during the **Fillet** or **Chamfer** command automatically forces a **radius setting of 0.** Good for cleaning up corners!

- If you have set running object snaps but you want to turn them off for **one point**, you can hold down **F3 Key to set Osnaps to none**. When you release the **F3 Key** the running object snaps are restored.

- Hold down the **Shift key** to **toggle Ortho mode** while you're placing a line. As soon as you **release** the Shift key, Ortho toggles on/off again! This is much easier than finding the F8 key or clicking the ORTHO button on the active status bar.

- Hold down Shift+A to temporarily **turn off Osnap**. Again, much easier than F3 or the OSNAP button.

- Hold down Shift+E to temporarily turn on the Endpoint Osnap.

- When **DYNMODE** is set to anything but 0, you can use the F12 Key to **turn off all features** temporarily until released.

## MISCELLANEOUS INTERFACE ENHANCEMENTS

- AutoComplete functionality for commands and system variables entered at the command line by using the TAB Key.

- A new Recent Input option (shown as a cascade on the CMCOMMAND context menu) lets you access recently commands and values including points, distances, angles, and strings in a context-sensitive fashion, depending on the type of prompt being displayed that the time of the right-click.

# COORDINATE INPUT REFERENCE SHEET

1.  Absolute Cartesian

    Syntax:     **3,4**

2.  Relative Cartesian

    Syntax:     **@4,2**

3.  Absolute Polar

    Syntax:     **9<75**

4.  Relative Polar

    Syntax:     **@5<60**


5.  Spherical coordinate input (3 dimensional applications)

    Syntax:     **7<35<53**

6.  Cylindrical coordinate input (3 dimensional applications)

    Syntax:     **60<35,30**

7.  Dynamic mode World Coordinate System (WCS) coordinate input

    Syntax:     **#60<35,30**

8.  Dynamic mode World Coordinate System (WCS) coordinate input

    Syntax:     **#4,6**