AutoCad for Surface Mining

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GD41-2 This course focuses on how to use AutoCAD® effectively, with an emphasis on the mining industry. We will cover topics including techniques for creating entities and managing files. In the process, we'll cover general set up and standards for drawings using geometric tools; how to compile files for presentation and evaluation; and exporting projects to other users for collaboration purposes. We'll illustrate these techniques with exercises.

About the Speaker:
Jill has been drafting since the pre-CAD days. She learned construction drafting in the Army National Guard, and has worked in the gold mining industry for 16 years. When AutoCAD® came to her site, she was given the job of training CAD users. In 1991, she organized the Northeastern Nevada AutoCAD Group and has been a leader ever since. She also taught AutoCAD at the local college. Jill's love of teaching inspired her to teach through the AUGI® Training Program and now at Autodesk University®.

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DATA ANALYSIS

One of the most important concepts in industry is that one drafter or designer is rarely the only person to use a drawing. Pre-planning can be crucial to a successful project. Evaluating drawings using layers, linetypes and blocks with attributes will demonstrate the importance of standards and the process that takes place in creating drawings for design purposes.

DATA TRACKING AND DISPLAY

We will explore how to create entities and files that are easily amenable to change. The easiest way to allow change is to group entities with like characteristics together. Using layers may be fundamental to elicit change. Evaluate how the entities are going to be used in a drawing. Will text accompany the entity? Is it useful to have the text as an “attribute” or is a “text” entity enough and more useful? Many entities in mining drawings have uses for an “attribute” to be attached. If AutoCAD Map is used at your site, attributes for information stored in tables or databases is very handy.

Creating blocks on layer zero compared to creating them on specific layers will allow different kinds of data controls resulting in different visual effects and filter techniques. Sometimes keeping track of the evolution of an entity is important, but other times where the entity exists currently is the only important factor. For example data can be tracked by the year no matter what layer it is on, or tracked visually with the name reflecting the year. In the second scenario the block cannot carry color nor linetype characteristics unless they are created on the layer for that year. Apply this theory when analyzing all data for your drawings using blocks, linetypes, entity types etc.
QUICK DRAW

Drawing easily, accurately and quickly is a drafter/designer’s specialty. We will investigate the use of geometry and CAD tools including bhatch, UCS, array and point filters to develop designs and new drawings.

GEOMETRY

Sometimes it is expedient to make a layer for construction lines that will not be thawed at plot time. Other times using existing geometry to construct new entities can shortcut creating construction lines.

HATCH

The ansi hatch patterns are made at scales for mechanical parts and do not lend themselves to mapping scales as well as some of the other hatch patterns. If slanted lines are needed try using the “line” hatch with an angle. The scales used will be more like the scales for linetypes and text in mapping drawings. The creation of SOLID hatch has given us a powerful visual tool with less overhead in the drawing.

LIMITS

Similar to the ansi hatch situation, when a new drawing is begun with the “start from scratch” option the default area of the drawing is 12x9. Mapping units are usually feet, miles, meters etc. If no template drawing is used that defines a drawing area, a zoom, extents will be necessary to see the information dxfed, xrefed or inserted into the drawing.

UCS

When using the UCS command with the OB or Entity option set the “ucsicon” to “Origin”. The icon will follow each setting on the new lines as they are constructed. All angle references are from 0 (zero) on the x axis.

Notice the ucsicon is at the lower left endpoint of the line and the "x" axis is aligned with the line. Having the icon follow the geometry is a visual reference helping to ensure the control of drawing new entities.
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POINT FILTERS

Whenever you are working with as-built data there are always non-zero "z" values assigned to that data. If no special settings have been made, the default "z" of any new geometry defaults to 0(zero) elevation. Therefore it is possible that a drawing could contain lines extending, for example, from 3720 elevation to 0(zero) elevation. When this occurs the line type is on such an angle that a dashed line may appear continuous. If you desire to put a design on a particular elevation use the elevation command or change the design entity's elevation property after creation. Use the elevation command with caution. Once set, the elevation will default to that value until changed or returned to 0(zero). The move command WILL NOT correct "z" value errors on a line that has one endpoint at a different elevation than the other endpoint. Using "point filters", lines can be drawn with endpoints at the same elevation regardless of the surrounding information.

DATA ASSEMBLY

Drawing size is still the key to performance with AutoCAD and data can be stored in small files then assembled to create a product for evaluation or presentation. To enhance the pre-planning of projects, use of AutoCAD variables and filters allow the drafter/designer control and speed.

SEARCH PATHS

Having search paths available in AutoCAD helps with inserting blocks and externally referencing files, as well as running LISP and scripts. Be sure to set up the Search Path for your directories containing your projects. In the TOOLS pulldown, under PREFERENCES or OPTIONS, on the FILES tab; use the ADD and BROWSE buttons to add the required directories. Use the MOVE UP button to increase the speed of the search. AutoCAD will search your project directory first to find the files needed if your directories are listed before the AutoCAD program directories. You can have several profiles containing different SEARCH PATHS and change profiles per project. You also can put all the paths for all projects into the same profile.

VISRETAIN

VISRETAIN is the variable allowing visibility settings in drawings that contain externally referenced files. Following are the definitions from AutoCAD help.

VERSION 14 HELP

Controls visibility of layers in xref files.

0 (zero) The xref layer definition in the current drawing takes precedence over these settings: On/Off, Freeze/Thaw, color, and linetype settings for xref-dependent layers.

1 On/Off, Freeze/Thaw, color, and linetype settings for xref-dependent layers in the current drawing take precedence over the xref layer definition.

VERSION 2002 HELP

Controls the visibility, color, linetype, linewidth, and plot styles (if PSTYLEPOLICY is set to 0) of xref-dependent layers; specifies whether nested xref path changes are saved.

0 (zero) The layer table, as stored in the reference drawing (xref) takes precedence. Changes made to xref-dependent layers in the current drawing are valid in the current session only and are not saved with the drawing. When the current drawing is reopened, the layer table is reloaded from the reference drawing and the current
drawing reflects those settings. The layer settings affected are On, Off, Freeze, Thaw, Color, Ltype, LWeight, and PStyle (if PSTYLEPOLICY is set to 0). This setting also specifies that changes made to the paths of nested xrefs are for the current session only and are not saved with the drawing.

1 Xref-dependent layer changes made in the current drawing take precedence. Layer settings are saved with the current drawing’s layer table and persist from session to session. Nested xref path changes are saved with the current drawing and persist from session to session.

Essentially setting VISRETAIN to 1 will give you visibility control in the current session and save it with the drawing created in that current session. Sometimes a combination of saving the files to be externally referenced with the preferred entity stacking and then controlling the layering or stacking of the xrefs themselves will be necessary to achieve the display required.

The DRAWORDER command will help display entities in front of other entities. When a drawing is externally referenced the “whole” drawing or block is stacked in the current session, not individual layers or entities.

ENTITY STACKING

Stacking entities for visibility is important for the published copy. In AutoCAD the last objects created in the drawing are listed last in the database, consequently those objects are displayed on top of all the other objects. The “OBJECT SORT METHOD” can be set in the TOOLS/PREFERENCES OR OPTIONS pulldown to help with “picking on the screen” and will allow for speed in revising or editing. If PICKSTYLE is set to 2 or 3 the Control Key will allow toggling through stacked entities to select the correct one for editing.
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When dealing with Xrefs, opening a new drawing and re-stacking the xrefs as they are attached is not a bad option. Generally speaking though, the compiled project will have too many xrefs to re-stack efficiently. Base information does not have to be stacked in a particular order and only a few key xrefs have to be shown in a certain order. Just dealing with those few items and using the copy command a few times will probably accomplish what you need fairly quickly.

Knowing your data and your project becomes an important concept. When you update the small files that will eventually go into the final product make sure that the data looks as close to what you will need later as you can. Having to re-open files to re-stack the entities or hatching can be eliminated if you remember what you will be doing with the information. Consider how many files you may be updating to be externally referenced or inserted into the final product that may have to be re-stacked.

POINTS OR BLOCK SYMBOLS?

One more idea to keep in mind when you are deciding how to put a project together is, when to use point entities or block symbols. Some survey data is dumped from the data collectors using a “point” entity. Check the data before you use it in other drawings. If you already have “point” entities in drawing1 the new data will take on the properties you have previously set for the points in the point style dialog box. Sometimes that means the point symbols will not display properly. Remember you can only have one pdmode and pdsize set in a drawing. If drawing1 contains point entities and is going to be xrefed or inserted into drawing2 the points within the drawing1 will take on the qualities of drawing2. Block symbols may be more flexible in some cases. Block sizes must be considered though. If the drawing information encompasses several miles and the area to be displayed and plotted is only several hundred feet, check the size of the block symbol and verify that it will display and plot properly before you begin the compilation of the data.
PLOTTING

Model space plots and screen dumps of images are useful for quick plots for fieldwork or planning discussions.

HARD COPY PLOTS

- You can plot the model space “display” (screen limits) or a model space “window” of the data.
- You can plot a paperspace layout.
- Always plot to some usable and standard scale. Remember to use the 1/???xp (version 14) or the new scale pulldown (version 2000+) to set the scale for each separate paperspace viewport. AutoCAD versions 2000 and above have convenient scales already programmed, but the 1”=500’ mapping type scales may have to be typed in or programmed in.
- Always include a grid.
- Most mapping is done with at least one coordinate grid referenced.
- Run a check print and verify that the important data is clear and legible and a color that is easily differentiated from the base information.

DIGITAL PLOTS

To make a digital check plot, one option is to use the “saveimg” command to save a screen capture to Paint or other image software. Images can be contained in a text document or attached to an email and printed by the recipient. The clarity may not be the best with this option so later we’ll look at other image options. It is helpful to have the largest view of the area of interest visible before saving the image. If cropping must be done afterward there will be room for fine-tuning the crop. It also helps keep the line quality when reducing the image in paint.
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LEGENDS

In version 14 there is a Bonus Tool called "WIPEOUT". It is very useful for creating a mask for text and legends. It must be created at an elevation ("z" value) higher than all other data in model space. The legend must be inserted at or above that elevation to display it on top of the wipeout. You may have to restack the legend text above the wipeout, using "copy" or "draworder" commands, to ensure it comes out on top every time. To position the wipeout the frame must be visible. Entering the "WIPEOUT" command and choosing FRAME/OFF will visually remove the frame of the wipeout.

In versions 2000+ the Region command will accomplish the same masking effect as the wipeout. The difference in using a REGION instead of a WIPEOUT is that you must use the HIDE command or HIDE PLOT to get the entities behind the REGION to be covered with the REGION. Using the hide sometimes causes the filled text to be wireframe. You could use the plot style to configure pen widths to plot filled text instead of wireframe.

If I know I am going to use a legend on a wipeout or region inside modelspace, as opposed to inserting it into paperspace, I like to create my legends in modelspace and wblock them out to their own file. That way the symbols, text height and linetypes match the model. Sometimes I will even keep the legend in modelspace in an area away from the drawing limits to be viewed through a separate paperspace viewport.

THIS IS THE LEGEND AREA WITH BLOCK SYMBOLS LINETYPES AND SPECIAL NOTES
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PLOT STAMP

Even on quick work-up plots it is important to include the date, scale, path and filename and creator’s initials somewhere on the plot so that the revisions marked on the plot or image can be incorporated into the original AutoCAD file.

In AutoCAD 14 use “hpconfig” with HP plotters to select what will be printed in the stamp.

In AutoCAD 2002 in the “plot” command there is a stamp available. You need to load a Plot stamp parameter file ie: inches.pss or mm.pss file and save it as a name such as “general site” or something specific to the plot setup before you can configure what the stamped information is. Then the date, time etc. that is to be printed on the stamp can be check marked in the dialog box.
Get it OUT!

The reason for all the previous work of drafting/designing is to communicate completed or final plans to someone else. There are several options for exporting drawings to other drafter/designers as .dwg or other formats, including images for use as slides or plates in reports.

FILE TRANSFER AS .DWG

**Pack 'n Go** The Bonus/Tools/ from version 14 or the File/eTransmit option in version 2002 are great for assembling all the text fonts, blocks and external references together.

Version 14

Pack 'n Go assembles all the files in a specified “assembly” directory allowing a zip file to be created containing all the files in that directory. If you have created a .plt file with specifications for a plotter that can be utilized by the customer or a .pc2 file, then copy those files, using My Computer or Microsoft Explorer, to an “assembly” or “zip” directory. Use Pkzip or Winzip to compress the files.

In AutoCAD version 2002, **Etransmit** will assemble AND create the zipped file.

Version 2002

On the AutoCAD pull down choose **File/eTransmit**. There are many new options in the dialog box. The **General** tab allows descriptive notes to be included, the kind of assembly defined (folder or .exe file) the directory the file will be written to defined, whether or not the paths will be included etc.

**NOTE:** Many companies have firewalls that will not accept an .exe file. You will want to verify what format is acceptable and then choose the appropriate .zip as your **standard default** to eliminate problems.

The files created by the previous 2 methods can now be attached to an email (if you didn’t already specify email in version 2002) OR they can be loaded onto a network drive or ftp site. If the recipient keeps all the support data together the drawing “should” re-assemble just like it did for the originator.

**DWF**

In the version 2002 Plot command there is an option to create a .dwf file using the **ePlot** and **eView** drivers.

The following information is from the AutoCAD help file:

*With ePlot and eView, you can generate electronic drawing files that are optimized for either printing or viewing. The files you create are stored in Drawing Web Format (DWF). DWF files can be opened, viewed, and plotted by anyone using Volo View or Volo View Express. With Volo View or Volo View Express, you can also view DWF files in Microsoft Internet Explorer 5.01 or later. DWF files support real-time panning and zooming as well as control over the display of layers and named views.***

*Warning! If you try to view DWF files created with ePlot or eView in an Internet browser with the WHIP! 4.0 plug in, some information may not display correctly or you may experience errors. However, the DWF file will not lose data or be modified in any way.***

*As the name suggests, ePlot and eView create virtual electronic plots. When creating ePlot and eView DWF files, you can specify a variety of settings such as pen assignments, orientation, and paper size, all of which control the appearance of DWF files.***

*DWF files are created in a vector-based format (except for inserted raster image content) and are typically compressed. Compressed DWF files can be opened and transmitted much faster than AutoCAD drawing files. Their vector-based format ensures that precision is maintained.*
DWF files are an ideal way to share AutoCAD drawing files with others who don’t have AutoCAD. Because the Volo View and Volo View Express interfaces are easy to use, even those with no CAD knowledge can easily view and navigate a DWF file.

If you have Volo View open a .dwf and try zooming in and out, check out the markup utility and with the help of the following images turn on and off layers. Unlike the bitmap images the resolution is excellent no matter how far zoomed in you are.

Try the ePlot driver and send a plot to your plotter from Volo View. I think you will find the results very good compared to other image plotting techniques.

IMAGES IN PRESENTATIONS

Some drawings must be shown in presentation form. Important concepts to remember when creating images for presentation:

- The more light that comes through the projector the harder it is to see the image. A medium gray is helpful when many topo lines are shown. For simplified cross sections blue, green and black sometimes show off the lines and text effectively.
- Use fat plines for the important information and solid filled text for labels.
- Use several slides to get the point across if the information is too dense.
- Keep the drawing very clean and simple.
- The quality visible on the screen does NOT equal what will be projected. Make several different slides of each drawing and project them BEFORE the presentation. This allows time to choose the slide with the best projectability and time to change the slide if necessary.

Good Luck in your file transfers and presentations! Hope to teach you again!