PDHonline Course G371 (2 PDH)

Minimal AutoCAD® 2011

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Minimal AutoCAD™ 2012

Thomas Mason, PE

It is illegal to do a screen-shot of Google Maps and print copies to send along with invitations to your son's graduation party. Instead, spend 30-minutes on AutoCAD™ and make your own map.
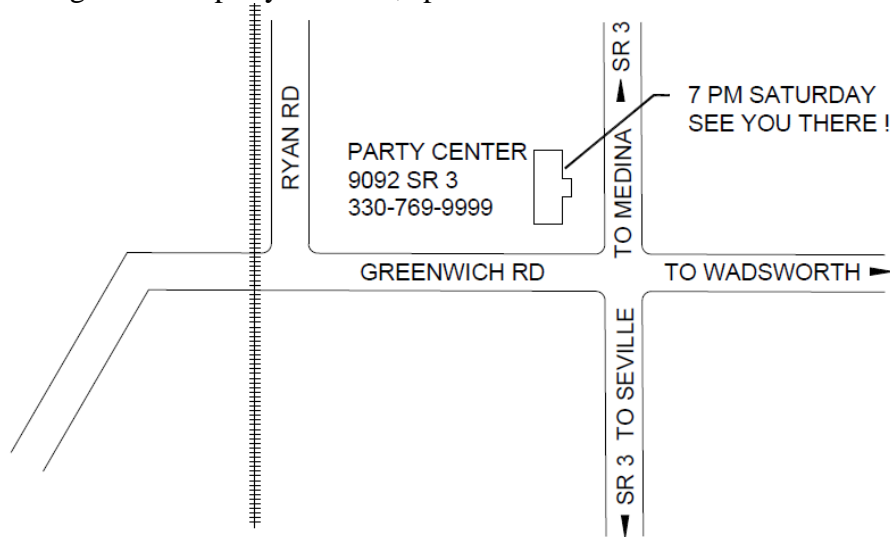

The Plant Engineer wants to get prices for a brick security building for the truck entrance. The contractors know how to build an insulated shed but you have to tell them what you are looking for. An AutoCAD™ sketch looks better than felt pen on a napkin, and it takes less than 60 minutes.
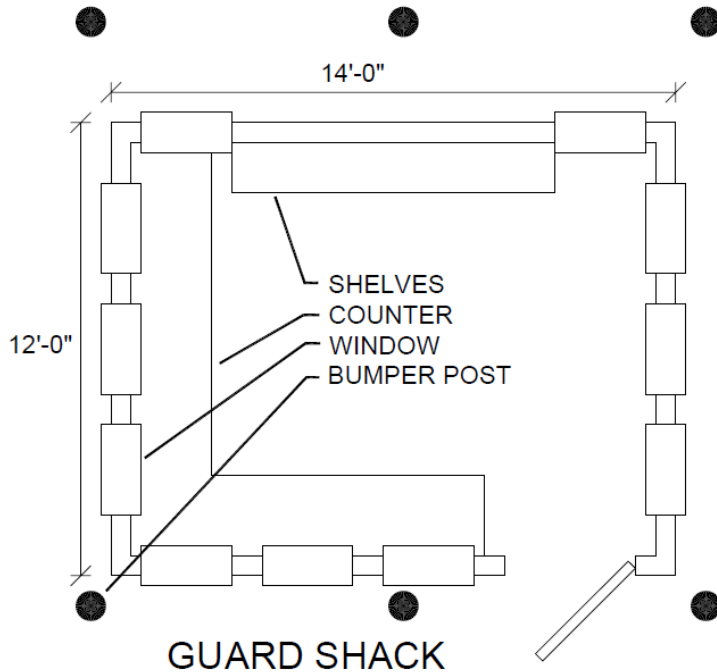
The Architect asks what the maintenance light is going to look like and how you are going to mount it. Instead of forwarding a catalog image from the internet, you make a line drawing for her.
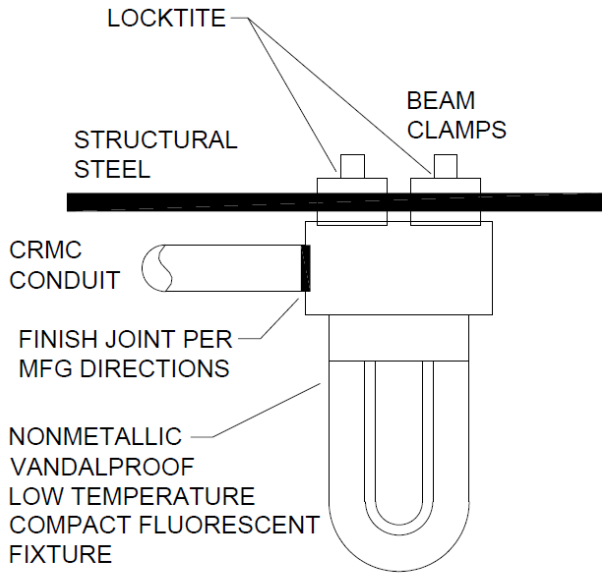
MAINTENANCE LIGHT

The first step in an Arc Flash Study is collecting data on the plant electrical system. Existing documentation is only partial and needs field verification. Of course, you have to make a drawing.
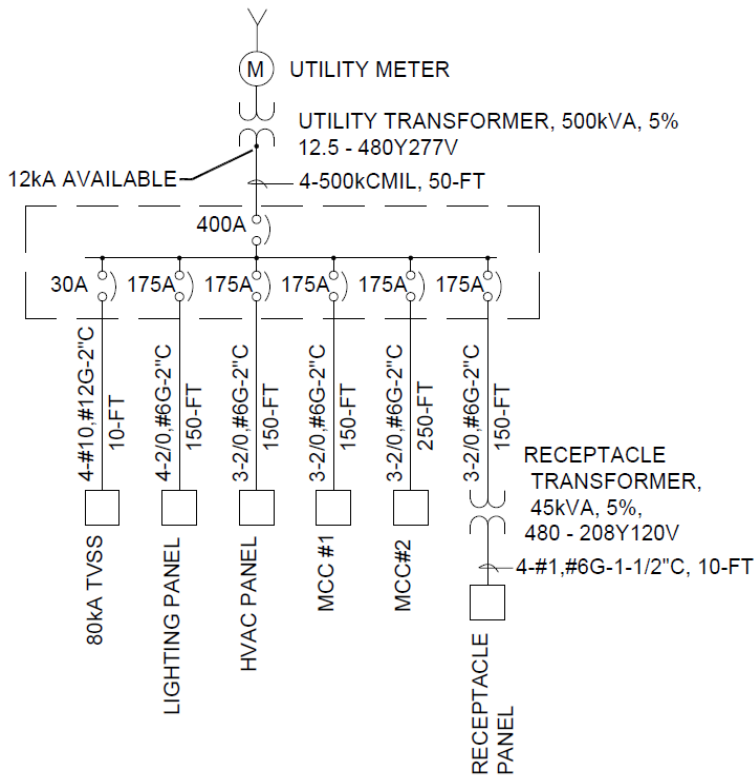
PLANT ONE-LINE DIAGRAM

There is an HVAC problem in the lab and you want to make sure you are looking at all the pieces-and-parts that are supposed to work together.
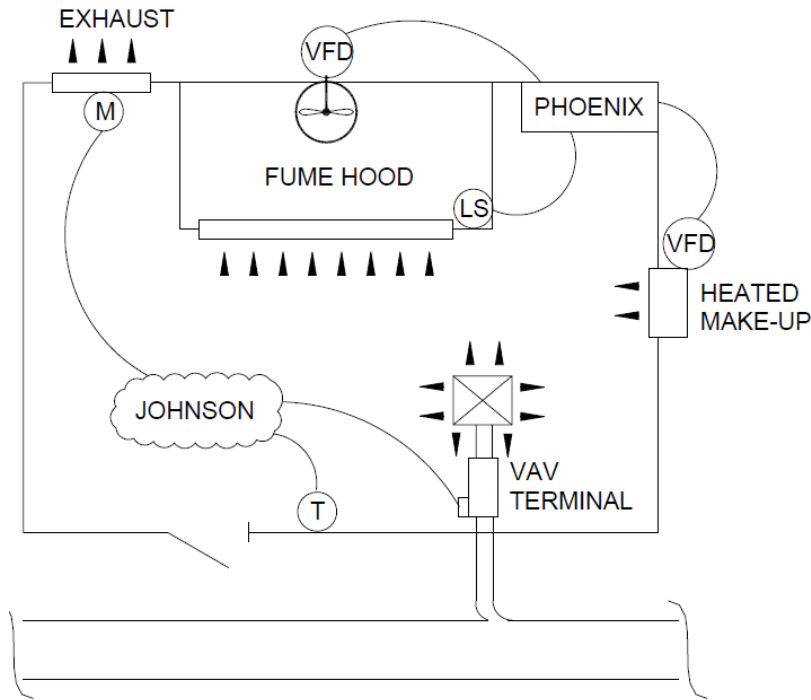
LAB HVAC

Why is this course organized this way and what is this box ?

First, the box.

This course contains *hard content*, as commands, discussion and examples. It also contains *asides*, interesting useful information that is not central to immediate productivity. Such asides will be placed in boxes like this, called *sidebars*, to offer the opportunity to skip them completely, if you wish.

Now, regarding the examples.

To convince you that this course provides immediate value in your job, we gave examples of the product we promise to deliver. If you wanted to learn to do structural design connection details or architectural soffit details, this is the wrong course. We will refer to our examples to show how the basic commands of AutoCAD™ can be used to rapidly produce drawings.

There is always a struggle between making a summary list and discussing and showing how to use the commands. Both approaches are addressed by this course. Please refer to the Command Summary .pdf that you downloaded separately. Make sure you understand that author's meaning along with the words used in the narrative content of the course. The Command List should have long-term value.

DRAWING COMMANDS

Remember, sometimes the drawing commands don't work or work in strange ways - because of the setting on a hidden variable. This is rare on a new drawing, but common on an existing drawing, prepared by a sophisticated drafter. We will look at setting up a new drawing and resetting the hidden variables later.

LINE and POLYLINE - Type "L <ret>". Left-click the mouse at the start position. Move the mouse. Left click at the finish position. Done. If you are happy with the result, move on to the next line. If not happy, "U <ret>" for UNDO.

"<ret>". Left-click the mouse at the start position. Move the mouse. Left click at the finish position. Note that AutoCAD™ usually lets you repeat a command without re-entering it. To get out of drawing lines, "<ret> <ret>".

Type "PL <ret>". Left-click the mouse at the start position. Move the mouse. Left click at the finish position. Ready to continue the line. Left click at the next position. If you are happy with the result, "<ret>".

"<ret>". Left-click the mouse at the start position. Move the mouse. Left click at the finish position. Note that AutoCAD™ usually lets you repeat the PL command without re-entering it. To get out of drawing lines, "<ret> <ret>".

If all that makes sense, then skip this box and goto the next command.

If this doesn't make sense, then it is going to be a lot harder for you.

AutoCAD™ is somewhat flexible on command input. "L" = "LINE" = "l" = "line" and mixed-case works. "PL" = "POLYLINE" = "pl" = "polyline" and mixed-case works.

In older versions of AutoCAD™, line and polyline were different. Some more complicated commands only worked with LINE and some only worked with PL. More recent versions prompt you to convert to the required form or automatically convert. You have to do some drawing and see how the system you are using responds.

Note that when you sit at a different computer, you may get different results. Many firms keep an old version on an old pc for ease of accessing old jobs. Many firms do not upgrade until they are forced.

CIRCLE and DONUT - Type "C <ret>". Left-click the mouse at the center of the circle. Move the mouse. Left click at the radius. Done. If you are happy with the result, move on to the next circle.

"<ret>". Left-click the mouse at the center of the circle. Move the mouse. Left click at the radius. Done. To get out of drawing circles, "<ret> <ret>".

Type "DONUT <ret>". Enter the inside diameter, as, "1 <ret>". Enter the outside diameter, as, "1.1 <ret>". Left-click the mouse at the desired center of the donut. If you are happy with the result, move on to the next donut. To get out of drawing circles, "<ret> <ret>". AutoCAD™ usually expects inches, but you can enter "1.1'" and it gives you 12.12 inches.

Isn't this unnecessarily complicated ?

Yes, and it will be eliminated in a future revision of AutoCAD.

The reason we have circle and donut as separate commands is from the origin of AutoCAD™. It started as a simple program and grew. Sometimes the growth was integrated and sometimes the growth showed up a new, disparate commands. As the software has matured, many of the disparate commands are being integrated.

Note three important features - growth, integration and backward compatibility. AutoCAD™ is famous for listening to users and incorporating changes to meet voiced needs - usually the next year or two years later.

Second, they improve the product from their own observations of improving productivity. It is common for a revised command to incorporate two functions which previously required separate commands.

Third, they work hard to maintain backward compatibility. Yes, they introduce new and better commands, but they keep the old commands and try to keep them operating the way they used to operate. For example, when I learned AutoCAD™, the only way to enter commands was from the command line - the way I am showing you. Since then, they added drop-down menus, icons and now, a context-sensitive ribbon. The command line still works and it works on all versions of AutoCAD™ (but not on other AutoDesk products).

MTEXT – Type "MT <ret>". Left-click the mouse at the start position. Left-click and drag down-and-right to the approximate size of text you want. Left-click. You should get a flashing text-cursor (I). The machine is ready to type, using the size and font that was left by the last user. On my machine, presently, it want huge letters in "Annotative (style) romans (font)". I usually enter "TXT", then click on OK on the edit bar that appeared. I fix the result using edit commands. Since the text is really ugly until you edit it, we will jump to that subject now.

EDIT COMMANDS

SCALE - Type "SCALE <ret>". Left-click on the object to be sized up or down, "<ret>". Left-click on the reference point for the scaling (doesn't matter much since you can move it when you are done). Type ".1 <ret>".

If that size isn't right, do it again. 1.2 is a nice, small increment for upsizing. .8 is a nice small increment for smallifying. [Getting text exactly the right size is very, very important to some people. We will do that before we are done.]

MOVE - Type "M <ret>". Left-click on the object to be moved, "<ret>". Left click on the starting point, usually the piece of the object that you want to align with something else. Mouse over to the

desired destination, "<ret>".

ROTATE - Type "ROTATE <ret>". Left-click on the object to be rotated, "<ret>". Left click on the starting point, usually the piece of the object that you want to align with something else. Mouse around and watch the result on the screen, left-click and "<ret>".

MTEXT FORMATTING BOX - There is tremendous power available through the mtext formatting box. We will examing only style, font, alignment and numeric list. Style is the far-left box and font is next to it. My machine comes up "Annotative" style. We don't want that, it changes between model space and paper space (which we will discuss later). Anything except "Annotative" is ok because we are going to select ARIAL font. When we select ARIAL, it only affects the words we enter, not existing text on the sheet. In 2012, Arial is believed to be the most readable font. In 1990, it was required that all drawing text (except architects') be in ROMANS, an especially ugly font.

Double-left-click on some text. The Mtext formatting box should come up. (If not, you found some legacy straight-text. Move on to another piece of text.) Left-click the top-left style box. Click on STANDARD. Move to the next box, font. Click on TT Arial. Everything you type after this will be in Arial.

Double-left-click on some text. The Mtext formatting box should come up. Select the existing text by left-click and drag across it. Left-click the top-left style box. Click on STANDARD. Move to the next box, font. Click on TT Arial. Everything already typed should be in Arial.

Just below the font box are four icons for alignment. Far-left of the four is left-aligned, which is right almost all of the time.

Two groups to the right of alignment is a vertical 1,2,3 with lines after each. This lets you quickly make numbered lists in a consistent fashion. We are not going to exercise this feature, but is works very, very well. Most existing drawings have each item number and each text entry a separate object. This was necessary because of clunky alignment capabilities in AutoCAD™.

We have accomplished a lot.
You could move on from here, exploring the available commands, by yourself. You picked up on the structure of the commands, "VERB, OBJECT, START, FINISH".   Before you step off on your own, however, let's make things easier, by introducing some nit-picky detail stuff.

SPACEBAR - Usually, the spacebar works just like <ret>, but it is easier to find and quicker.

WINDOW and CROSS - If you have a single object, you select it by clicking on it. If you want to move a bunch of objects and keep their spacing, there are two ways to select them. Window draws a window around the objects and is the default if the selection click doesn't hit an object.

"M <ret>". Left-lick on an open area. Mouse to another point, making a box that includes only the objects of interest. Left-click to complete the window. Left-click a start point for the move, perhaps a

point you want to align with something. Left-click at the destination for the group. Note that you can watch the objects move as you move them.

Cross draws a window around and through the objects and is the default if the selection click for up-left or down-left if the cursor doesn't hit an object. This is not confusing. Both window and cross do the same thing. Cross includes objects "touched" while window does not. This is very useful when selecting only a few pieces of a group.

"M <ret>". Left-click on a lower-right open area. Mouse to another point, making a box that includes only the objects of interest. Left-click to complete the window. Left-click a start point for the move, perhaps a point you want to align with something. Left-click at the destination for the group. Note that you can watch the objects move as you move them. The only difference is objects "touched" by the box.

ADD and REMOVE - We can add objects to a window or cross selection group by hitting "A <ret>" instead of simply "<ret>".

"M <ret>". Left-lick on an open area. Mouse to another point, making a box that includes only the objects of interest. Left-click to complete the window. **"A <ret>". Click on additional objects or make a new window or cross. "<ret>" to finish selection.** Left-click a start point for the move, perhaps a point you want to align with something. Left-click at the destination for the group. Note that you can watch the objects move as you move them.

Remove works the same way.

AutoCAD™ is very good at handling long lists of selected, added and removed objects. I am not. I get confused very quickly. Window and cross are essential. Add and remove once makes things easier. More than that increases the likelihood of a following undo command.

The UNDO command.
We slipped this powerful command in right at the beginning. There are very, very few commands that cannot be undone. This brings up an important caveat - saving your work. In early versions, when you saved your work, it reset the undo list. This is no longer true. The reasoning was that you can go back to the saved work directly and didn't need repeated undo's to get there. It rarely caused trouble then and shouldn't cause trouble now. What about saving? Save when you are at a point that you don't want to repeat the work since the last save. When I complete a list, with a lot of alignment move commands, I save it because I don't want to go through that again. Any time you do something really tricky, like get alignment exactly correct, save it.

SAVE, QSAVE, SAVEAS and CLOSE - There are three roughly identical save commands. Save works as you would expect it to. Quick save was introduced when files got huge and pc's were slow. They are not discernably different on a modern pc with 4GB of RAM. Saveas lets you change the filename or AutoCAD™ version. Quit is mentioned here because it is the no-save command. There

are times when you want to mark-up a drawing and plot the comments, but not save the comments. This is probably a bad idea, but just say, "CLOSE <ret>". [It is recommended, rather, that you do a SAVEAS and append the title with your initials and the date.]

AutoCAD™ versions and TrueView™

As I mentioned, I work on a lot of projects for a lot of employers on a lot of very different AutoCAD™ workstations. Also, I have a setup at home. Periodically, I get into trouble for saving in a newer version than that supported by the firm. I resave, using the SAVEAS command or do a conversion using TrueView™. Note that there are genuine new features that don't convert to earlier formats. As I am working on old .dwgs and just doing lines, circles and text, I don't get into these.

TrueView™ is a free download from AutoDesk [http://usa.autodesk.com/adsk/servlet/pc/item?siteID=123112&id=9078813]. It is easy to use and very reliable in converting newer .dwg files to older formats. Sometimes it complains, but the results are usable.

EDITING COMMANDS CONTINUE

COPY - "COPY <ret>". Left-click on the object to be copied, "<ret>". Left click on the starting point, usually the piece of the object that you want to align with something else. Mouse around and watch the result on the screen, then left-click Window, cross and undo work. In newer versions, the copy command keeps copying until you say, "<ret>". In older versions, you must re-start the copy command to make multiple copies.
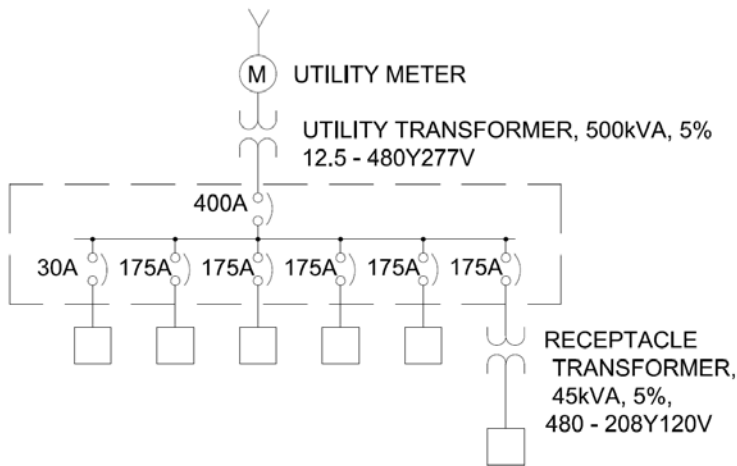

The power of COPY.

I have a lot of trouble with text. On an existing drawing, there will be multiple sizes, styles and fonts.

I can copy the first text entry I did that looks right or I can copy the previous drafter's text entry that looks right. It is common that I must add a detail. Drafters spend a lot of time choosing the size and font for the legend and for the notes and for the text. Copy a similar detail and edit it and it comes out matching.
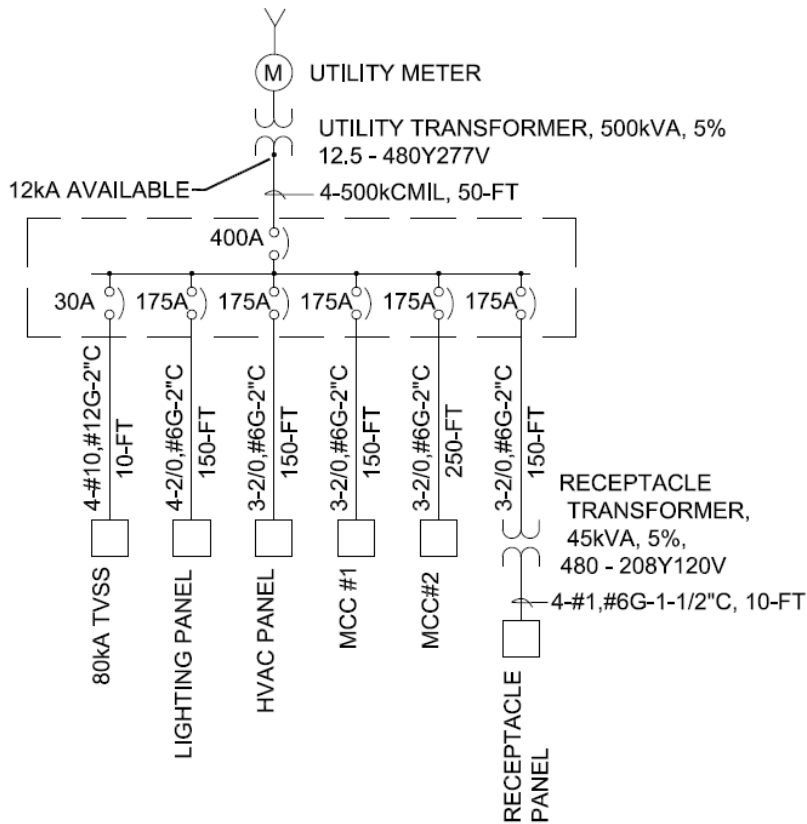

STRETCH - "STRETCH <ret>", left-click a cross box to include the pline vertex that you want to move; ""<ret>"; left-click the start point; left-click the end-point; "<ret>". You need to stretch objects made of polylines to make them bigger or to make them point a different direction.

When I am doing a one-line diagram, it always grows. As I add more information, I stretch the lines to make room.

CAUTION - Life is better when you have SNAP ON (F9) when you do a STRETCH.

Y

(M) UTILITY METER

UTILITY TRANSFORMER, 500kVA, 5%
12.5 - 480Y277V

400A

30A   175A   175A   175A   175A   175A

RECEPTACLE
TRANSFORMER,
45kVA, 5%,
480 - 208Y120V

PLANT ONE-LINE DIAGRAM
WITHOUT WIRE SIZES AND LABELS

Y

(M) UTILITY METER

UTILITY TRANSFORMER, 500kVA, 5%
12.5 - 480Y277V

12kA AVAILABLE   4-500kCMIL, 50-FT

400A

30A   175A   175A   175A   175A   175A

4-#10,#12G-2"C   4-2/0,#6G-2"C   3-2/0,#6G-2"C   3-2/0,#6G-2"C   3-2/0,#6G-2"C   3-2/0,#6G-2"C
10-FT   150-FT   150-FT   150-FT   250-FT   150-FT

80kA TVSS   LIGHTING PANEL   HVAC PANEL   MCC #1   MCC#2   RECEPTACLE PANEL

RECEPTACLE
TRANSFORMER,
45kVA, 5%,
480 - 208Y120V

4-#1,#6G-1-1/2"C, 10-FT

PLANT ONE-LINE DIAGRAM
WITH WIRE SIZES AND LABELS

MIRROR – "MIRROR <ret>". Left-click on the object to be mirrored, "<ret>". Define the mirror surface by left clicking, moving and left-clicking again. Answer "Y" to delete the original or "N" to keep the original. Window, cross and undo work. There is a hidden variable associated with MIRROR. Text sometimes mirrors in a hinky fashion, or you may want text to mirror in a hinky fashion.

MIRRTEXT - "MIRRTEXT <ret>". "0 <ret>" retains text direction. "MIRRTEXT <ret>". "1 <ret>" mirrors the text. What this doesn't mention is the text alignment. If you do it a lot, I am sure you can get text mirroring to work for you. For an occational user, it is easier to mirror everything except the text and re-do the text.

FORCING PROPER ALIGNMENT AND SIZE

CURSOR, GRID, SNAP (F9), OSNAP (F3), POLAR TRACKING AND INCH - The cursor on the AutoCAD™ screen is adjustable. What you see is how the last user left it. It makes drawing much easier to stretch the horizontal and vertical cursor lines to fill the screen.

"CURSORSIZE <ret>". Type in, "100 <ret>". You can align objects to one screen pixel by getting the cursor on top of an existing object and watching the "greyness" change. This is good enough for many purposes and can be improved by zooming in (which we will discuss later).

Grid puts reference dots on the screen. It has become flaky in newer versions of AutoCAD™ but is still useful, and essential for persons with astigmatism. You can choose dot spacing, but the program adjusts it when you zoom.

"GRID <ret>". "ON <ret>", works.

Snap is for forced alignment. It only lets the cursor land on permitted locations. I often use "SNAP <ret>". "4"<ret>", for four-inch spacing in plan drawings. You turn on snap with F9 and turn it off again with another F9.

I often use "SNAP <ret>". ".125"<ret>", for 1/8-in spacing on details I am copying from quad-paper.

Osnap is object snap. Instead of snapping the cursor to a fixed grid, it snaps to a point on a nearby object. "OSNAP <ret>" brings up the list of object points you can choose from. 99% of the time, I want to extend a line by snapping onto the end of an existing line. I key "F3" to turn on osnap, then do a standard PL command, but "F3" again before the final point or it will look for an object to snap on to.

The undo command works within PL, if you haven't been too fast in hitting your final <ret>.

Polar tracking limits cursor motion to +/- 0-degrees or +/- 90-degrees or any other angle. The 0 and 90-degree choices are easiest, angles well-off 0 and 90 are easy and angles near 0 and 90 are difficult (possible, but better to turn it off). I turn polar tracking on and off from the circle-angle icon on the bottom border of the screen. You can do it from the command line, "DSETTINGS <ret>". This is the same setup box you saw for the osnap alignment point selection.

Inch is a command I created when AutoCAD™ went hinky on distance displays. In the old days, you could ask for a distance and it would report in inches, to the 10,000th. Now, it rounds off to 1/16ths. You need to know the exact size of an object when you are trying to match mtext fonts using the scale

command. The easy way is to measure your text, measure the correct text, divide the numbers and use scale. Only, AutoCAD™ eliminated the useful distance report. So, I created a block that I can overlay on the text samples and get the exact height, at least to two places.

"INSERT <ref>". "INCH <ret>." Left-cursor on insertion point. "CHRPROP <ret>". Left-click inch block. "LA <ret>". "DEFPOINTS <ret>".
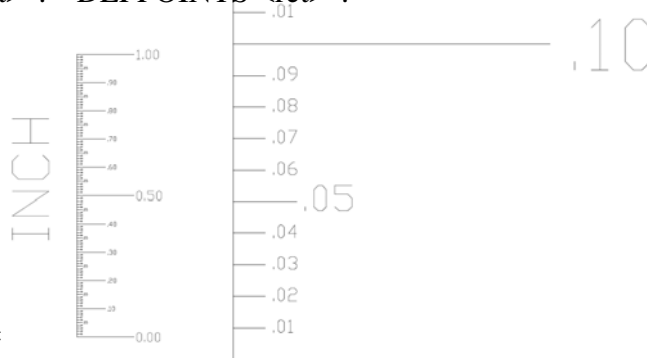
Where, INCH =

Slow down, Hoss. You're making fast and loose with some heavy-duty commands there.

You are right, of course. I introduce my inch scale because I got a lot of grief from a major client who kept complaining about my font size when I couldn't see the problem. To him, it was a big deal if the letter was .10, .11 or .12 high. I had to find a way to determine the precise height, with the distance command unusable.

This exercise starts with WBLOCK. Write block lets you save a piece of a drawing as a block. I use it whenever I have to invent something. I figure I will probably need it again, so I do a write block and e-mail it to myself. The command is "WBLOCK <ref>". Select object, select insertion point, select folder and filename, "OK <ret>".

After you have a saved block, you insert it by saying, "INSERT <ret>" browsing through your folders, selecting it and choosing a place to insert it.

Clients yell and scream if you leave the inch scale on the plotted version of the drawing. To avoid this, hide it. CHPROP changes the properties of an object. LA is the layer upon which the object resides. This is complicated, but we don't care, except for one very special layer, DEFPOINTS. Defpoints is reserved for defining insertion points, or anything else you want to use it for, but not plot. Defpoints does not plot.

Note that you can leave big text on defpoints for the next person who opens the .dwg file, but it will not plot and the Client will never see it.

Now, we have a measuring scale that we can see but won't plot and we know how to move it to the text we want to measure. We need PAN and ZOOM.

NAVIGATING THE SCREEN

Up until now we were working on a small part of the screen and everything we wanted to do fit in that small part. As the drawing grows, we must be able to move around the workspace, zoom in on details and zoom out to see how it is coming together.

PAN and ZOOM - On the oldest functioning versions of AutoCAD™ you had to say, "P <ret>" to pan and "Z <ret>" to zoom. Those commands still work. Today we use the scroll-wheel on our mouse. If you don't have a scroll-wheel, you can investigate P and Z on your own. The only remaining use is "Z <ret>"; "E <ret>"; that zooms to the extents. You do this at the end of an editing session to see if you accidentally dropped an object outside the border of the drawing. "Z <ret>"; "P <ret>"; takes you back to the previous view.

To pan, "Press the scroll-wheel and cursor around." To zoom, "Spin the scroll-wheel."

**FIVE FUN COMMANDS – PEDIT, MLINE, FILLET, TRIM AND EXPLODE**

There is probably a formal category for the following commands. They are very specialized editing commands. We group them here as fun commands.

PEDIT - Pline edit does exactly what it promises, but you are always surprised and almost always have to undo and try again. The first thing it does is convert whatever you select into a polyline (except circles). Next, it gives you eight choices of things to do with the new polyline. We will only examine WIDTH and SPLINE.

WIDTH changes the width of the polyline. This is forbidden by professional drafters. They recommend that each line be on a unique layer and each layer be defined by a unique line width. This works well for them, but I get confused. Instead, I draw my object and change the width to what I need. On the illustrations that introduced this course content, I changed the leader lines from the default of "0" to ".06-in". "0" really means the lightest that will reproduce. ".06-in" means that on a 1:1 reproduction, the actual line width will be .06-in. I usually try different numbers until I get something that I like.

By the way, real drafters do the same thing. You can create an underline on text from within the mtext formatting box, but real drafters don't like that line. They draw their own underline and modify its width.

I often force the width of the polylines on my illustrations so that they will reproduce better. I always use a donut where a circle would do because I can force the width of a donut but PEDIT doesn't work on circles.

SHELVES
COUNTER
WINDOW
BUMPER POST
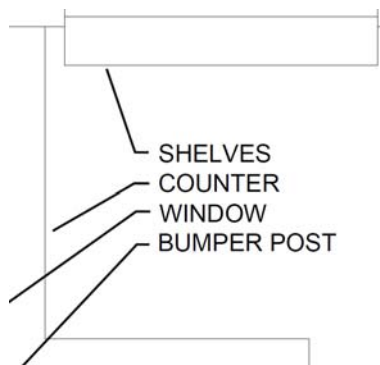
This closeup from the guard shack example shows plines, mtext and pedit on some plines.

PEDIT SPLINE fits a spline curve to the vertices of a polyline. A spline-fit curve is some kind of mathematical thing. I have seen demonstrations of different types of curve fits. Spline fit is the most attractive. It draws a smooth curve loosely approximating the underlying plines, somewhat near the vertices.
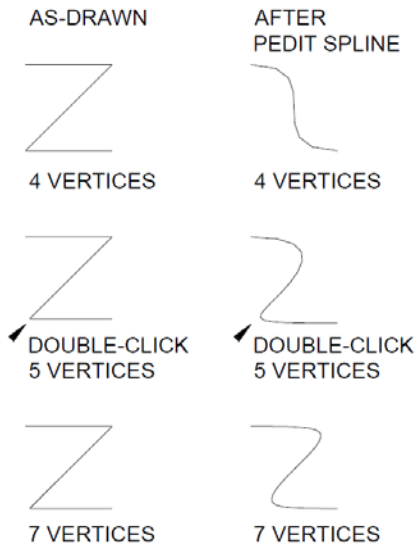

MLINE - Multiple polylines is the forbidden command. Every drafting supervisor I have worked for insists that we not use it. There are sub-commands within PL that do the same thing. Only, I have never learned the sub-commands within PL and MLINE is so much fun.

"MLINE <ret>". Left-click and drag down.
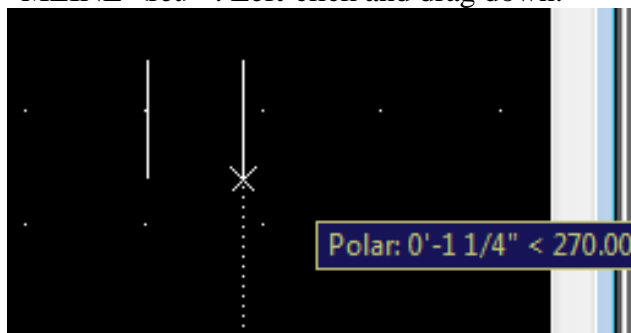
On my current pc, it starts drawing two lines. The cursor is turned off when I do a screen print, but you can see the floating information box, which is telling me that, so far, it has drawn something 1-1/4" in the 270-degree direction. The choice of two lines and the spacing was left over from the roads I drew for the graduation party map.
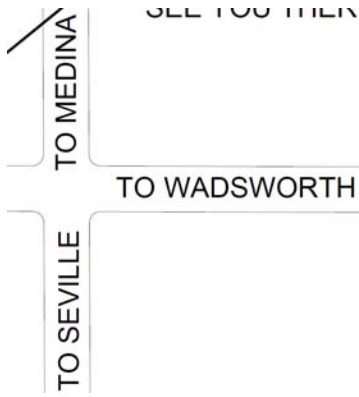
The intersection of SR 3 and Greenwich Road was done with two mlines, a circle, TRIM and FILLET, which we will look at soon.

This course is not going to go through the capabilities of the MLINE command, beyond telling you that it is forbidden and fun.

FILLET - Fillet refers to something machinists do with metal. For us, it is an arc connecting two lines. By default, the radius is zero, giving you a sharp corner. I usually pick a radius by checking a distance on the drawing I am doing.

"DIST <ret>". Left-click start; Left-click end; read the distance on the command line.
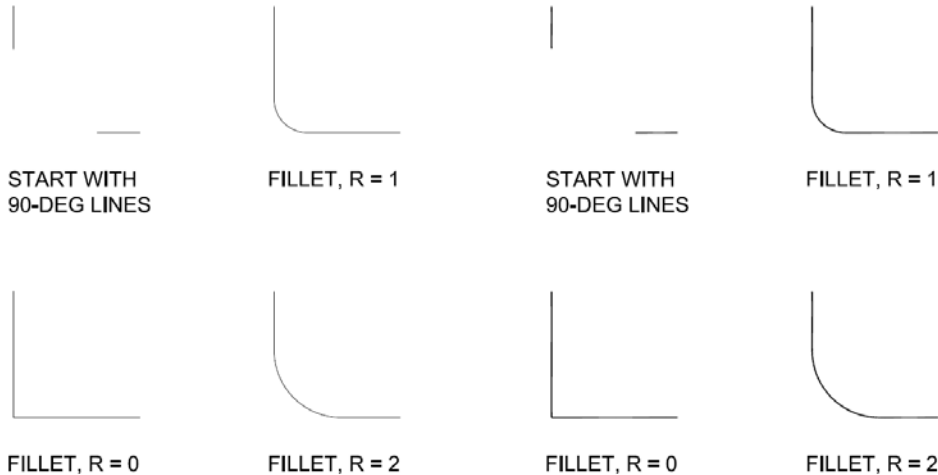

"FILLET <ret>"; "R <cr>"; "2 <cr>". Left-click first end-point. Left-click second end-point. You can get some very interesting figures from clicking in reverse order or clicking the other ends of the lines. I have always considered these forms wrong, but they certainly are interesting.
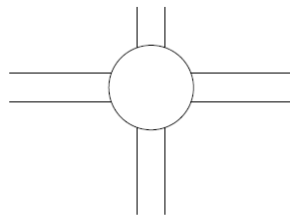
TRIM and EXPLODE - You can trim (almost) anything. You can only explode blocks (and some things are blocks that would surprise you). When you explode a block, you return it to the elements

that make it up. (An inserted block is an object and cannot be edited unless exploded.)
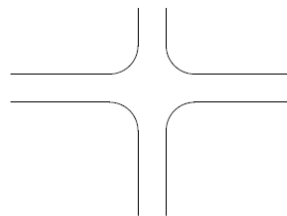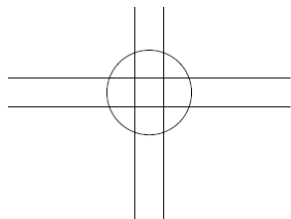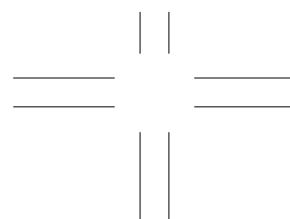
The intersection was created as follows:
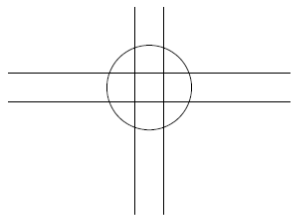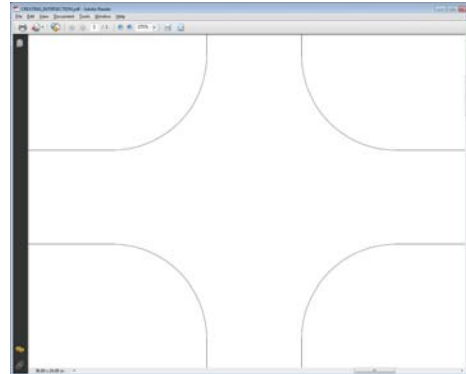

CREATING THE INTERSECTION

"X <ret>". Left-click on one mline; left-click on second mline; <cr>. (X is the command for explode.)

"E <ret>". Left-click on the circle. (E is the command for erase.)

Fillet as shown earlier.

The inserted blow-up of the intersection is to answer the question, "What is going on where the fillets meet the mlines?" Nothing is going on. It plots correctly, but Adobe and MicroSoft® have some trouble reproducing it correctly.

Adobe is one of the most quirky products on the market. To pursue this topic, download the content from PDHonline course G210, National CAD standards. I did a lot of tests for that course to find out how to get Adobe to do decent AutoCAD™ plots. And, they weren't transferrable to other plotters or printers. Adobe errors are called, "artifacts". MicroSoft® errors are called, "undocumented features".

PLOTTING
In paper space, first, do a "Z <cr> E <cr>. This zooms to extents. (This is a chance to clean up all objects that were accidentally dropped outside the drawing border.) You may only be interested in a small part of the drawing, but AutoCAD™ takes care of that. Exactly the same procedure is followed to plot the whole sheet or to plot only a portion.

Say, "PLOT <cr>". On my jobs, a lot or warning notifications come up. Answer, "No" or "Continue" to each. Our AutoCAD™ guru's won't explain to me what they mean except that it works fine without correcting whatever is bothering the software.

Next, a very complicated control screen comes up, similar to the following:
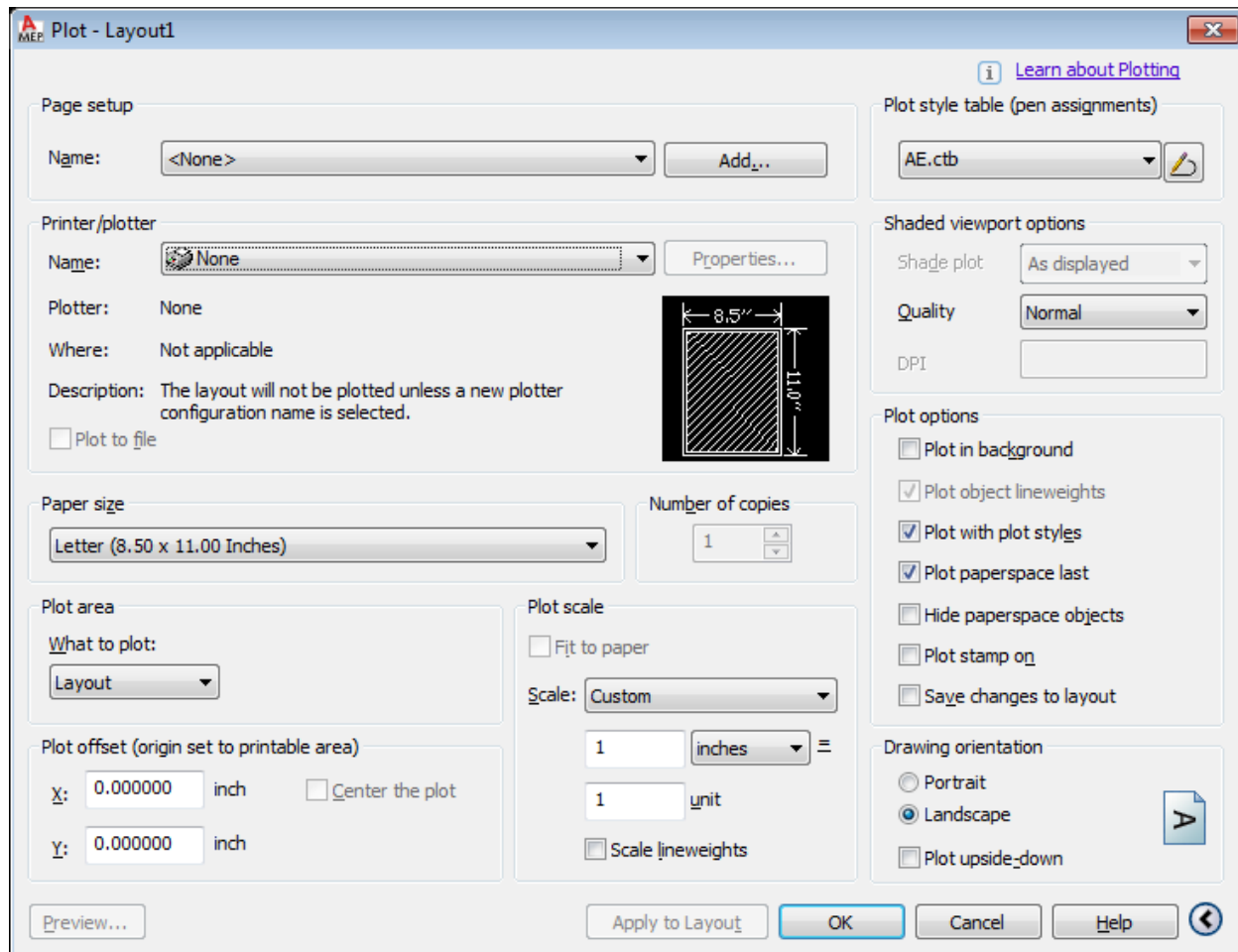
This is how it was left by the last plot, or at least, what is saved with the drawing. Note the second selection down, PRINTER / PLOTTER and the choice, NONE. It's not going to work without some adjustment.

The first and easiest choice it to go to NAME and choose PREVIOUS PLOT and see what comes up. We are not going to demonstrate that because there only a chance it will work for you. Instead, we will force values to make it do what you want. You want a .pdf that you can e-mail out. (You can also plot easily from a .pdf, so this is a good generalized solution.)

Plot to .pdf

The procedure is as follows:

1) Click on NAME. You want *CutePDF Writer*, or *do PDF* or *DWG to PDF*. CutePDF Writer is most reliable for me. It is a free download, but you want an AutoCAD™ specialist to install it if it is not present. Do PDF usually works, but sometimes it misses layers. That is, everything looks right, at a glance, but the VAV boxes are missing. It, too, is a free download. I have never used DWG to PDF.

2) Choose a PLOT STYLE TABLE. If you don't know about a special table for the project, use AE.ctb. It comes with AutoCAD™ and works very well. If there are defects, I suspect neither you or

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the recipient will be able to identify them.

3) Choose a paper size. CAUTION, Adobe uses low-resolution for small paper and high-resolution for big paper. Big paper is a better plot - both for the screen and for printing. I recommend size ARCH E1. There is no problem printing the result to 8-1/2 x 11 or 11 x 17 or 30 x 42. There is no problem using the ZOOM on Adobe and printing the current view to 8-1/2 x 11 or 11 x 17.

4) Choose what to plot. The safest is WINDOW. You use the mouse to select the entire drawing or whatever part is of interest.

5) Click CENTER THE PLOT and FIT TO PAPER.

6) Click PREVIEW and see if you are happy. There is a good chance you will want to click WINDOW and try again. Very rarely, the previous user has entered PLOT OFFSETS which move the information you want off the plot. Go back and set offsets to 0.

7) Click OK and wait. AutoCAD™ isn't very good at telling you that it is working. Making a .pdf takes some time (30-seconds or more). CutePDF Writer or do PDF will ask you what folder and what file name you want. I put mine in DOCUMENTS and move it later. I usually leave the .dwg file name, which the .pdf engine offers, but add my initials and the date.

8) Wait some more. The file should appear in your DOCUMENTS folder. Sometimes CutePDF Writer opens it for me on the screen. This is a good idea, whether automatic or manual. If the plot is bad, now is the time to do a good one.

THEORY - PAINFUL THE FIRST TIME
If you understand model space, paper space, viewports, layers and x-refs, skip this section and go to the next section.

For persons working with AutoCAD™ the first time, there are five very foreign concepts - model space, paper space, viewports, layers and x-refs. Almost all the time, model space is where we do design. If you start a new drawing, it opens a black sheet in model space.

Look at the lower-left corner of the drawing space and you should see two tabs. On my current machine, they are called, "Model" and "Work". For many years, they were called, "Model" and "Paper", then "Model" and "Layout". Regardless, usually, the design is in model space and the plotting border, or "frame" is in paper space.
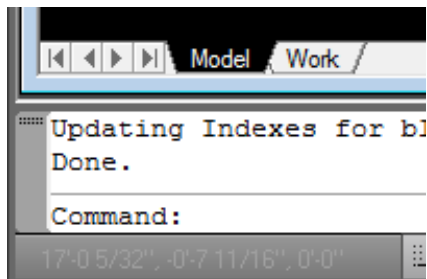
CRITICAL CONCEPT - Correct design problems in Model space; Correct border problems in Paper space.

For example - It is 5:30 PM Friday and you learn that power for the school is to come from a different pole than the one shown on the electrical site plan. The set goes to the printer Monday morning.

Open the site plan drawing. Go to model space. Grab the pole-end of the incoming power conduit and STRETCH it to the correct pole. Go back to paper space and save the drawing.

For example - It is 5:30 PM Friday and you notice that the title blocks haven't been updated from DD submission to CD submission. The set goes to the printer Monday morning.

Open the first drawing. [It is forbidden to open all the drawings at the same time. Some places have the machines locked so that you can't. On the other hand, it makes a task like this MUCH easier. Let's compromise and shift-select the first 10 that you have to revise.]

Title blocks are done two ways - with text or with block attributes. If it is text, you will know because when you move the cursor over the title block, each data entry gets highlighted individually. If they are blocks, then ALL data gets highlighted at the same time.

If it is text, then you copy the last submittal entry to the next open entry. Double click on the text and edit the revision number, date and say "CD SUBMITTAL". Leave the initials because you didn't change the design.

You are supposed to do this for each sheet, in turn. If you have naughty thoughts, then copy the new data to the pc clipboard [Ctrl-C left-click a window around the text and <ret>.] (Use of the Windows clipboard is another forbidden feature.)

Save the first corrected drawing.

Continue to the second drawing, and so on.

If it is a block, then you must edit the data content of the block. This isn't hard, but this example may be the only time you will encounter the command. On current versions, just double-click on the data and an edit window will open, letting you choose each line that you must correct or enter. On an older version, the command is "DDATTE <cr>; left-click a piece of the data" and a similar edit window should come up. Left-click OK when you are done.

Copy to the clipboard only works for one of the data values, but it does speed things.

Wait a minute. The drawing opened in model space, with no border. How do I get into paper space ?

"PS <cr>" takes you into paper space.

Also, you can click on the Work tab in the lower-left. Some drafters change the name of the

Work tab to the individual drawing number, as, "P-001". This is meaningful if the firm uses automated plotting. It is a nicety otherwise. To rename the Work tab, or to add more Work tabs, right-click on the Work tab.

We now know how to work in model space and in paper space, but a little discussion of the theory helps understanding. Model space for construction drawings is almost always a building background from the Architect, with plumbing fixtures, supply piping and waste piping added by the discipline designer. The term "model" is appropriate. As we advance to 3-D representation, it is even more meaningful.

We found the plotting border in paper space and that kinda makes sense. But, why separate the two. Why not make the border a block and insert it over top the building in model space ? That would work on a log cabin, but not on a 50,000 sq-ft school. The first floor of the school is huge. And, it is nice to do all the plumbing layout in one place, not start a new sheet for each 75-ft section of the first floor.

So, we do all the plumbing design on a single model space and cut a window into the paper space to look at the portion being documented in that sheet number. You can see now why you might want a number of different paper spaces for a single model space.

"Cut a window in paper space to look at a piece of model space?" Yes, it is called a viewport. It is the way we can have 10 different paper space drawings for a single model space building. Each paper space has a unique viewport. In fact, you can put two or more viewports in a single paper space.

Example - Making a plumbing drawing from a food service drawing.

The Architect goes to bed with a food service supplier (Kitchen Equipment Contractor – KEC). The supplier gets the contract on a no-bid basis. The project gets a finished Kitchen Equipment Plan, with exact spacings, model numbers, electrical connections and requirements and plumbing requirements. The plumbing designer makes a plumbing kitchen plan.

First, start the drawing. (In the real world, I always take an existing drawing and erase the content.) In paper space, x-ref in the border. You can identify the border by opening any drawing in the project and saying, "XREF <cr>" to see the list. One will be labeled BORDER or something similar. Copy the title block information from another drawing and edit it for the plumbing kitchen plan.

Second, go into model space and x-ref in the food service drawing. Usually, there is an architectural drawing of that part of the building that has the food service plan x-ref'd in. Go to the x-ref table and see the name and the full path used to import it. (You will use that path. Cut-and-paste doesn't work, though.) X-ref it in at 1.00 scale to location 0,0.

Go back to paper space and make the first viewport for the kitchen plan, "VIEWPORTS <cr> 1 <cr>" and left-click to define where you want the window, maybe the left 2/3 of paper space. It can be stretched later. Note that the viewport frame will be in whatever layer was current. Leave it that way. Towards the end, you will want to switch it to DEFPOINTS for a cleaner plot.

When you create the viewport, it is automatically opened - that is, your cursor is in model space through the window. You can work in model space, but not in paper space. Use the pan and zoom

controls on your scroll wheel to get the image you want. Usually, you end up doing a little bit of "Z <cr> 1.1 <cr>" and "Z <cr> .95 <cr>" to get exactly what you want. Say, "PS <cr>", to close the window into model space. You can double-click in the window any time to get in, but it is easy to screw up the zoom factor. It is better to use the Model tab.

Third, create a new viewport for the Plumbing Schedule. Same procedure as the plan view port. Again, a single view port. This time zoom in on the Plumbing Schedule that the KEC left for you.

All done.

Also, since it is an x-ref, whenever the food service drawing is changed or replaced, it is automatically changed on your plumbing sheet.

Did you pick up on the part of using the Architect's background for all of the electrical, plumbing, HVAC and technology drawings ? That sounds essential. But, wait! Architects change their minds. Do we have to erase the current background and insert a new one with each architectural change?

No. There is a command to "pull in" the Architect's background and update it every time it is saved. An "x-ref" is an external reference drawing that you want to put in model space. On my machine, I say, "XREF <cr>", and it opens a big table which lists all the of x-ref's presently in use. I want to add one, so left-click on the far-left icon which looks something like a sheet of paper with a paper clip. I get fours choices, but the first is ATTACH DWG. That's what I want. I left-click on it. It brings up a "Select File" window exactly like every other "Select File" window. Choose a folder and file name. Then it asks you if you want to scale the x-ref and where you want to drop it. Almost always, we scale to 1.00 and drop it in at 0,0.

Is this dangerous ?

No. It cannot harm the source file and anything you do to your own file should be recoverable. Worst case, CLOSE the drawing without saving. That takes you back to the last time you saved.

There are common mistakes, however. Three will be discussed here - paper space, layer and location. If you are in paper space when you remember to load an x-ref, it will load into paper space. You will notice because paper space is ~24-in x 36-in and model space is ~1,000-ft x 1,000-ft. The building background will be large-beyond-huge in paper space. Do a Z <cr> E <cr> to see. You can undo the zoom and undo the x-ref. If you get confused, go to the x-ref table and right-click on the x-ref you don't want. Choose DELETE.

Go into model space and try again.

Layers can be a difficult concept. The basic idea is that life is better if you can segregate the HVAC trunk ducts from the HVAC branch ducts from the HVAC terminal boxes from the thermostats from the DDC panels from the air handler enclosures from the air handler dampers from the air handler motors.

So, we do. The software that is AutoCAD™ keeps a separate list for trunk ducts, for branch ducts, for terminal boxes, for thermostats, for DDC panels, for air handler enclosures, for air handler dampers and for air handler motors. Each separate list is called a layer. It can be turned on to show the trunk ducts or turned off to hide them. You can make trunks a different color or line type or line width than

branches. Keen? Huh?

The bad part is that you can only work in a single layer at one time. That is the current layer. When you drop in a building background, it goes into the current layer. Maybe you don't want the building background to turn off and on with branch ducts.

There are four popular answers to the question, "What layer for the building x-ref?" The first is a unique layer, maybe named, "X-REFs". Some people do this; it works well. I usually drop in the x-ref, then, "CHPROP <cr>; LA <cr>; X-REFs <cr>".

The second answer is Layer 0. In the very, very old days, Layer 0 was the "Construction Layer". The idea was to do the drafting in Layer 0, then CHPROP it to the correct layer. Anyway, Layer 0 is still there and a good place for x-refs.

The third answer is TRUNK_DUCTS. That is, many drafters just drop in the building background wherever they happen to be working. Really.

The fourth and most interesting is DEFPOINTS. As mentioned earlier, DEFPOINTS looks good, but doesn't plot. Whenever the plot comes out missing the building background, you can guess it was inserted on DEFPOINTS.

Getting more complicated with x-refs.

The procedure for making a plumbing set of drawings which was described above is as follows:

1) Architect draws building walls and doors and names each space.
2) Plumbing designer pulls background into model space as background.
3) Plumbing designer places urinals and commodes and sinks and waste piping and supply piping on individual layers on top of the building background.
4) Plumbing designer creates a separate paper space for each section of the building. Each gets a unique paper space tab, unique drawing number on the border he pulls in and a unique viewport for the section of the building being documented on the particular sheet.

This works well. Actually it works super-well if the firm uses automated plotting, which is built into AutoCAD™.

However, this is not how it has been done for about 5-years on the jobs I work on. Instead of a single drawing with multiple paper spaces for all the plumbing, there is an intermediate model, called the "model", for the design. Each of the paper spaces can be on a separate .dwg file or on lots of different .dwg files.

The power of this procedure is that one person can work on the design, model space drawing, while someone else works on the paper space. Or a different person can work on each paper space .dwg. Because of x-refs, it all stays coordinated. AutoCAD™ actively supports this by telling anyone working on a drawing with x-refs any time any of the x-refs is updated. In the information balloon at the bottom, there is a link to force update to the newest x-ref.

This procedure is as follows:

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1) Architect draws building walls and doors and names each space.
2) Plumbing designer pulls background into model space as background.
3) Plumbing designer places urinals and commodes and sinks and waste piping and supply piping on individual layers on top of the building background.

STOP. This .dwg file with all the plumbing design is saved as, "First_Floor_Plumbing_Model.dwg".

4) Plumbing designer creates a separate .dwg file with a paper space for each section of the building. She pulls in First_Floor_Plumbing_Model to model space for each .dwg file.   Each gets a unique paper space tab, unique drawing number on the border she pulls in and a unique viewport for the section of the building being documented on the particular sheet.

This procedure works well and division-of-labor is supported.

What do you do in paper space beyond the border and sheet information ?

This is the footnote for the comment at the beginning of this discussion that "most of[1]" the design is done in model space.

This is theory and provided here only to explain the benefit of separating the design model from the paper space sheet.

We referred to plumbing design as spotting urinals and commodes and sinks and waste piping and supply piping. Actually there is more - call-outs and general notes and sheet notes. Further, there is a feeling that call-outs, general notes and sheet notes should be in paper space, not model space.

To the extent this is followed, there is a LOT of design in paper space.

The alternate thought is that this is unnecessary complication. These people put the call-outs, general notes and sheet notes on the model drawing and use viewports to bring them into each paper space, as needed.

Specifically, call-outs are on the objects being identified, in mtext. General notes are in a column far off to the right of the background. (Most firms have drafting standards which call for notes to be along the right border in paper space.) Individual sheet notes are anywhere convenient, off the background, and identified by the sheet where they go. When you put in a viewport for the sheet notes, you do not include the sheet identifier.

GOTCHA's

This section describes inscrutable problems and helps you scrute them.

LOCKED VIEWPORT - It was mentioned that Architect change things on the background. This creates a need to change the viewports on the electrical, mechanical and technology sheets.

It should be easy to stretch a viewport, "STRETCH <cr>"; left-click a cross-box on the area to be stretched; <cr>; left-click and drag; <cr>. Only, sometimes, AutoCAD™ ignores the stretch command. It almost always turns out that someone locked the viewports.

The command to unlock the viewport is readily available in current versions of AutoCAD™ but was hidden previously. If you can see the viewport border, left-click on it to select it then right-click for a menu and choose PROPERTIES. On my machine, I come up in the DESIGN tab and in the third group, MISC, is a line, " Display locked   YES". Left-click on the YES and you get a pull-down. Select NO and close the properties table.

If the viewport box is on DEFPOINTS and DEFPOINTS is turned off, then you have to learn a little about the LAYER MANAGER. We have avoided this topic because it can get very complicated, and most of the time, you don't care. Unfortunately, now we do.

Type, "LA <cr>". A really complicated screen comes up. Sometimes it has been minimized, so you have to expand it on the tree at top-left. Left-click on all the + symbols to see all the layers.

There must be a layer DEFPOINTS. AutoCAD™ won't let you delete it. The first three columns after the name are ON, FREEZE and LOCK. We need to see the lightbulb, see the sun and see the open end of the lock. Click on the icons until it looks this way. Close the Layer Manager. On some versions you can only X-out at the top, on others, you must say OK at the bottom. If the viewport box didn't come on, go back to the Layer Manager and make sure it heard you on DEFPOINTS, then look for a layer named VIEWPORTS, or something similar.

Be careful. You can't get into trouble on DEFPOINTS because it is physically incapable of printing. VIEWPORTS or something similar will print if you turn it on. Make sure you turn it off again when you are done.

Keep being careful. As you were fussing with DEFPOINTS, you may have made it the current layer, with the green checkmark in front of it. That means that anything else you do will be on the non-printing DEFPOINTS layer. It is safest to double-click Layer 0 to make it current.

FORCED LINE WIDTH - Sometimes you try to draw a polyline and you get a monster-wide line. That means that the last person locked the line-width. This usually comes from one of two sources - retained PL setting or forced property. Say, "PL <cr>"; left-click a start point; STOP. Look at the command line:
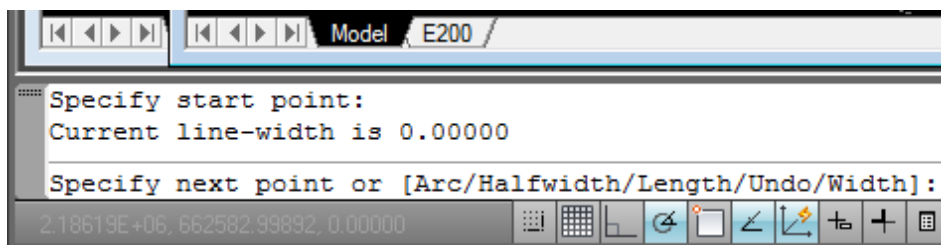
Now, we are talking about a problem with a huge-wide polyline. That means that, "Current line-width is NOT 0.00000". But, it gives you a choice for width, so, say, "W <cr> 0 <cr>". It will ask for end-width, and "0" is the answer again. Now polylines will be the normal zero width.

The second source of huge-wide polylines is a forced value in the line properties. You fix this just like you fixed the locked view port. Left-click on a line to select it. Right-click on it to get the menu. Left-click on PROPERTIES. On my machine, I have 9 GENERAL properties. Color should be ByLayer. Lineweight should be ByLayer.

FORCED LINE COLOR - This is a line property which has been forced on the PROPERTIES list. As above, get to the list; change color to ByLayer.

This is especially interesting because color forced on the layer list cannot be modified by CHRPROP.

Color may seem to be a trivial attribute, but some persons have a lot of trouble seeing colors other than yellow, green and cyan (light blue). Dark blue and magenta are invisible against the model space black background. (The Kitchen Equipment Contractor who is sleeping with our Architect uses forced-magenta. The designer I replaced used a lot of dark blue.)

Table (lc) - (bad). Table, lower-case, refers to drawing lines to make a form and creating text to enter data. You MUST use SNAP. If you are willing to invest hours of effort, you can get a presentable table. Save it using WBLOCK.

TABLE - (bad). Table, upper-case, is a new command in recent versions of AutoCAD™. The bad part is that it does strange things on its own and is difficult-to-impossible to edit. For a small, quick-and-dirty tables, it produces a product which is "almost right". As the table gets larger and you do more edits, it deteriorates.

I turned in a really ugly table as part of a recent project. For unknown reasons, some of the data cell text changed size. We could find no way to edit it back to match and didn't have time to erase and re-enter it. I had several very sharp drafting supervisors helping.

EXCEL OLE - (tolerable). Most designers I deal with do equipment schedules in Excel, then cut-and-paste them into paper space on a drawing. One fellow pastes into model space then does a viewport from paper space. This usually works. Editing usually works.

As best I can tell, AutoCAD™ saves the data and format with the .dwg file. You can take it to a different machine, on a different network, double-click on the table and it will open the file in Excel, where you can edit it. When you go back and click anywhere on the .dwg screen, it updates the AutoCAD version. That is when it works.

When it doesn't work, you can't get AutoCAD™ to open Excel. This is on bad days on my normal workstation, not when I am visiting down the street. Often, AutoCAD™ insists on opening the properties list. Close that and double-click again. After a few tries, the swirling MicroSoft "Wait" icon appears, then Excel opens on the file.

Sometimes, it won't save to the AutoCAD™ .dwg file. You click a different cell in the Excel to make sure the machine has recorded your last change. You click on the .dwg screen, and, it doesn't update. I

was in this condition for 30-minutes one day. I panicked, thinking that I had lost my changes and would never be able to edit tables on my favorite workstation ever again. I don't remember if it finally did transfer or if I shut it down and went home. Anyway, it mostly works right since then.

My current disaster with AutoCAD™ and Excel is down-the-street. I am doing a school design for a small firm. I took my Excel Panel Schedules with me on a thumb drive. I opened one in Excel and did cut-and-paste to make – Panel schedules on a sht. I edited the headers to identify each panel, the location, the AIC rating, the main breaker and the voltage level. I entered the TVSS on each of the 208Y120 panels. It was looking good.

The next time I opened the sheet, on the same machine, all the panel schedules were squooshed horizontally. They don't stretch horizontally, only by maintaining the wrong lxw ratio. I turned in that sheet as part of the last submittal and haven't heard back or figured out how to fix it.

Don't mess with PE seals.
There are laws and regulations and firm rules on the use of PE seals on a drawing. They are not consistent or widely enforced. However, there is significant liability if something goes wrong and there is a seal on a sheet which shouldn't have it or if there isn't a seal when it should have had it. Use your best judgment or check with someone you trust, but my rule is to "leave it the way I found it".

Finding your own AutoCAD™ solutions.
First, AutoCAD™ documentation is terrible. You can be pretty sure you won't find your topic, won't find an answer and won't understand the answer if you find one. Until recently, most firms had a copy of a third-party book on AutoCAD™. They cost $40-60 and are very readable. Except for the time to use the Table of Contents and Index, this is a good solution. Today, however, the internet can answer your question, usually within the first three hits from a Google search. Use the form, "AUTOCAD LOCKED VIEWPORT". Google will fix it if you spell it wrong. When the answer really is tricky, I cut-and-paste from the site into an e-mail, which I send myself. For example, there is a command within AutoCAD™ to block out part of a drawing. It can be used in model space or paper space. It can be tiny or huge. You can enter new text or objects on top of it. I keep remembering it as "WHITEOUT". That is wrong. The correct command is, "WIPEOUT". If you do a Google search for, "AUTOCAD WHITEOUT", it comes back with hits which include details on how to use WIPEOUT.

This concludes the content portion of the PDHonline course on Minimal AutoCAD™-2011.