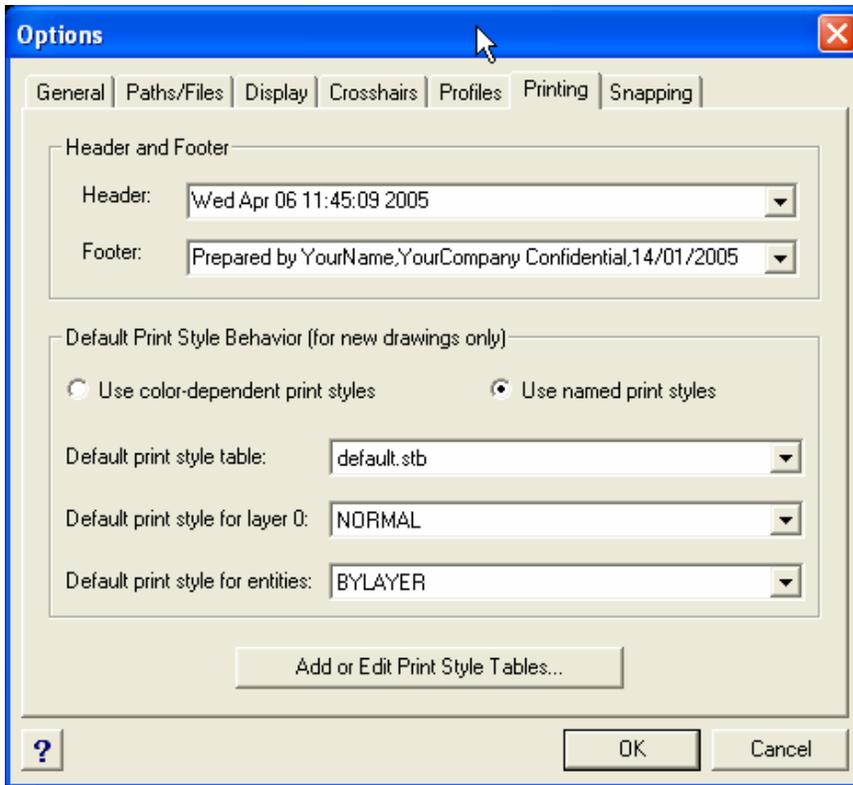


## Printing / Plotting in MSCAD2005

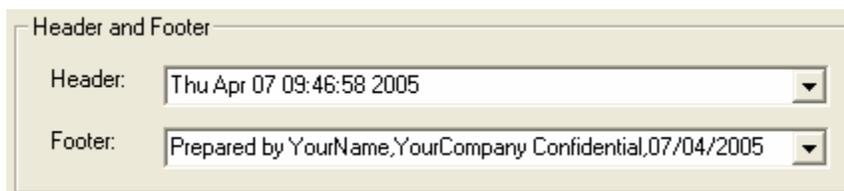
Printing has changed from some of our previous versions of MSCAD. Below is an overview of the options available for printing or plotting in MSCAD2005.

Before you run the print command, there are a few options found under the CAD Tools menu -> Options (or type in CONFIG) that we need to review. Pick on the Printing Tab to see the following.

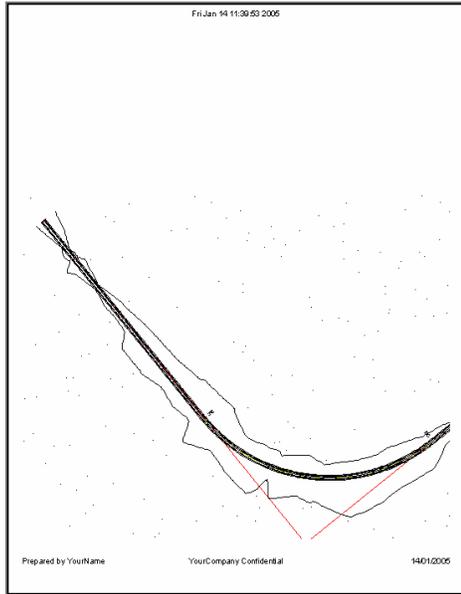


### **Header and Footer:**

The Header and Footer permit you to add some text to the outer edge of the print paper with each print. You can type in almost anything and it will be added to the print.



Example: This print has both a Header (at the top) and a Footer (at the bottom)



## Default Print Style Behavior

MSCAD2005 uses print styles to change the appearance of your printed drawing without modifying the actual entities in your drawing. Assigning print styles allows you to customize the color, pen width, linetype, and lineweight that are used to print your drawing.

Print styles help you control what your drawing looks like when it is printed. Rather than describe what an entity looks like on your screen, print styles describe what an entity will look like when you print it. For example, you can map all yellow entities in your drawing to print in blue without modifying the actual entities. You could also map all yellow entities to print with whatever lineweight, linetype, or pen width that you specify. Because print styles are saved in print style tables, which are files located on your computer, disk, or server, you can reuse them to help eliminate the need to reconfigure your print settings each time you print a drawing. For example, you may have multiple clients who have their own printing preferences. You can save print styles in a named file for each of your clients. You can even share the file with co-workers, or store the files on a network to ensure that everyone in your office uses the same standards.

Default Print Style Behavior (for new drawings only)

Use color-dependent print styles       Use named print styles

Default print style table: Autodesk-Color.stb

Default print style for layer 0: NORMAL

Default print style for entities: BYLAYER

This will determine if the new drawing will use the CTB (Color Dependant Print Style) or the STB (Style Dependant Print Style) when you get to the print/plot stage in the drawing.

You can choose from the different styles that you may have created and saved on your system. The CTB file format can be generated from inside the Print dialog – detailed info follows below in the Advanced Tab section of this document.

**\*NOTE\***

If you are working in a drawing that was saved from MSCAD2004 (and some other sources as well), then the default will be to use STB files. To change this you can set a variable in MSCAD2004 prior to saving the DWG file. You have to set PSTYLEPOLICY to 1 for CTB and set it to 0 for STB.

You can also type in the command CONVERTPSTYLES in MSCAD2005 to convert the plotting routine to use CTB instead of STB. Save the drawing, once converted, so you will not need to do it again in this drawing. When you run CONVERTPSTYLES you will see this dialog – pick OK to complete the command.



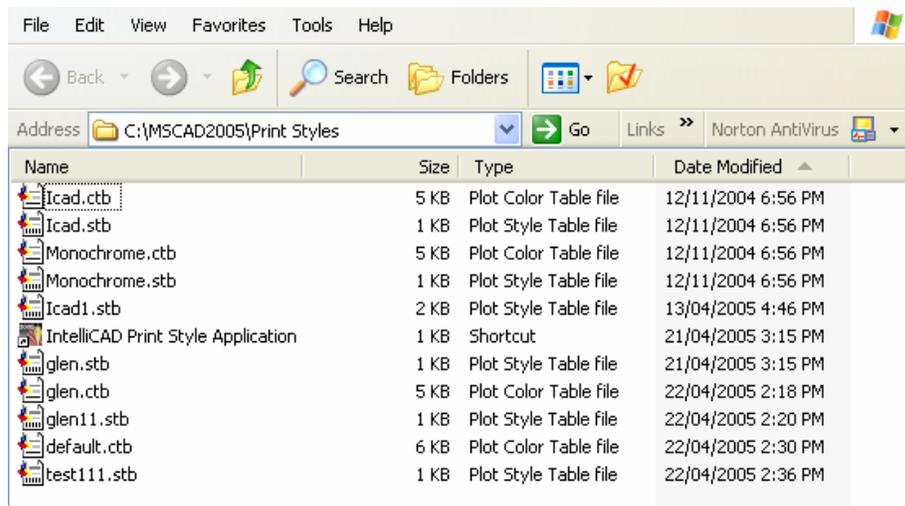
**Color-dependent plot style tables (.CTB file)** use an entity's color to determine characteristics such as lineweight. Use color-dependent plot styles to ensure that all entities that share the same color are plotted the same way. While you can edit plot styles in a color-dependent plot style table, you cannot add or delete plot styles. There are 255 plot styles in a color-dependent plot style table, one for each MSCAD2005 color. In versions prior to MSCAD2005, color was often used to control an entity's plotted appearance (most often lineweight). This technique was similar to using color-dependent plot style tables. When you create a color-dependent plot style table, you can import previous configuration information found in the registry or contained in a PCP file.

**Named plot style tables (.STB file)** contain user-defined plot styles. When you use a named plot style table, entities that have the same color may be plotted differently, based on the plot style assigned to the entity. A named plot style table can contain as many or as few plot styles as required. Named plot styles can be assigned to entities or layers, in the same way that linetype and color are assigned to entities. An entity whose plot style is set to BYLAYER inherits the plot style assigned to its layer.

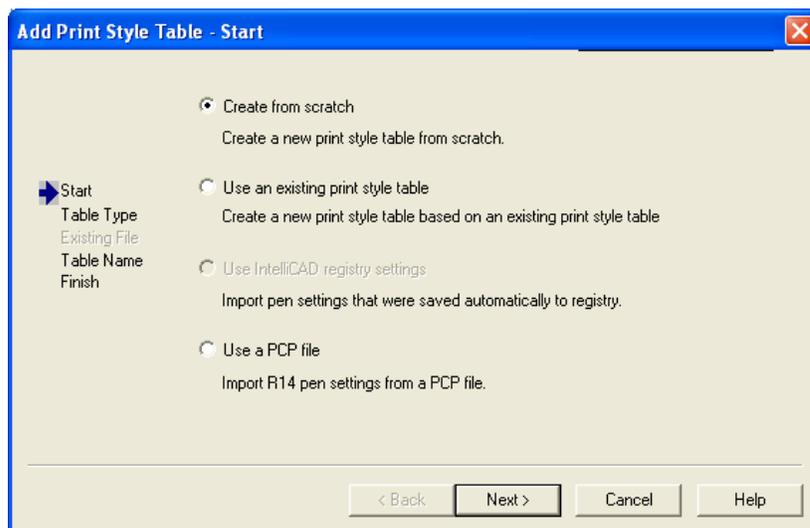
## Add or Edit Print Style Tables

Add or Edit Print Style Tables...

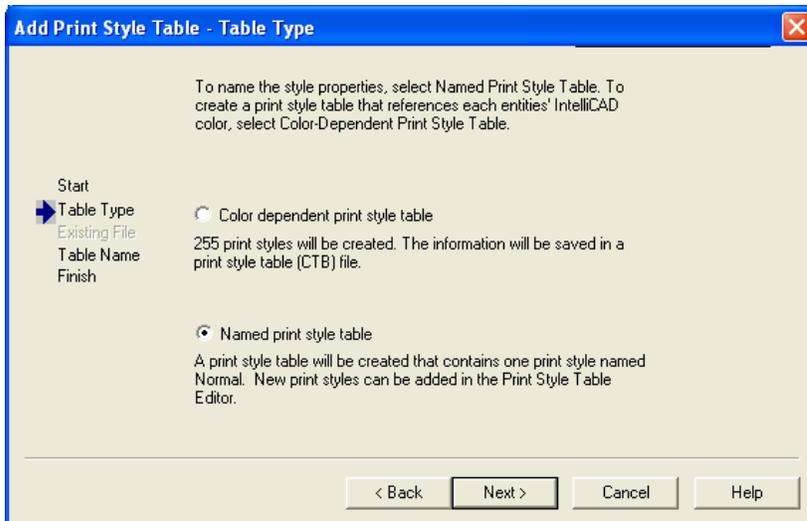
This opens the Windows Explorer program to allow you to create or copy STB or CTB files, as required, so we can use them.



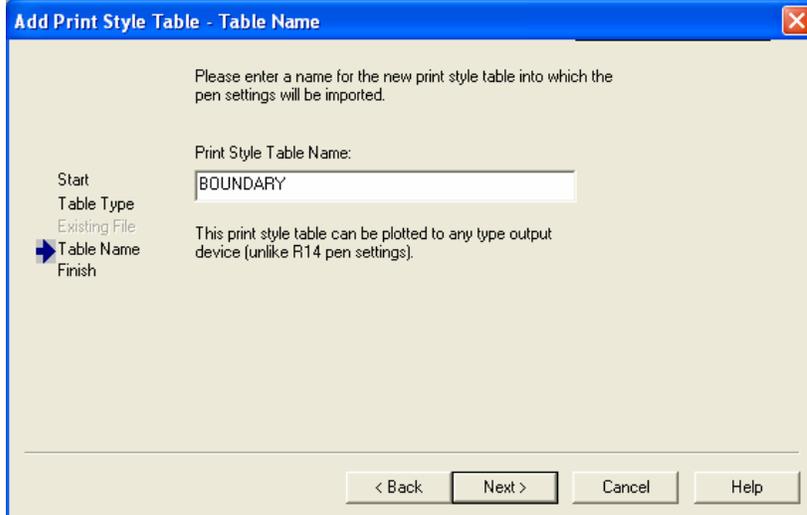
The shortcut:  IntelliCAD Print Style Application  
Will allow you to create a new STB or CTB file



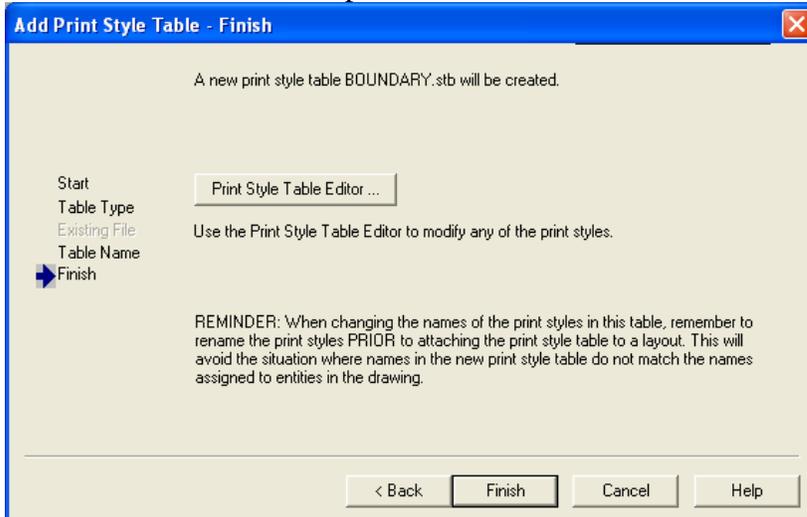
Creating a new CTB file is discussed further down in this document:



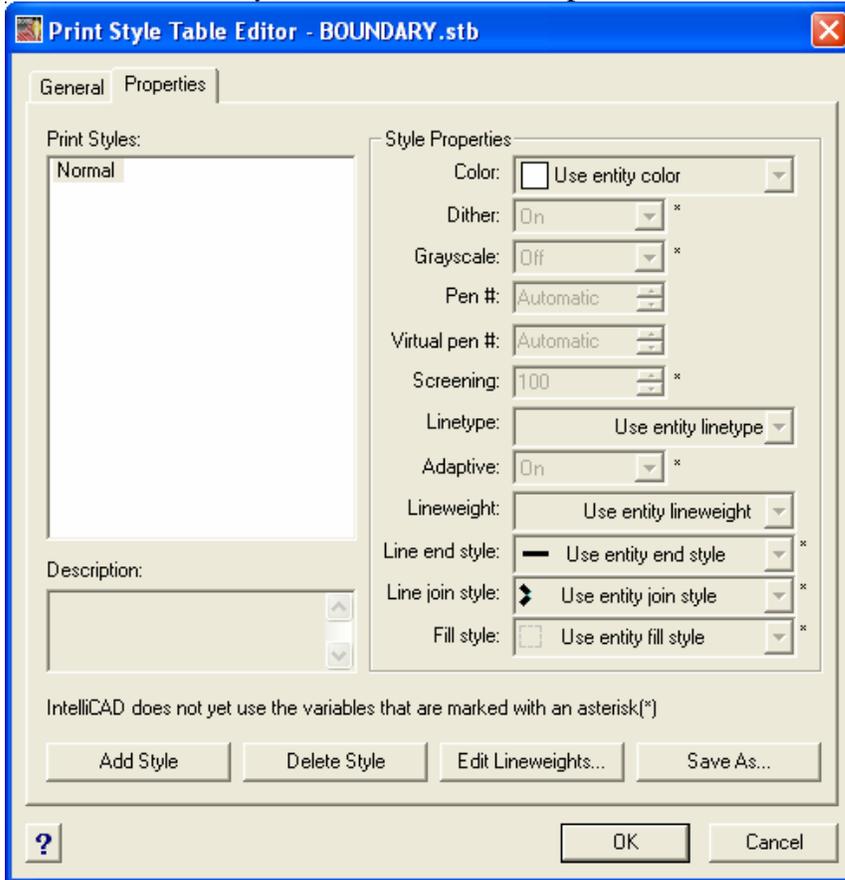
### Named Print Style Table (STB)



Give the table a name and pick Next to continue:



Pick on the Print Style Table Editor, the Properties Tab, to see the following:



**Add Style:** this button allows you to create a new style and name it.



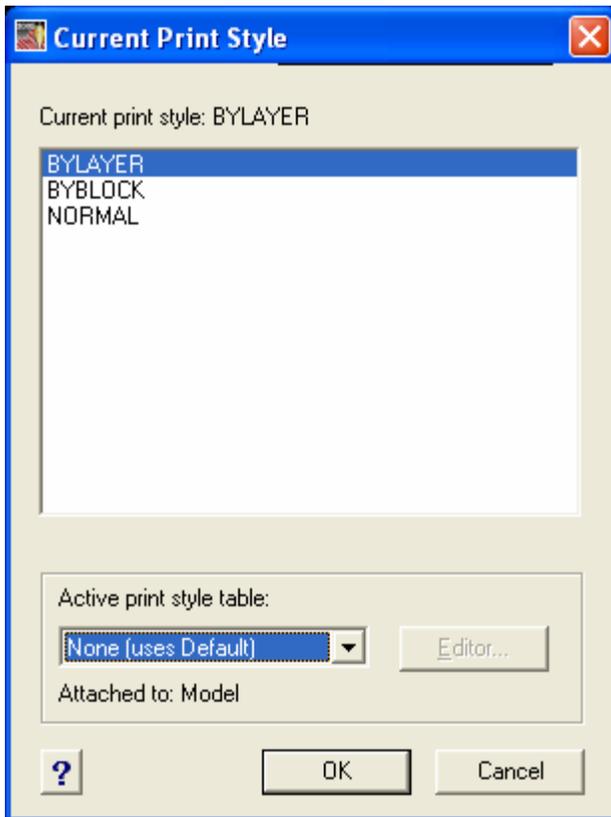
**Delete Style:** will erase the currently highlighted print style.

**Style Properties:** discussed further down in this document under the CTB table settings.

**PRINTSTYLE** command, type it at the command prompt:  
If the current drawing is set to use CTB files, then you will see this dialog:



If you are set to use STB files then you will see this dialog:



Active Print Style Table allows you to pick from any table available.

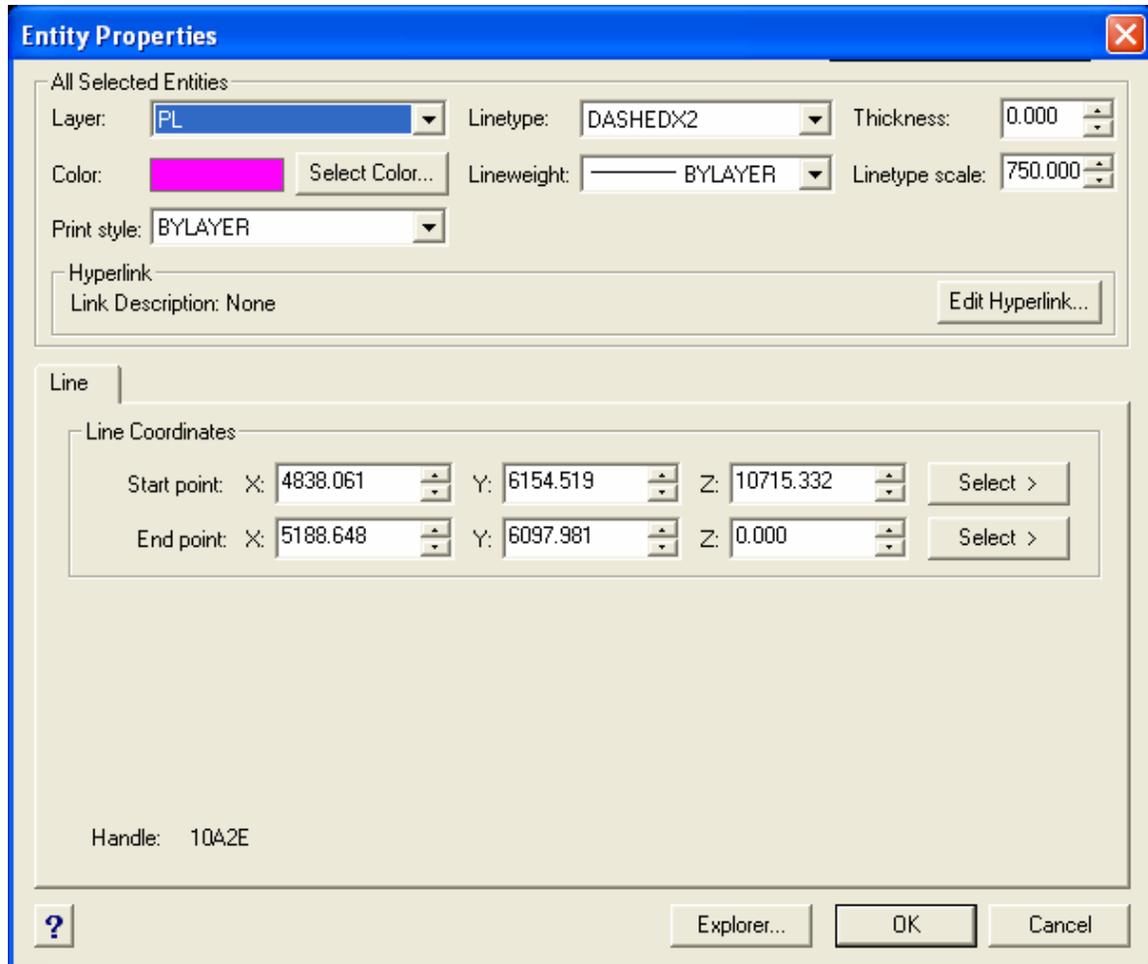


The Editor button allows you to edit the Style Settings (discussed elsewhere in this document)

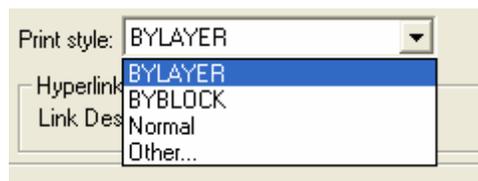
When using a STB file, you can then run the ENTPROP command (Modify menu -> Properties).

Every entity in the drawing can have it's own sprint style attached to it.

Select the entity to edit and you get a dialog like this:



The Print Style for this entity can be changed by picking on the Print Style option:



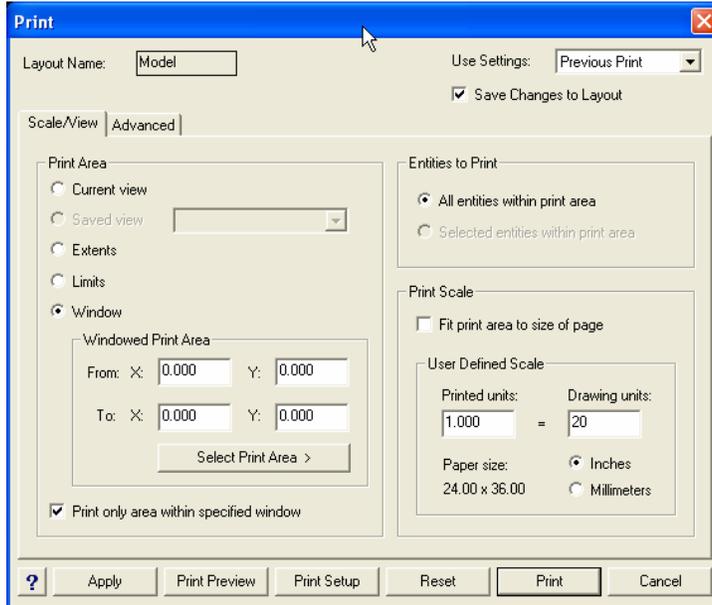
Other takes you to the same dialog as shown in the PRINTSTYLE command discussed above.

## Now we can run the PRINT command.

To print the current drawing, go to the File pull down menu and pick on PRINT.

Or you can pick on this button on the toolbar 

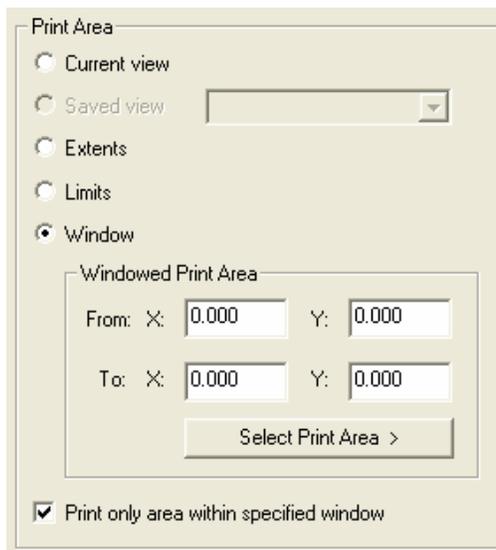
Either way, this is the dialog you will see:



There are a lot of options available, so lets break it down and explain each of them;

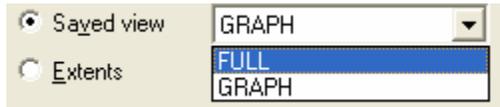
### Scale/View tab

#### **Print Area:**



Current View: The current view is exactly what you saw on screen when you picked the print command. This includes any blank areas around the drawing. It basically is a screen capture of the drawing window.

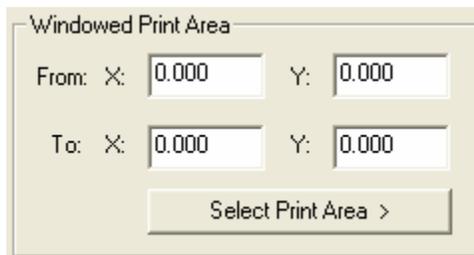
Saved View: If you have used the VIEW command in your drawing, to save different views, then they will be available to select to print.



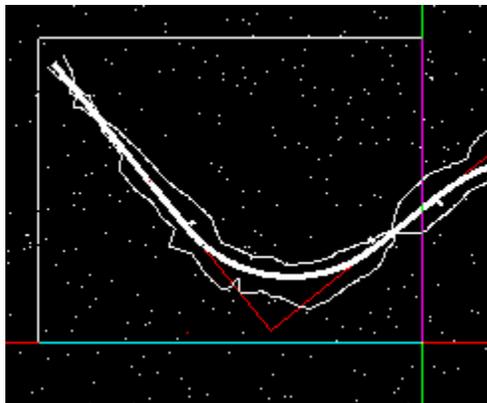
Extents: This will select the entire drawing. (for the layers that are on and thawed)

Limits: If you have set your limits, then you can print only what appears inside the limits.

Window: This allows you to accurately pick what portion of the drawing you wish to print. Picking this option will activate the Windowed Print Area.



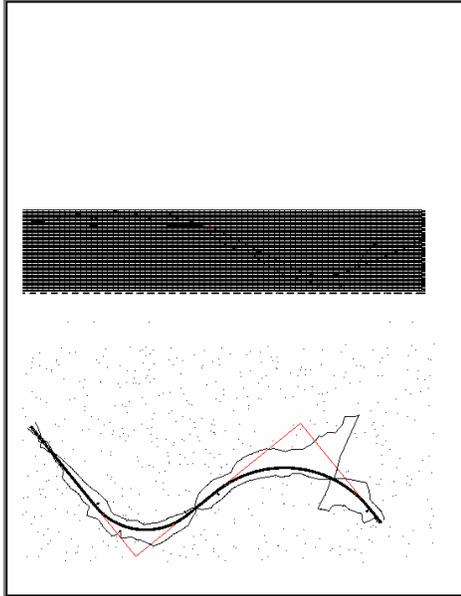
This allows you to either type in the X and Y values to give the upper and lower corners of the print area, or you can pick the Select Print Area> button to pick them on screen.



Pick the top left and move the mouse to the bottom left – use your object snaps for accuracy, if required. When you pick the second location, you will be returned to the dialog, and the coordinates you picked will be in the Windowed Print Area.

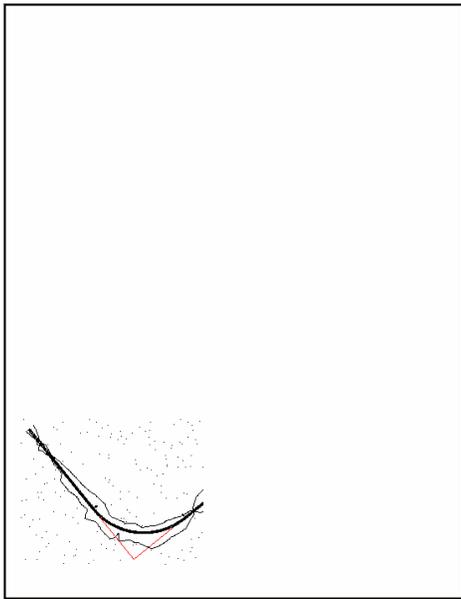
Print only area within specified window:

When the print paper is larger than the area to be printed, if you turn this option OFF – then anything else in the drawing, outside the selected area, will also be printed at the scale set. If this option is turned ON then only the windowed portion will be printed, leaving blank paper.

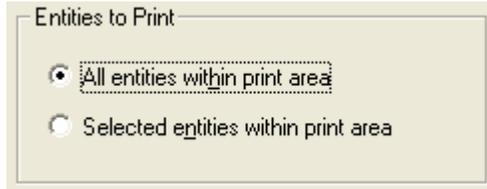


ABOVE – Option turned OFF

BELOW – Option turned ON

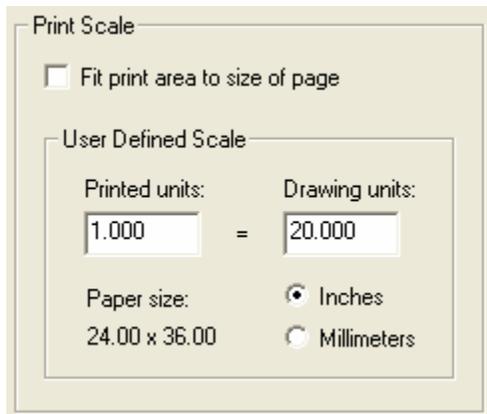


## Entities to print:



If you have selected some items in the drawing, prior to running the print command – then you have the option to print only the selected items or all of the entities within the print area.

## Print Scale:



Fit print area to size of page: The Fit option, when checked ON, takes the objects in the print area and scales them so they will fit on the size paper sheet currently selected. The drawing will be scaled within it's self but you will not know what the actual scale is.

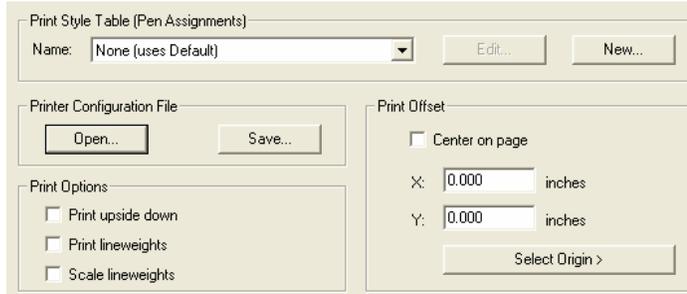
User Defined Scale: You can pick between inches and millimetres for the units. Printed Units will normally be set to 1 and Drawing units will be set to control the print scale.

### *Examples:*

inches: 1 = 50 would be 1 inch = 50 feet  
inches: 1 = 100 would be 1 inch = 100 feet

millimetres: 1 = 1 would be a ratio of 1:1000  
millimetres: 1 = 0.25 would be a ratio of 1:250

## Advanced tab



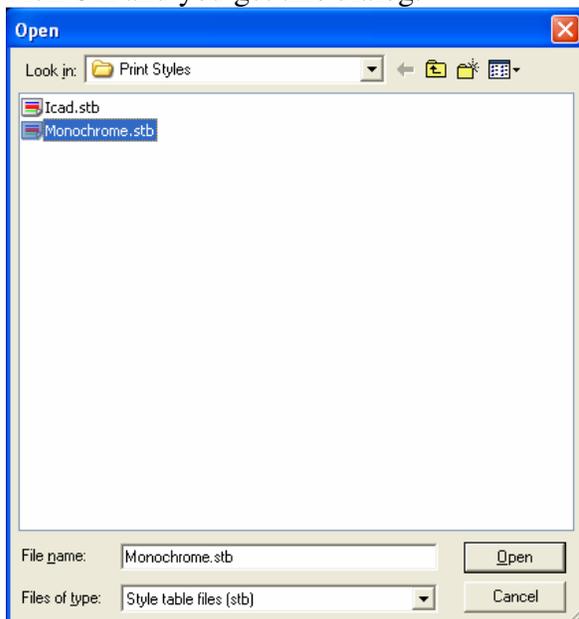
### **Print Style Table (Pen Assignments):**

**\*NOTE\*** If you are working in a drawing that was saved from MSCAD2004, then the default will be to use STB files. To change this you can set a variable in MSCAD2004 prior to saving the DWG file. You have to set PSTYLEPOLICY to 1 for CTB and set it to 0 for STB.

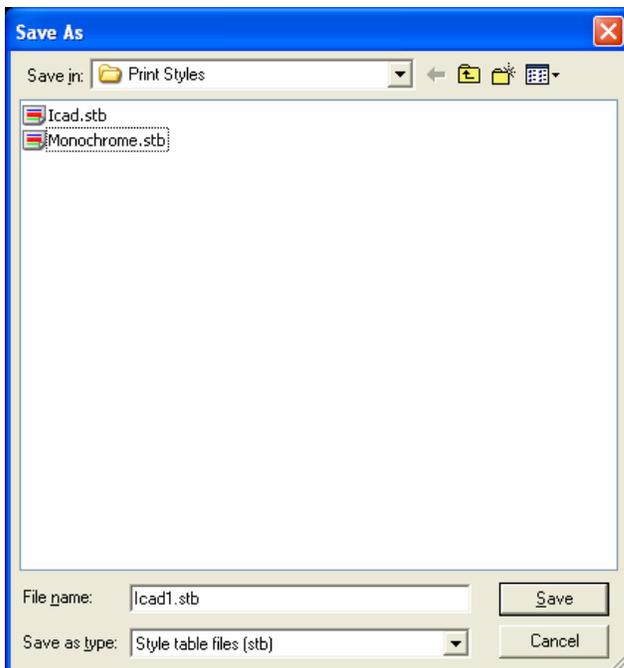
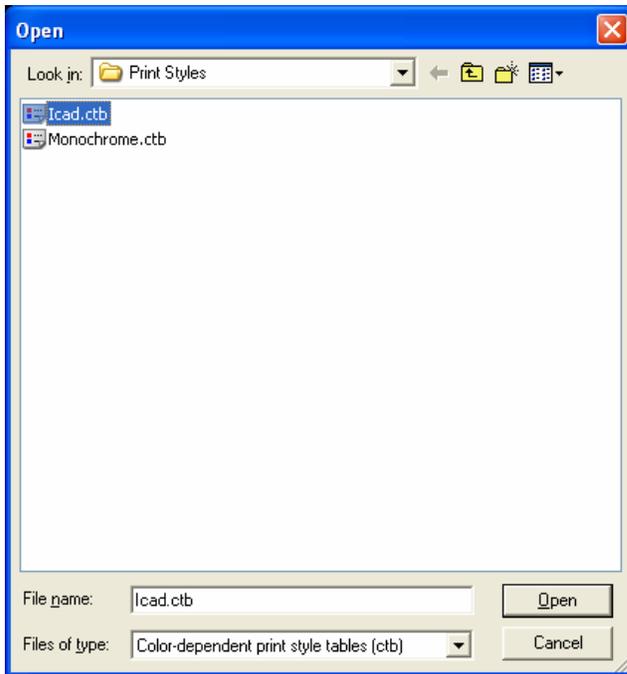
You can also type in a command in MSCAD2005 to convert the plotting routine to use STB instead of CTB. Type in CONVERTPSTYLES and you will see this dialog:

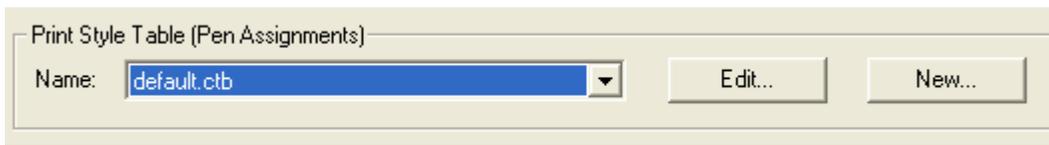


Pick OK and you get this dialog:



The CONVERTCTB command converts the CTB file into a STB format.





You can create either a named plot style table (STB) or a color-dependent plot style table (CTB). With a named plot style table, you can add and define plot styles as you like. A color-dependent plot style table creates 255 plot styles based on color.

By using color-dependent plot styles to control how objects are plotted, you ensure that all objects that share the same color are plotted the same way.

When a drawing uses color-dependent plot style tables, you cannot assign a plot style to individual objects or layers. Instead, to assign plot style properties to an object, you change the color of the object or layer.

You can assign color-dependent plot style tables to layouts. You can use several predefined color-dependent plot style tables, edit existing plot style tables, or create your own.

### Name

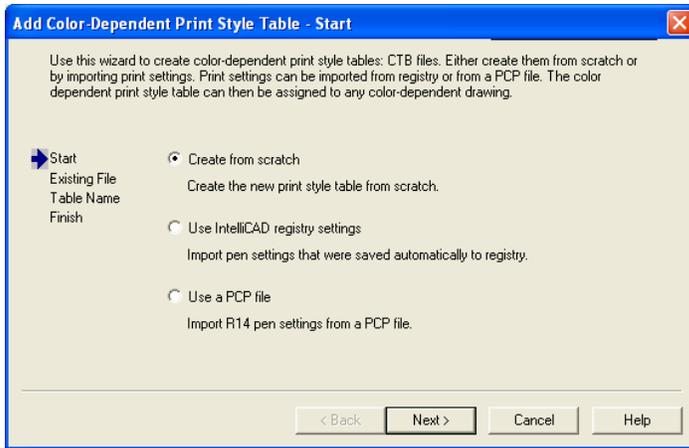
Choose from any saved styles.

### Edit

Takes you to the same stage as shown under the New button, with the dialog "Print Style Table Editor".

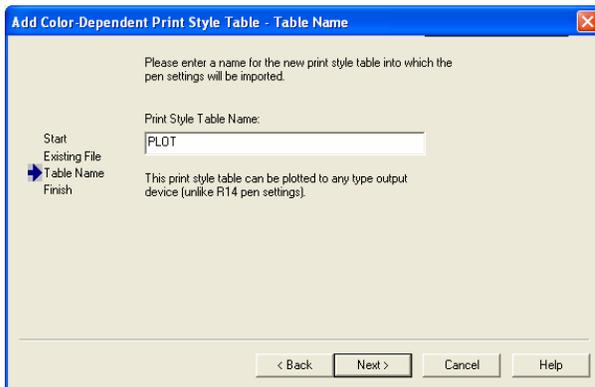
### New

Allows you to create a new CTB file in 1 of 3 ways.

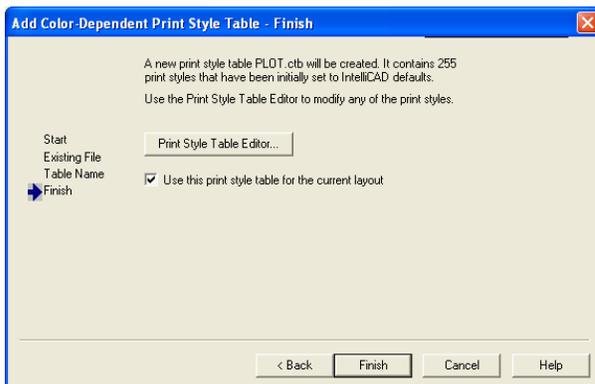


## 1) Create from Scratch:

Pick on Next and you get this dialog:



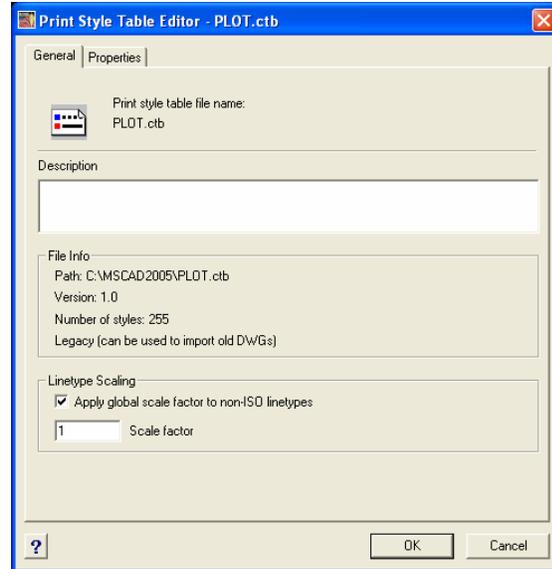
Give the CTB file a name and pick on Next:



You can then edit the table by picking on the Print Style Table Editor, or simply pick Finish to use the defaults. If you have the check mark for “Use this print style table for the current layout” turned on then it uses it for the current layout only. If not then it is simply being setup for future use.

## Print Style Table Editor

### General Tab:



### **Description:**

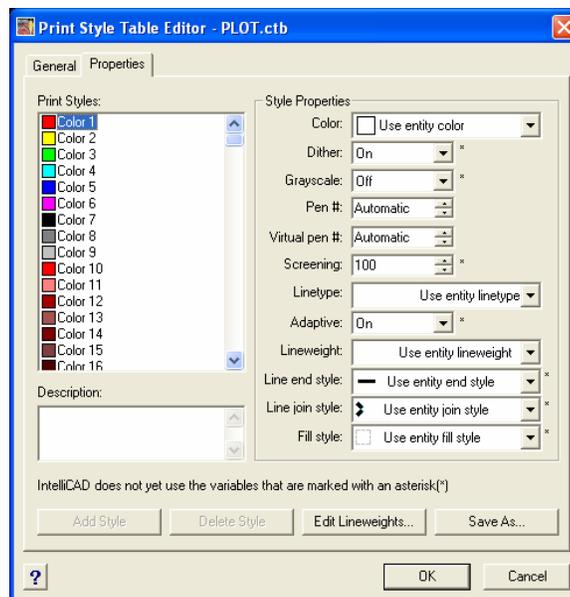
Allows you to add some information to allow you to know what this print style was used of created for.

### **File Info:**

Displays information about the print style.

**Linetype Scaling:** Scales all the non-ISO linetypes in the plot styles of objects controlled by this plot style table.

### Properties Tab:



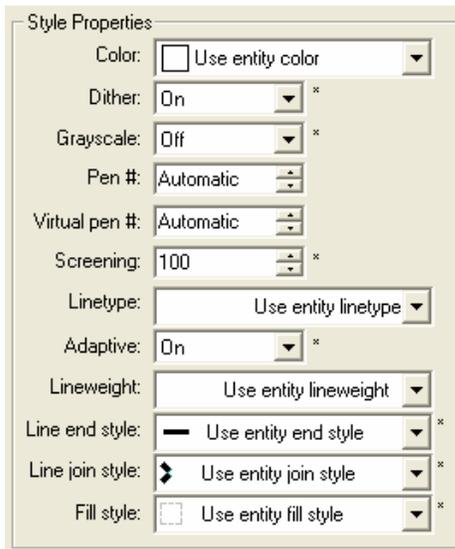
This tab has many settings to control how the colors will be used to plot the linework from your drawing. Anything with an \* beside it is currently not supported in this version. The fields are there for compatibility issues with AutoCAD (which can use the same CTB files for plotting)

### Print Styles:



There are 255 colors that can be setup individually. Pick on the color to edit, then the Style Properties options will be active.

### Style Properties:



The Style Properties are the same for CTB and STB files.

### Color:

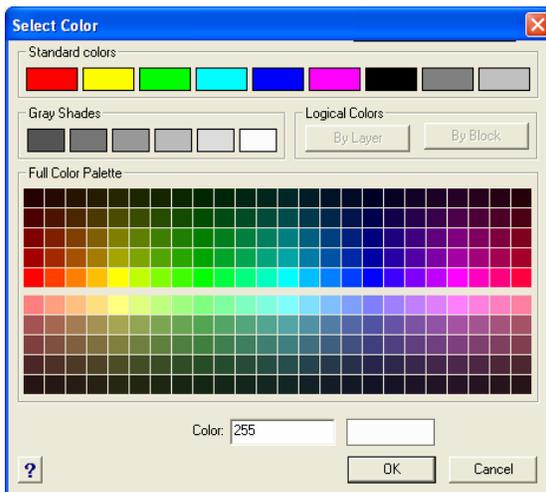
Picking on the little black arrow on the right pulls down the color options available.



Use Entity Color means the color selected and show in the drawing, will be used by default. So if you have something drawn in RED in the drawing, it will also plot in RED on paper, if you have a color printer/plotter.

Picking another specific color means you are changing the color to print to paper. So everything in RED on screen will not plot with whatever color you had selected.

Picking Other brings up the standard color dialog:



Which allows you to pick any color you wish. True Color is not supported at this time – added for compatibility with AutoCAD.

Dither: This option is not used by MSCAD2005, but is recognized and saved for compatibility with AutoCAD. Turn on dithering to print colors using dots and patterns, combining to create custom colors. Turn off dithering to print colors using only the ink colors available in your printer or plotter. Some printers or plotters do not support dithering, in which case, this setting is ignored.

Grey Scale: This option is not used by MSCAD2005, but is recognized and saved for compatibility with AutoCAD. Turn on grayscale to print colors in grayscale. Turn off grayscale to print colors normally. Some printers or plotters do not support converting colors to grayscale, in which case, this setting is ignored.

Pen #: Use pen numbers when your output device is a plotter with physical pens used for plotting your drawing. Automatic uses a color index to determine the pen width automatically (this is the default). Setting the pen number to 0 (zero) is the same as Automatic. Specific pen numbers range from 1 to 32 and determine the width of the pen used for the print style. The way in which pen numbers are assigned for plotting depends partly on your plotting device and its settings.

Virtual Pen #: Use virtual pen numbers when your printer does not use physical pens, such as a laser or inkjet printer. Automatic uses a color index to determine the virtual pen width automatically (this is the default). Setting the virtual pen number to 0 (zero) is the same as Automatic. Specific pen numbers range from 1 to 255 and determine the width of the virtual pen used for the print style. The way in which virtual pen numbers are assigned for printing depends partly on your printing device and its settings.

Screening: This option is not used by MSCAD2005, but is recognized and saved for compatibility with AutoCAD. Type or scroll to the screening value (0 to 100) for the selected print style. Lower values print with less ink, producing faded colors. Higher numbers print with more ink, producing more true colors. Dither must be On for screening to work.

Linetype: Allows you to pick a linetype to use for that color. Will over-ride the setting in the drawing. If set to Use Entity Linetype – then the drawing settings for the linetype will rule.

Adaptive: Not supported at this time – added for compatibility with AutoCAD

Lineweight: Allows you to set a lineweight for that color. Everything drawn with this color will use this lineweight. If set to Use Entity Lineweight – then the drawing settings for the lineweight will rule.

Line End Style: Not supported at this time – added for compatibility with AutoCAD

Line Join Style: Not supported at this time – added for compatibility with AutoCAD

Fill Style: Not supported at this time – added for compatibility with AutoCAD

## **2) Use IntelliCAD Registry Settings:**

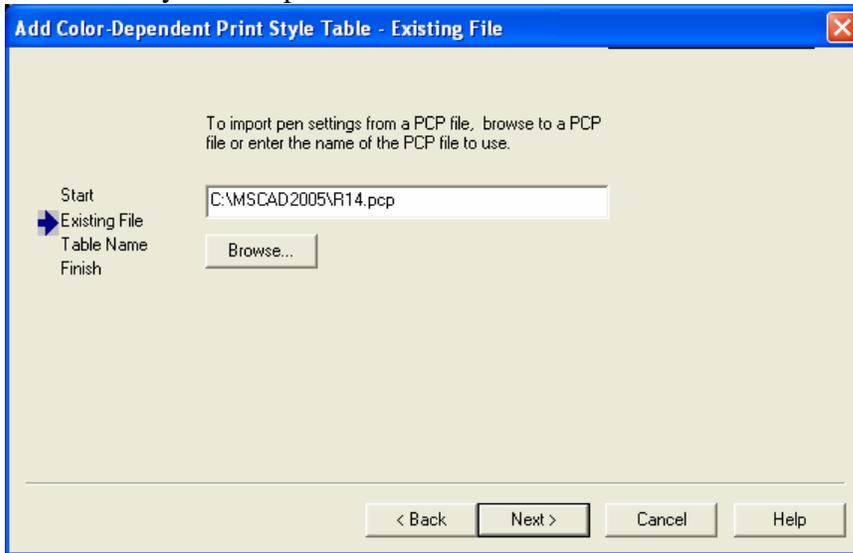
Follows the same steps as shown above, but starts with the last saved defaults from the registry instead of from scratch.

## **3) Use a PCP File:**

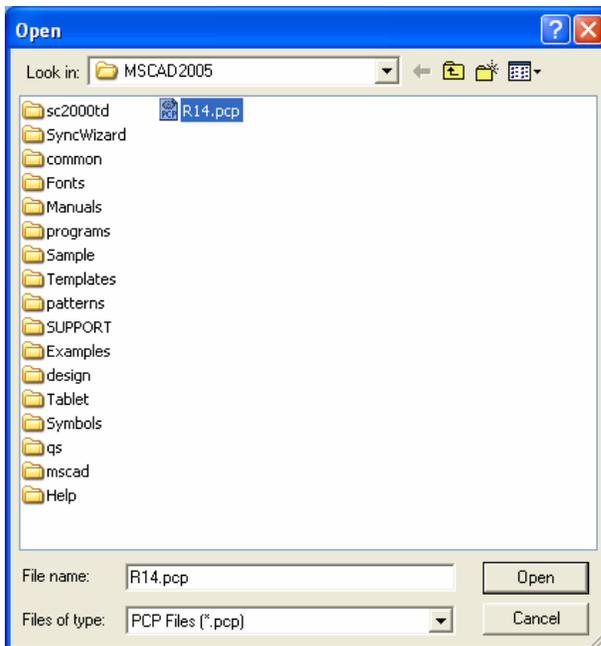
Printer configuration files store the printer information you create for specific drawings, which eliminates the need to completely reconfigure your print settings each time you print a drawing. MSCAD2005 supports the plot configuration (PCP) file format used by

AutoCAD. This feature makes it possible to use existing PCP files saved in AutoCAD, as well as to save your MSCAD2005 print configuration settings to a PCP format.

This allows you to import an AutoCAD R14 format file that contains pen settings.



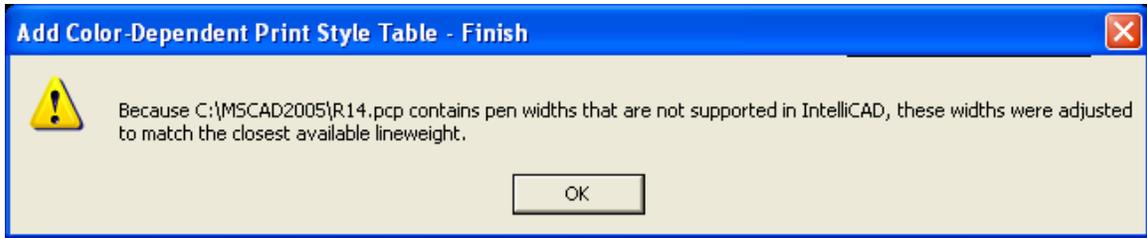
Pick on the Browse button to search and locate the desired file on your hard drive (if you have one).



Once you locate the file, pick on it and then pick on Open to return to the previous dialog.

Picking on Next will then take you to the same dialog as in the Create from Scratch method.

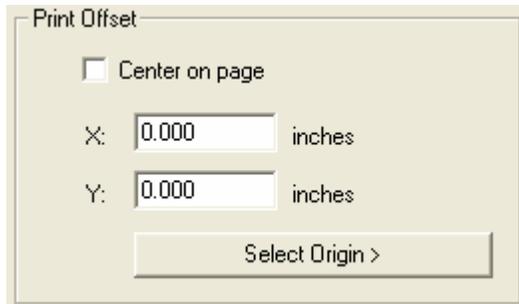
When you pick on the Print Style Table Editor button you may see the following dialog:



Simply pick ok to continue, as it will adjust what it needs to so the settings can be used.

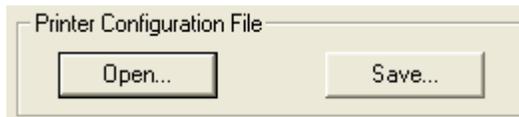
The remainder of the steps are the same as in the Create from Scratch method, above.

### **Print Offset:**

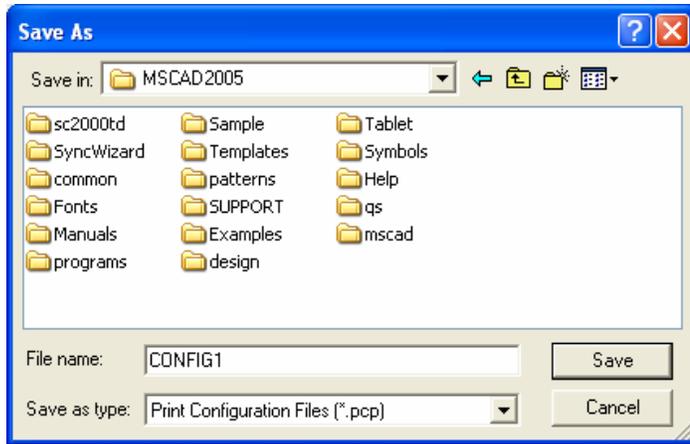


This option allows you to shift the print on the paper, if required. This may help you fit to pre-printed sheets so your drawing merges with the linework on the sheet. You can either Center it on the page or you can type in the X and Y or select the origin on screen by picking the Select Origin button.

### **Printer Configuration File:**



You have the ability to Save and Open print configuration files using the PCP format that AutoCAD has used for years.



This allows you to copy the configuration file to another machine, if desired.

### **Print Options:**



Prints upside-down: Some plotters require this to be set to orient the drawing on the paper correctly.

Use Lineweight: Each layout in your drawing can specify whether to print lineweights. If entities are assigned lineweights, you can control whether they print with the assigned lineweights. If you turn off lineweight printing, entities print with a default outline.

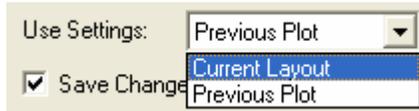
Scale Lineweight: prints lineweights in proportion to the scale you set on the Scale/View tab. If not selected, lineweights print at their assigned size without adjustments for the print scale. (A Layout tab must be active to scale lineweights.)

### *Top Portion of Print Dialog*



Layout Name: You are shown which drawing layout you are currently printing. If you have not worked in Layout Mode (also known as Paper Space) then the Layout Name will default to Model. This means you are currently working in Model Space (which a vast majority do automatically, and don't even realize it). If you have setup 1 or more Layout Sheets then the current sheet name is displayed.

Use Settings: You can choose between using the Current Layout, as you have just finished setting it up, or using the Previous Plot settings.



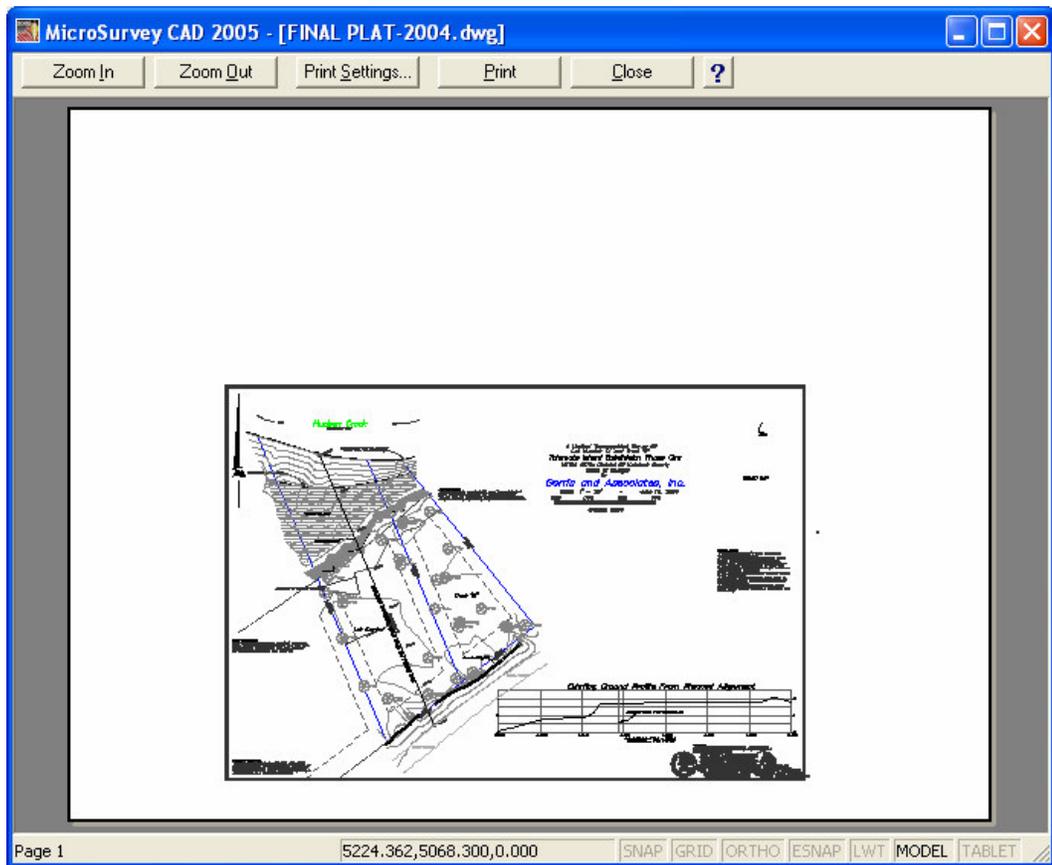
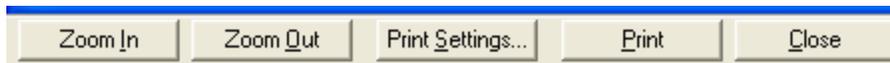
Save Changes to Layout: when turned on, will do as it says. If off then the changes you are making are for this drawing only.

*Bottom Portion of Print Dialog*



Apply: applies any changes you have made, so you can do a Print Preview.

Print Preview: Once in the Print Preview, you will see the drawing in the middle of the screen and buttons at the top left of the screen.



Zoom In: Allows you to zoom in on the print preview screen.

Zoom Out: Allows you to zoom out on the print preview screen.

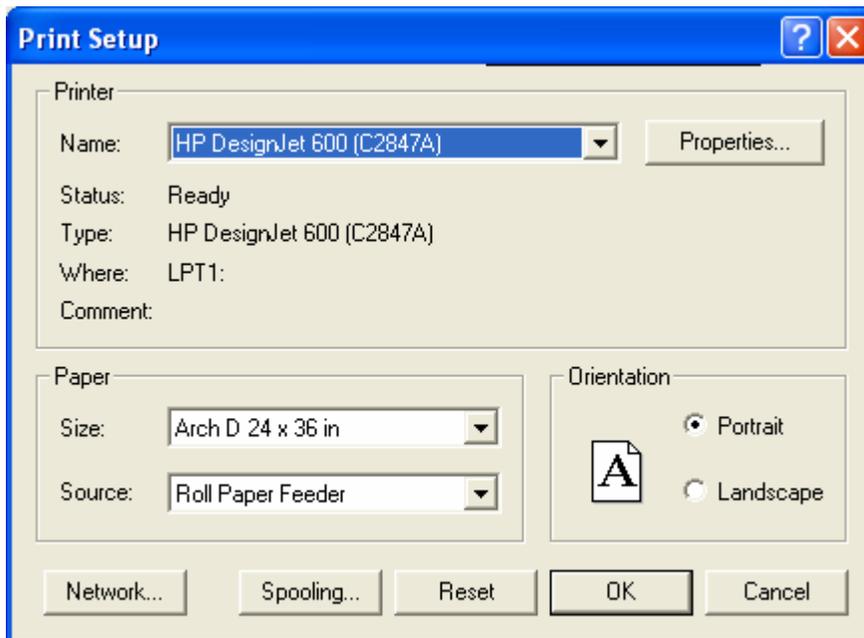
If you pick on the preview screen with your mouse you will see this icon  – this allows you to zoom between 3 different zoom levels, just keep picking to cycle between them.

Print Settings: Returns you to the previous dialog:

Print: Sends the job to the print driver.

Close: Exits from the Print Preview.

Print Setup: This is a link to your Printer selection options for Windows.



You can choose which printer/plotter to go to, the paper size, orientation, etc. Pick OK once you have what you need, and return to the Print dialog.

Reset: Takes all of the changes you have made and wipes them out – replacing them with our defaults. This allows you to get back to what we ship so you can redo your settings.

Print: Sends the job to the printer driver to go to paper (or a file, if the print driver is configured to do so)

Cancel: Takes you out of the command.