

Day 2 Seminar:

Appendix E

Handbook Supplement

This supplement is intended to provide basic instruction in CAD file manipulations that can be performed with AutoCAD or AutoCAD LT. Although the examples refer specifically to AutoCAD LT 2002, the demonstrated manipulations can also be performed in later versions of AutoCAD software [and corresponding functions are available in other CAD software such as *TurboCAD*; even more limited CAD utilities such as *Print2CAD* include many of these functions (see page hb23)]. All page number references preceded by "hb" in this PDF supplement refer to pages located in the 2016 - 2017 *Day 2 Seminar Handbook*.

by Michael E. Cope

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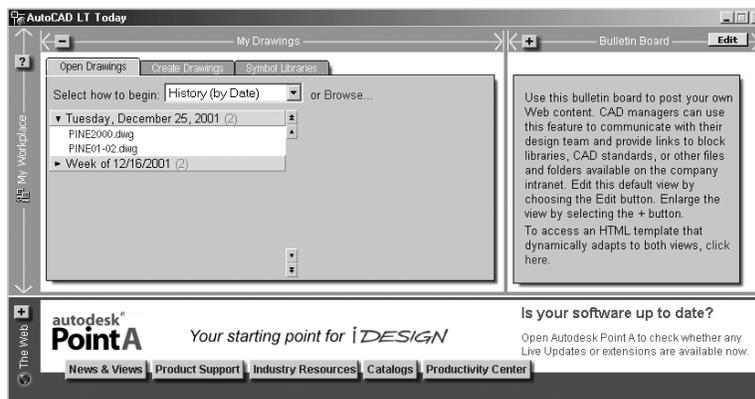
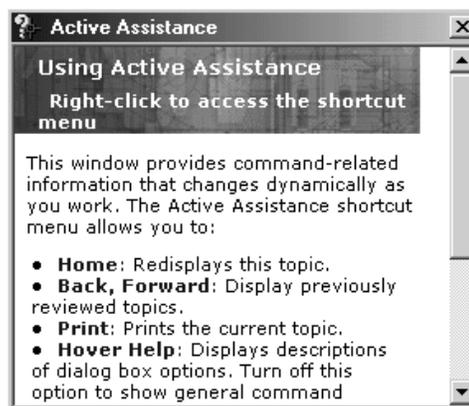
Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT Getting Started with AutoCAD LT

Back in the early 2000s, the following AutoCAD LT techniques were routinely required to overcome problems encountered with AutoCAD (DWG/DXF) files and maximize the amount of useable CAD data available to AGTEK (with current versions of AGTEK, however, these steps are typically no longer needed).

- ◆ To start AutoCAD LT, double click its shortcut icon on the Windows desktop or click on **Start > Programs > AutoCAD LT 2002 > AutoCAD LT 2002** from the Windows taskbar.

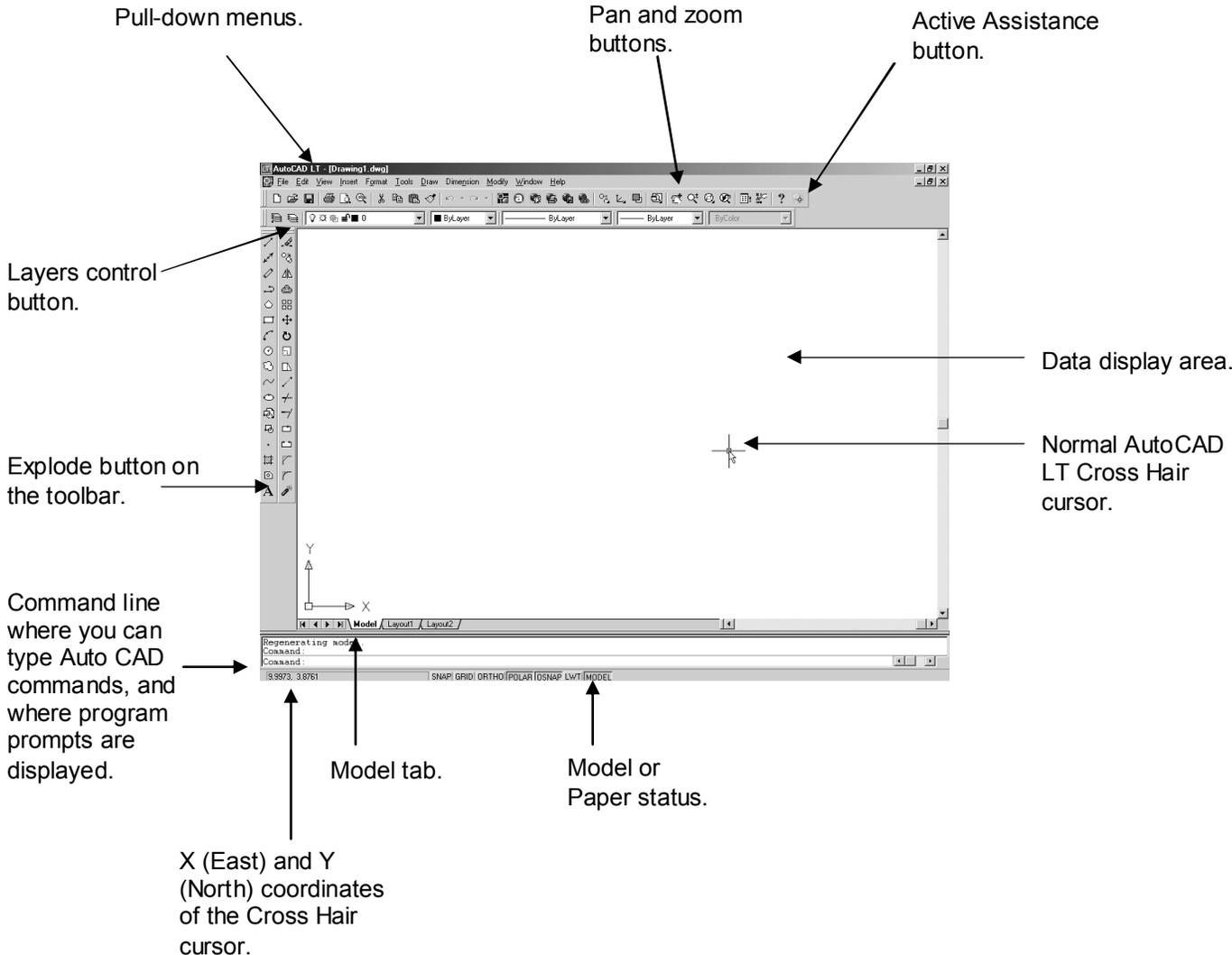
- By default, the Active Assistance dialog will pop up automatically with most program functions. It can be disabled for the current session by moving the Windows Arrow cursor over the Active Assistance icon on the Windows taskbar, **Right-clicking**, and selecting **Exit** (it can be restored by clicking the **Active Assistance** button at the upper-right of the screen).

- The AutoCAD LT Today dialog can be permanently disabled by selecting **Tools > Options** from the menu, clicking the **System** tab, and setting **Startup** to **Do not show a startup dialog** (it can be restored through the same procedure).



Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT Getting Started with AutoCAD LT (Cont.)

This is what the main AutoCAD LT screen looks like as configured out-of-the-box . . .



See next page for instructions on opening CAD files with AutoCAD LT.

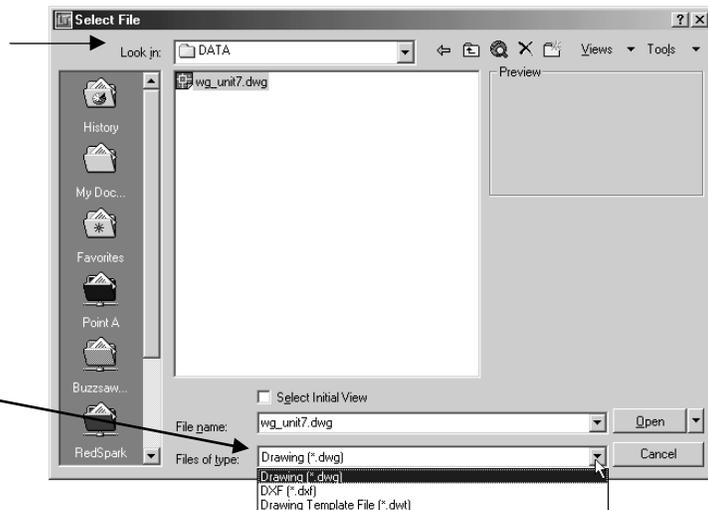
Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT Message Dialogs when Opening Files

To open a file with AutoCAD LT select **File > Open** from the menu . . .

Step 1: Choose the drive location and data folder of the file you want to open (if the file was e-mailed to you, it may be on your local hard drive or a network drive depending on where it was saved; if you received it on a disk or CD, choose the corresponding drive location).

Step 2: Identify the file type (.DWG or .DXF).

Step 3: Once the file is located, double click it to open it (or click it once, and then click on **Open**).



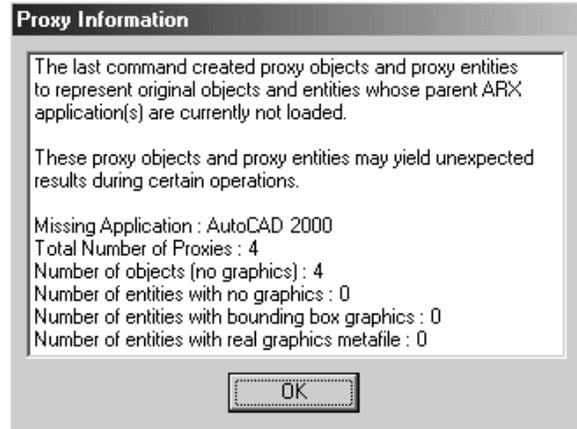
If you get an **incompatible version** message, you are not using the latest version of AutoCAD LT, you will not be able to open the file, and you have three choices:

- Make yourself a nuisance to the engineer by requesting the file be resent in a compatible older version—see page hb235.
- Upgrade to AutoCAD LT 2013/2014/2015/2016 (or get a free copy of DWG TrueView—see pages hb23, hb29).
- Send the file to AGTEK Support (see page hb187).

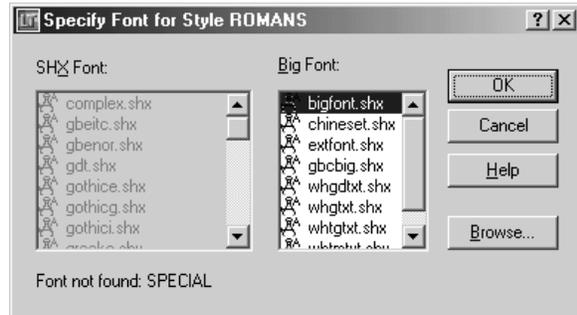


Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT Message Dialogs when Opening Files (Cont.)

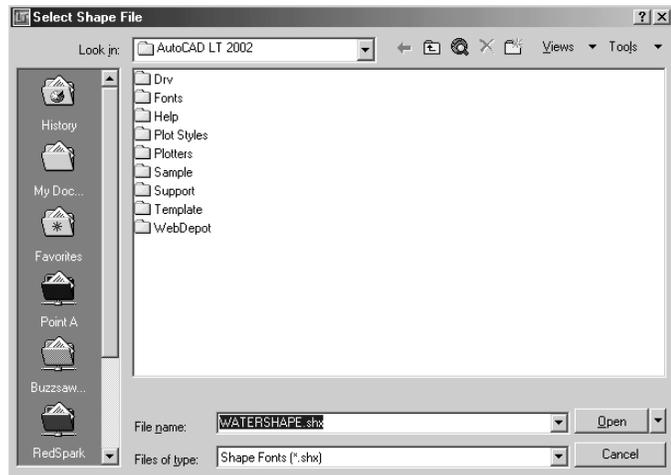
If you are using an older version of AutoCAD LT you may get a **Proxy Information** message that indicates a newer AutoCAD application is missing. Click **OK** to continue loading the file. Even though you can open this file, some newer AutoCAD objects were not translatable to the older file format in which they were saved. This is another reminder to upgrade to the latest version of AutoCAD LT. (See the Note about Object Enablers on the next page.)



If you get a **font not found** dialog, just click **OK** to continue.

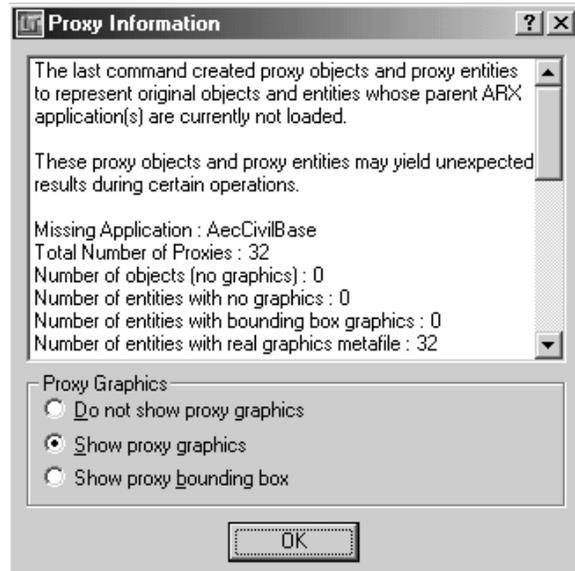


If you get a **Select Shape File** dialog, just click **Cancel** to continue.



Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT Message Dialogs when Opening Files (Cont.)

If you get a **Proxy Information** message that indicates AecCivilBase as a missing application, the file may contain AecCivil Contours that require special processing. Be sure the **Show proxy graphics** radio button is on, then click **OK** to continue loading the file (see AecCivil Contours on page 19).



Note: Downloading and installing Autodesk's free AutoCAD **Object Enablers** (see page hb23) may reduce the occurrence of Proxy Information warnings when opening DWG/DXF files with AutoCAD and AutoCAD LT. Object Enablers can be found on Autodesk's website but direct links are available on the Resources page at www.EarthworkSoftwareServices.com (see page hb29 of the *Day 2 Seminar Handbook*).

Tip: After any message dialogs have displayed and the file has finished loading, use the Purge command to delete any empty layers from the file. To use the command, type **Purge** at the command line and press the **Enter** key. (Purge will only delete empty layers, so you don't have to worry about losing any valuable data.)

Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT Saving Changes to CAD Files

Most of the troubleshooting techniques in this chapter require changes to the CAD file that must be saved before you exit AutoCAD LT.

Typically, it is better not to save these changes back to the original CAD file (if you make a mistake, you may need the original CAD file again).

The following procedure is a good way to save changes without affecting the original file . . .

Step 1: After completing any required modifications (as discussed in the following sections of this appendix), Select **File > Save As** from the menu (**File > Export** if you are using a version earlier than AutoCAD LT 2000/2002).

Step 2: Select a save destination (such as C:\Agtek\Data).

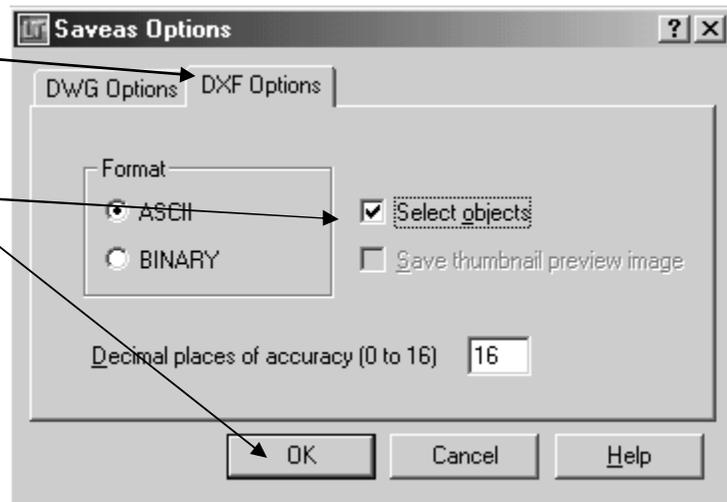
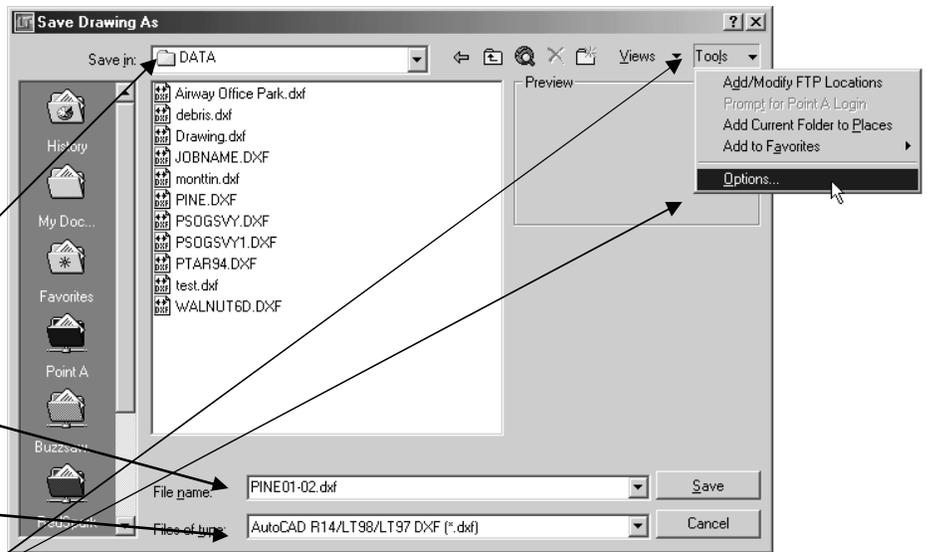
Step 3: Name the DXF file.

Step 4: Select **AutoCAD LT 2010 (or earlier) DXF** for the file type.

Step 5: Click on the **Tools** button and select **Options**.

Step 6: Click the **DXF Options** tab.

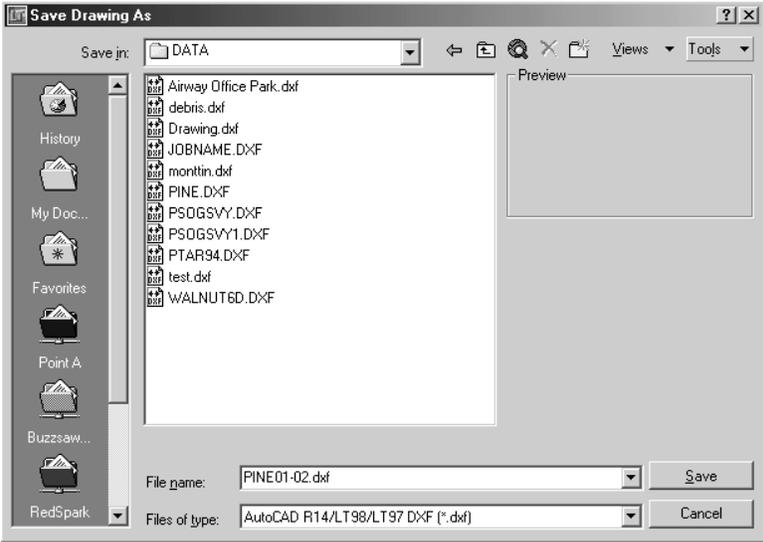
Step 7: Check the **Select Objects** option (don't change the format from ASCII or the number of decimal places from



Tip: These steps will save your changes to a DXF file. If you prefer to save your changes to a DWG file, you can use the **WBLOCK** command and a similar set of steps (see your AutoCAD LT users guide for details).

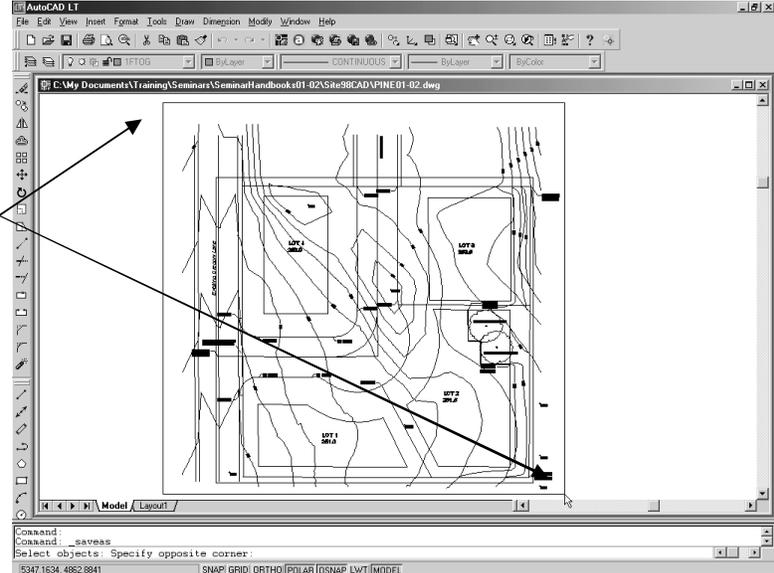
Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT Saving Changes to CAD Files (Cont.)

Step 8: Click on **Save**.



Step 9: Window select around all visible data desired for export by **Left-Clicking** once in the upper left corner, then once in the lower right corner, making sure all data is *completely* within the selection window.

[If you window select from right to left, data partially within the selection window is also selected.]



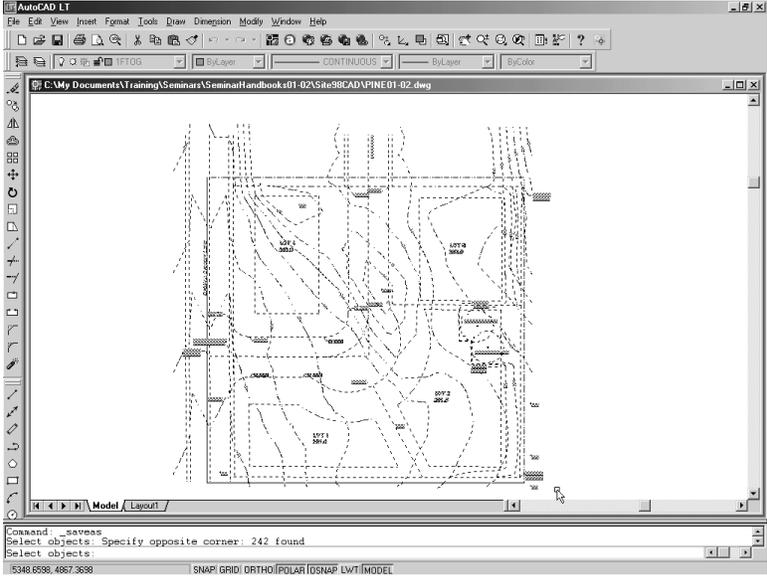
Day 2 Handbook - Appendix E Supplement

Manipulating CAD Files with AutoCAD LT

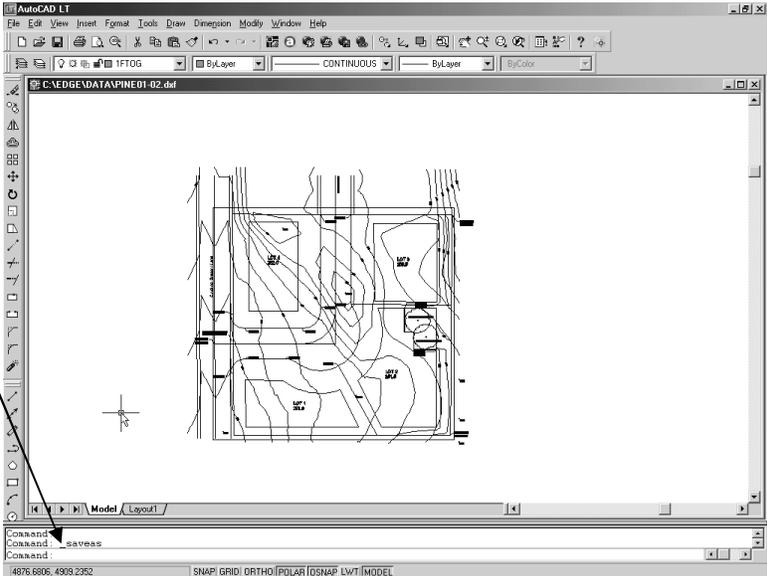
Saving Changes to CAD Files (Cont.)

Step 10: The selected data is highlighted and the program prompts to select more objects. **Right** click or press **Enter** to indicate the selection is complete.

[Note: The main advantage of using window select in this process is to exclude data on any frozen layers (see pages 11-12) and any extraneous data that may have horizontal coordinates considerably different from those of the actual data that you wish to use.]



Step 11: The export process is complete when the blank Command prompt returns to the lower command line. You are done with AutoCAD LT, so select **File > Exit** from the menu (**DO NOT** save the file when prompted).



Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT Missing Data?

Once a file is loaded in AutoCAD LT, compare the screen to the paper grading plan. Sometimes data shown on the paper plan does not appear on the screen . . .

- ◆ The data may actually be excluded from the file. It is not uncommon for data such as existing ground contours to be contained in a separate CAD file. If you do not already have the separate file containing the missing data, you should contact the engineer and request it.
- ◆ The data may be part of the current file but it may be in a form that AutoCAD LT cannot process and display. A good example of this is proxy information related to AecCivil Contours (see page 19).
- ◆ If the screen looks empty, select **View > Zoom > Extents** from the menu, then look carefully for two tiny “specks” on opposite edges of the screen (the file may contain two sets of data separated by millions of feet on the horizontal and each speck can include huge amounts of data). If you see the specks, zoom in on them for a closer look.
- ◆ Finally, the data may be part of the current file but it is on a layer that is **Off** and/or **Frozen**. Such data can be identified as follows:

Step 1: Select **Format > Layer** from the menu or click on the **Layers** button in the upper-left corner of the AutoCAD LT screen to access the **Layer Properties** dialog (see next page).



Layers button

Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT Missing Data? (Cont.)

The first column of the Layers Properties dialog lists the **Name** of each layer in the file.

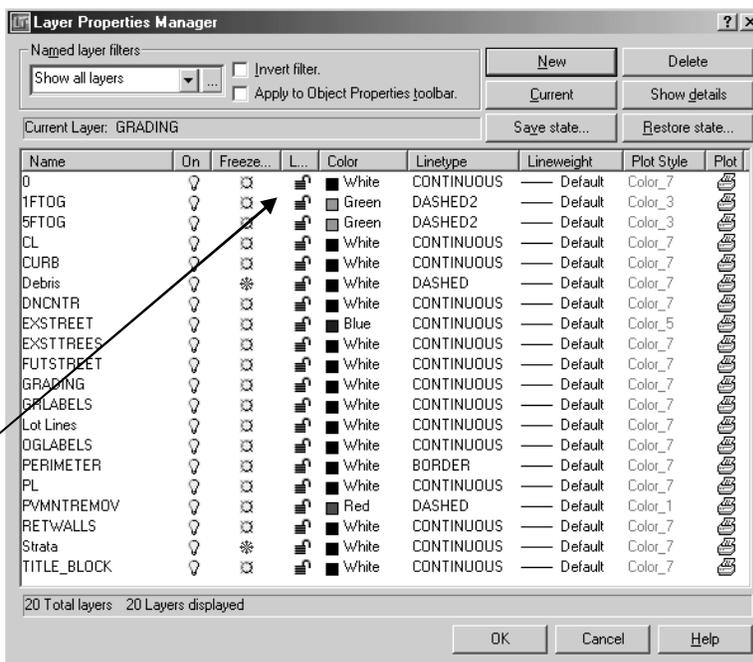
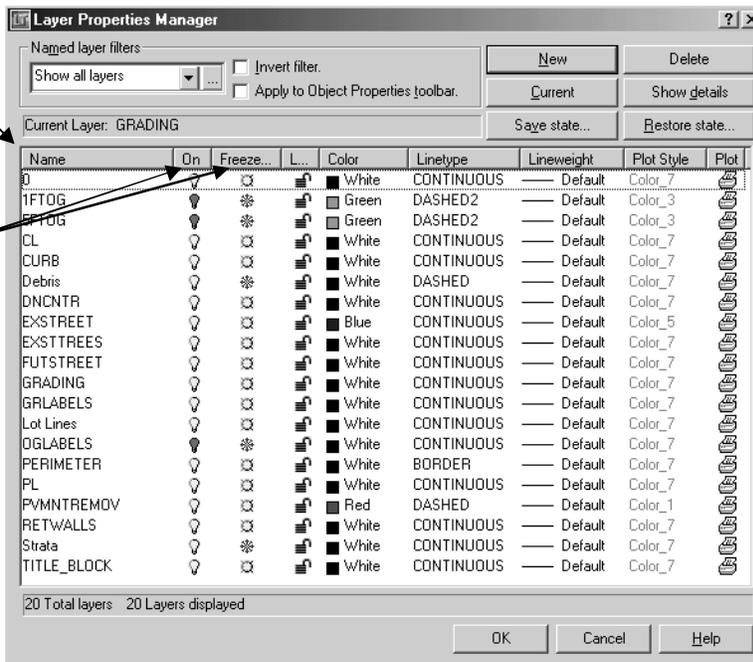
The second and third columns indicate the **On/Off** and **Freeze/Thaw** status of each layer. To be visible, a layer must be **On** (yellow light bulb) and **Thawed** (yellow sun).

Step 2: Look at the names of layers that are **Off** (darkened light bulb) and/or **Frozen** (snowflake) to identify the layers that may contain the missing data (**1FTOG**, **5FTOG**, and **OGLABELS** in this example).

Step 3: Turn **ON** the identified layers by clicking on their darkened light bulbs, and **Thaw** the identified layers by clicking on their snowflakes.

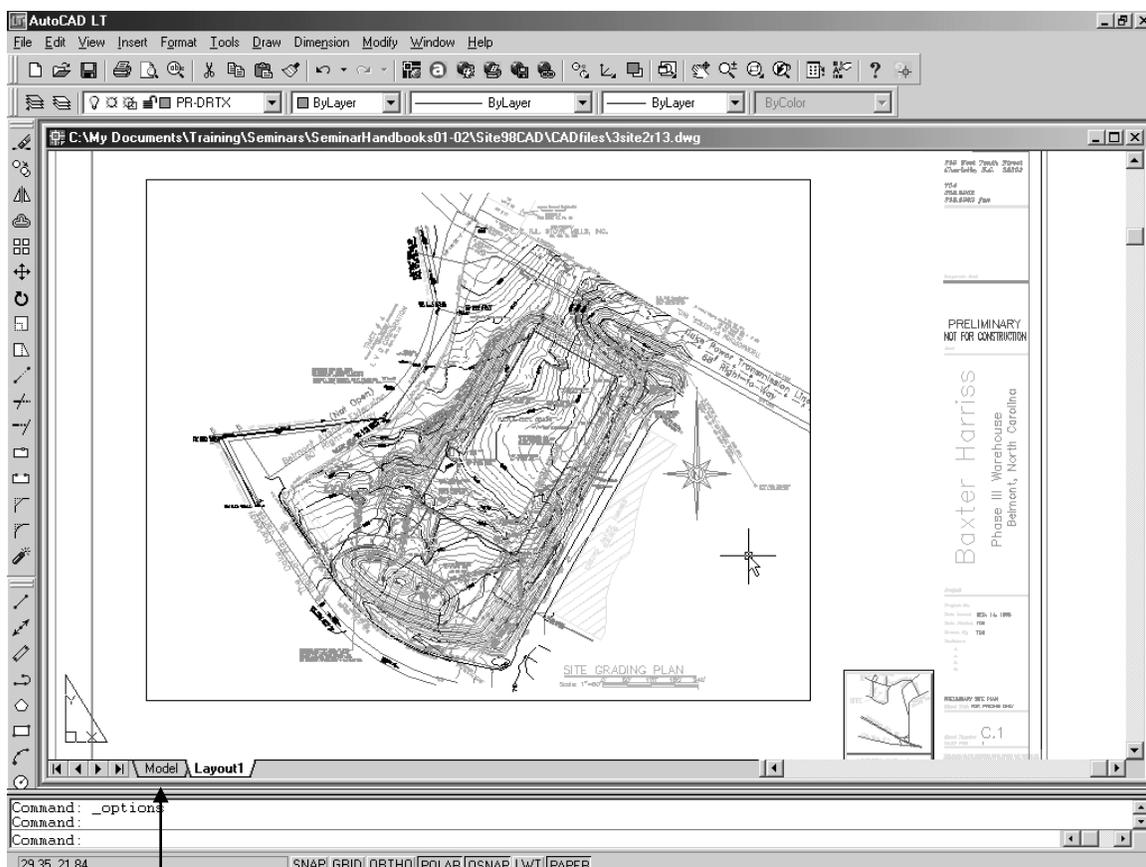
Step 4: Click on **OK** to regenerate the plan view and see the previously missing data. Now that you can see the data, you can troubleshoot it if necessary.

*[Note: The fourth column indicates **Locked** (closed padlock) or **Unlocked** (open padlock) layer status. A **Locked** layer cannot be modified until it is **Unlocked**. Some of the problems described next require layers to be modified by **Exploding** them—those layers must be unlocked first. Incidentally, importing CAD files with Blocks (page 14) on Locked layers may crash AGTEK--using a CAD program to Unlock all layers will solve the problem.]*



Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT Paper Space vs. Model Space

There are two primary working environments in AutoCAD: Model Space and Paper Space. Model Space is where drafting and design takes place, while Paper Space is where layout for the paper plot takes place. The only problem with Paper Space is that it will not allow the selection of individual objects for troubleshooting purposes. To avoid this problem, switch to Model Space . . .



The best way to switch from Paper Space to Model Space is to click the Model tab here.

You can also click here to toggle between Paper Space and Model Space. [In AutoCAD LT 98 or earlier versions, select **View > Model Space [Tiled]** from the menu.]

Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT Block References

Early versions of AGTEK could not import Block References unless they were first exploded . . .

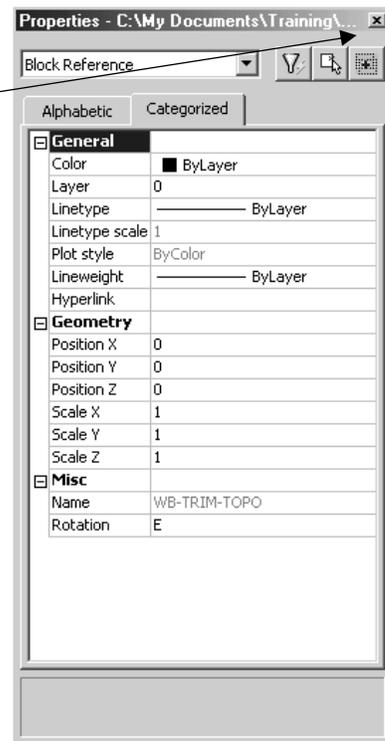
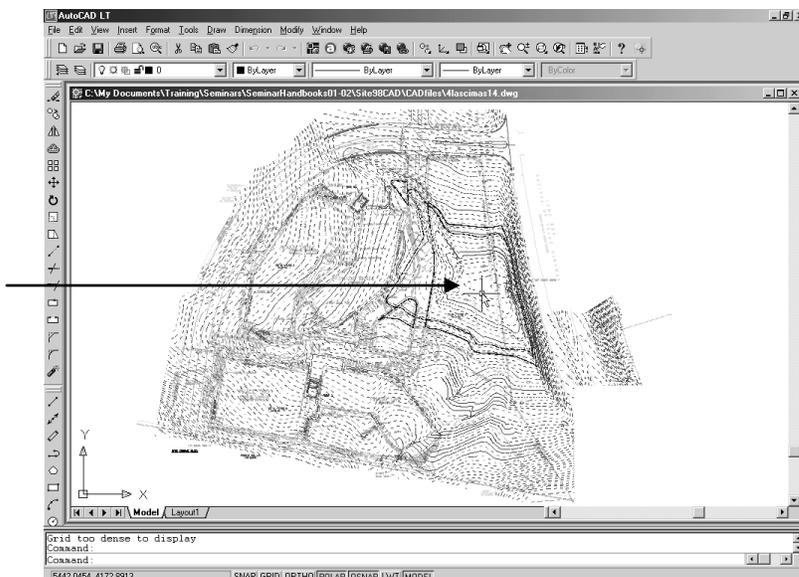
FYI: A Block is a CAD object that contains other CAD objects. Exploding a Block (see next page) frees the individual objects within the Block.

Step 1: Press the **Esc** key several times to clear any previous selection, and then click on the object in question (any existing contour in this example). If each point on the contour is highlighted by a small blue square, that object is “AGTEK friendly” and it is not a Block Reference. If (as in this example) the clicked contour and every other contour are highlighted by dashed lines, there may be a problem.

Step 2: Identify the selected object by **Right-clicking** and selecting **Properties** from the pop-up menu (or type **LIST** or **LS** at the command line).

Step 3: The Properties dialog indicates that the object is a Block Reference.

Step 4: Close the Properties dialog by clicking the **X** button.



Note: AGTEK automatically explodes Block References as the CAD file is being read. To minimize any problems, place all related CAD files in the same AGTEK job folder before importing them in AGTEK. Some Block configurations, such as Blocks in External Reference files (see page 16), Blocks on Locked layers (page 12), or *nested* Blocks (Blocks within Blocks), can present problems. Troubleshooting steps for Blocks include using a CAD program to Unlock layers, explode Blocks (see next page) and possibly convert the CAD file to an earlier version file, then re-import the modified CAD file in AGTEK (DWG TrueView—see pages hb23, hb29—can be used for the file conversion but it does not include an explode function). If that fails, send the problem CAD file to AGTEK support—see page hb187—or plot the CAD file to PDF (with DWG TrueView) and import the resulting vector PDF file in AGTEK per the steps starting on page hb34—the PDF accuracy will often be very good (see page hb22).

Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT Block References (Cont.)

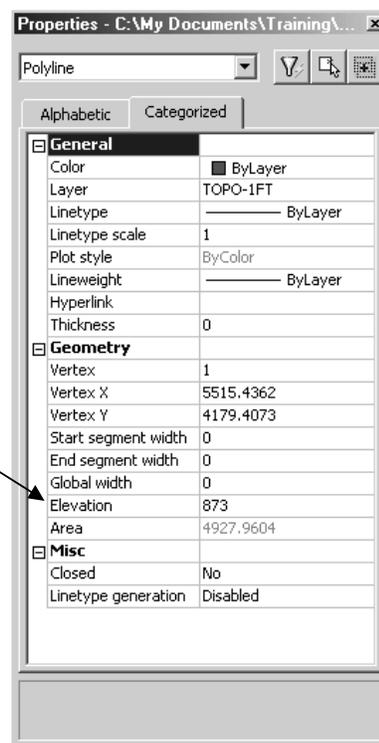
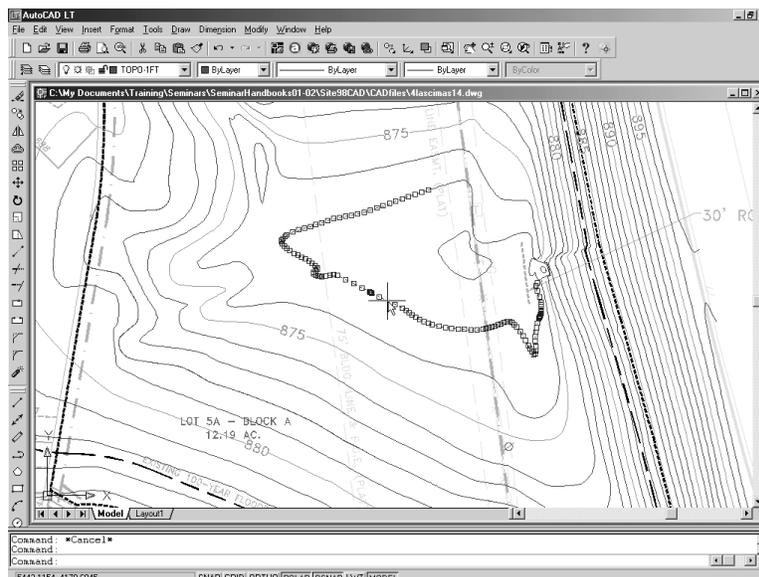
Step 5: Explode the selected block by selecting **Modify > Explode** from the menu or by clicking on the **Firecracker** toolbar button.

Step 6: Click on a contour now (zoomed here) and it should highlight with blue squares.

[If a block will not explode, verify that it is not on a Locked layer—see page 12. If the layer is not locked, try exploding it a second time—sometimes blocks contain other blocks (nested Blocks).]

Step 7: **Right-click** and select **Properties** from the pop-up menu. This contour is now a 3D polyline.

Step 8: After Exploding the block, save your modifications per page 8 and try re-importing it in AGTEK.



Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT External References

Early versions of AGTEK required the following manipulations for
External References . . .

FYI: An External Reference (or Xref) links one DWG file (the Xref file) to another DWG file (the Master or Base file). The Xref objects are visible in the Master/Base file, but the Xref objects are not physically in the Master/Base file.

Step 1: Press the **Esc** key several times to clear any previous selection, and then click on the object in question (any existing contour in this example). If each point on the contour is highlighted by a small blue square, that object is "AGTEK friendly" and it is not an External Reference. If (as in this example) the clicked contour and every other contour are highlighted by dashed lines, there may be a problem.

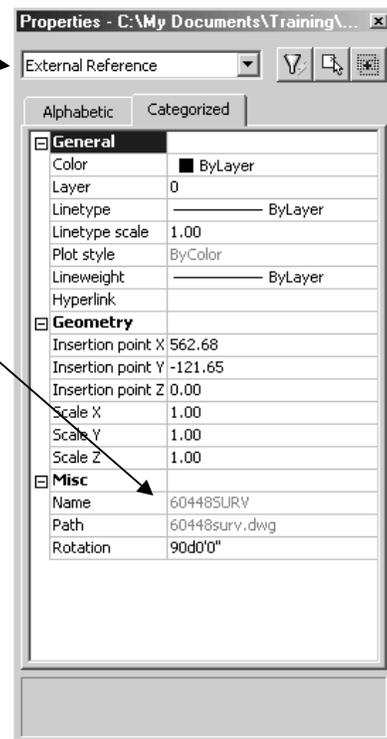
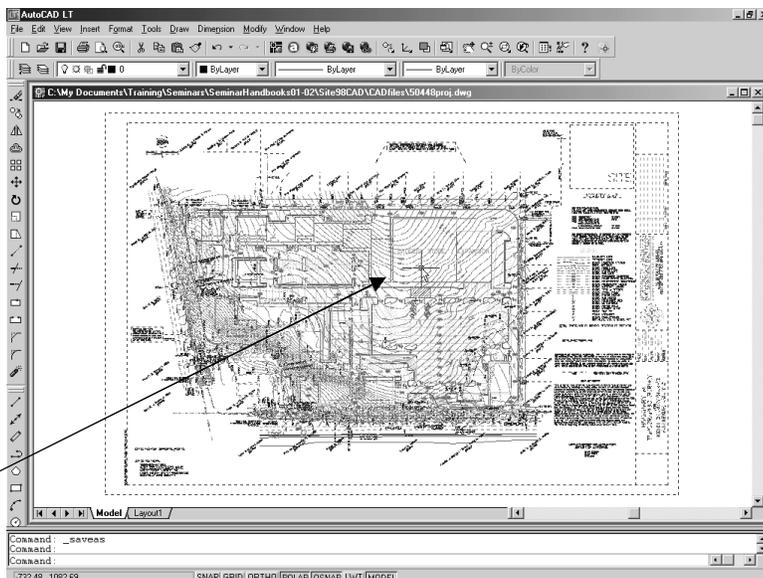
Step 2: Identify the selected object by **Right-clicking** and selecting **Properties** from the pop-up menu (or type **LIST** or **LS** at the command line).

Step 3: The Properties dialog indicates that the object is an External Reference (note the name of the External Reference file).

Step 4: Close the Properties dialog by clicking the **X** button.

*[Note: If the files are on a CD/DVD they may need to be copied to your hard drive and opened from there. After copying the files, you may also need to disable the Read Only attribute—in Windows Explorer, **Right click** the file and select **Properties**, then **Uncheck** the **Read Only** attribute and click **OK**.]*

Note: As long as all related CAD files are located in the same AGTEK job folder, AGTEK should read most External Reference files without problem; however, AGTEK may have problems with External References that contain blocks (see page 14) or with other issues (page hb59). If so, a potential solution is to *bind* the External References and (if necessary) *explode* the resulting block (see page 15 above and pages hb236, hb238 for examples) and possibly convert the modified file to an earlier version (with a CAD program or with DWG TrueView—see pages hb23, hb29), then re-import the modified Base CAD file in AGTEK. If that fails, send the Base and External Reference CAD files to AGTEK support and they will fix it for you—see page hb187—or plot the CAD file to PDF (with DWG TrueView, hb239) and import the resulting vector PDF file in AGTEK per the steps starting on page hb34—the PDF accuracy will normally be very good (see page hb22).

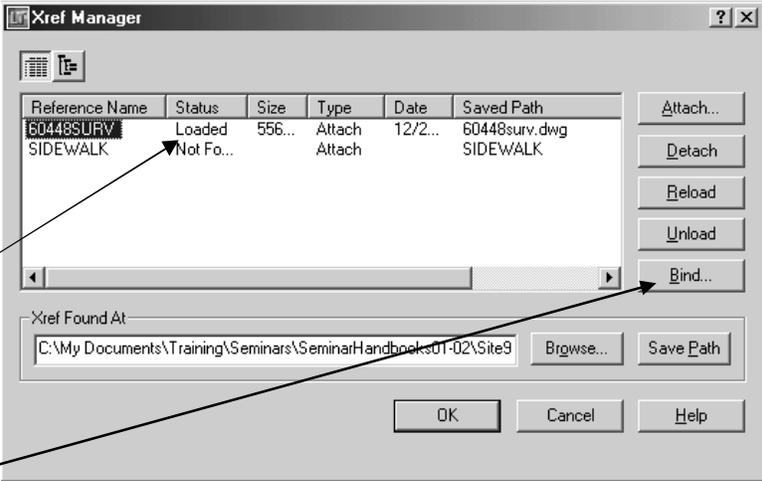


Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT External References (Cont.)

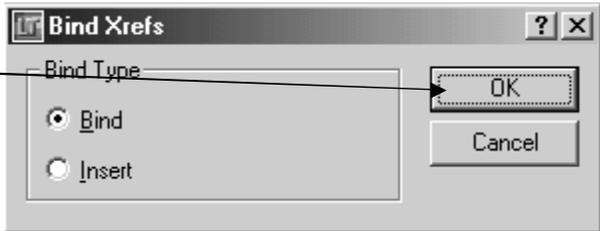
Step 5: Select **Insert > Xref Manager** from the menu or type **XREF** at the command line.

Step 6: Check the Status of the file name identified in Step 3 above. If it is **Loaded**, you have the file and can proceed. If it is **Not Found**, you do not have it and should request it from the engineer.

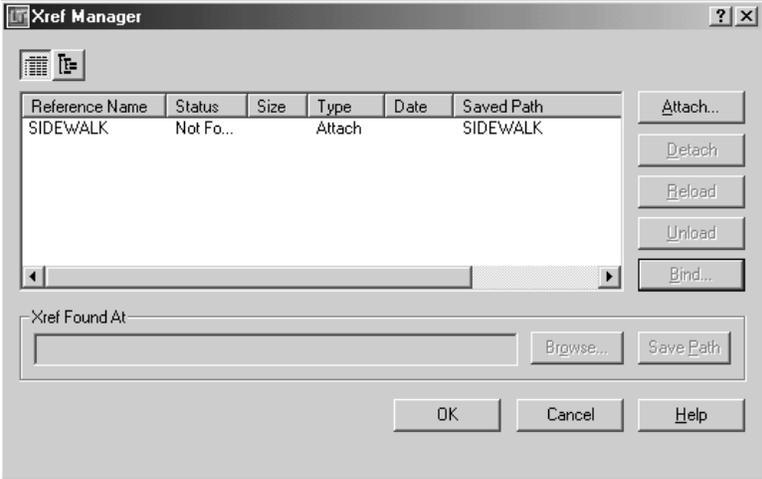
Step 7: Click on the file name (**60448SURV** in this case), and then click on **Bind**.



Step 8: Select the **Bind** option and click **OK**.



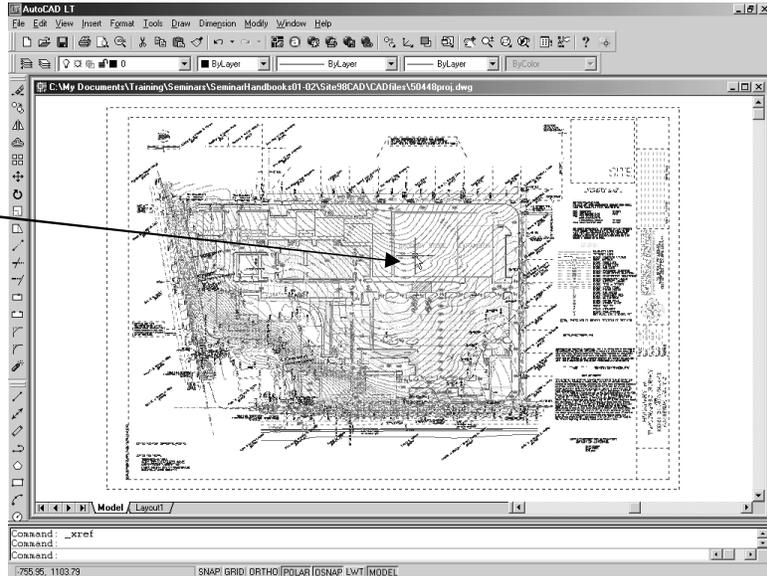
Step 9: The file is no longer listed in the Xref Manager because it is now physically part of the current file. Click on **OK**.



Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT External References (Cont.)

Step 10: Press the **Esc** key several times to clear any previous selection, and then click on an existing contour. The clicked contour and every other contour are again highlighted by turning to dashed lines, so there may still be a problem.

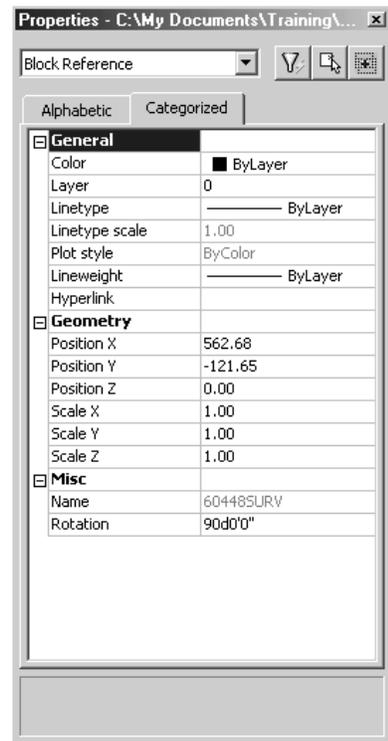
Step 11: Identify the selected object by **Right**-clicking and selecting **Properties** from the pop-up menu (or type **LIST** or **LS** at the command line).



Step 12: The Properties dialog indicates that the object is a Block Reference.

Step 13: **Explode** the block. Close the Properties dialog by clicking the **X** button, pick the block, and then select **Modify > Explode** from the menu.

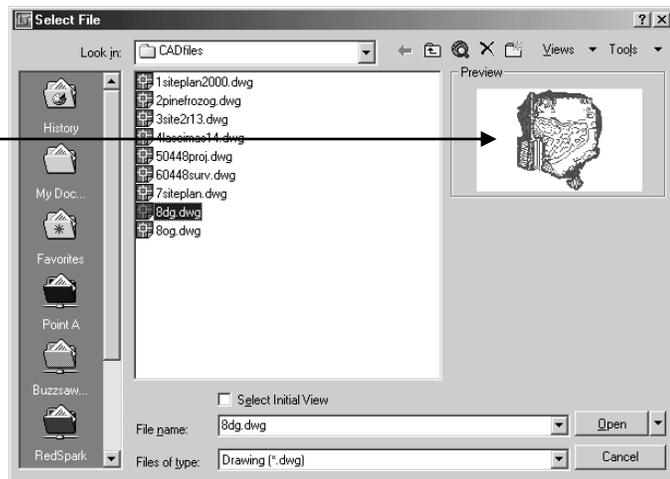
Step 14: After Exploding the block, save your modifications per page 8 and re-import the modified CAD file in AGTEK.



Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT AecCivil Contours

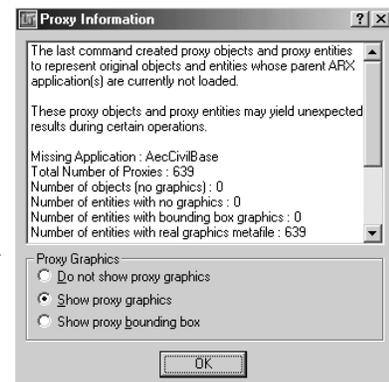
Various AutoCAD add-on applications can be used by designers to produce site grading plans (e.g., Civil 3D). These add-on applications produce custom CAD objects that AutoCAD LT does not recognize and which are replaced by proxy graphics when the file is opened. Depending on how the proxy graphics were saved, AGTEK may not be able to read them (always ask the engineer to “enable and save proxy graphics” in the furnished CAD files) . . .

Step 1: In the file open dialog, click on the CAD file you want to open and take note of the **Preview** image (are contours displayed in it?). Click on **Open**.



Step 2: A Proxy Information message will be displayed if the file contains proxy objects; be sure the **Show proxy graphics** option is selected, and then click **OK**.

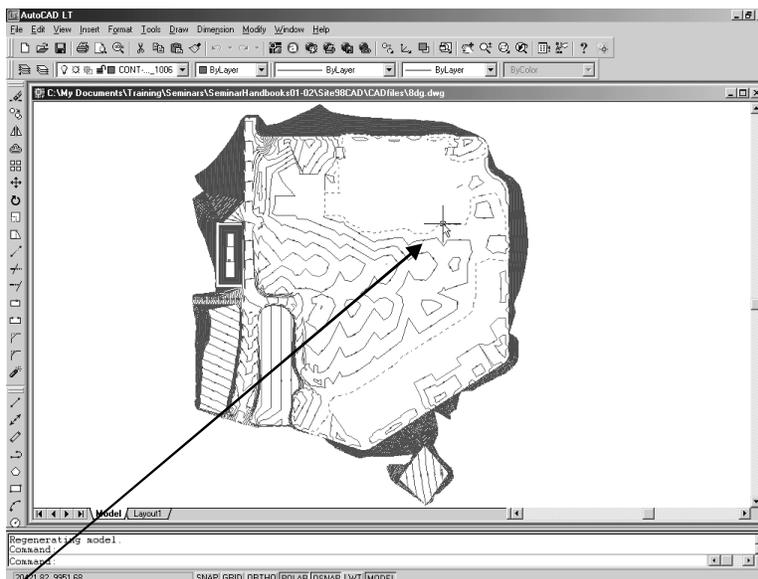
[The proxy message is the first hint that the file may contain AecCivil Contours.]



Tip: Depending on how they are saved in the CAD file, AGTEK can read AecCivil contours, but it can be helpful to install the free **Civil Object Enablers** for your copy of AutoCAD (see Note on page 7). If you have problems with AecCivil contours from any version of AutoCAD, you can try using AutoCAD LT to perform the steps beginning on this page or (better yet) send the CAD file to AGTEK Tech Support or ask the designer to convert the AecCivil contours to Polylines for you).

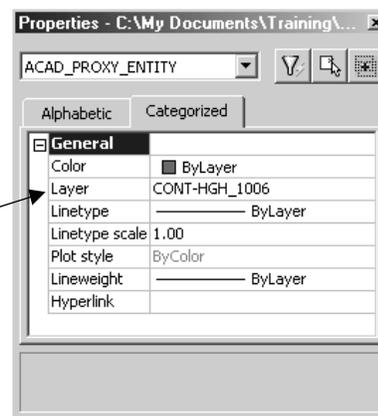
Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT AecCivil Contours (Cont.)

*[If a proxy objects message was displayed when the file was opening but you do not see any contours after the file is opened (and the contours are not on a layer that is **Off** and/or **Frozen**) and/or you see unexpected rectangles on the screen, you may be unable to use the AecCivil Contours in this file. If so, Ask the engineer to convert the contours to polylines and resend the file to you, or send the file to AGTEK's support department and they will do the conversion for you—see pages hb187-188.]*

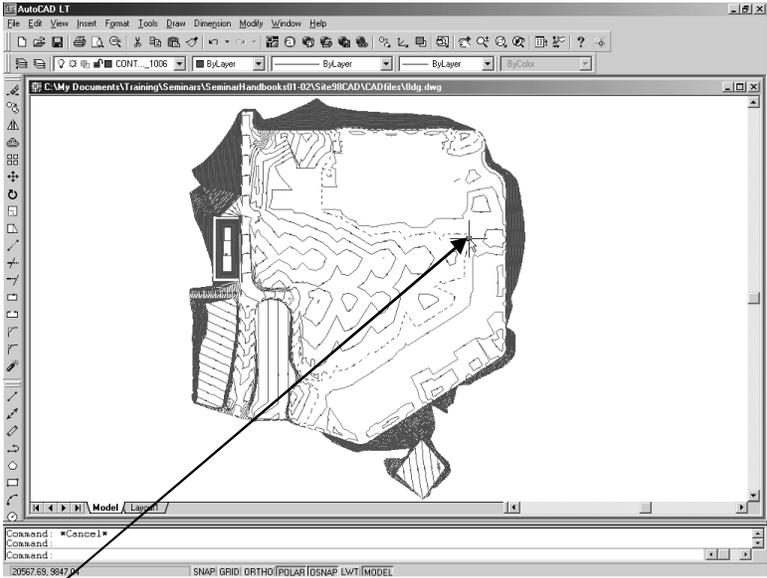


Step 3: If you see the contours after the file is opened, click on one contour. If only the clicked contour highlights as a dashed line (with no blue squares), it is probably an AecCivil Contour.

Step 4: With the contour still selected, **Right-click** and select **Properties** from the pop-up menu. The Properties dialog confirms the proxy status of the contour (note the layer name for future reference).

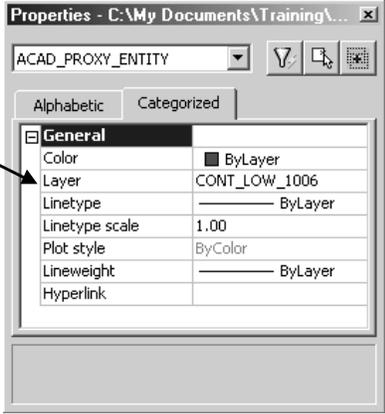


Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT AecCivil Contours (Cont.)



Step 5: Contours are normally drafted on two layers, one of which is a labeled index layer (typically the 2-, 5-, or 10-foot contours). So repeat Steps 3 and 4 above until you have identified and checked the second contour layer.

*[Remember to press **Esc** several times before clicking on the next contour.]*

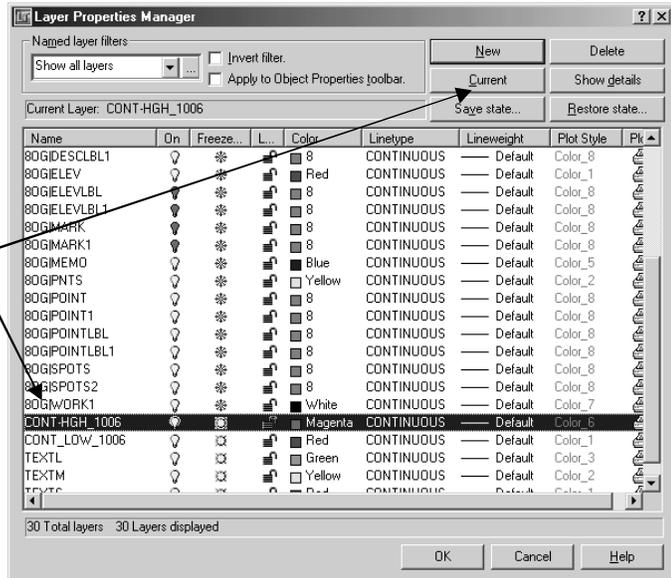


Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT AecCivil Contours (Cont.)

Two layers of proposed contours have been identified as containing AecCivil Contours. These layers will need to be exploded before they can be used in your AGTEK program. Each contour is a separate object so the most efficient means of exploding all the contours on both layers is to use a window selection. But, because we do not want to indiscriminately explode other layers in the file, the two contour layers should be isolated first. This is accomplished by freezing all layers but the two of interest ...

Step 6: Select **Format > Layer** from the menu or click on the **Layers** button in the upper-left corner of the screen to access the **Layer Properties** dialog.

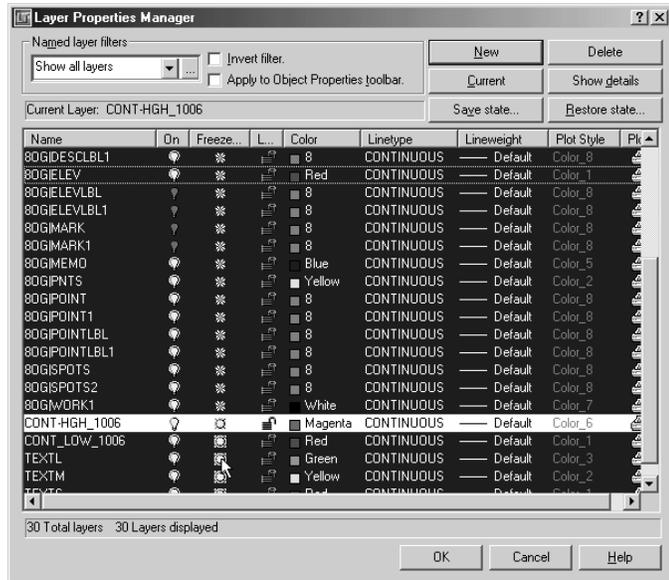
Step 7: Click on one of the contour layers identified in Steps 3 - 5 above (**CONT-HGH_1006** in this case), and then click on **Current**.



Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT AecCivil Contours (Cont.)

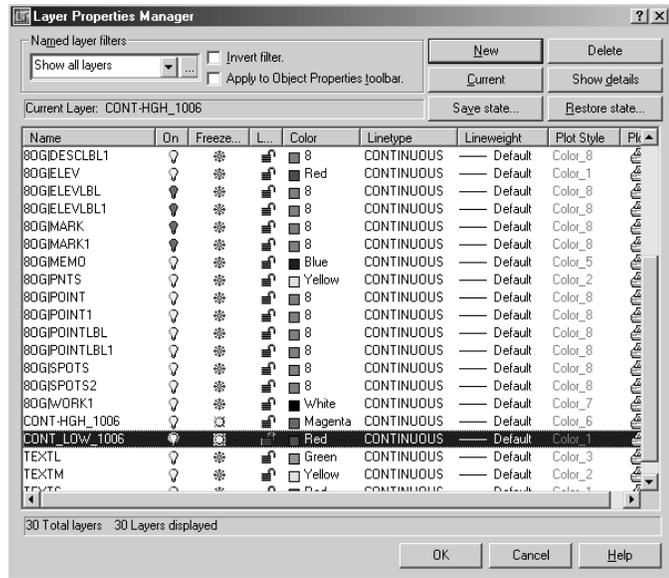
Step 9: Click on a yellow sun icon in the third column of any layer other than the one selected in Step 7 above (if you start by clicking on a snowflake, you will have to click it a second time).

[This step freezes all layers but the current, CONT-HGH_1006, layer.]



Step 10: Click on the name of the other contour layer identified in Steps 3 – 5 above (**CONT_LOW_1006** in this case), and then click on the snowflake for that layer to **Thaw** it. Click on **OK**.

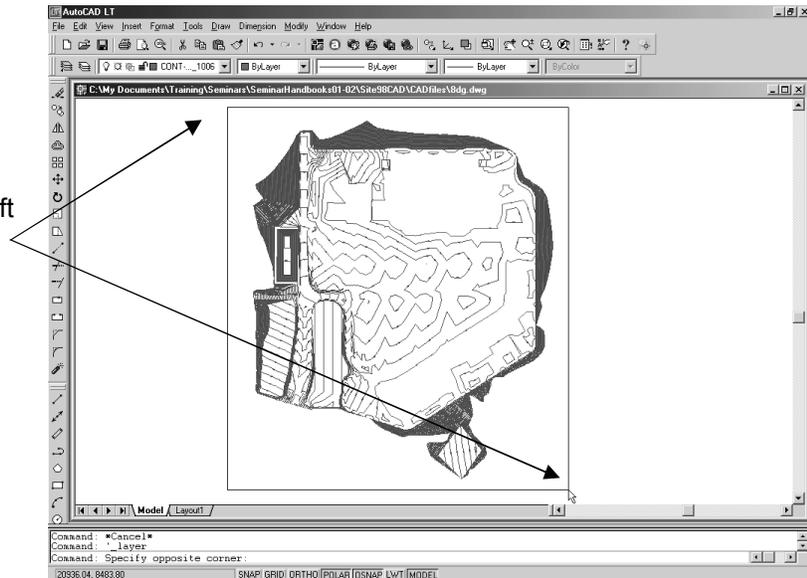
[All layers are now Frozen except for the two layers of AecCivil Contours.]



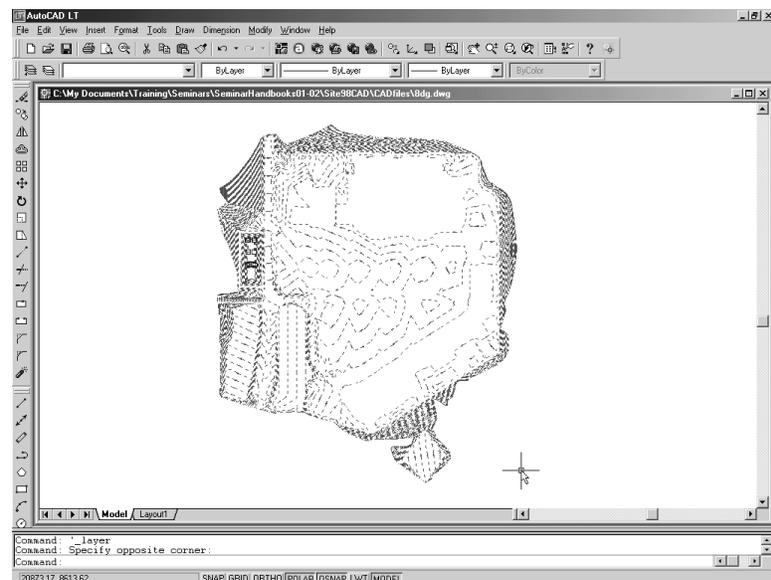
Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT AecCivil Contours (Cont.)

Step 11: Window select around all visible data by clicking once in the upper left corner, then once in the lower right corner, making sure all data is *completely* within the selection window.

[If you window select from right to left, data partially within the selection window is also selected.]

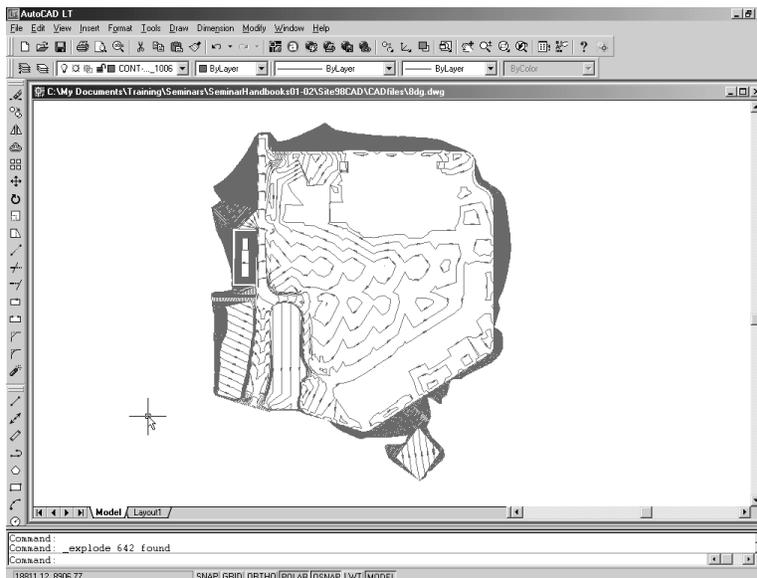


Step 12: Explode the selected contours by selecting **Modify > Explode** from the menu or by clicking on the **Firecracker** button → here.



Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT AecCivil Contours (Cont.)

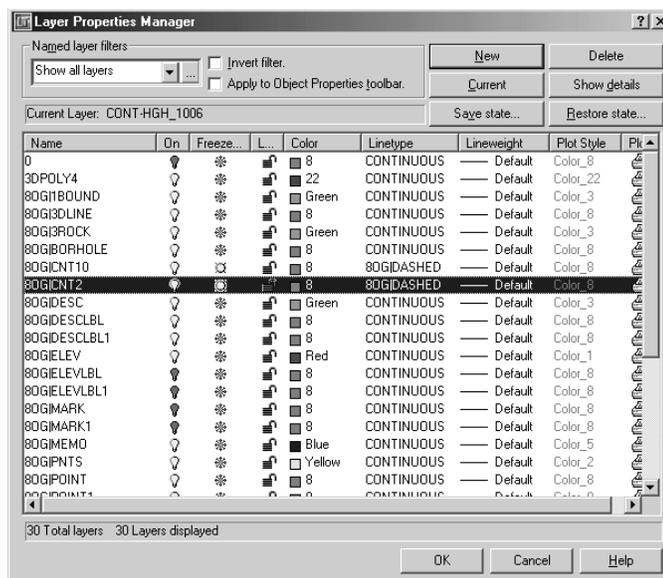
Step 13: If you pick one of the exploded contours it will still highlight as a dashed line without blue squares. And if you check its properties it will still indicate that it is a proxy entity. But the exploding process has modified it so that AGTEK can process it.



Step 14: Use the Layer Properties Manager to **Thaw** any of the **Frozen** layers that you need for AGTEK.

[The existing contours in this file are actually an External Reference that will need to be bound, and then exploded—see page 16 above and pages hb236, hb238.]

Step 15: Save your modifications per page 8.

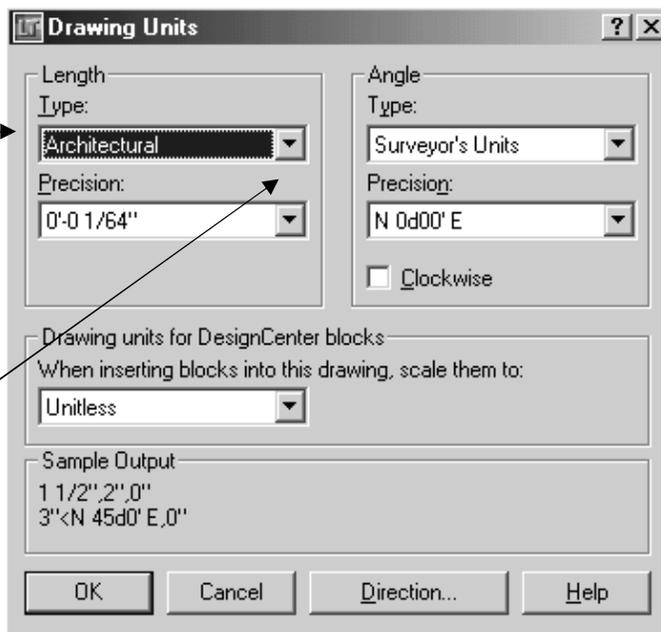


Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT Files in Architectural Units

CAD files for site grading are typically created with decimal drawing units (1 CAD unit = 1 foot) but occasionally a grading design will be drafted in architectural units (1 CAD unit = 12 inches). By default, AGTEK processes imported CAD data for decimal units so any architectural unit CAD data will be processed incorrectly unless you know it is in architectural units and compensate for it.

- ◆ To determine a file's Unit Type in AutoCAD LT, open the file and select **Format > Units** from the menu:

- Note the Drawing Units Type under Length (in this case, the file is in **Architectural** units and steps will need to be taken for correct processing in AGTEK.



Warning: If you click on the display button to the right of Architectural, you can select Decimal units from the list. Do not do it—it will not correct the problem!

Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT Files in Architectural Units (Cont.)

There are two ways to approach using CAD data in architectural units with AGTEK . . .

- ◆ The first (and more complicated) approach is to use AutoCAD LT to convert the CAD file's architectural units to decimal units as described below:

Step 1: Select **File > New** from the menu (or type **NEW** at the command prompt and press **Enter**).

Step 2: Type a "." and press **Enter** (to indicate no template file).

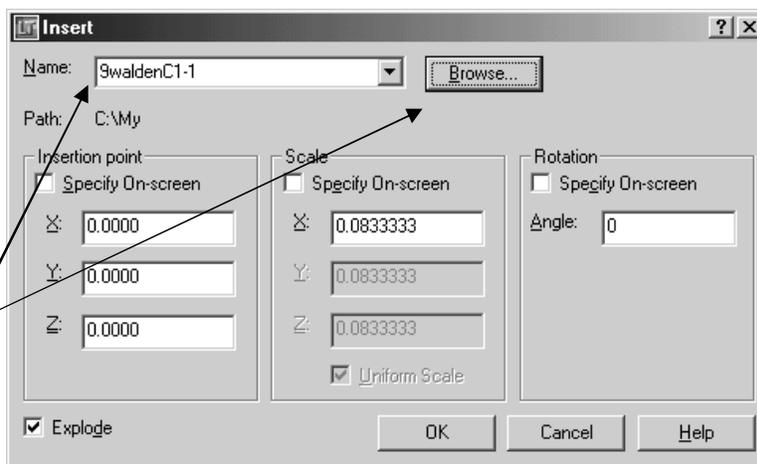
Step 3: Select **Insert > Block** from the menu (or type **DDINSERT** at the command prompt and press **Enter**).

Step 4: Click on **Browse** to locate the architectural unit CAD file (**9waldenC1-1.dwg** in this case).

Step 5: Complete the rest of the Insert dialog **EXACTLY** as shown here, and then click **OK**.

Step 6: Select **View > Zoom > Extents** from the menu.

Step 7: Correct any other problems with the CAD file, and then save it by selecting **File > Save** from the menu. This file can now be imported by AGTEK with no further adjustments required.



Note: A corresponding conversion can be made with TurboCAD (see hb240).

Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT Files in Architectural Units (Cont.)

- ◆ The second (and much simpler) approach is to let the AGTEK program do the units conversion automatically when it imports and transfers the CAD layers as follows (see page hb100 for a more detailed example of this method):

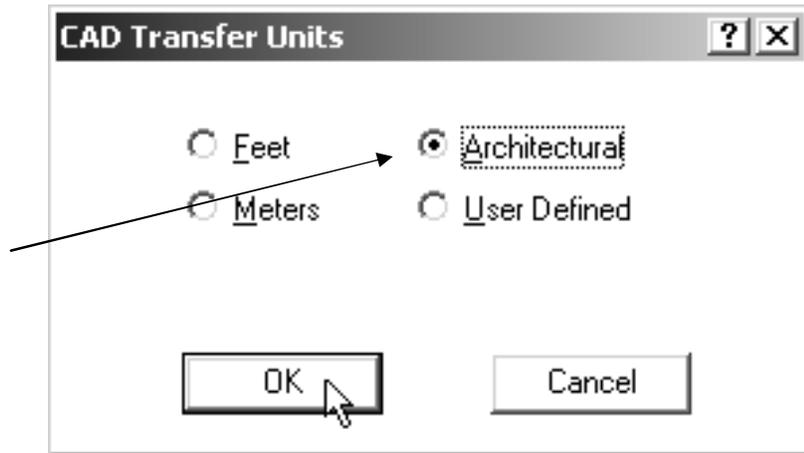
Step 1: In AGTEK 3D/4D, select **File > Open** from the menu and select the architectural unit CAD file.

Step 2: In the CAD Transfer mode *BEFORE* transferring any layers, select **Data Transfer > CAD Transfer Units** from the menu.

Step 3: In the CAD Transfer Units dialog, select **Architectural** and then click **OK**.

[This setting change will only affect the current session of AGTEK; the next time the program is started, the CAD Transfer Units will default back to Feet (decimal units).]

Step 4: Begin transferring the CAD layers. The AGTEK program will automatically compensate for the architectural units of the CAD data.



Metric Unit Conversions: Substituting the appropriate scale factors, the **DDINSERT** command described on the previous page could be used to convert CAD data from metric units to decimal units before importing the CAD data into AGTEK. But a metric-to-decimal units conversion can also be made in AGTEK as follows: (1) Import and process the metric CAD data with AGTEK (using **Meters** as the *CAD Transfer Units*, per page hb60) and save the resulting metric AGTEK job file, (2) create a new AGTEK job file, selecting **Feet** for *Units* in the *Job Information* dialog, (3) import (select **File > Import** from the menu) the metric AGTEK ESW file created in Step 1, (4) in CAD Transfer Mode (but before actually transferring any data), select **Data Transfer > CAD Transfer Units** from the menu, (5) in the *CAD Transfer Units* dialog, select **Feet and User Defined** (enter “**0.3048**” for the *Units per Foot* value), and click **OK**, and (6) transfer all data to the appropriate surfaces/layers and it will be converted to feet. A very good online video documenting these steps is available on AGTEK’s website at www.agtek.com/trainingvideos/Earthwork3DMetric.html.

Day 2 Handbook - Appendix E Supplement Manipulating CAD Files with AutoCAD LT CAD Survey Data (Spot Elevations)

Sometimes spot elevations are furnished in a CAD file in lieu of a contoured topo (the utility of CAD spot elevations will depend on how they are represented in the CAD file) . . .

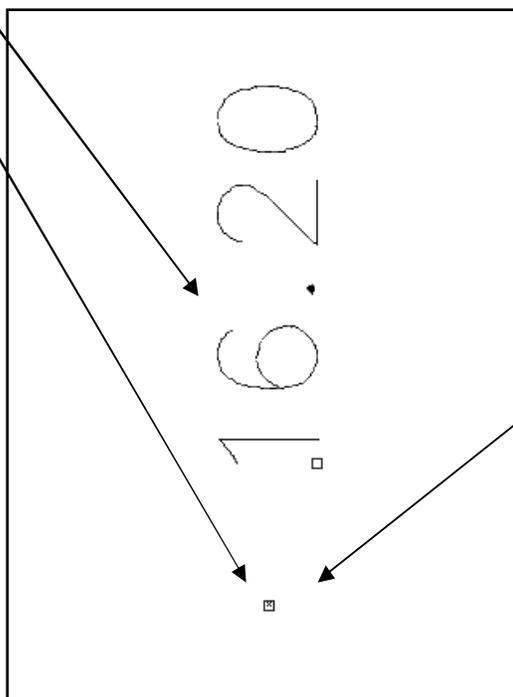
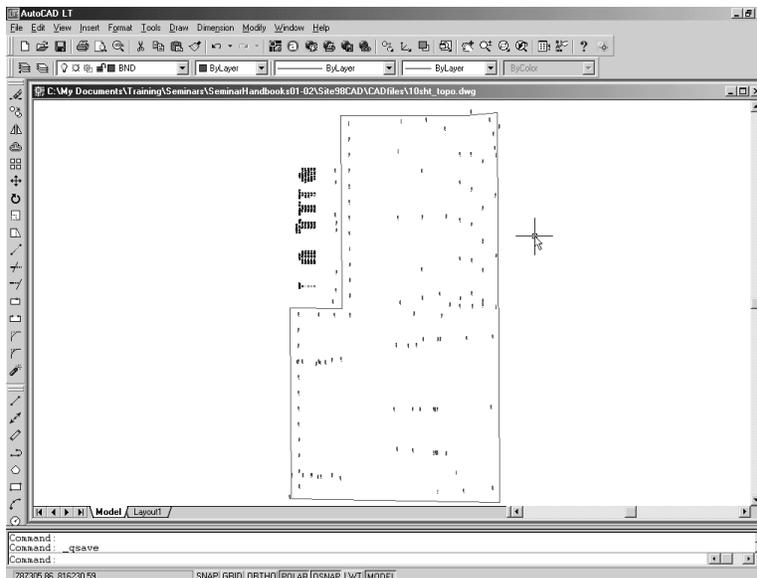
Step 1: If there are layers containing other data in the file, use the Layer Properties Manager (as described on pages 22-23) to isolate the layer(s) containing the spot elevation data.

Step 2: Identify what types of CAD objects represent the survey data (pick an object, **Right-click**, and select **Properties**). In the best case, each survey shot will be a single 3D **POINT** object at the correct horizontal ground position. In the next-best case, each survey shot will consist of at least two CAD objects:

- A **TEXT** object (2D) representing the elevation value, and
- A **POINT** object (2D) representing the horizontal ground position of the shot.

In the worst case, each survey shot will be represented by 3 or 4 objects or attributes (elevation, point number, description, ground position, etc.) combined in a **Block Reference**. You can try exploding the block (see page 14-15), but some or all of the data may disappear and you may have to manually digitize the points from the paper or PDF plan sheet.

Tip: If the spot elevations are represented in blocks and exploding them causes the loss of required data, the AutoCAD **Burst** command can be used to convert the attributes to text objects. [Only available in full-blown AutoCAD with the Express Tools option.]



Tip: If the spot elevation ground position point is represented as a "+" or "x" with no horizontal coordinates at the center, use AutoCAD LT to change the point display style (**Left click** on one of the ground points, then select **Format > Point Style** from the menu and choose the dot option rather than "+" or "x").

Step 3: Save any modifications per page 8.