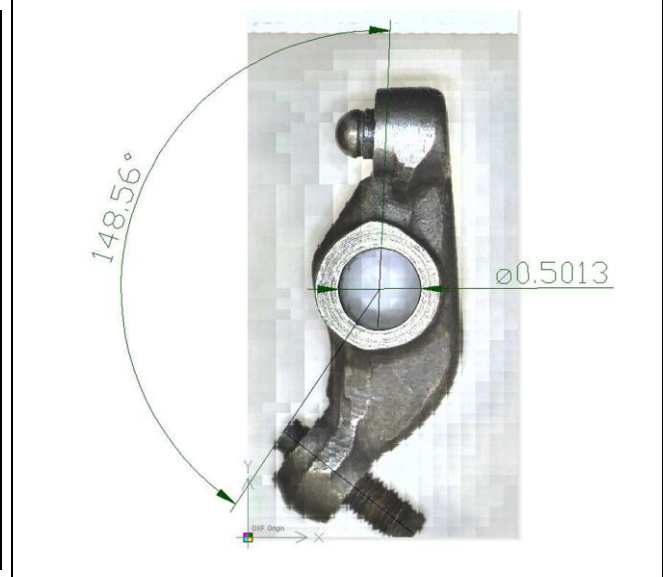
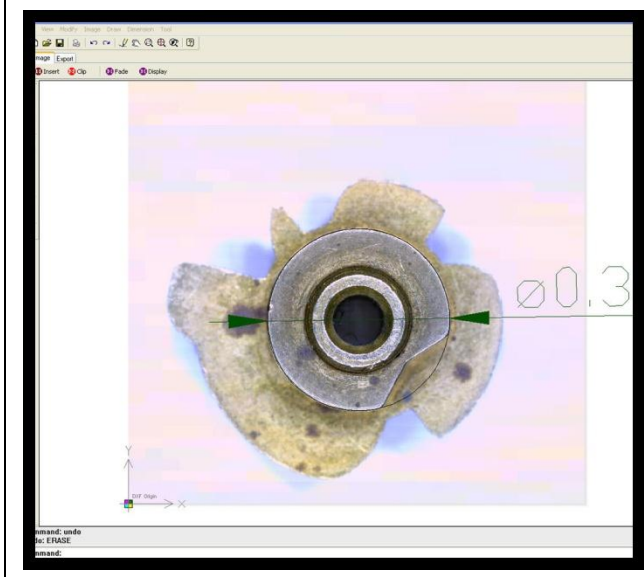
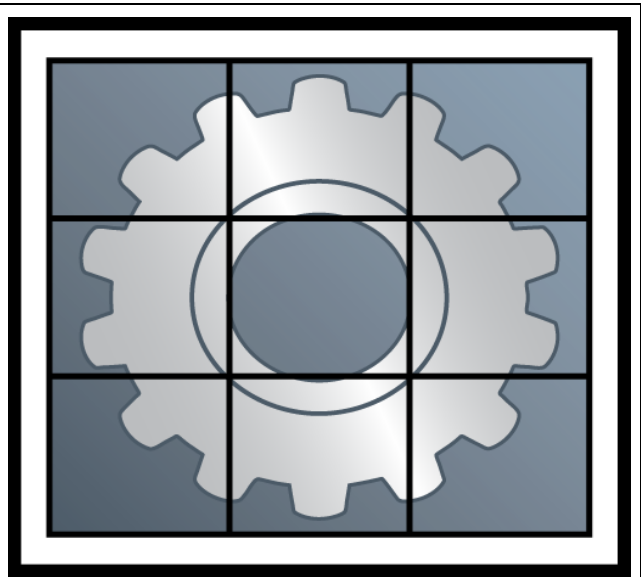


Tormach® CNC Scanner

Tormach® CNC Scan CAD

Operator's Manual



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Questions or comments?
Please email us at:
info@tormach.com



Tormach CNC Scanner Manual
Part Number 31734 – Rev 1

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I Preface

I.1 Introduction

Tormach CNC Scanner is a high resolution 2D dimensional scanning system. CNC Scanner includes 3 modules:

- Optical centering Module
- High Resolution 2D dimensional Scanner
- Tormach Scan CAD – 2D drafting and measuring software.

CNC Scanner is designed to be used with a Tormach PCNC or any other machine that uses Artsoft's Mach3 PC-based motion control software and an R8, Morse Taper #3, or other $\frac{3}{4}$ " spindle collet.

I.2 The CNC Scanner Concept

CNC Scanner is designed as an affordable measuring and reverse engineering tool to expand the utility of the PCNC milling machine. It combines an affordable digital USB microscope with simple software control for analyzing 2D and 2.5D objects.

CNC Scanner can be used for:

- Locating work offsets
- Accurate measurement of small or difficult-to-measure dimensions
- Reverse engineering parts
- Simple qualitative metrology
- Recreating 2D designs

I.3 Performance Expectations

As with any precision tool, accuracy is a combination of equipment resolution, careful setup, and technique. With this understanding, we believe the following accuracies are representative of what can be achieved with the CNC Scanner, although this will vary greatly depending on the details of each situation.

- Optical Centering ± 0.005 " for X and Y positioning
- CNC Scanner: ± 0.001 "

I.4 Requirements

- CNC Mill controlled by Mach3 (such as Tormach® PCNC or similar)
- 1GB Storage Drive (minimum)

2 What's in the Box

2.1 Packing List

Hardware:

- USB Microscope
 - *(Includes Software Driver CD-ROM)*
- TTS Mount
- O-Rings (x2)
- Set Screws (x4)
- Storage Case

Software:

- Tormach CNC Scanner CD-ROM
 - *(Includes install for both CNC Scanner Plug-ins and Tormach CNC Scan CAD)*
- Dongle Key
 - *(if license was purchased)*

2.2 Assembling the Scope

To set up the scope:

1. Pull USB cord through the TTS mount
2. Secure the Scope to the mount with O rings
3. Loosely attach the 4 set screws

2.3 Installing the USB Microscope Software Driver

Insert the Software driver CD-Rom into the computer and following the instructions on screen to install the driver.



Figure 1. Assembling the Scope

3 Installing CNC Scanner/Tormach Scan CAD

3.1 Running the Set Up Wizard

Follow the directions on the screen to install the software. This will install both CNC Scanner and Tormach Scan CAD.

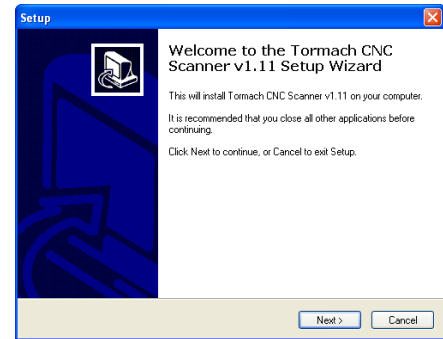


Figure 2. Tormach CNC Scanner Setup Wizard

3.2 Using CNC Scanner for the first time

Depending on your specific configuration of Mach3, you may have configure the CNC Scanner plug-ins prior to first use. To do so, take the following steps:

1. Choose *Config*→*Config Plugins* from the pulldown menu

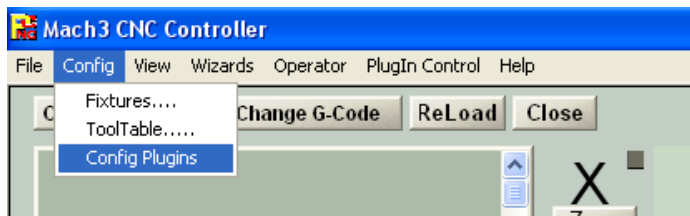


Figure 3. Config Plugins Menu in Mach3

2. Ensure that the plugin *topstools* is enabled (displays Green Check Mark next to the name). If a Red X appears instead, click on the box to enable and switch to a Green Check Mark.

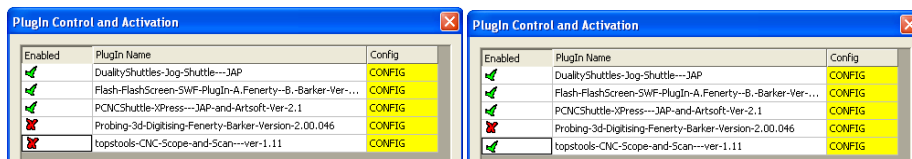


Figure 4. Enabling the topstools Plug-ins for CNC Scanner and Scope

3. The CNC Scope and CNC Scan plug-ins should now appear under the *Plugin Control* menu. You may have to restart Mach3 before the new plug-in settings take effect.

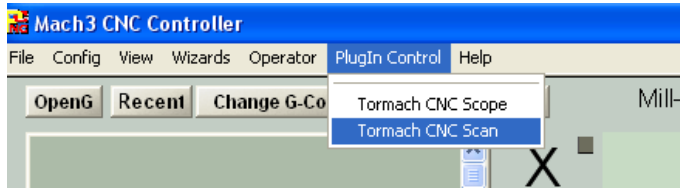


Figure 5. Accessing CNC Scope and CNC Scanner in the Plugin Control Menu

4 CNC SCOPE

4.1 Using CNC Scope

To use CNC Scope choose *Plugin Control*→*Tormach CNC Scope* from the pulldown menu.

4.2 Working with CNC Scope

The CNC Scope window is shown below.

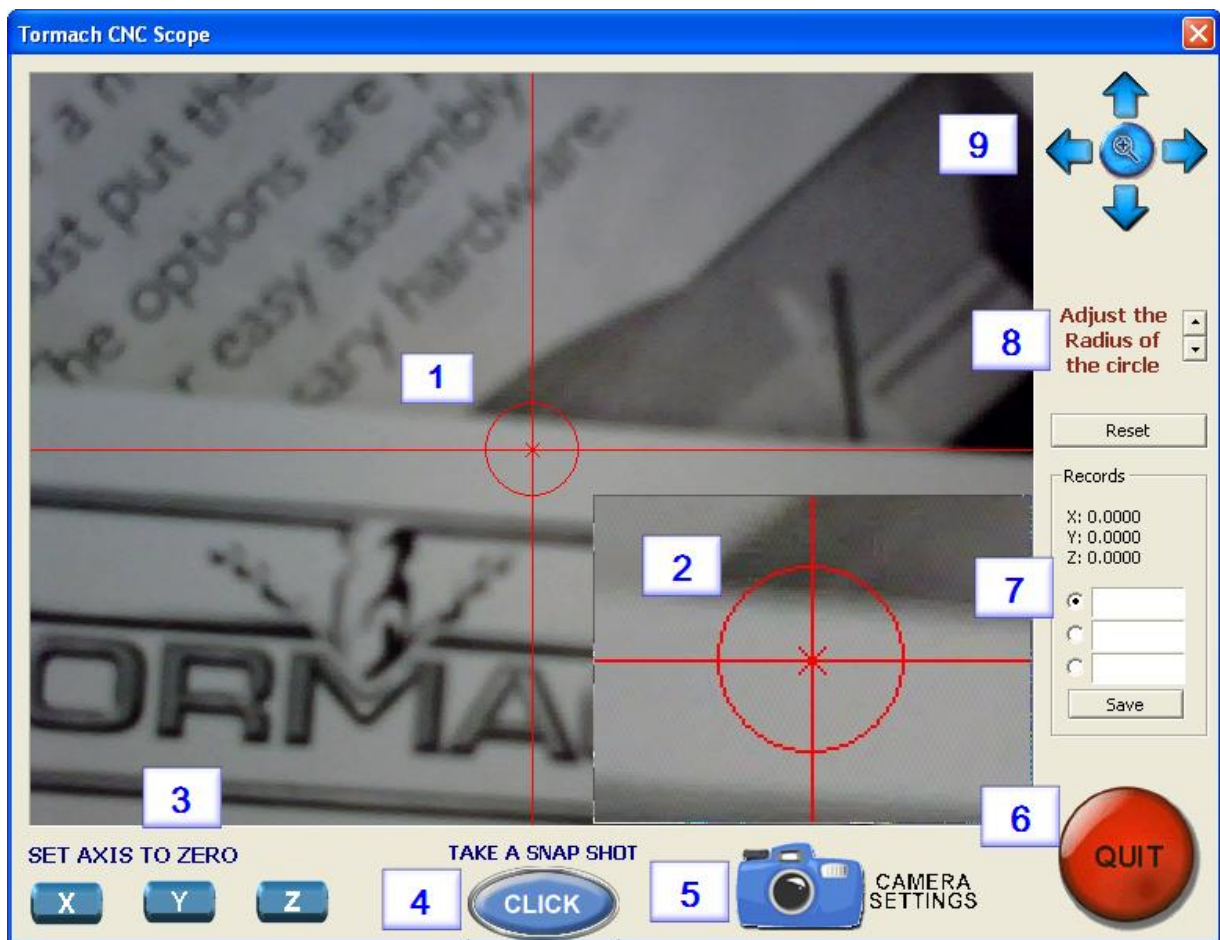


Figure 6. CNC Scope Operators Screen

1. Image Window. Displays the current image of the scope. The center is indicated by the crosshairs
2. Close-up Window. Magnified view of current image near the crosshairs
3. Set Axis to Zero. Changes the corresponding X,Y, or Z DRO to a value of "0". This is helpful for establishing program reference points.
4. Take Snapshot. Saves the current image as a bitmap (.bmp) file
5. Camera Settings. Adjusts camera settings (See section 5.4)
6. Quit. Exits CNC Scope
7. Record Scope Setting. You can save up to three scope settings that can be recalled later
8. Adjust Crosshair Size. Changes the size of the crosshair circle. This is helpful for centering over a hole.
9. Pan Crosshairs. Moves Crosshairs with respect to center of image.

4.3 Centering the Scope

If you intend to use CNC Scope as an optical centering device the following procedure is recommended for the most accurate results.

1. Using the machine, drill a hole in a piece of stock with a good sharp drill bit (a .25 drill bit works well). Then set the X and Y coordinates to zero.
2. Mount the camera into the camera housing, using the bands provided.
3. Insert the 4 set screws into the 4 locations on the camera housing.
4. Using an Allen wrench, tighten or loosen the screws until the camera appears to be centered in the camera housing.
5. Mount the camera housing in the machine spindle.
6. Make sure the machine is at the zero coordinate for X and Y, then start the CNC Scope Plug-in within the Mach 3 control software.
7. Move the Z axis of the machine close to the surface of the material that you drilled a hole into, and focus the camera using the focus wheel.
8. Using the set screws in the side of the housing adjust the screws in or out to place the red crosshairs as close to the center of the hole as you can.
9. Using the CNC Scope software, increase or decrease the diameter of the circle on the crosshairs to closely match the drilled holes diameter.
10. Using your hand slowly rotate the machine spindle while watching the screen. If it appears that the red circle rotates concentrically around the edge of the drilled hole, then the camera is centered; otherwise continue to adjust the screws until the circles are concentric.

To test your centering, move the machine away from the drilled hole. Then, by using the machine jog controls or MDI interface move the crosshairs back over the drilled hole until you think it is over center and check your X and Y position. If it is close to zero, the scope is centered. If it is not very close, go back to step 4.

Please be aware that any further focusing or refocusing of the scope after completing the alignment procedure can have an adverse effect on its performance as an Optical Centering device. For the most accurate results, the centering procedure should be repeated for each instance.¹

¹ CNC Scanner Design Analysis

5 CNC Scanner

5.1 Before You Begin

Take care in the following preparations. The following recommendations will increase your chance for a successful scan

- Place the object that you wish to scan on a flat surface. A flat plate resting on top of a vise works well. Plate glass works well for small objects.
- If possible, place the object on top of a contrasting background. White paper works well.
- Make sure the object will remain still and cannot move while the scan is in progress. In general, this is a problem with lightweight and/or small objects. Larger objects can be held directly in the vise. Double-sided tape is also helpful. Putty can be used as well.
- The scope does not need to be centered for accurate scanning; however, it is helpful if the scope is rotated within the spindle so the image axes are aligned with the machine axes – that is, horizontal movement in the image corresponds to X motion of the machine and vertical movement corresponds to Y motion of the machine.
- Ensure the scope is positioned at a safe distance above the object.
- Make sure that the complete area that you wish to scan resides within the work envelope of the machine. If this is not the case, you will trip a limit switch and the scan will be lost.

5.2 Starting a Scan

To start a scan choose *Tormach CNC Scan* from the PlugIn Control menu.

5.3 Planning a Scan

When you open CNC Scanner, the Scan Setup window is displayed.

5.3.1 Naming the Scan File

Type a name for the scan and choose *Create File*. The raw scan data is saved in the directory indicated as `filename.tsi`.

5.3.2 Determine the Scan Area

Here, you must specify an area to be scanned. Choose a width and depth that defines a rectangular box around the area of interest.

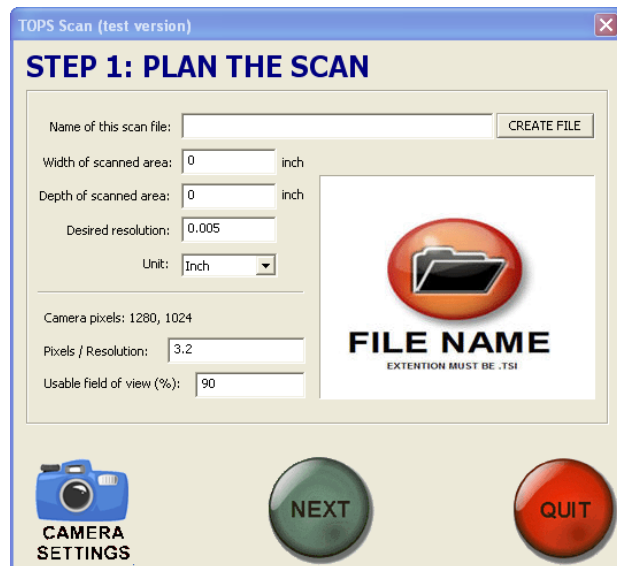


Figure 7. Planning the Scan Menu

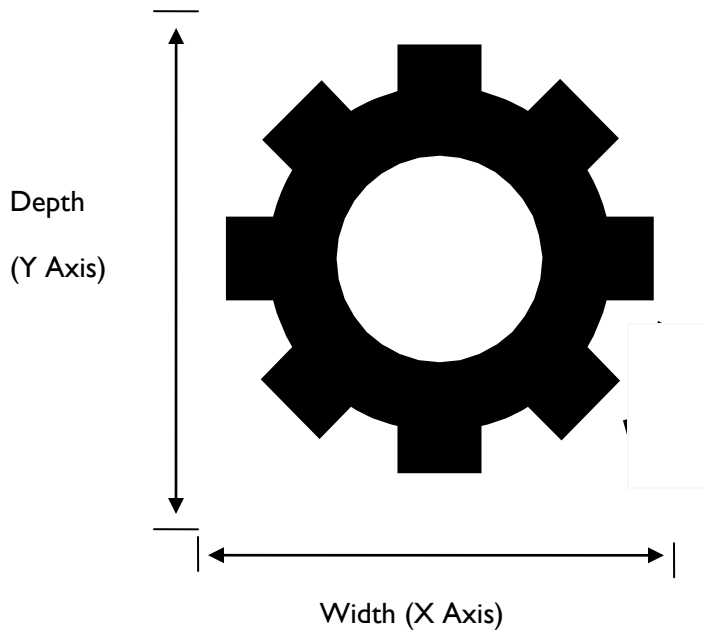


Figure 8. Determining Width and Depth of Scan area

5.3.3 Choose a desired resolution

Desired resolution is analogous to a $\pm$ dimension tolerance in a technical drawing.

Choosing a smaller tolerance will result in a more accurate scan but also results in longer scan times and larger file sizes. For example, you may ± 0.01 " resolution for a precision mechanical connection; however, ± 0.005 " might be okay for recreating a cast surface and ± 0.015 " might be just fine for purely aesthetic recreation.

5.3.4 Choose Units

You can choose either Inches or Millimeters. This information will be embedded in the .tsi image.

5.3.5 Pixels / Resolution

Minimum number of pixels needed to determine the presence of a feature – for example an edge. This is used by CNC Scanner during mosaic assembly. The default value is 3.5.

You can adjust this value. Increasing this value allows you to increase the working distance of the camera; however, this will ultimately be at the expense of the accuracy you can achieve from measurements in the scan.

5.3.6 Usable Field of View

The percentage of each photograph used to assemble the mosaic image. A larger percentage results in faster scan times as the entire image is used for mosaic assembly; if a small percentage is used, the area between photographs overlaps. For example, if a 50% field of view is used, the mosaic will consist of 4x the amount of photographs compared to a 100% field of view; a 10% field of view is 100x.²

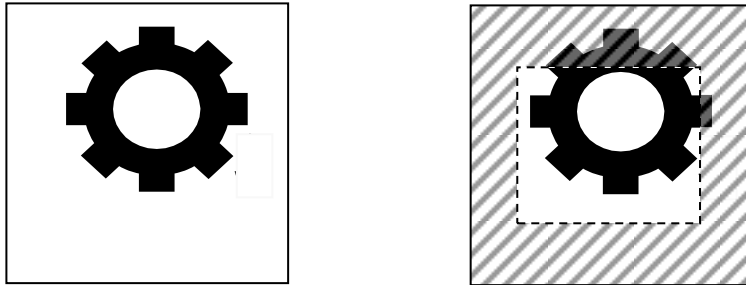


Figure 9. Comparison of 100% FOV vs. 50% FOV

5.4 CNC Scope Camera Settings



Figure 10. Camera Settings Function

5.4.1 Brightness

The overall intensity of reflected light in an image. Lower brightness will be darker and higher brightness will be lighter

5.4.2 Contrast

The amount of gradient between the darkest pixels and lightest pixels in the image. Higher contrast will lighten the lightest pixels and darken the darkest pixels.

5.4.3 Gamma

Signal correction between Scope and Display monitor.

² CNC Scanner Design Analysis

5.4.4 Hue

The distinguishing attribute of color.

This is what makes red different than blue, for example. When moving this slider, think about moving around a color wheel from 180° to -180°.

5.4.5 Saturation

The amount of grey in a color. Lower saturation has more grey, higher saturation has less grey.

5.4.6 Sharpness

The clarity of detail in an image. Higher sharpness means more abrupt transitions between different colors

5.4.7 Snapshot Size

Depends on Hardware. For CNC Scope, 1.3MP

5.4.8 USB Bandwidth

The rate of data transfer between from the scope in Mb/s. Choose the highest rate possible; however, it may be necessary to reduce bandwidth if you notice problems with the function of other USB devices, such as the Tormach Jog Shuttle. This is dependent on the performance of the control computer.

5.4.9 Exposure

Total amount of light collected during image capture. Higher exposure improves images taken in low light, but too much can “washout” features.

5.4.10 White Balance

Adjusts images for the differences in spectral composition of white light originating from different sources (i.e., LEDS, sunlight, florescent tubes, etc.)

5.4.11 Flicker

Number of frames per second. Depends on Scope. For CNC Scope, max is 30Hz, 15Hz at 1.3MP

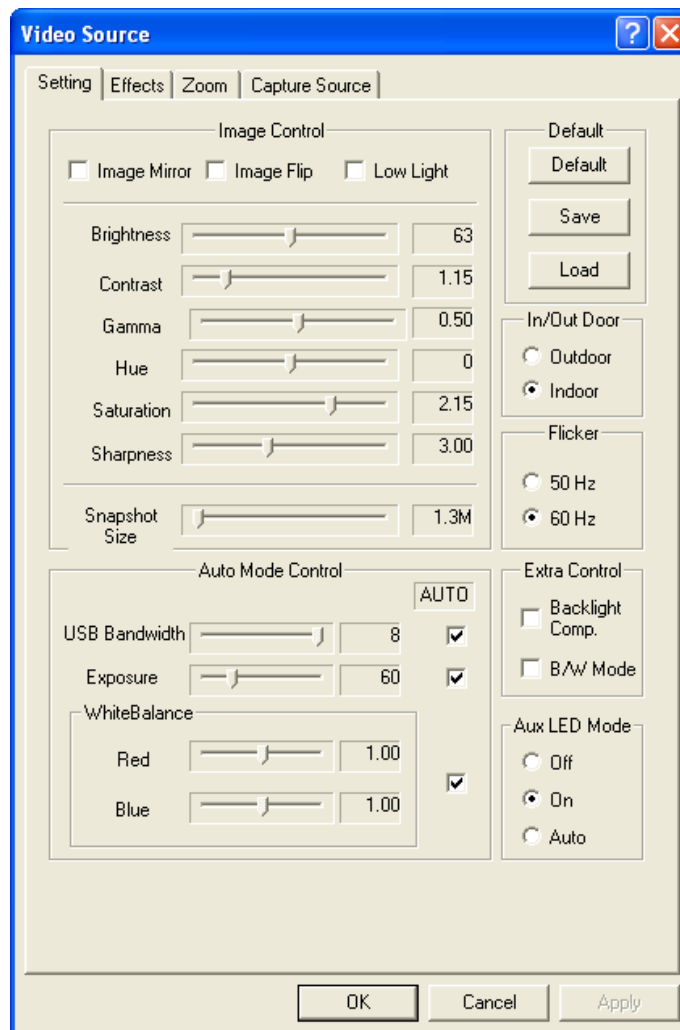


Figure 11. Camera Settings Menu

5.5 Picking a Start Point

The next step is to pick the start point for the scan. This point corresponds to the lower lefthand corner of the image. To do so, simply position the crosshair cursor over the start point of the scan by moving the X and Y axes of the machine. When you are over the start point, click “set start point”.



Figure 12. Establishing the Start Point

CNC Scanner assigns the start point a value of (X0.050, Y0.050). This assures that the area directly around the start point is included in the scan..

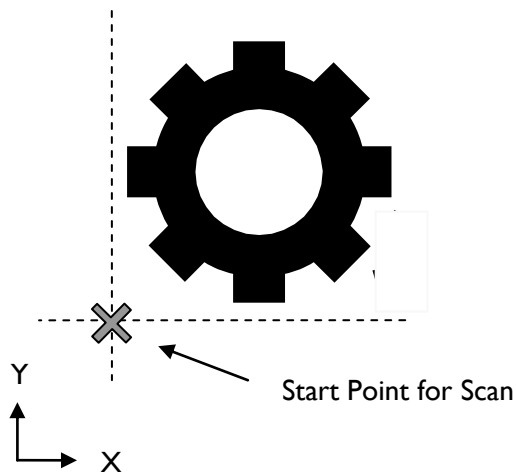


Figure 13. Setting the start point for the Scan

You can also use the arrows on the screen to jog the X and Y axis in discrete step amount specified in the box. To use the previously stored start point, choose “Go To Previous Start Point”.

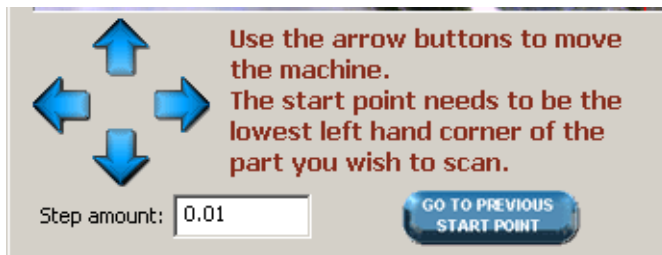


Figure 14. Setting the Start Point

5.6 Establishing Image Scale

After you have selected the Start Point for the image, you must choose a scaling method. There are three ways to assign a scale for a scanned image: Use *Previous Scale*, *Scale by Image*, and *Scale by Motion*.

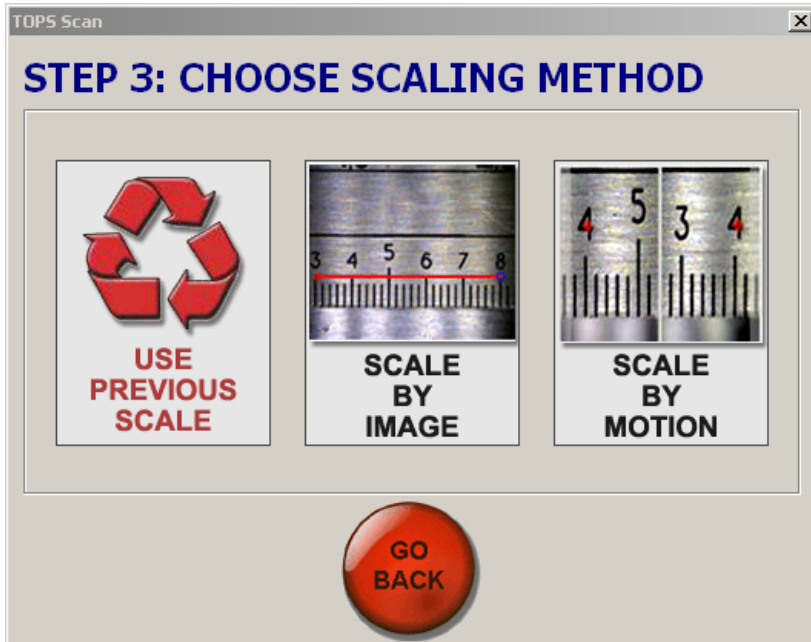


Figure 16. Choosing a Scaling Method

5.6.1 Use Previous Scale

This option reuses the previously defined scale. The scale will only be correct if the focal plane and magnification has not been changed from the previous scan. It can be useful for:

- Repeating a scan
- Comparing nearly identical objects; i.e., a series of coins. This will ensure the scale remains identical in each image.



5.6.2 Scale by Image

You can use a known dimension in the field of view to scale the scan. This dimension could be, for example:

- Ruler
- Drilled or, better yet, reamed hole of known diameter
- Gauge block

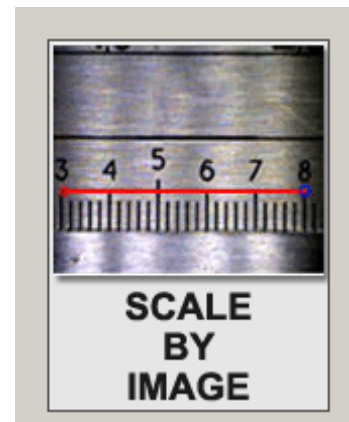


Figure 17. Scale by Image

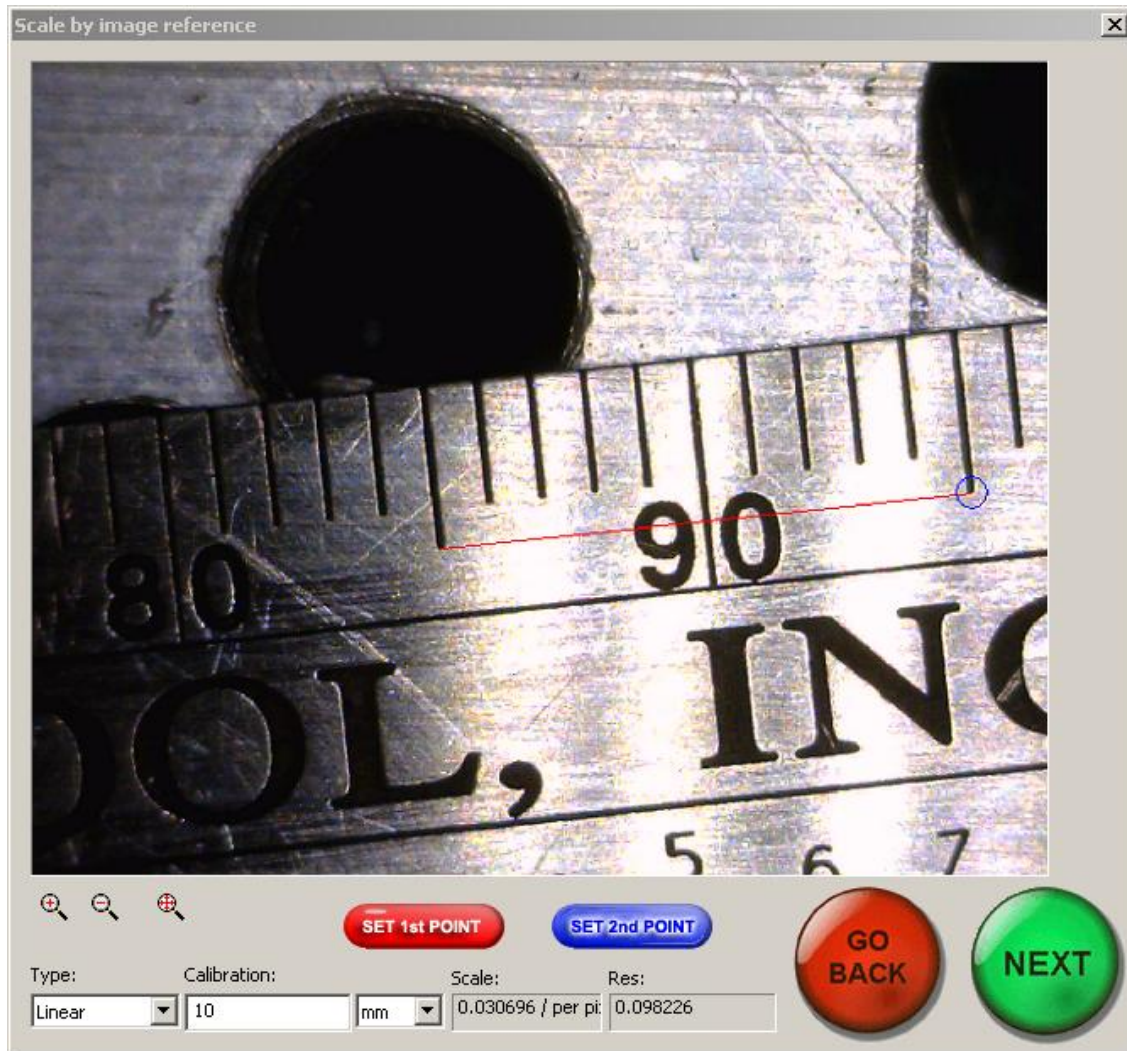


Figure 18. Scale by Image Operator's Screen

5.6.3 Scale by Motion

A unique feature to CNC Scanner is the ability to scale by motion. With this method, scale is established by the distance that the machine moves when a distinct point is repositioned from one end of the field of view to the other end. This point can be:

- Machined corners or other geometric features
- Dust, pores, grains, scratches, corrosion – any recognizable feature



Figure 19. Scale by Motion

Scaling by motion will provide improved scaling accuracy of the scanned image when compared to the traditional scaled by image method. This is because scaling accuracy is determined by the positioning accuracy of the CNC machine. In the case of the Tormach PCNC, this is better than .001”.

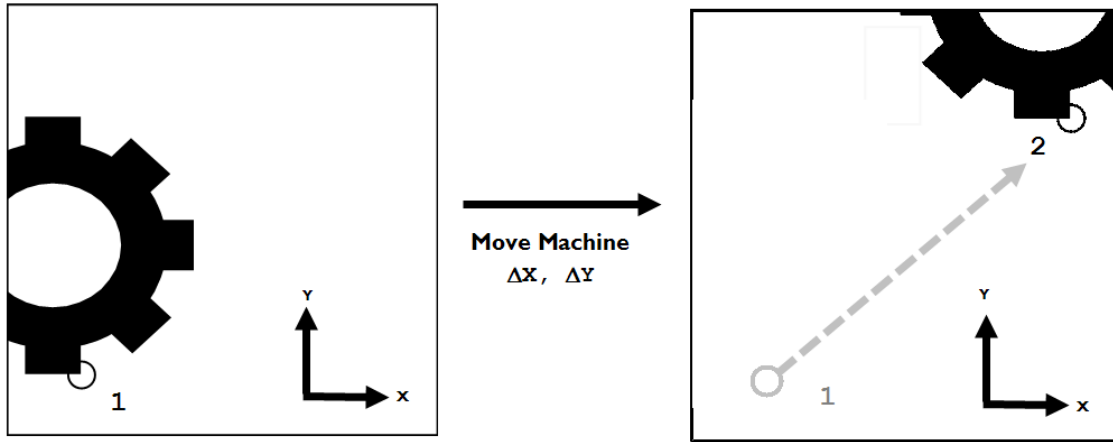


Figure 20. Calibrating for Scale by Motion

Scale by motion establishes scale by calibrating the change in position of a point in the camera's FOV (from 1→ 2) to the actual change in position of the machine ($\Delta X, \Delta Y$).

To scale a scanned image by motion, choose “scale by motion”

A new camera viewport will open. The viewport is further broken down into 3 regions

1. Field of view (FOV). This area represents 100% FOV of the camera
2. Usable Field of View (UFOV) . This is the area that will be used to build the photomosaic. This is specified in the planning phase as a percentage of the FOV (See Section 5.3.6)
3. Snapshot window. To establish scale with the scale by motion technique, CNC Scanner takes two snapshots. The snapshot window represents the area that will be captured in each snapshot. You can reposition the snapshot window within the FOV by pressing and holding the center mouse button and dragging the mouse.

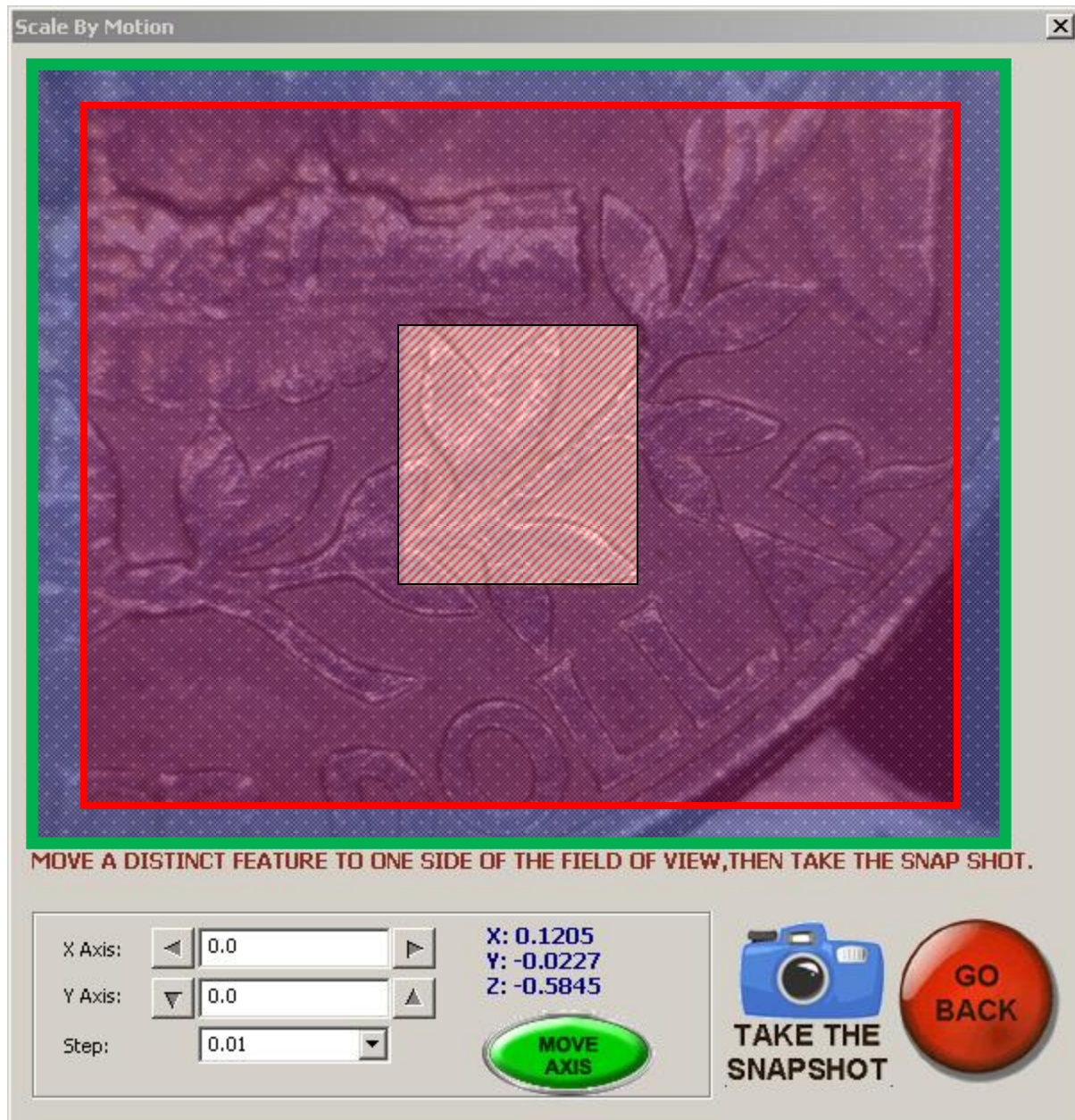


Figure 21. Scale by Motion viewport. 100% FOV is shown in green. UFOV is shown in red. Snapshot Window is shown in center.

When the snapshot window is positioned where you want it, click *Take The Snapshot*.

Now, reposition by jogging the axes of the machine and repeat to for the second snapshot. To achieve best results, position the point at opposite corners of the each viewport.

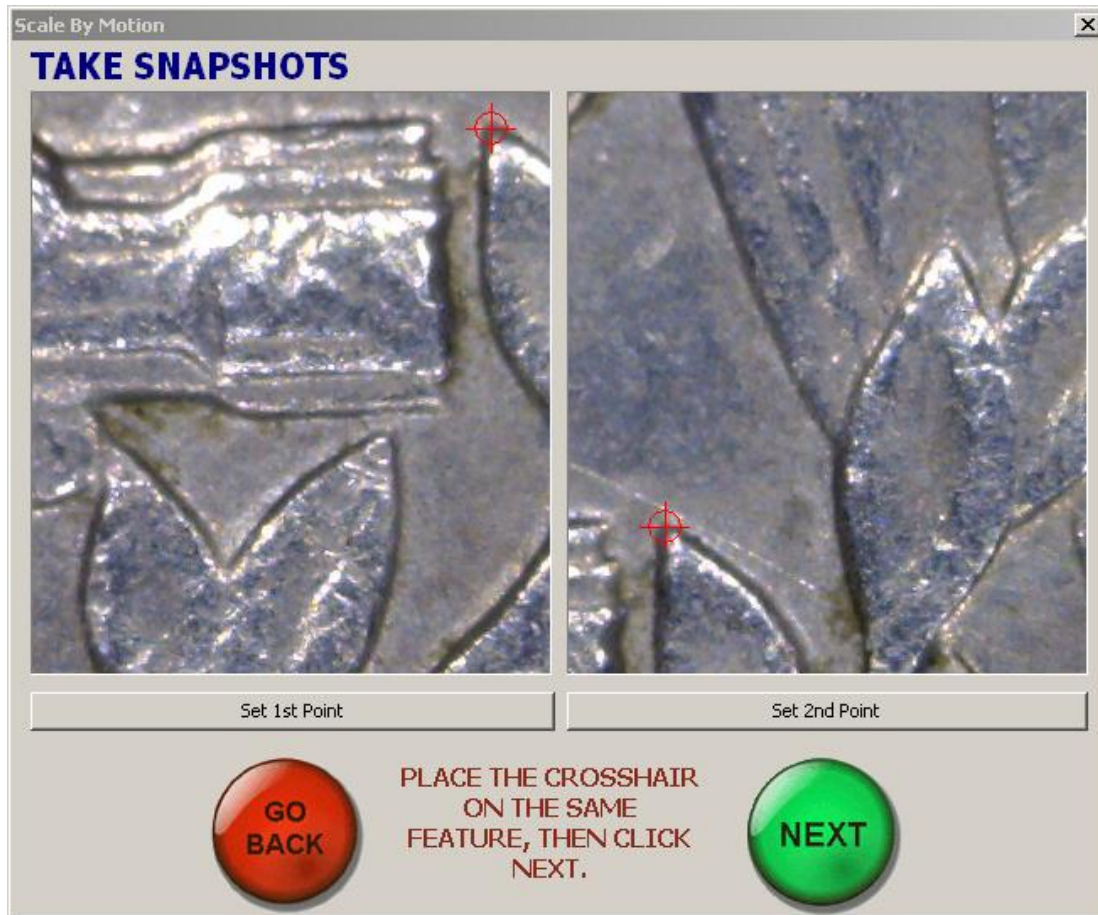


Figure 22. Identifying the same point in two frames of reference for Scale by Motion.

With both snapshots taken, the next step is to position the crosshair cursor on the point of interest in each snapshot viewport. To do so, simply click on the point. When you have successfully selected the same point in both snapshots, choose "Next" to start the scan.

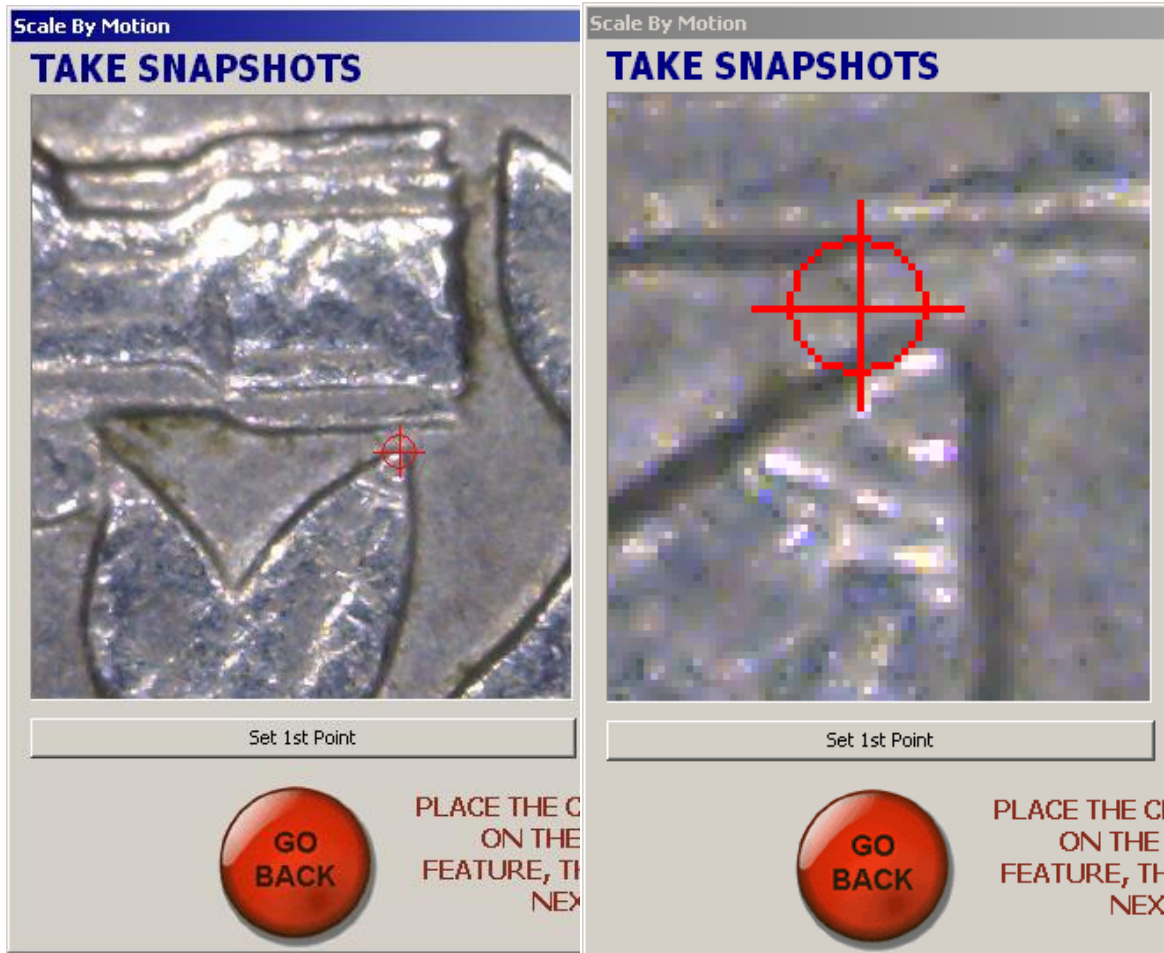


Figure 23. Zooming in to identify points with Scale by Motion

You can also zoom in by using the center wheel of the mouse. This allows you to zoom into the individual pixels of the snapshot to set each point. Press and hold the center mouse button while moving the mouse to pan the zoomed snapshot within the viewport.

5.7 Completing the Scan

Once you have successfully set the image scale, you are ready to begin the scan. Press “Start the scan”. CNC Scanner will begin to collect images for constructing the photomosaic. Depending on the number of pictures needed to build the photomosaic, this process may take some time to complete.

When the scan is complete, choose one of the following options:

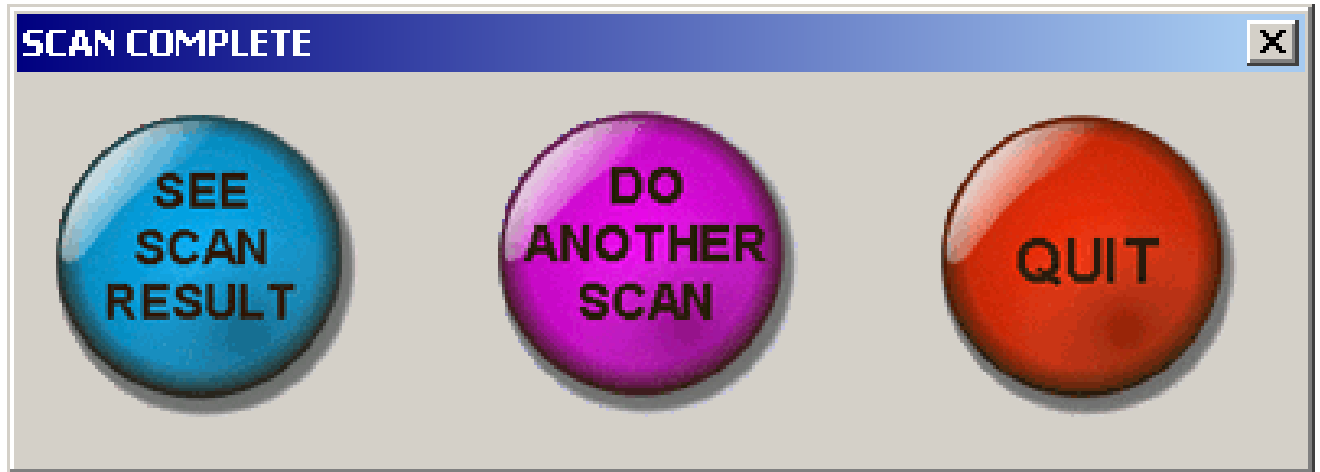


Figure 24. After completing the scan, the following options are available

- *See Scan Result:* This will open the image in Tormach® CNC Scan CAD
- *Do Another Scan:* Returns to *Plan the Scan* screen
- *Quit:* Returns to Mach3 control screen

Regardless of which option you choose, the raw image file is saved as a .tsi in the location chosen during the *Plan the Scan* stage.

6 Getting Started with Tormach® CNC Scan CAD

6.1 Introduction

This section guides one through the installation of CNC Scan CAD software and explains the basics that a user needs to know.

NOTE: This guide assumes that you have used other Windows based programs previously and are familiar with Windows terminology and techniques.

6.2 System requirements

The following operating systems and hardware are recommended to install and run CNC Scan CAD.

6.2.1 Operating system

- Microsoft Embedded Standard : Tormach® Mach OS
- Microsoft Windows XP
- Microsoft Windows Vista

6.2.2 Processor

- Pentium III 800 MHz or higher

6.2.3 RAM

- 512 MB (recommended)

6.2.4 Video

- 1024 x 768 VGA with True Color (minimum)

6.2.5 Hard disk

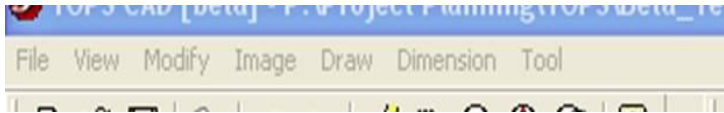
- 1GB minimum for CNC Scan CAD
- 20GB or greater recommended for CNC Scanner

6.2.6 Pointing device

- Mouse, trackball or other devices

6.4.1 Menu bar

Use menus and shortcut menus for access to frequently used commands, settings, and modes.



6.4.2 Toolbars

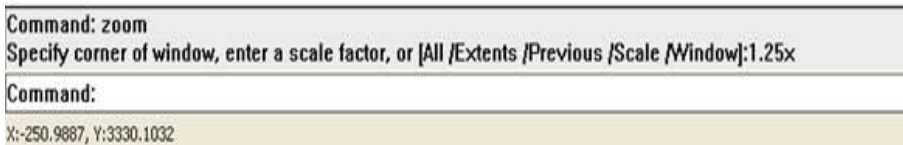
Use buttons on toolbars to start commands, display fly-out toolbars and tooltips.



6.4.3 The Command Window

The command window displays commands, system variables, options, messages, and prompts. The Command Window can be resized as needed.

The command bar is a window where you can type CNC Scan CAD commands and view prompts and other program messages.



6.4.3.1 Enter Commands through the Command Line

Type the full command name on the command line and press ENTER or SPACEBAR, or right-click the pointing device to start the command.

6.4.3.2 Execute Commands

To execute commands, press SPACEBAR or ENTER, or right-click the pointing device when the command names have been entered or in response to prompts.

6.4.3.3 Cancel Commands

To cancel a command in progress, press ESC.

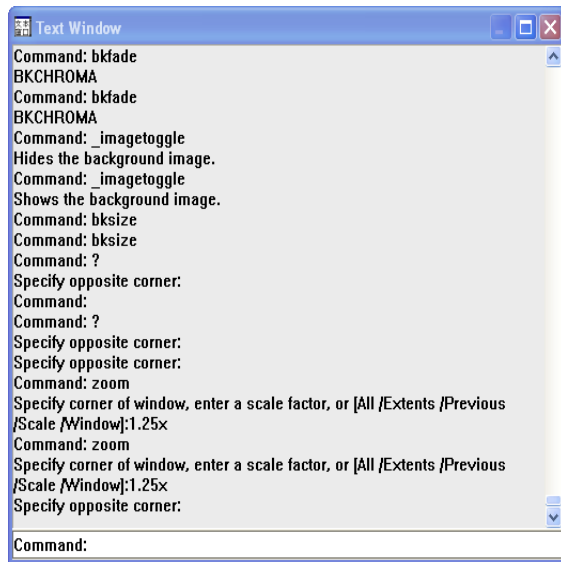
6.4.3.4 To repeat the last command

Do one of the following:

- Press the spacebar.
- Press Enter.
- Click the right mouse button in the drawing.

6.5 Commands History Window

The text window is a window similar to the command window in which the user can enter commands and view prompts and messages. A complete command history for the current CNC Scan CAD session is displayed.

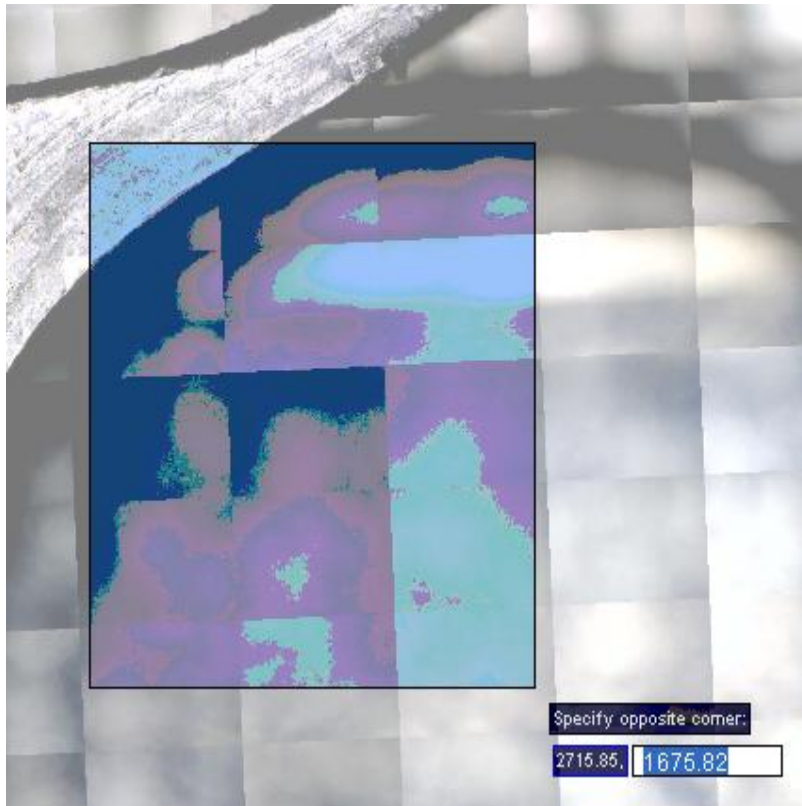


6.5.1 To display or close the Prompt History window

Press F2.

6.6 Dynamic input

The dynamic input box moves with the cross cursor and displays the command interface near the cursor. When dynamic cursor is active, you can toggle between input fields by using TAB.



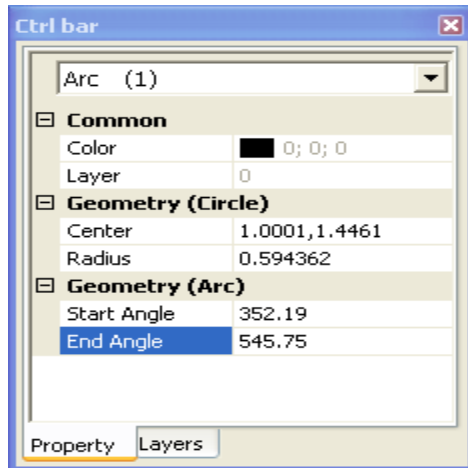
6.7 Layer tab

The layer tab displays the current layers and their properties



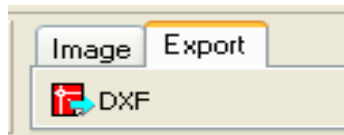
6.8 Property tab

The property tab displays to properties of each object. You can either select the objects from the pulldown menu or select them by choosing them with the cursor. You can change object geometries by highlighting their values and entering new values.



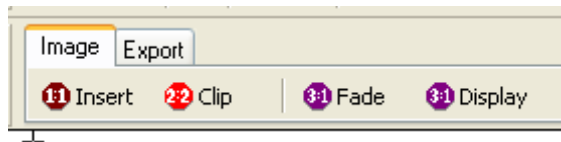
6.9 Export tab

The Export Tab is used to export the current CNC Scan CAD drawing as a .DXF



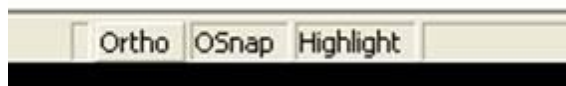
6.10 Image tab

You can use the Image Tab to access the Image functions of CNC Scan CAD. See Chapter I for more details.



6.11 Status bar

You can see drawing mode status by observing which buttons are depressed on the status bar. These modes are active.



6.11.1.1 Ortho

When Ortho mode is active, you are restricted to drawing only horizontal or vertical lines

6.11.1.2 OSnap

When OSnap mode is active, the active snap modes are toggled on/off

6.11.1.3 Highlight

When Highlight mode is active, objects are highlighted when the cursor is moved over the object.

7 Working with Drawing Objects

7.1 Selecting Objects

You have a wide range of options when selecting objects for editing.

You can create a selection set that consists of one or more objects before modifying them.

Use any of the following methods to create a selection set:

- Choose a command or tool first, and then select objects.
- Select objects first, and then choose a command or tool (most objects).
- Select objects by pointing, and then use grips to modify them.

7.1.1 Select Objects Individually

You may select one or more objects individually.

You can select one or more objects under the prompt of select entities on the command line. The selected highlights on the screen and can be removed from the selection set as needed.

7.1.2 Select objects with pickbox

You may select objects using the mouse or other pointing device. The mouse or other pointing device is represented by a square pickbox on the drawing area. When selecting an object, the cursor square pickbox should overlap the object. The size of pickbox can be changed on the 'Options' dialog box.

7.1.3 Remove Selection from Objects

If an entity is no longer needed in a selection set, one can deselect it to remove it from the selection set.

7.1.3.1 To remove an entity from the selection set

Press Shift, and then select the entity again (must be running a command).

NOTE Pressing Shift while selecting entities, one may use a crossing window (?) to remove all entities from the specified selection set.

7.1.3.2 To remove all entities from the selection set

Press Escape.

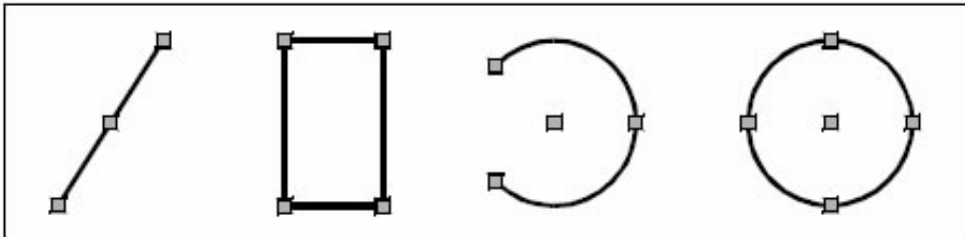
7.1.4 Prevent Objects from Being Selected

You may prevent objects from being selected by using a locking layer. The locking layer means that you cannot do any operations except for editing objects on this layer.

7.1.5 Select Objects First

You may select objects first and then choose how to modify them. As one selects each object, it is highlighted with small squares called grips, which appear at strategic points on the object.

The locations of the grips depend on the type of object selected. For example, grips appear at the endpoints and midpoint of a line, at the quadrant points and the center point of a circle, and at the endpoints, midpoint, and center of an arc.



Examples of grip locations for different objects.

After one or more objects are selected, you may choose an object modification command, such as Copy or Move, from the Modify menu or toolbar.

7.1.6 Select a Command First

After typing a command, the cursor becomes a pickbox and prompts the user to select objects. There are several methods to select objects:

- Select objects using the mouse or other pointing device.
- Click on the drawing area and drag the cursor to define a square selection area.
- Enter ? to display a list of all selection methods
- Use multiple methods together. For example, to select most of the objects on the drawing, you may select all objects and then remove the undesired ones.

When objects are selected, these objects are added to the selection set. If at least one object is selected, the removal of one of them is available from the selection set. To finish adding objects to the selection set, simply press Enter. Most object-modification commands act on the entire selection set.

7.1.7 Object selection methods

When you choose a command that requires selecting an object (such as deleting or changing object properties, for example), you may use any of the following selection methods by choosing them in the command line prompt or entering them in the command bar:

- **Select all entities (ALL)** Selects all entities in the current drawing.
- **Add to set (A)** Adds one or more entities to the selection set.
- **Remove from set (R)** Removes one or more entities from the selection set.
- **Previous selection (P)** Selects entities included in the previous selection set.
- **Window (W)** Selects entities contained entirely within a rectangular selection window.
- **Crossing window (C)** Selects entities contained within or crossing the boundary of a rectangular selection window.
- **Window polygon (WP)** Selects entities contained entirely within a polygon selection window.
- **Crossing polygon (CP)** Selects entities contained within or crossing the boundary of a polygon selection window.
- **Window circle (WC)** Selects entities contained entirely within a circular selection window.
- **Crossing circle (CC)** Selects entities contained within or crossing the boundary of a circular selection window.
- **Box (B)** Selects all entities completely inside a box.
- **Fence (F)** Selects all entities crossing a selection box.
- **Remove(R)** Remove specified entities from the current selection set.
- **Undo (U)** Cancel the last action.
- **?** Displays all selection methods, from which, you may specify the desired one.

7.2 Select Multiple Objects

You can select multiple objects at one time using a rectangular area or fence.

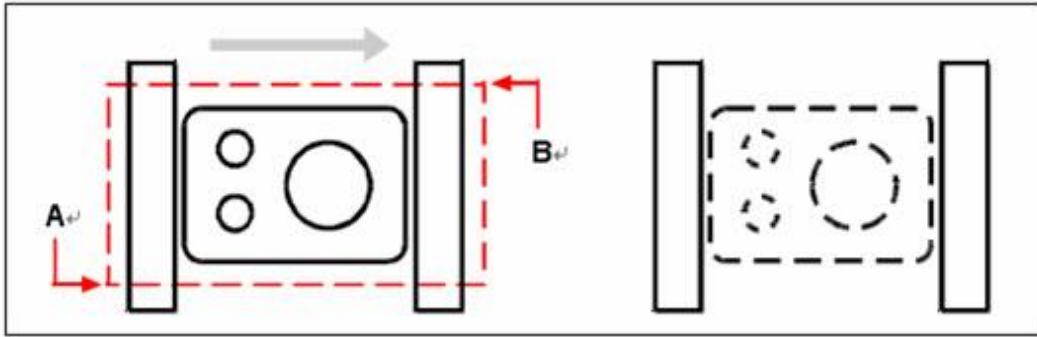
7.2.1 Specify a Rectangular Selection Area

You may select all objects inside the rectangular selection area. When specifying the first corner, there are two methods to define a rectangular selection window:

- Drag the cursor from left to right to create an enclosed window selection. Only the objects entirely contained in the rectangular selection area are selected, or
- Drag the cursor from right to left to create a crossing selection. The objects contained or passing through the rectangular selection area are selected.

7.2.1.1 To create a Window

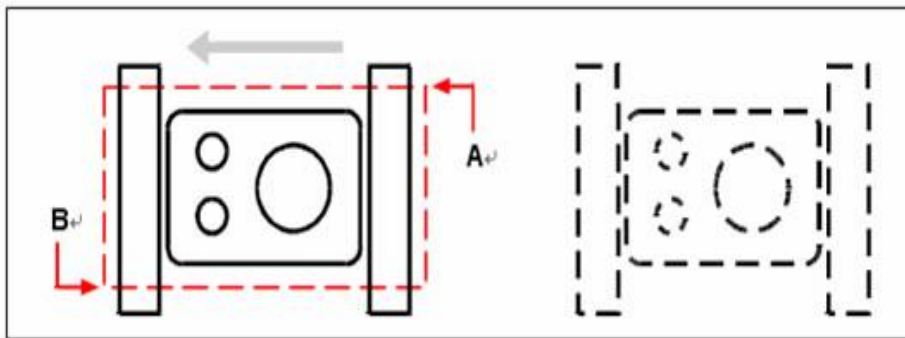
1. Click to select a point in the drawing.
2. Click to the right of the first point to select a second point in the drawing.



Window by selecting the first (A) and second (B) points.

7.2.1.2 To create a Crossing Window

1. Click to select a point in the drawing.
2. Click to the left of the first point to select a second point in the drawing.

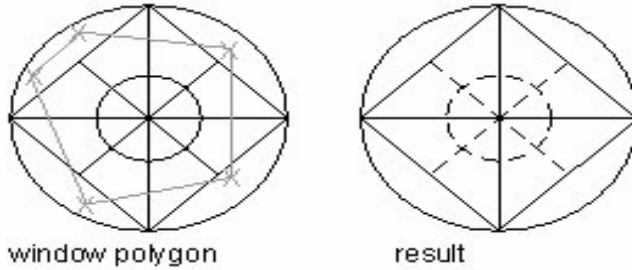


Crossing Window by selecting the first (A) and second (B) points

In addition to a rectangular window, you may define a selection window using other shapes such as a polygon, circle, or fence (a multi-segmented line that selects objects it crosses).

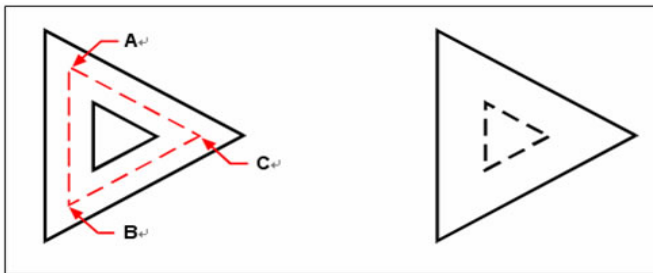
7.2.2 Specify an Irregularly Shaped Selection Area

You may select entities with an irregularly shaped area defined by the specified points. Under the prompt of SELECT command, you may select entities entirely enclosed by the selection area with the WPolygon, or the entities enclosed or crossed by the selection area with the CPolygon.



7.2.2.1 To select objects using Window Polygon

1. Activate an object modification command.
2. On the command line, type wpolygon.
3. Specify the vertices of the polygon.
4. To complete the selection polygon, press Enter.



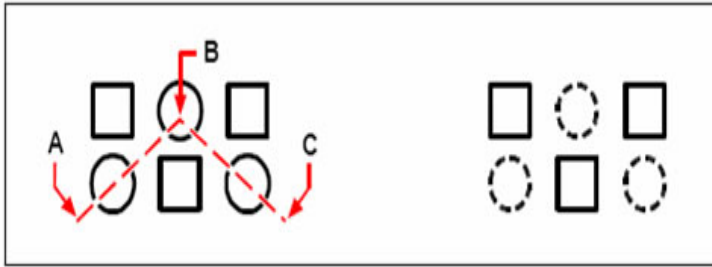
Window Polygon by specifying the vertices of the polygon (A, B, and C)

7.2.3 Specify a Selection Fence

It may be much easier to select entities in a complex drawing by using a selection fence. A selection fence appears like a polyline and chooses only the entities it passes through.

7.2.3.1 To select objects using Fence

1. Activate an object modification command.
2. On the command line, type f.
3. Specify the endpoints of the Fence segments.
4. To complete the Fence, press Enter.



Fence by specifying the endpoints of the fence segments (A, B, and C)

7.2.4 Use Other Selection Options

Enter by typing ? This displays all selection options, from which you may choose a proper one.

7.2.5 Remove Selection from Multiple Objects

Under the prompt of Select entities, you may type R and choose a selection to remove objects from the selection set. You also can enter A to add objects to the selection set.

By holding down SHIFT and selecting objects again, the selected objects will be removed from the selection set.

8 File Commands

8.1 Create a New drawing

Command line NEW

Click on the menu icon New to create a new drawing.

8.2 Open an Existing Drawing

Command line OPEN

1. Choose *File > Open*.
2. Choose the folder containing the desired file.
3. Choose the desired drawing and then click *Open*, or
4. Double click the drawing you want to open.

Choose the type you want to open. CNC Scan CAD can open .TSP files.

8.3 Save a Drawing

Command line QSAVE

You may save drawing files for later use just as one does with other Microsoft Windows applications. This command will overwrite the existing drawing.

8.3.1 To save a drawing

1. Choose *File > Save*.

TIP: When a drawing is saved for the first time, the program displays the *Save Drawing As* dialog box so that the user can choose a directory and type a name for the drawing.

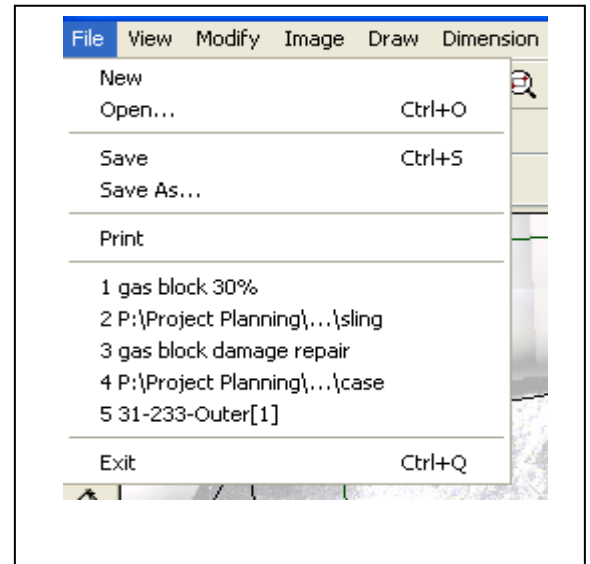
8.4 Save As

Command line SAVE

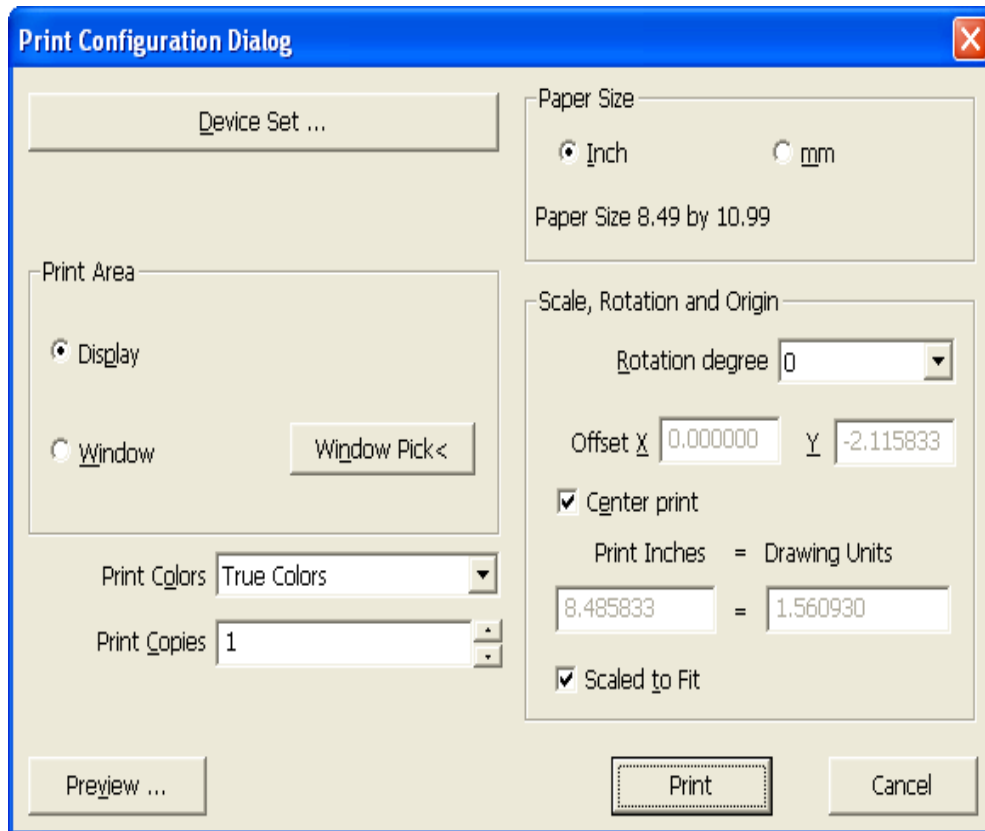
Use the *Save As* function to save the file with another name.

8.5 Print

Command line PRINT



Use the Print command to print the screen or change print settings.



8.6 Exit

Command line QUIT

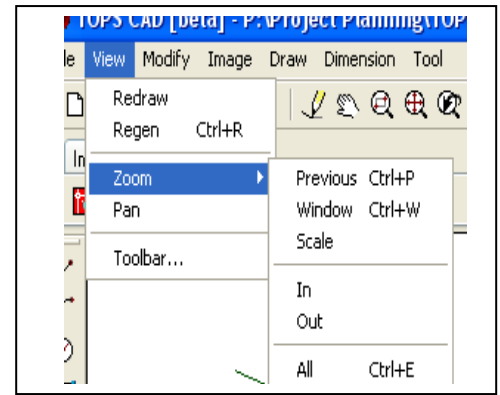
Use the Exit function to close CNC Scan CAD.

9 View Tools

9.1 Redrawing and regenerating a drawing

Command line: REDRAW

As you work on a drawing, visual elements may remain after the completion of a command. You may remove these elements by refreshing, or redrawing, the display.



9.1.1 To redraw (refresh) the current window display

- I. Choose *View > Redraw*.

9.2 Regenerating a drawing

Command line: REGEN

Information about drawing entities is stored in a database as floating point values, ensuring a high level of precision. Sometimes a drawing must be recalculated, or regenerated, from the floating point database to convert those values to the appropriate screen coordinates. This occurs automatically. You may also manually initiate a regeneration. When the drawing is regenerated, it is also redrawn.

9.2.1 To regenerate the current window

- I. Choose *View > Regen*

If more than one window is displayed, type `regenall` to regenerate all the windows.

9.3 Magnify a View (Zoom)

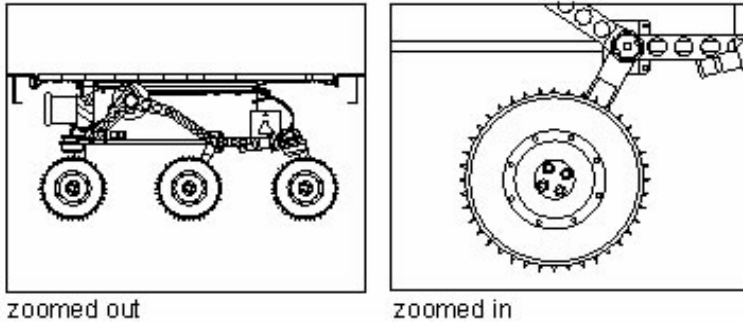
Command line ZOOM

You may use ZOOM command to zoom in or zoom out the drawing view. ZOOM does not change the absolute size of objects in the drawing; it changes only the magnification of the view.

With the zoom feature, you may change the magnification of the drawing at any time. The cursor changes to a magnifying glass when a zoom tool is active. Zoom out to reduce the magnification so you can see more of the drawing, or zoom in to increase the magnification so you may see a portion of the drawing in greater detail.

9.3.1 Zooming in and out

One of the easier ways to change the magnification of the drawing is to zoom in or out by a preset increment. On the Zoom toolbar, the Zoom In tool doubles the current magnification of the drawing. The Zoom Out tool reduces the magnification of the drawing by half. The portion of the drawing locating at the center of the current viewport remains centered on the screen as you zoom in and out.



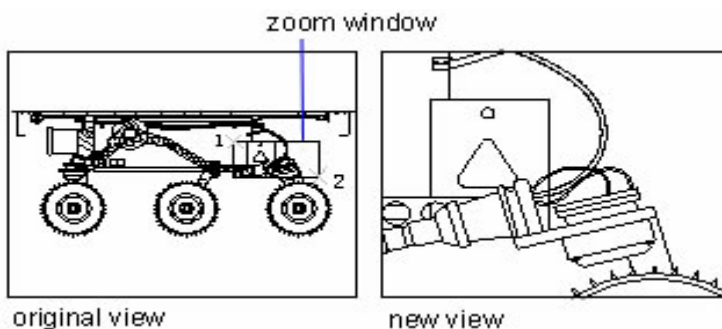
9.3.2 Zooming methods

To zoom, you may use any of the following methods:

- Create a window to define the portion of the drawing to zoom
- Use a mouse with a wheel and rotate the wheel to zoom in and out.
- Zoom to Magnify a Specified Rectangular Area. You may specify a rectangular area defined by two corners to display the objects within that area as large as desired.

9.3.3 To zoom in to an area using a window

1. Choose *View > Zoom > Window*.
2. Select one corner of the window around the area you want to magnify.
3. Specify the opposite corner of the window around the area you want to magnify.



To specify a rectangular window around the area you want to magnify, select first one corner (A), and then select the opposite corner (B)

9.3.4 To zoom using a mouse with a wheel

Rotate the wheel to zoom in or out. Each rotation of the wheel away from the user zooms out .8 times; each rotation toward the user zooms in 1.25 times.

9.3.5 Displaying the previous view of a drawing

After zooming in or panning to view a portion of the drawing in greater detail, you may want to zoom back out to see the entire drawing. On the Zoom menu, the *Zoom Previous* tool allows you to restore the previous view. Selecting this tool repeatedly steps back through up to 25 successive zoomed or panned views.

9.3.6 Zooming to a specific scale

You may increase or decrease the magnification of the view by a precise scale factor measured relative to the overall size of the drawing or in relation to the current display. When you change the magnification factor, the portion of the drawing located at the center of the current viewport remains centered on the screen.

To change the magnification of the view relative to the overall size of the drawing, type a number representing the magnification scale factor. For example, if the user types a scale factor of 2, the drawing appears at twice its original size. If the user types a magnification factor of .5, the drawing appears at half its original size.

You may also change the magnification of the drawing relative to its current magnification by adding an x after the magnification scale factor. For example, if the user types a scale factor of 2x, the drawing changes to twice its current size. If user types a magnification factor of .5x, the drawing changes to half its current size.

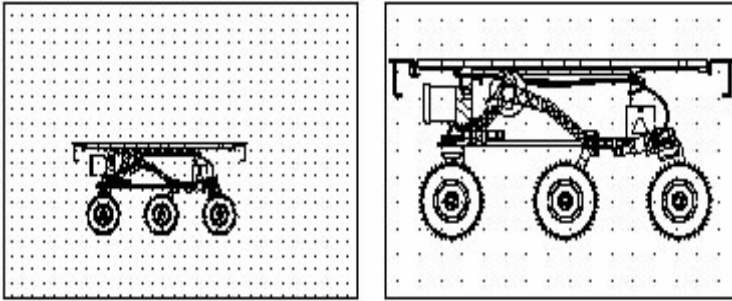
9.3.7 To zoom to a specific scale relative to the current display

1. Do one of the following:
 - On the Zoom toolbar, click the Zoom Scale, or
 - Type zoom and then press Enter.
2. Type the scale factor, followed by an x (such as 2x).
3. Press Enter.

9.3.8 Displaying the entire drawing

ZOOM Extents displays a view that includes all of the objects in the drawing as large as possible. The view can display the objects on layers that are turned off but cannot display the objects on frozen layers.

ZOOM All displays all of objects within either the user-defined limits or the drawing extents, whichever view is larger.



You may use the Zoom All tool on the zoom toolbar to display an entire drawing.

9.4 Pan a View

Command line: PAN

Use the PAN command to shift the location of the view. PAN does not change the location or magnification of objects on the drawing; it changes only the view.

9.4.1 Using the Pan command

You may move the drawing in any direction using the pan command. Panning shifts or slides the view of the drawing horizontally or vertically. The magnification of the drawing remains the same, as does its orientation in space. Only the portion of the drawing displayed is changed.

To pan, you may use any of the following methods:

- For precise panning, specify two points defining the magnitude and direction of the pan. The first point, or the base point, indicates the starting point of the pan. The second point indicates the amount of pan displacement relative to the first point.
- If you have a mouse with a wheel, press and hold the wheel, and then move the mouse.

9.4.1.1 To pan

1. Choose *View > Zoom > Pan*.
2. Specify two points defining the magnitude and direction of the pan.

9.4.1.2 To pan using a mouse with a wheel

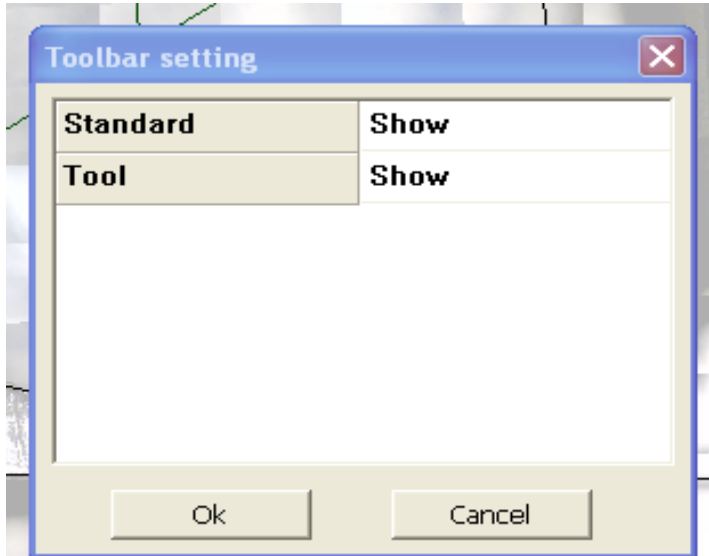
Press and hold the wheel, and then move the mouse in the direction in which one wants to pan.

(The MBUTTONPAN system variable controls this feature.)

9.5 Toolbar

Command line: SHOWTOOLBAR

This command allows you to toggle the toolbar displays on/off. Choose the either show or hide each tool bar by clicking on the right hand column.



10 Modify Tools

You may easily modify the size, shape, and location of objects. Either enter a command first and then select the objects to modify, or select the objects first and then enter a command to modify them.

10.1 Undo

Command line UNDO

To restore the deleted objects, use the *UNDO* command.

10.2 Redo

Command line REDO

To reverse the *UNDO* command, use the *REDO* command

10.3 Delete/Erase Objects



There are several methods for removing objects from the drawing.

Removes objects using one of the following methods:

- Delete objects with *ERASE* command.
- Press *DELETE* to remove the selected objects.

You may use the *ERASE* command to delete objects, regardless of which of the methods one uses to select objects.

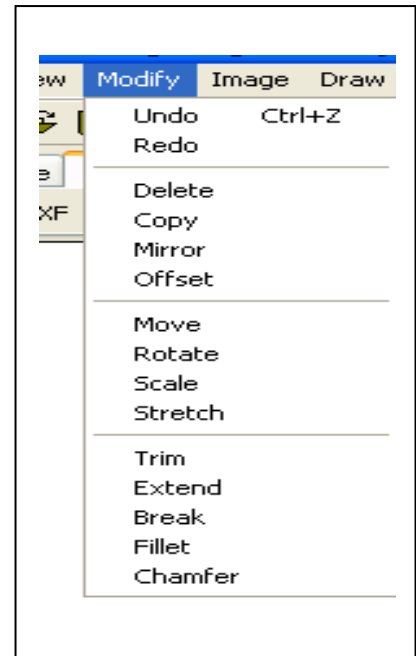
10.4 Copy Objects

Command line COPY

Create exact duplicates of objects at a specified distance from the original.

You may copy one or more entities, making one copy or multiple copies within the current drawing. You may also copy entities between drawings.

You may duplicate objects within the current drawing. The default method is to create a selection set and then specify a base point, and a displacement point, for the copy.





Object to copy (A), base point (B), and displacement point (C)

While specifying two points, you can enter two coordinates to copy objects with the relative distance. The location of copies is determined by the relative coordinate. You do not have to enter coordinate with the @ as the relative coordinate is assumed.

10.4.1 Create only one copy of object one time

1. Choose *Modify > Copy*.
2. Select the objects, and then press Enter.
3. Specify the base point.
4. Specify the displacement point.

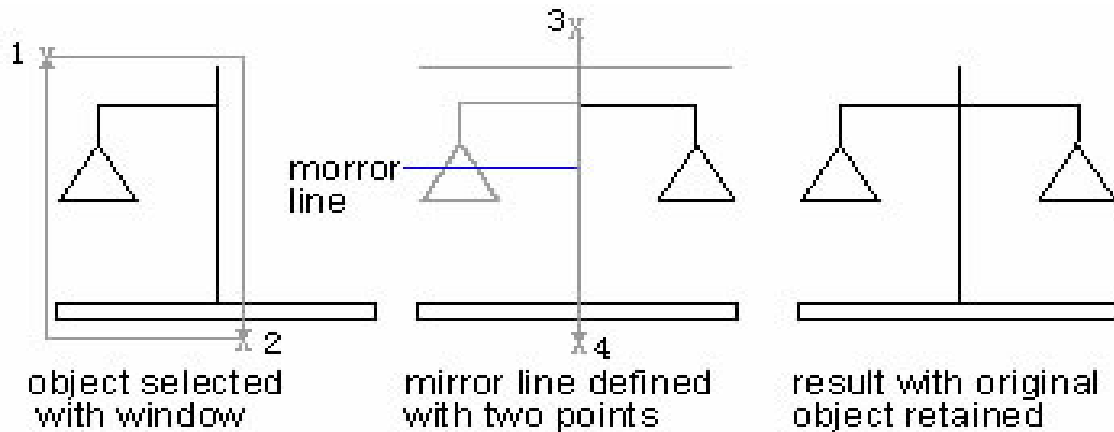
10.5 Mirror Objects

Command line MIRROR

Flip objects about a specified axis to create a symmetrical mirror image.

You may specify a baseline to create a mirror image using MIRROR command. The mirror image is symmetrical with the original one. So if one wants to create a symmetrical object, one only need to draw half of it, and then create the full object using the MIRROR command.

When creating mirror objects using MIRROR command, you should specify a mirror line around which the mirror object is created.



10.5.1 To mirror objects

1. Choose *Modify > Mirror*.
2. Select the object, and then press Enter.
3. Specify the first point of the mirror line.
4. Specify the second point of the mirror line.
5. On the command line, if one wants to retain the original objects, type N; if one wants to delete the original objects, type Y.

10.6 Offset an Object

Command line OFFSET

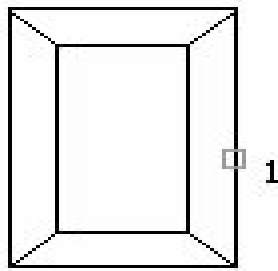
Offsetting an object creates a new object whose shape parallels the shape of a selected object.

You may use the offset feature to copy selected objects and align the offset to the original objects at a specified distance. You may make offset objects using arcs, circles, lines.

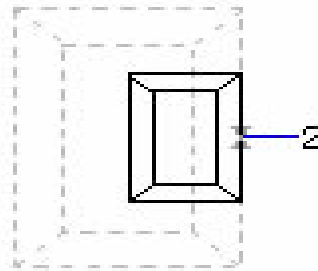
Making offset copies of curved objects creates larger or smaller curves, depending on which side of the original object the user places the copy. For example, placing a offset copy of a circle outside the circle creates a larger concentric circle; positioning the copy inside the circle creates a smaller concentric circle.

10.6.1 To make a offset copy by specifying the distance

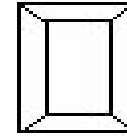
1. Choose *Modify > offset*.
2. Specify the distance by selecting two points or by entering a distance.
3. Select the object to copy.
4. Specify on which side of the object to place the parallel copy.
5. Select another object to copy, or press Enter to complete the command.



object selected



object scaled by
factor of 0.5



result

This image shows how to move an object. Perform MOVE command and then select object (1) to move; specify a base point (2) and Displacement point (3). The object is moved from point 2 to the point 3.

10.7 Move Objects



Command line MOVE

You may move objects to a new place without changing the objects.

You may move objects to a new place without changing the objects. You may type a coordinate value for the new position in the dynamic input box.

In addition to specifying base point or displacement point on the drawing, You may also type coordinate value on the command line. The Vector option can also be used to define a new position.

10.7.1 To move entities

1. Choose Modify > Move.
2. Select the objects, and then press Enter.
3. Specify the base point.
4. Specify the displacement point.



To move an object, select it (A), and then specify the base point (B) and the displacement point (C)

You may also move objects using grips. To move an object by dragging its grips, select the object to highlight its grips, and then click the desired grip and drag it to a new position. The grip one selects depends on the type of object the user is going to move. For example, to move a line object, select the midpoint grip. To move a curved object, such as an arc, circle, or ellipse, select the center point grip. Not all objects can be moved using grips.

10.7.2 To move an object using grips

1. Select the object.
2. Click a grip to select it.
3. Drag the object to where it needs to be relocated.
4. Click to release.

10.8 Rotate Objects

Command line `ROTATE`

Rotate objects around a specified point.

You may rotate the selected objects around a specified point.

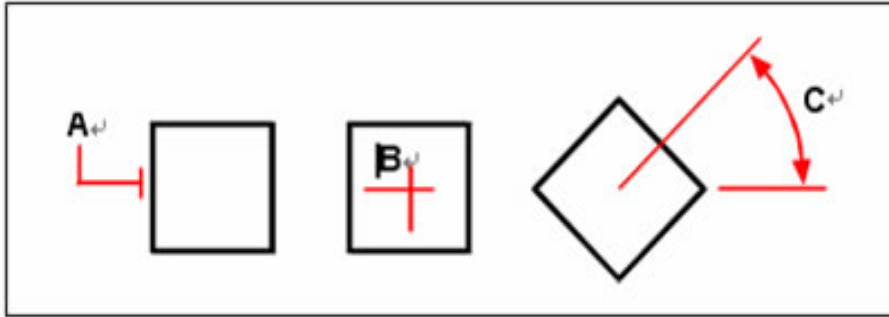
You may specify the rotated angle by specifying a point on the drawing or entering the angle value directly. The direction that objects rotate depends on whether the value the user has entered is positive or negative.

10.8.1 Rotate an Object by a Specified Angle

You may rotate objects by specifying a base point or an angle.

10.8.2 To rotate objects

1. Choose Modify > Rotate.
2. Select the objects, and then press Enter.
3. Specify base point.
4. Specify the rotation angle.



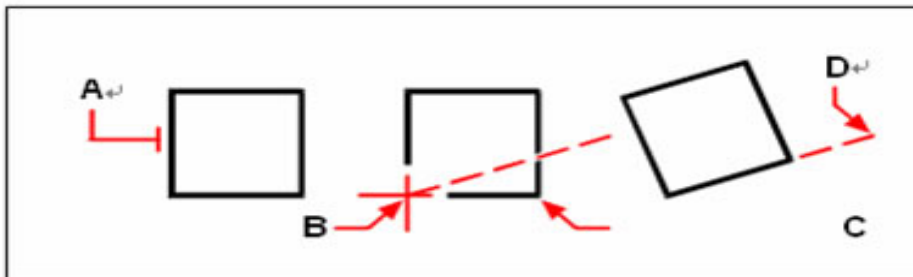
To rotate an object, select the object to rotate (A), and then specify base point (B) and the rotation angle (C)

10.8.3 Rotate an Object to an Absolute Angle

You may also rotate objects by specifying an absolute angle.

10.8.4 To rotate a selection set in reference to an Absolute Angle

1. Choose Modify > Rotate.
2. Select the objects, and then press Enter.
3. Specify base point.
4. On the command line, type Reference.
5. Specify the reference angle.
6. Specify the new angle.



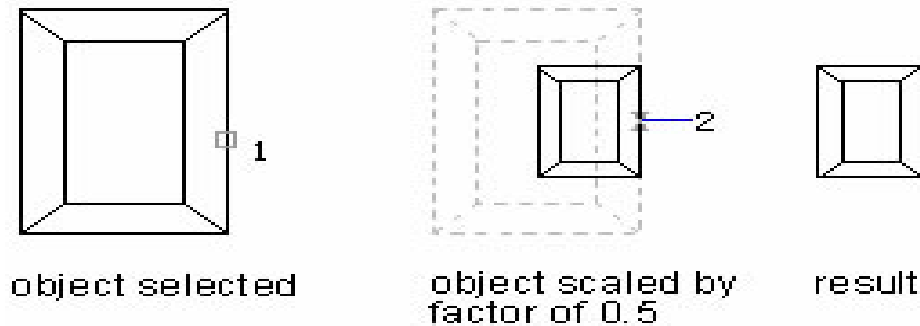
To rotate an entity in reference to a base angle, select the entity (A), specify base point (B), select the reference angle and pick point (B) again (or type the @ symbol), specify the second point (C), and then specify the point representing the new angle (D)

10.9 Scale Objects

Command line SCALE

Resize objects to make them longer or shorter in only one direction or to make them proportionally larger or smaller. You may also stretch certain objects by moving an endpoint, vertex, or control point.

You may use SCALE command to zoom the selected objects. You may type a scale factor or specify a base point and distance to zoom objects. A scale factor that is greater than 1 enlarges the object; and a scale factor that is smaller than 1 shrinks the object.



You may also scale objects by comparing the basic scale and new scale. The objects become larger if the new scale is large than the basic scale.

The dimensions will be scale along with the objects to scale.

10.9.1 To scale a selection set by a scale factor

1. Choose Modify > Scale.
2. Select the objects, and then press Enter.
3. Specify the base point.
4. Specify the scale factor.



To scale an object by a scale factor, select the object (A), and then specify the base point (B) and the scale factor

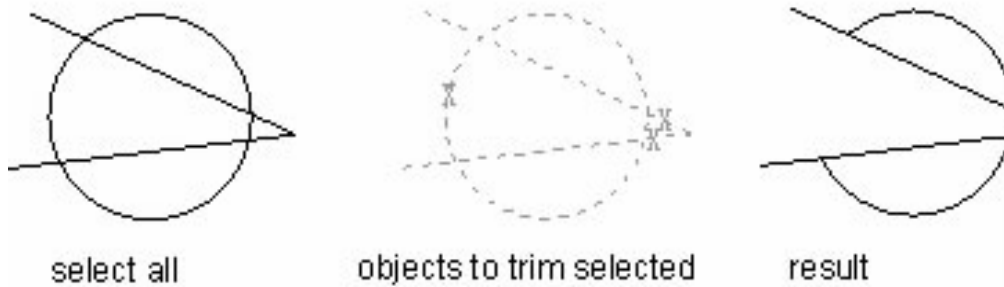
10.10 Trim or Extend Objects

Shorten or lengthen objects to meet the edges of other objects

You may connect edges of objects through trimming or extending.

When trimming or extending objects, you will need to specify cutting edges to trim or a boundary to extend, then you may select objects to cut or extend. By pressing ENTER at the prompt for selecting entities, all entities are the possible boundaries.(?)

If the user wants to select blocks as cutting edges, one can only use the single selection, Crossing, Fence, and Select All options.



10.11 Trim Objects

Command line TRIM

You may clip or trim, objects so they end at one or more implied cutting edges defined by other objects. You may also trim objects to the point at which they would intersect an implied cutting edge. When trimming objects, one first selects the cutting edges and then specify the objects to trim, selecting them either one at a time or using the fence selection method.

You may trim arcs, circles, lines, unclosed polylines, and rays. In which, arcs, circles, lines, polylines, rays, infinite lines, and viewports on a Layout tab can act as cutting edges.

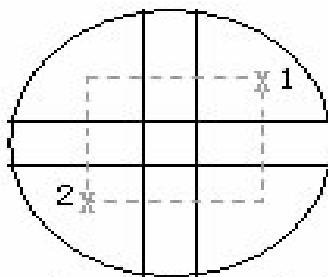
10.11.1 To trim an object

1. Choose Modify > Trim.
2. Select one or more cutting edges, and then press Enter.
3. Select the object to trim.
4. Select another object to trim, or press Enter to complete the command.

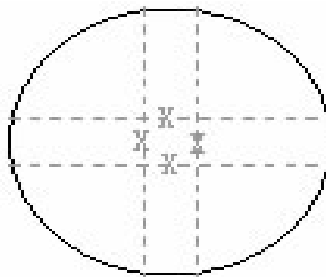


To trim objects, select the cutting edge (A), and then select the objects to trim (B)

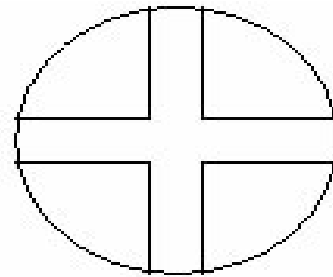
You may trim objects to the nearest point of other objects. By selecting objects and clicking objects to trim, the system specifies the nearest objects as the cutting edges. The following picture shows that the crossing boundary is smooth after trimming.



cutting edges selected
with crossing selection



objects to trim
selected



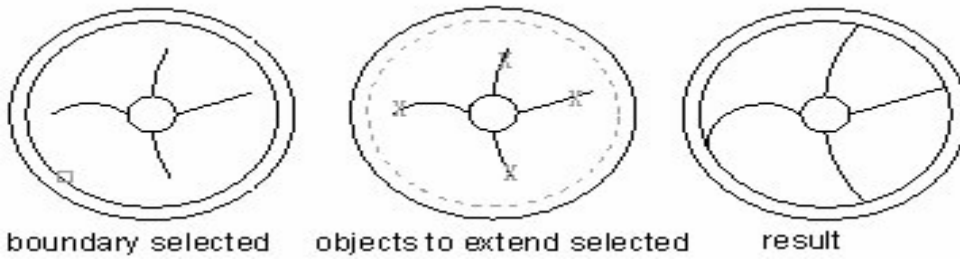
result

10.12 Extend Objects

Command line: EXTEND

You may extend objects so that they end at a boundary defined by other objects. You may also extend objects to the point at which they would intersect an implied boundary edge. When extending objects, you may first select the boundary edges, and then specify the objects to extend by way of selecting them one by one or using the fence selection method.

You may extend arcs, lines, two-dimensional polylines, and rays. Arcs, circles, ellipses, lines, splines, polylines, rays, infinite lines, and viewports on a Layout tab can act as boundary edges.



In this example, lines are extended precisely to a circle, which is the boundary edge.

10.12.1 To extend an object

1. Choose Modify > Extend.
2. Select one or more objects as boundary edges, and then press Enter.
3. Select the object to extend.
4. Select another object to extend, or press Enter to complete the command.



To extend objects, select the boundary edge (A), and then select the objects to extend (B)

10.12.2 To extend an object to an implied boundary

1. Choose Modify > Extend.
2. Select one or more boundary edges, and then press Enter.
3. On the command line, type edge mode.
4. On the command line, type extend.
5. Select the object to extend.
6. Select another object to extend, or press Enter to complete the command.



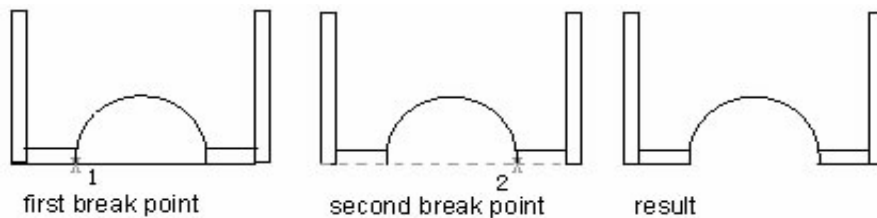
Select the boundary edge (A), and then select the objects to extend (B)

10.13 Create Breaks

Command line BREAK

Create a gap in an object, resulting in two objects with a space between them.

You may break arcs, circles and lines. When breaking entities, you must specify two points for the break. By default, the point used to select the entity becomes the first break point; however, you may use the first break point option to select a break point different from the one that selects the entity.



10.13.1 To break an object

1. Choose Modify > Break.
2. Select the object.
3. Specify the second break point.



Select the object (A), and then specify the second break point (B)

10.13.2 To select an object and then specify the two break points

1. Choose Modify > Break.
2. Select the object.
3. On the command line, type first break point.
4. Specify the first break point.
5. Specify the second break point.



Select the object (A), and then specify the first (B) and second (C) break points

TIP: To break an object in two without removing a portion of the object, specify the same point for the first and second break points by typing the at sign (@) and pressing Enter instead of specifying the second break point.

10.14 Create Chamfers

Command line: CHAMFER

Chamfering creates a flattened or beveled corner where two lines meet.

You may connect two nonparallel objects by extending or trimming them and then joining them with a line to create a beveled edge. You may chamfer lines, polylines, rays, and infinite lines. When creating a chamfer, you may specify how far to trim the objects back from their intersection (distance-distance method), or may specify the length of the chamfer and the angle it forms along the first object (distance-angle method).

When chamfering a polyline, you may chamfer multiple segments between two selected polyline segments, or may chamfer the entire polyline.

If two objects to chamfer on the same layer, the chamfer line lies on the layer as the object does; otherwise, the chamfer line lies on the current layer which controls the properties of the line. The multiple option can help you chamfer multiple objects at one time.



10.14.1 Chamfer by Specifying Distances

The chamfer distance is the amount that each entity is trimmed or extended to meet the chamfer line or to intersect the other. If both chamfer distances are 0, chamfering trims or extends the two entities until they intersect instead of creating a chamfer line.

The default setting for the first distance is the last one the user has specified, and the second distance is the same as the first one. However, you may reset the desired chamfer distance.

10.14.2 To chamfer two objects using the distance-distance method

1. Choose Modify > Chamfer.
2. On the command line, type D.
3. Specify the first and second chamfer distances.
4. Click OK.
5. Select the first object.
6. Select the second object.



Select the first (A) and second (B) objects

10.14.3 Trim and Extend Chamfered Objects

If the objects to chamfer are two intersecting lines, and one selects the Trim option to chamfer objects, the system automatically trims the lines to the endpoint of chamfered line. If the selected lines are not intersecting with each other, system extends them to get intersection.

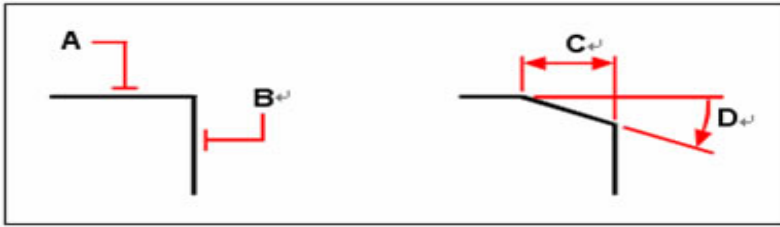
If you select No Trim, the system will create chamfer directly and not trim.

Chamfer by Specify Length and Angle

You may chamfer objects by specifying a start point and an angle.

10.14.4 To chamfer two objects using the distance-angle method

1. Choose Modify > Chamfer.
2. On the command line, type A.
3. Specify the chamfer distance and angle.
4. Click OK.
5. Select the first object.
6. Select the second object.



Select the first (A) and second (B) objects. The chamfer is drawn, based on the distance (C) and the angle (D)

10.15 Create Fillets

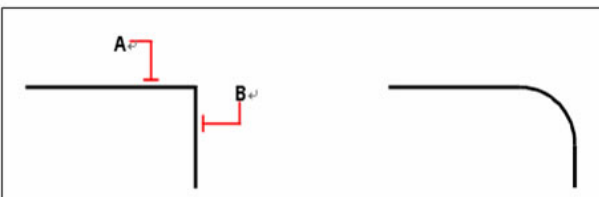
Command line `FILLET`

Filleting connects two objects with a smoothly fitted arc of a specified radius.

You may create a fillet using the `FILLET` command. This command can apply to objects such as arcs, circles, ellipses, lines, polylines, radials, splines or construction lines as well as 3D solids. A fillet is an arc that connects two objects smoothly and has a specified radius. An inside corner is called a fillet; an outside corner is called a round.

10.15.1 To fillet two objects

1. Choose *Modify > Fillet*.
2. On the command line, type R.
3. Specify the fillet radius.
4. Click OK.
5. Select the first object.
6. Select the second object.



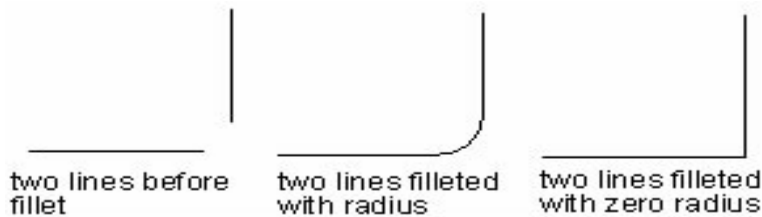
Select the first (A) and second (B) objects

Using the `FILLET` command one can create an arc that is tangent to the selected objects easily.

When the objects to fillet are on the same layer, the fillet arc is created on that layer; otherwise, the fillet arc is created on the current layer that affects the properties of objects such as color and linetype. The Multiple option can create fillets for more than one set of objects until the user exits the command.

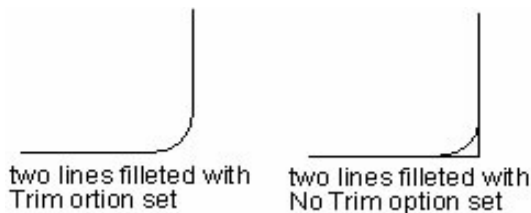
10.15.2 Set the Fillet Radius

You may specify the radius of a fillet arc connecting two objects. The latest value is always the current radius value. This setting is only available when you create a new fillet arc. When the radius is set to 0, the objects to fillet are trimmed or extended to get intersecting, as oppose to actually creating a fillet arc.



10.15.3 Trim and Extend Filleted Objects

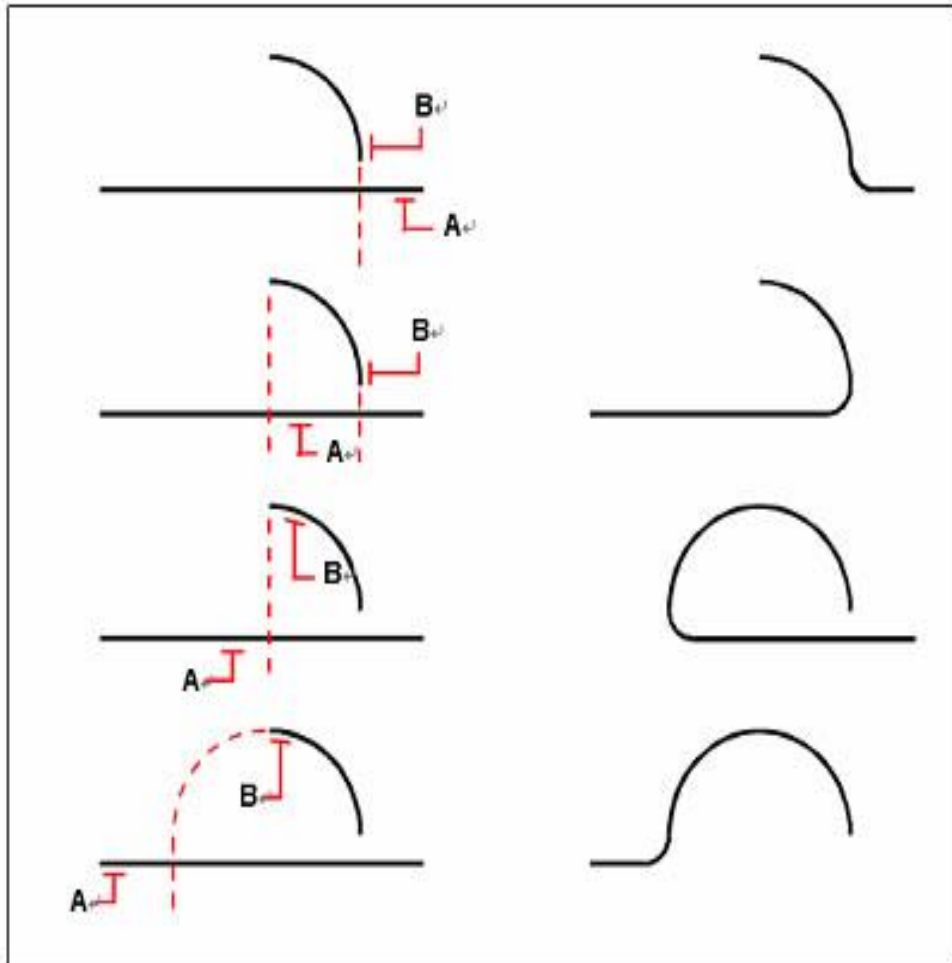
While performing fillet, you may select the Trim option to decide whether the selected edges are extended to the endpoints of fillet arcs. By default, all objects except for circles, ellipses, closed polylines and splines are trimmed or extended when filleting is performed.



When mode is set to TRIM, FILLET trims the intersecting lines to the endpoint of the fillet arc. If the lines do not intersect, the system extends or trims them to get an intersection.

10.15.4 Specify the Location of the Fillet

When the objects to fillet are a line and an arc or a circle, there are several potential fillets and the system creates the fillet with the endpoint of arc nearest to the selected point. The location of fillets depends on the objects you specify, as shown in the following picture:



Select objects (A and B)

11 Image Tools

CNC Scan CAD contains several simple tools that can be used to trace over images. These tools can be accessed with using either tool bar buttons or the *Draw* pulldown menu.

11.1 Insert a Picture

Command line INSERTIMAGE

You can use this command to insert an image file. CNC Scan CAD can import the following image types

- .tsi
- .bmp
- .jpg
- .tif
- .tga
- .pcx

11.2 Adjust Angle

Command line BKANG

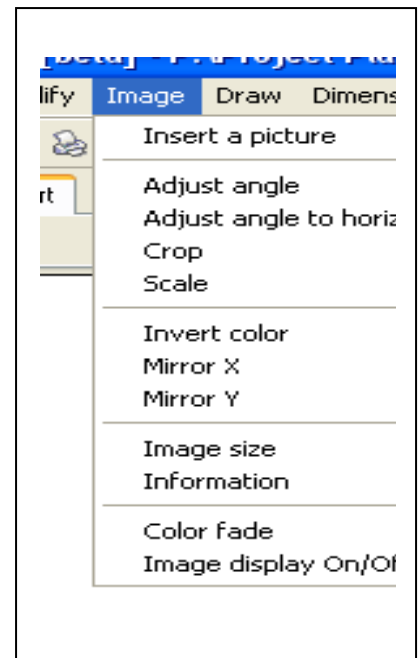
This command rotates the image around its center by the angular amount specified

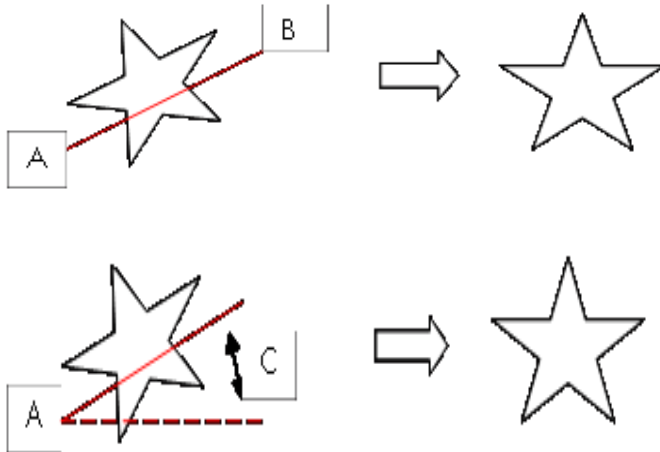
11.3 Adjust Angle to Horizontal

Command line BKANG2

This command allows you to rotate an imported image by specifying its horizon. This can be done in either of two ways:

- Choose two points along a straight line
- Choose one point and one angle that specify a straight line





11.4 Crop

Command line `BKCROP`

This command allows you to crop an imported image by specifying opposite corners of a rectangular box. Any part of the image that is not contained in the box will be discarded

11.5 Scale

Command line `BKSCALE`

With this command, you can change the scale of the image in either the horizontal (X) or vertical (Y) dimension by drawing a line and reassigning its dimension.

The scaling factors do not need to be the same for both X and Y directions

11.6 Invert Color

Command line `BKINV`

This command will invert colors of the image, providing a photonegative effect.

11.7 Mirror X

Command line `BKMIR`

This command will mirror the image in the X direction. The center of the image remains in the same place.

11.8 Mirror Y

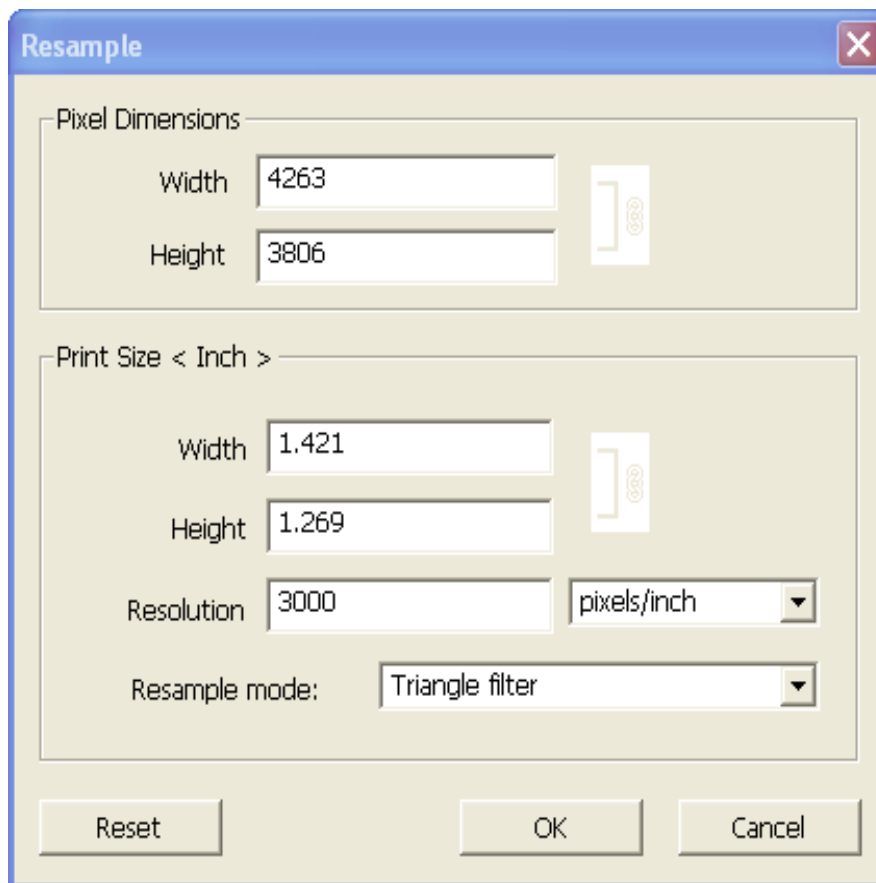
Command line BKFLP

This command will mirror the image in the Y direction. The center of the image remains in the same place.

11.9 Image Size

Command line BKSIZE

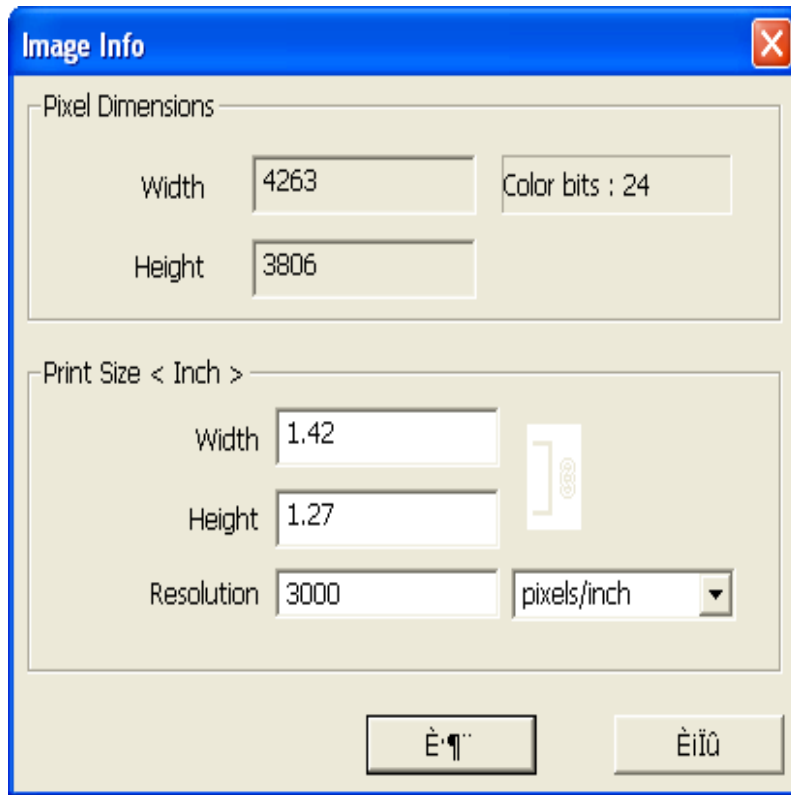
This command is used to resize the image.



11.10 Information

Command line BKINFO

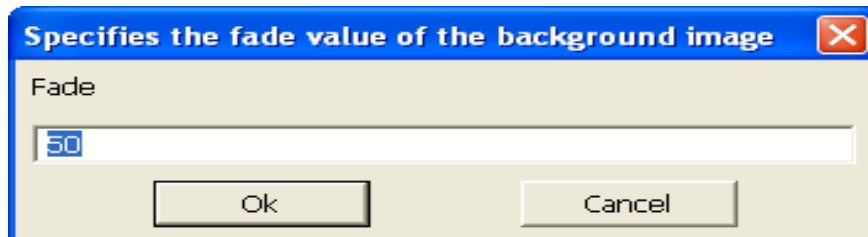
This command displays information about the image size. You can also adjust the image resolution in this menu.



11.11 Color Fade

Command line BKCHROMA

This command is used to fade the image. The native value of an imported image is 100. The image can be faded from values ranging from 100-10, where 10 is the lightest image.



11.12 Image Fade On/Off

Command line _IMAGETOGGLE

This command toggles the image display on and off.

12 Drawing Tools

CNC Scan CAD contains several simple tools that can be used to trace over images. These tools can be accessed with using either tool bar buttons or the *Draw* pulldown menu.

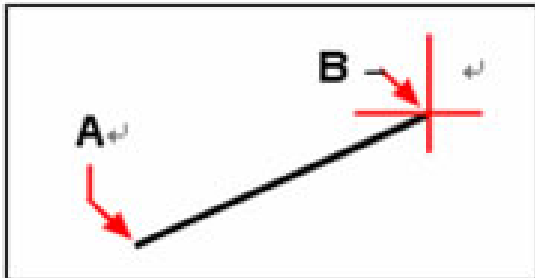
12.1 Line Tool



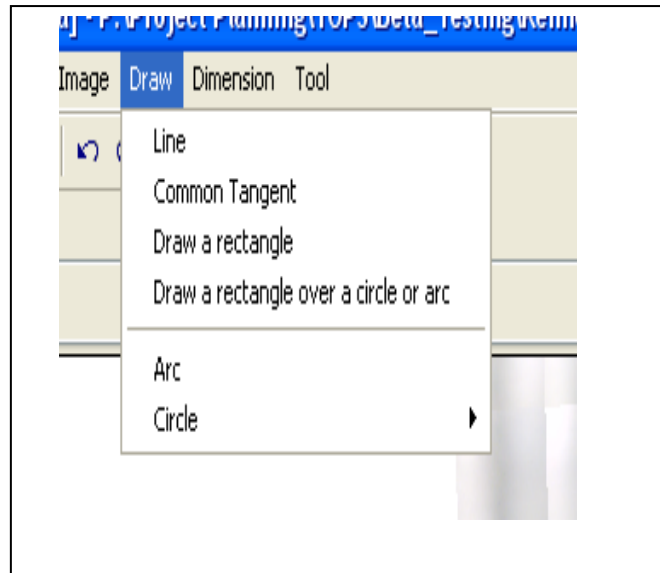
Command line `LINE`

Like most CAD programs, in a simple line with connected segments, each segment is a separate line object.

A line consists of two points - a start-point and an end-point. You may connect a series of lines, but once again, each line segment is considered a separate line object.



Start point (A) and endpoint (B).



12.1.1 To draw a line

1. Choose *Draw > Line*.
2. Specify the start point
3. Specify the end point
4. Press Enter to complete the command

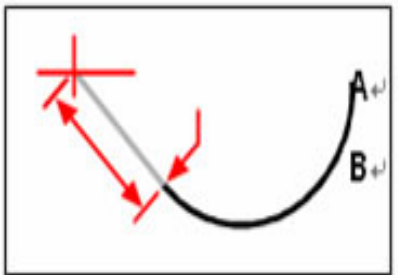
To start a new line at the endpoint of the last line drawn, start the Line command again and press Enter at the Specified Start Point prompt.

To continue drawing a line segment from the endpoint of your last line drawing, you may start over with the Line command, press Enter, and then click your mouse to specify your start-point.

If the last object one drew was an arc, one can also draw a line connecting to and starting from the endpoint of the arc.

12.1.2 To draw a line as a continuation from the end of an arc

1. Choose Draw > Line.
2. Take the end of an arc as the start point of line.
3. Specify the length of the line.



Endpoint of previous arc (A) and length of the line (B).

12.2 Arc Tool

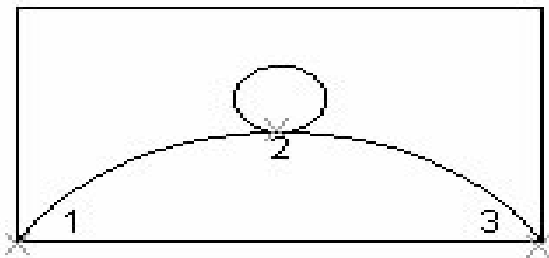


Command line ARC

An arc is a portion of a circle. The default method for drawing an arc is to specify three points—the start point, a second point, and the endpoint. You can draw arcs using any of the following methods:

- Three points on an arc
- Start point center point endpoint
- Start point center point included angle
- Start point center point chord length
- Start point endpoint included angle
- Start point endpoint starting direction
- Start point endpoint radius
- Center point start point endpoint
- Center point start point included angle
- Center point start point chord length

12.2.1 Draw Arcs by Specifying Three Points



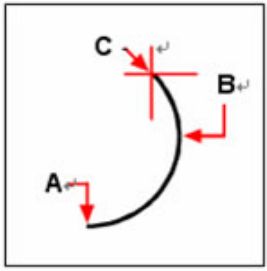
draw an arc by specifying
three points

Drawing an arc by specifying three points. The start point of the arc (1) snaps to the endpoint of a line. The second point of the arc (2) snaps to the middle circle in the illustration.

You may draw an arc by specifying three points: the start point of the arc, a point anywhere on the arc and the endpoint of the arc.

12.2.2 To draw an arc by specifying three points

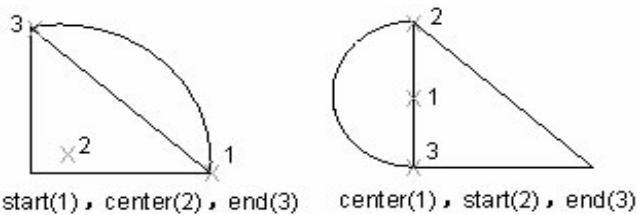
1. Choose *Draw > Arc*
2. Specify the start point.
3. Specify a second point.
4. Specify the endpoint.



Start point (A), second point (B), and endpoint (C)

12.2.3 Draw Arcs by Specifying Start, Center, End

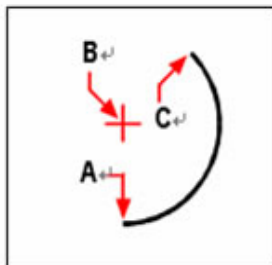
Under command line prompts for drawing arc, draw an arc by specifying center, start point and end point.



Drawing an arc by specifying three points

12.2.4 To draw an arc by specifying its start point, center point, and endpoint

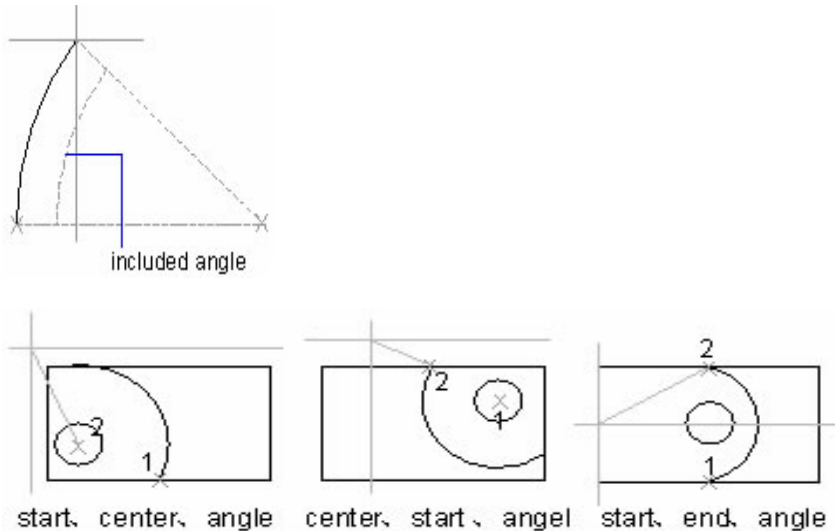
1. Choose *Draw > Arc*
2. Press 'C' key to specify the center point.
3. Specify the center point.
4. Specify the start point.
5. Specify the endpoint.



Start point (A), center point (B), and endpoint (C)

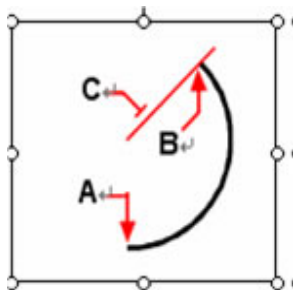
12.2.5 Draw Arcs by Specifying Start, Center, Angle

You can draw an arc by specifying start point and included angle. The included angle determines the endpoint of the arc. Use the Start, End, Angle method when you know both endpoints but cannot snap to a center point.



12.2.6 To draw an arc by specifying two points and an included angle

1. Choose *Draw > Arc*
2. Specify the start point.
3. Specify the end point.
4. Press 'A' key to specify the include angle.
5. Specify included angle.

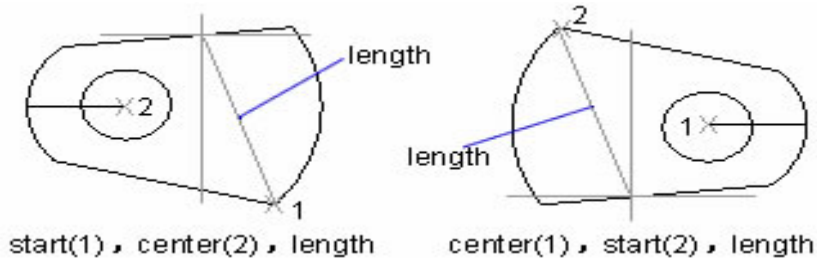


Start point (A), endpoint (B), and included angle (C)

To draw an arc in a counterclockwise direction, enter a positive value for the included angle. To draw an arc in a clockwise direction, enter a negative value for the included angle.

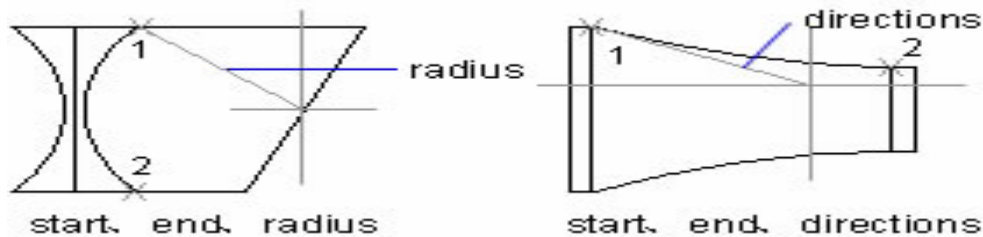
12.2.7 Draw Arcs by Specifying Start, Center, Length

Draw an arc by specifying the center of the arc after picking the start point of arc and then specifying the length of arc. Or use Center, Start, Length, which is actually equivalent to specifying the included angle.



12.2.8 Draw Arcs by Specifying Start, End, Direction/Radius

Draw an arc by specifying start, end radius/Direction, and entering a length. You can also click a line segment using any pointing device either clockwise or counterclockwise. If used the second way, move the cursor to direct the arc upwards or downwards.



Move the cursor downwards to draw a concave arc or upwards to draw a convex arc.

12.3 Circle Tool



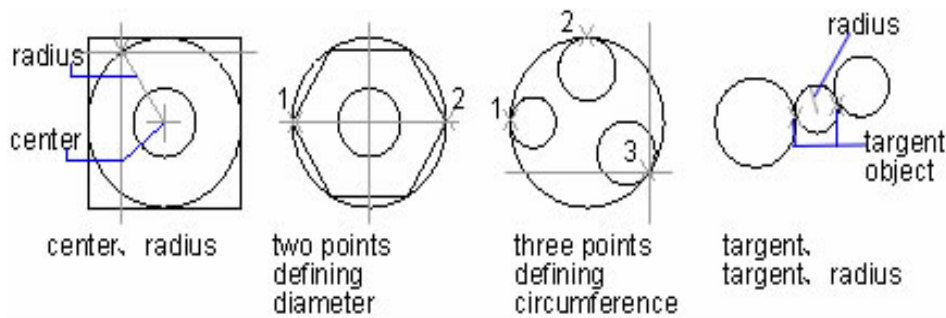
Command line: CIRCLE

You may specify various combinations of center, radius, diameter, points on the circumference, and points on other objects to draw circles.

The default method for drawing a circle is to specify a center point and the radius.

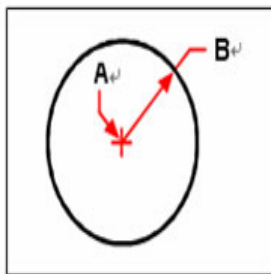
You may draw circles using any of the following methods:

- Center Radius
- Two points
- Three points
- Tangent Radius



12.3.1 To draw a circle by specifying its center and radius

1. Choose *Draw > Circle>Center, Radius*.
2. Specify the center point.
3. Specify the radius of the circle.

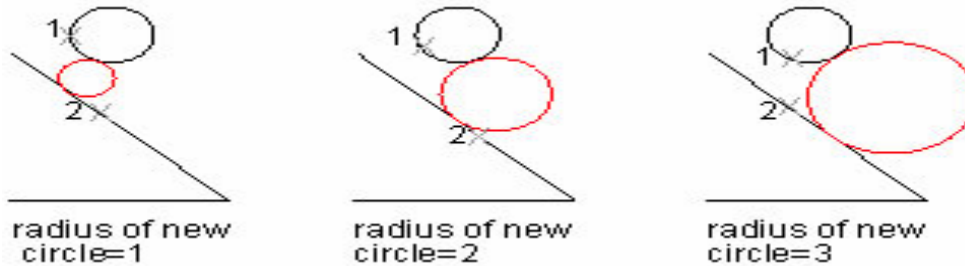


Center point (A) and radius (B)

12.3.2 Draw a Circle Tangent to Other Objects

A tangent point is a point where an object touches another object without intersecting it. To create a circle that is tangent to other objects, select the objects and then specify the radius of the circle.

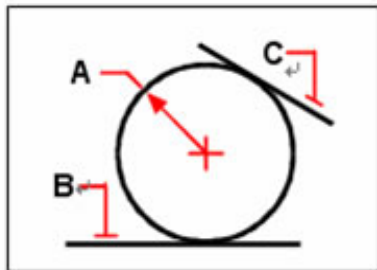
You may also turn on the Object Snap mode to set the snap point as Tangent. This allows circles that are tangent with other objects can be created easily. A three point tangent method is also available for creating circles with three tangent points.



The red circle is the one being drawn, and points 1 and 2 select the objects to which it is tangent

12.3.3 To draw a circle tangent to existing objects

1. Choose *Draw > Circle>Tan, Tan, Radius*.
2. Select the first tangent point on the object to be tangent with the circle.
3. Select the second tangent point on the object to be tangent with the circle.
4. Specify the radius of the circle.



Radius of circle (A) and tangent lines (B) and (C)

12.4 Rectangle Tool

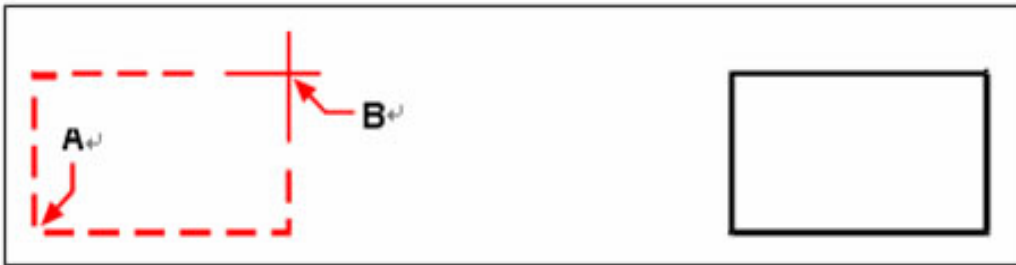
Command line: RECTANG



With CNC Scan CAD, rectangles are 4 closed lines. You may draw a rectangle by specifying either two of its opposite corners.

12.4.1 To draw a rectangle

1. Choose *Draw > Rectangle*.
2. Specify one corner of the rectangle.
3. Specify the opposite corner of the rectangle.



Specifying opposite corners (A, B) results in the rectangle shown on the right.

13 Dimension Tools

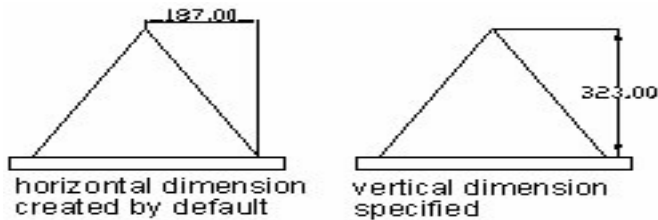
13.1 Create Horizontal and Vertical Dimensions

Command line: DIMLINEAR

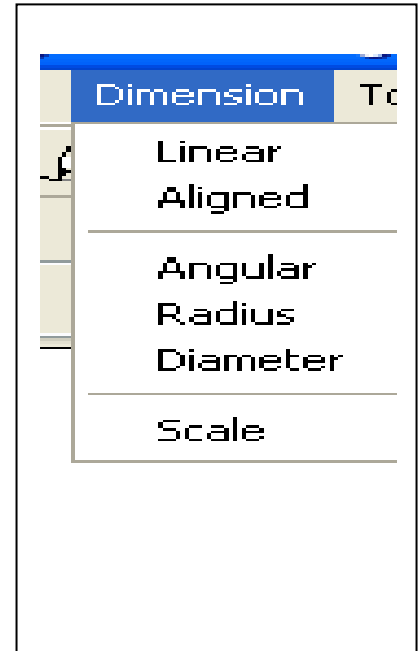


Linear dimensions annotate linear distances or lengths and can be oriented horizontally, vertically, or aligned parallel to an existing entity or to the selected extension origin points.

The system automatically applies a horizontal or vertical dimension depending on the specified extension line origins or the location where you select an object. However, you may create a dimension by specifying it to be horizontal or vertical.



A horizontal dimension is created unless you specify a vertical one.



13.1.1 To create a horizontal or vertical dimension

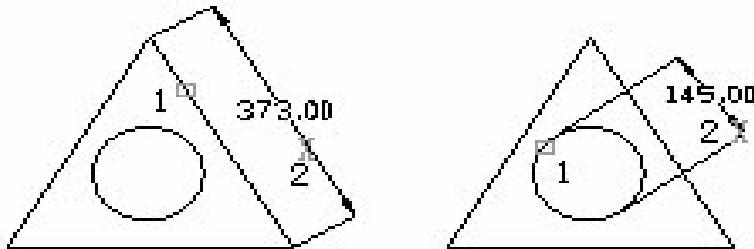
1. Choose *Dimensions > Linear*.
2. Press Enter, and then select the entity to dimension. Or, you may insert the dimension by specifying the first and second extension line origins.
3. Specify the dimension line location.

13.2 Create Aligned Dimensions

Command line: DIMALIGNED



With aligned dimensions, the dimension line is parallel to the line passing through the extension line origins. The aligned dimension is also parallel to the objects you specify. The following figure shows two examples of aligned dimensions. Select the object (1), and then specify the location of the aligned dimension. The system creates the extension lines automatically.



13.2.1 To create an aligned dimension

1. Choose *Dimensions > Aligned*.
2. Press Enter, and then select the entity to dimension. Or, you may insert the dimension by specifying the first and second extension line origins.
3. Specify the dimension line location.



To insert an aligned dimension by selecting the entity, select the entity (A) to dimension, and then select the dimension line location (B).

13.3 Dimension Arcs



Command line: DIMANGULAR

Angular dimensions are used to measure the angle between lines or three points. You may dimension objects including circles, arcs, and lines.

Angular dimension lines are typically represented with arc segments. To create an angular dimension for two nonparallel lines, the dimension line arc spans the angle between the two lines. As you specify the dimension line arc location, if the dimension line arc does not intersect one or two extension lines being dimensioned, the system automatically draws them. The dimension line arc between lines is always less than 180 degrees. However, for arcs, the dimension of the line arc can be up to 360 degrees.

13.3.1 To dimension an angle between two lines

1. Choose *Dimensions > Angular*.
2. Select one line.
3. Select the other line.
4. Specify the dimension line location.



Select one line (A), select the other line (B), and then specify the dimension line location (C).

If the object to be dimensioned is arc, the dimension line arc is placed between the extensions. The dimension line arc is also drawn between the extension lines when you use three points to create an angular dimension. The quadrant of the dimensioned angle depends on the location that you specify for the dimension line arc.

13.3.2 To dimension an angle encompassed by an arc

1. Choose *Dimensions > Angular*
2. Select the arc.
3. Specify the dimension arc location.



To dimension the angle subtended by an arc, Select the arc (A), and then specify the dimension arc location (B).

13.4 Create Radial Dimensions



Command line: DIMRADIUS

You may create radial dimensions and jogged dimensions for circles or arcs to measure their radius.

You may create radial dimensions for circles or arcs to measure their radius. The radial dimension is a radial dimension line with an arrowhead that points to a circle or arc.

13.4.1 To create a radial dimension

1. Choose *Dimensions > Radius*.
2. Select the arc or circle.
3. Specify the dimension line location.



Select the circle (A), and then specify the dimension line location (B).

13.5 Create Diameter Dimensions



Command line: DIMDIAMETER

You may create diameter dimensions for circles or arcs to measure their diameters. Creating diameter dimensions resembles creating radial dimensions.

13.5.1 To create a diameter dimension

1. Choose *Dimensions > Diameter*.
2. Select the arc or circle.
3. Specify the dimension line location.



Select the circle (A), and then specify the dimension line location (B).

13.6 Scale Dimensions

Command line: DIMSCALE

This tool is used to specify the size of the dimension text. The default value is 0.100. Simply specify a new value to resize the text.

13.6.1 To resize the scale of the dimension text

1. Choose *Dimensions > Scale*
2. Specify a new value

I4 Tools

I4.1 Help

Command line: `HELP`

Displays the online help menu

I4.2 Layer Bar

Command line: `ShowLayerCtrl`

Toggles the layer bar on/off

I4.3 Distance

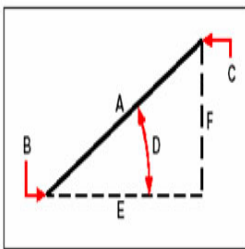
Command line: `DIST`

Calculate the distance between any two points you have selected. The following information is displayed:

- The distance between the points, measured in drawing units.
- The incremental change (delta) in position between the points in both X and Y
- The angle made by a line between the two points and the X Axis.

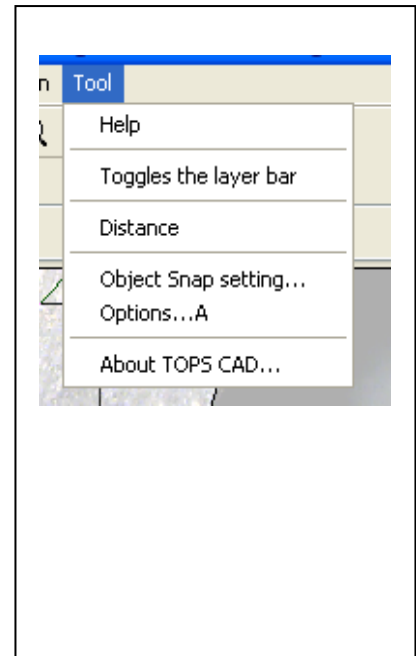
I4.3.1 To calculate the distance between two points and their angle

1. Choose Tools > Distance.
2. Specify the first point.
3. Specify the second point.



Use the Distance command to calculate the distance (A) between two points (B and C), the angle in the xy plane (D), the angle from the xy plane, and the delta x (E), delta y (F), and delta z distances between the two points.

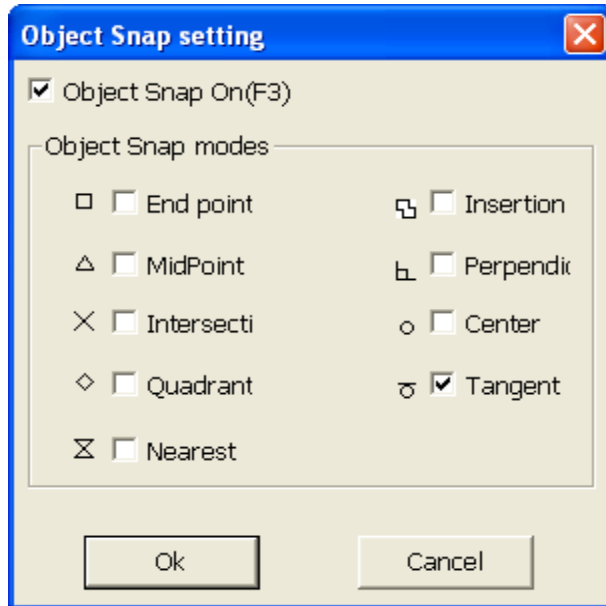
TIP: To use specific points on selected objects, use object snaps to precisely select the points on the objects.



14.4 Object Snap Setting

Command line: OSNAP

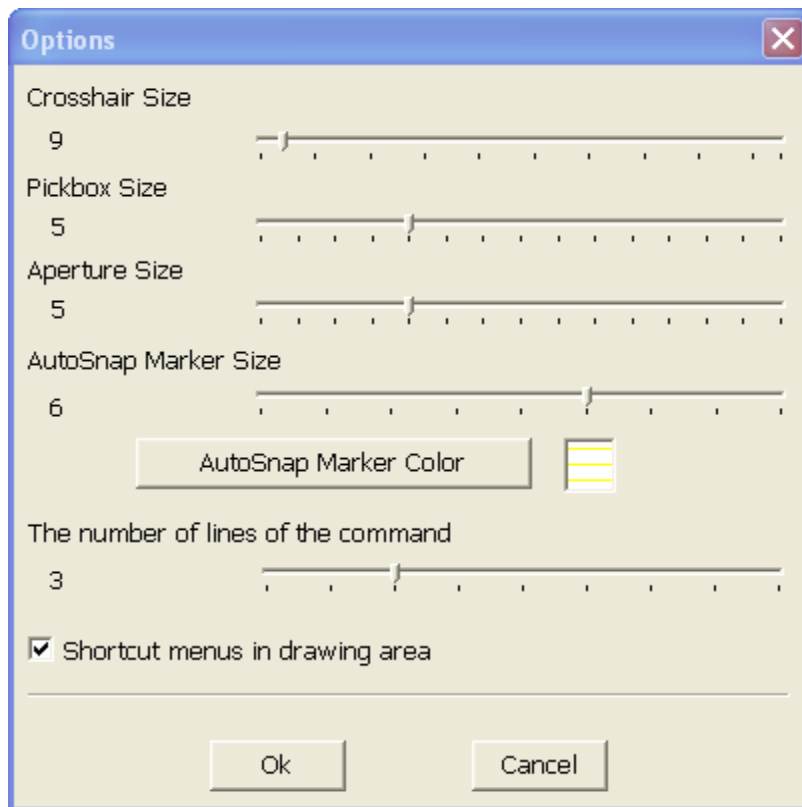
Displays Snap menu. You can select active snap points in this menu



14.5 Cursor Options

Command line: CADOPTION

Displays Cursor Options menu.



14.6 About CNC Scan CAD

Command line: ABOUT

Displays version information

I5 Troubleshooting

Troubleshooting CNC Scanner		
Problem	Solution	Discussion
CNC Scanner Plug-ins do not appear in Mach3 Menu	Ensure <i>topstools</i> is enabled in <i>Config</i> → <i>Config</i> <i>Plugins</i> menu and restart Mach3	
Camera does not display	Ensure Camera Software Driver is correctly installed	
Camera resolution is not correct.	Reinstall camera drivers. This may also be a problem with your USB port. Try another port	

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