CEE 4020 - Computer Applications in CEE

Labwork for Weeks 1 and 2

September 3, 2010

Displaying, Dimensioning and Scaling Information

For the most part, the standard dimension conventions are fine for our purposes. However, we can create a new dimension style for our drawing so that we have two different dimension styles to choose from. The new dimension style can be created by executing the *DIMSTYLE* command to create a new dimension style named *MyStyle*. As usual we will start with the Standard style as our template. The following steps are recommended:

- On the Lines and Arrows tab of the Dimension Style window, set the Dimension Line and Extension Line Color to ByLayer

- On the Primary Units tab, choose different unit formats, levels of precision, zero suppression and watch what happens in the drawing in the right corner of the window. Set the Unit format to Engineering, the precision to 1 decimal places. Make sure that none of the boxes in zero suppression are checked.

- On the Fit tab, change the Scale for Dimension Features by choosing an overall scale of 20.00

- On the Alternate Units tab, check Display alternate units, choose 0 decimal places of precision and type mm in the box following suffix.

- Do not change the text height - instead use *DIMSCALE* to scale all text size. This can be also done by setting the appropriate scale in *Overall Scale*. For this drawing use 20.

- Note changes in *DIMSTYLE* do not apply retro-actively. So please ensure you set the style up before you start dimensioning.

In the Dimension Style window, set the new dimension style to be the current dimension style. The most important thing about dimensioning is to set the dimensioning scale correctly before you start drawing the dimensions. Think about the scale at which you will be printing your drawings. For example, if you are drawing an object that is 6 in. by 4 in. and using letter-sized paper for your drawing, you will probably print the drawing at a scale of 1:1 (1 modelspace-unit equals 1 paperspace unit). If you are drawing an object that is 10 ft. by 8 ft. and letter-sized paper, you will probably print the drawing at a scale of 1:12 (1 in. in paperspace equals 1 ft. in modelspace). The easiest way to make sure that all of the dimension features (text, arrow heads, etc.) are sized appropriately is to use the standard sizes set as default in the Line and Arrows and Text tabs of the dimension window and to use the Scale for Dimension Feature in the Fit tab to scale all of the dimension features uniformly.

Linetype scaling:

- Use the *LTSCALE* command to control the appearance of different kinds of linetypes in Model space. For this drawing *LTSCALE* to 1.

- Use the *PSLTSCALE* command to determine if linetypes appear and plot the same way in layout as they do in Model space. If *PSLTSCALE* is set to 0 then the line scaling in a layout (and in a viewport) is the same as in the Model space. As a result, linetype scaling is exclusively controlled by *LTSCALE*. If *PSLTSCALE* is set to 1, then the linetype scale is entirely dependent on the viewport scale. Note: different layouts can have different scale settings. So when the drawing has more than one layout and viewports, then a setting of *PSLTSCALE* to 1 is suggested. Else it should be set to 0.

Lab Exercise 1:

Objective: To develop appropriately scaled layouts that can selectively display information on different layers: *class_09_01.dwg*. In this drawing you have already completed drawing the machine tool. Now continue with the following steps:

1. Import a title block from a drawing template:

    - Right click on the *Layout* tab and choose the option *From Layout ...*
    - Navigate to the *ce4020.dwt* file and select it.
    - Choose the *MyLayout* tab.

2. Dimension the drawing appropriately using the above information.

3. Create 2 independent polygonal viewports, and use scale factor of 0.04 in one and a scale factor of 0.02 in the other:

    - Make sure you are *in* the viewport (by double clicking inside it) and then from the command line use the *Zoom* command with *0.4 XP*.
    - Alternatively, in paper space, right click on the viewport and in the Properties tab set custom scale to 0.04. Note the following:
      1:25 implies: 1 unit in PS = 25 units in MS, or XP = 1/25, or XP = 0.04

4. In the first viewport do not display the circles layer: Open layer manager and freeze the circles layer by clicking on the *VP Freeze* button.

5. Produce two print-outs - as attached.

Some Commands Worth Noting

The AutoCAD commands *array* and *mirror* are advanced copy commands. The array command is a powerful alternative to simple copying. An array is the repetition of an image in matrix form. This command takes an object or a group of objects and copies it a specific number of times in mathematically defined, evenly spaced locations. There are two types of arrays. Rectangular arrays are linear and defined by rows and columns. Polar arrays are angular and based on the repetition of objects around the circumference or arc or circle. When we use arrays the following dialog box comes up.

```
Command: _mirror
Select objects: 1 found
Select objects:
Specify first point of mirror line: Specify second point of mirror line:
Delete source objects? [Yes/No] <N>:
```

A *spline* is a smooth curve passing through a given set of points. AutoCAD uses a particular type of spline known as a NonUniform Rational B-Spline (NURBS) curve. A NURBS curve produces a smooth curve between control points. The spline command allows you to identify the control points that are used to generate the spline, the tolerance used in fitting the spline through these points and the start and end tangents.

```
Command: spline
Specify first point or [Object]:
Specify next point:
Specify next point or [Close/Fit tolerance] <start tangent>:
Specify start tangent:
Specify end tangent:
```

The AutoCAD command *mline* draws multiple parallel lines. The command is similar to the line command (except multiple lines rather than a single line are drawn): you must specify two points that are the end points of the line segment and having drawn one segment, you may continue to draw line segments until you type escape or enter. With *mline* you have the option of specifying the line STyle (which defines the number and distribution of the of parallel lines), the Scale (which scales the spacing of the lines) and the Justification (which defines the position of the multiline relative to the endpoints you define in the drawing window either above, below or centered on the points you define). The command line trace is:

```
Command: mline
Current settings: Justification = Top, Scale = 1.00, Style = STANDARD
Specify start point or [Justification/Scale/STyle]:
Specify next point:
```

The AutoCAD command *mlstyle* is used to draw a new multiline styles. A particular mulitline style may have n parallel lines, of variable line type (this is probably the only place that it is appropriate to set a line type that is not defined by layer), with specific end caps, and may or may not be filled. These attributes, as well as the multiline name, are defined using the mlstyle command. To draw a new style, you need to draw the attributes of the new style and then use the Add button to add that style to the drawing. If you use the Save button you can save the line to a file for reuse in another drawing.

The AutoCAD command *pline* draws a polyline. The polyline is a sequence of lines and arcs. Unlike lines and arcs, with the polyline you can draw a continuous polyline that includes arc segments and straight line segments. Each polyline segment can have a start and an end width. A typical command line trace of the pline and pedit command is:

```
Command: pline
Specify start point:
Current line-width is 0.0000
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: a
Specify endpoint of arc or
[Angle/CEnter/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]:
Specify endpoint of arc or
[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: r
Specify radius of arc:
Specify second point:
```

The *pedit* command can be used to edit polylines and/or a set of multiple lines. The command line sequence for pedit is as follows:

```
Command: pedit
Select polyline or [Multiple]:
Enter an option [Open/Join/Width/Edit vertex/Fit/
Spline/Decurve/Ltypegen/Undo]:
```

Blocks

A block is a set of objects formed into a compound object. AutoCAD treats a block as a single object; thus the block can be copied, moved, rotated, scaled, etc. A block is defined by specifying the name of the block and then the set of objects that compose the block. The block name is used whenever you want to insert the compound object (block).

A block can be composed of objects that were drawn on several layers with several colors and linetypes. The layer, color and linetype information of these objects is preserved in the block. Upon insertion, each object is drawn on its original layer with its original color and linetype, regardless of the current drawing layer, object color and object linetype. There are three exceptions to this rule:

- Objects that were drawn on layer 0 are generated by using the current layer when the block is inserted, and they take on the characteristics of that layer.

- Objects that were drawn with color BYBLOCK inherit the color of the block (either the current drawing color when they are inserted or the color of the layer when the color is set by layer).

- Objects that were drawn with linetype BYBLOCK inherit the linetype of the block (either the current linetype or the layer linetype, as with color).

Typically, we will draw blocks on the 0 layer and set block object properties to be BYBLOCK. This gives us the most amount of flexibility in using the blocks.

Blocks defined with the *block* command (executed from the toolbar, menus and command line) are stored only in the current drawing and can be used only in the current drawing. The *wblock* command (executed only from the command line) is used to define blocks for use in multiple drawings, the wblock command saves the selected block as a separate drawing file.

Lab Exercise 2

In this lab exercise, we will use blocks to create the following block library of foundation elements 1 and the following drawing of a foundation plan 2.

Download the files *foundationElements.dwg* and *pileGroup.dwg* from the class website. These files contain drawings of the foundation elements shown Figure 1 and you will use these drawings to create blocks for each of the foundation elements shown in Figure 2. *pileGroup.dwg* contains a drawing of the two-pile foundation element shown in Figure 2. Make sure that all of the objects are drawn on the 0 layer and that all of the objects have color, linetype, linewieght set to BYBLOCK.

1. Use the block command to create a block for the foundation element, name the block *twoPileGroup* and choose a reasonable point on the pile cap (probably the center point, which you may need to locate with some construction lines) as the base point.

2. To confirm that the block was created, use the *insert* command to insert a copy of the foundation element in the file *pileGroup.dwg*. When the block insertion window opens you should see the block name in the box following Name. On which layer are you inserting the block? What are the properties of the piles and pile cap that comprise the block?
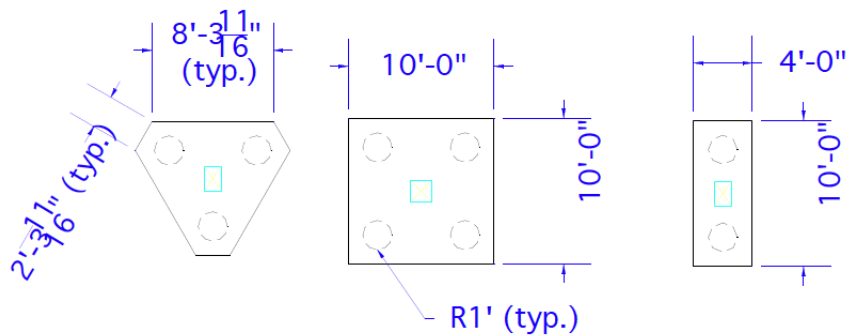
Figure 1: Typical foundation elements: Triangular, square and rectangular pile groups and pile caps

3. Use the wblock command to write the block so that it is available for use with other drawings. Make sure that you write the blocks in a file location from which you can easily retrieve it.

4. Open the file *foundationElements.dwg*. This file contains drawings of the other two foundation elements shown in Figure 2. Add the two-pile foundation to the drawing using the insert block command. This time you will need to browse to find the file *twoPileGroup.dwg*. Use the explode command to explode the block and change the properties of the piles and pile cap so that each is assigned to the correct layer. Create a new block for the two-pile foundation as well as two new blocks for the other two foundation elements in the file. Save the revised file.

5. Open a new drawing in which you will draw the foundation plan shown in Figure 2. Name the drawing *lastname_lab_2_foundationPlan.dwg*. Insert the block library using the insert command. To do this you will need to browse to find the *foundationElements.dwg* file that contains the blocks, Open the file for use and click the Okay button to indicate that you want to insert the foundation elements. You will see all of the foundation elements in the drawing window and will need only to click the mouse to insert all of the elements. If you click on the drawing window, all of the foundation elements will be inserted. Instead of inserting all of them, type Esc. Now, all of the foundation elements blocks should be available for you to use in the new drawing. To confirm this, use the insert command and verify that all of the block names appear in the box following Name in the insert window.

6. Use the foundation elements to create the foundation plan drawing shown in Figure 2 Before you start your drawing, think about how you can use the array, multi-copy, offset, and mirror commands to draw the foundation layout using only a few commands.
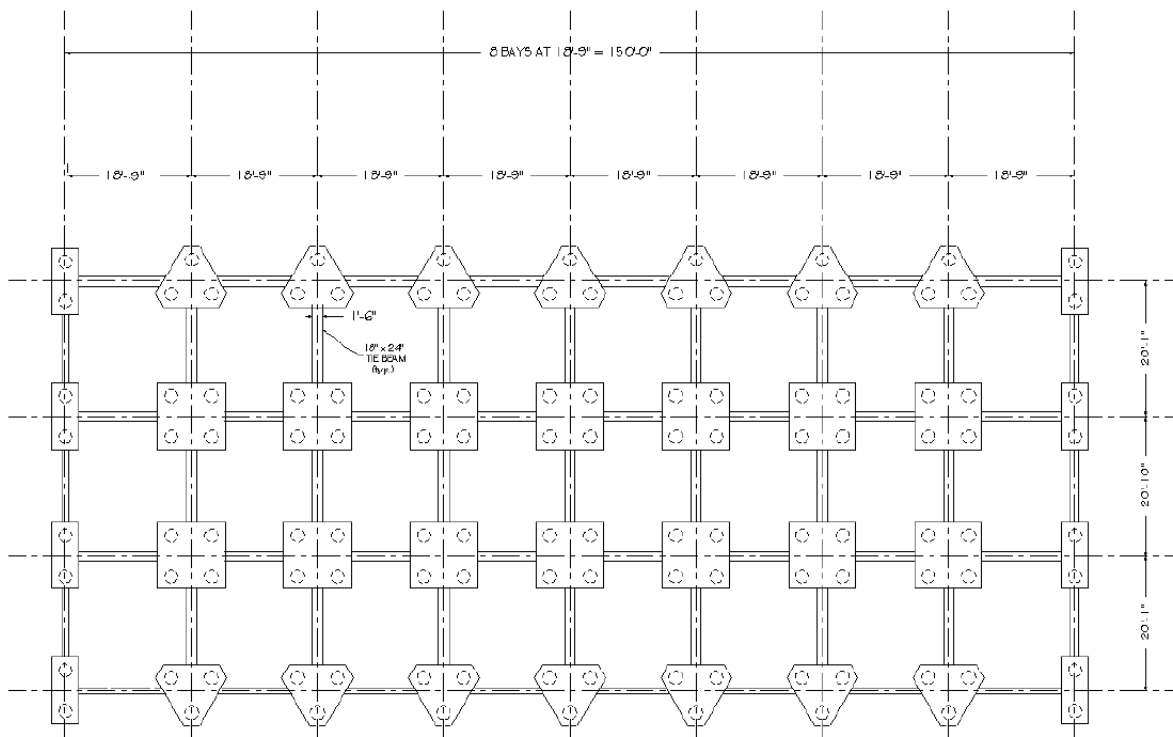
Figure 2: Foundation Plan