CAD Notes: The commands will be typed on the Command line located near the bottom of the workspace, and then press the Spacebar or Enter. You can also use the right-click key on the mouse to enter a command. If you right-click and go down to Recent Input, you can activate a previous command used.

1. **Text Style (ST)** – This command is used to open and modify the text styles within AutoCAD. Type (ST) > Enter > Modify the font, size and effect > Click Apply and then close out of the dialogue box. Choose fonts that are easy to read and ones that begin to stylize how you draw and add notation to a drawing.

2. **MText (MT)** – This command is used to create a multiline text object. Type (MT) > Enter > Click the starting point on the screen and then create a text window by clicking a second time. The dialogue box will open and you can begin typing.

3. **Purge (PU)** – This command is used to remove unused items, such as blocks definitions, layers, dimension styles, etc. Type (PU) > Enter > Check the unused items that you want to purge or delete from the drawing > Click Purge Checked Items or Click on Purge All as many times necessary in order to purge the drawing of all unused items.

4. **Audit (AU)** – This command evaluates the integrity of a drawing and corrects some of the errors within the file that are detected. Type (AU) > Enter > Type Y on the command line and press Enter again.

5. **Explode (X)** – This command is used to break a compound object into its component objects, such as an array, a block or some text. Type (X) > Enter > Select the object you want to explode > Press Enter.

6. **Scale (SC)** – This command is used to scale an object. You can scale an object using a scale factor. For example, if you wish to scale an object and reduce it to half of its size, Type (SC) > Enter > Select the object > Enter > Select the base point of the object > Then type 0.5, and the object should reduce to half of its original size. To scale an object based off of a reference length, Type (SC) > Enter > Select the object > Enter > Select the base point > Type R > Enter > Select the base point or first reference point, then the second reference point of the original length of the object, then simply type in the actual size you want the object to become.

7. **Dimensioning (DIMSTY)** – This command is used to open the Dimensioning styles within AutoCAD. Type (DIMSTY) > Enter > The dimension style manager will open. Select or highlight the "Annotative" style as a go by > Click New > Rename the new style "Dim-12" because this will be the starting point for our dimensioning. > Click Continue > Modifications can be found below for each tab below; however, the only thing that changes for new dimension styles will be the "Overall Scale" found on the Fit Tab: Architectural and Engineering Scale Factors can be found by following this link.