Starting a new drawing using any CAD software requires a series of steps. Measurement units, sheet size, layer designations, text fonts and text sizes plus many more items must be set. One setup is needed for fractional-inch drawings while a different setup is needed for metric drawings. There is a definite sequence because later settings may be influenced by earlier settings. Much of this work may be saved and re-used by creating template or “seed file” drawings.

All engineering drawings are placed on title blocks. Custom title blocks may be drawn and saved.
**Model Space** is used for creating drawings using real units and distances. The drawing screen must be set to a size large enough to accommodate a 3" x 4" machine part or a 100' x 300' building. All work is done using actual distances. At plot time the drawing is scaled to fit the page.

**Model Space setup is configured 1 to 1 with the World.**
A number of parameters must be set before beginning a drawing. AutoCAD and other CAD software must meet the needs of a wide variety of drawing styles and tasks. These settings are generally performed in order because some settings are dependent on other settings. This process is lengthy at first. Much of the work can be captured as a template. Prototypes may be created for decimal-inch, metric, architectural, surveying, etc., drawings. One set up, starting a new drawing is as quick as specifying the proper template.

**Layout (Paper Space) is configured 1 to 1 with the plotter.**
Custom title blocks may be created for each output device. One title block would be appropriate for a 8 1/2 x 11 laser printer while a completely different title block would be needed for a 34" x 60" Hewlett-Packard™ plotter. Each title block is optimized for size and print output. Multiple drawing views may be placed in a single page using **viewports**. Each view may be shown at a different scale.

**Drawing setup sequence.** This sequence is very important. Select:
- **Units.** Always work in the units appropriate for the job. Drafters should never have to convert units while drawing. Let the computer do the work.
- **Limits.** Always draw full scale. This is a different concept from paper drawing. AutoCAD software has the ability to create a drawing of any extent. (Hundreds or thousands of miles if needed.) No one should attempt to re-calculate dimensions in order to “draw to scale”. Scaling is the last step just before plotting. Again, let the computer do the math at that time.
- **Snap and Grid.** Snap is a tool not found on paper drawings. It will force points to an accurate, invisible grid if invoked. Snap is a very accurate guide. Grid is similar to the printed grids on engineering sketch paper. The grids are only a visual reference. Grids may be turned on/off as needed. Snap and grid may be the same value or each may be set to a different value.
- **Layers.** This is another concept not found on paper drawings. Layers are like clear plastic overlays, each of which may contain unique drawing information. Any number of layers may be used. Unique colors and linetypes may be assigned to layers. Standards have been set for layer names and uses.
- **Linetypes.** Drawings often use solid, dashed, center, etc. linetypes. Broken linetypes must be scaled according to drawing size.
- **Text.** Text height must be set depending on drawing size. Text styles may be set based on company standards.
- **Other settings.** Later, point styles, dimensioning, crosshatch patterns and other parameters will be introduced. Each of these functions require a setting related to drawings size and plot size.
Create a New Drawing

Starting a new drawing or selecting File New … will display a New Drawing dialog box.

On some versions of AutoCAD a dialog box will display startup options. Newer versions will display the options if the system variable Startup is set to "1". ("0" = no "1" = yes).

Command: startup

Enter new value for STARTUP <0>: 1

Use a Wizard has two options.

1. Quick setup with two steps.
   Units.
   Limits (drawing size)

2. Advance setup with 7 steps.
   Units
   Angle Measure Units
   Set Angle Zero direction
   Angle Measure Direction
   Limits (drawing area)
   Title Block
   Paper/Model space

Existing templates have many parameters pre-set. Some are for Model Space setup and others are for Layout (Paper Space) setup.

Acad.dwt is the default Model Space template when AutoCAD is started.

Acad.dwt has only the minimum settings required to start a drawing. New templates may be created by modifying this template and saving as a new name.
Use a Template

AutoCAD and AutoCAD LT include a number of pre-set drawing layouts. These are based on various U.S. and Metric standards. This makes starting a drawing with many parameters set very easy.

Custom Templates may be created with layer names, company name and other items included. Most companies will have their own templates ready for use. Exercises at the end of the chapter will show how to set up template files.

Start from Scratch

This option starts a drawing with minimum settings.

Two options are available:

- English units - about 9”x14”
- Metric units - about 300mm by 500mm

No title blocks are included.

To understand all the settings, the next pages will go through the steps in setting up a drawing template.
Start from Scratch - Creating new drawings and custom Templates

1. Units

Set Units first. Other settings depend on this choice.
Command: UNITS or, From the top menu pick Format.
Click on Units.
The dialog box shown allows quick choices for Units and Angle display.

Note: In AutoCAD, METRIC units are Decimal units.

Early versions of AutoCAD used a text display to format the settings. Type Units.
The window changes to a text screen as shown. Choices and values may be set from the text screen prompts.
If you do not need to change a setting, just press (Enter key).

Units may be changed during a drawing session. Always work in the appropriate units.
There should be no need to calculate conversions between units.

Command:
2. Limits

**Command: Limits**

**Limits** sets the drawing size in real units. The example sets the lower left corner to -1,-1 and the upper right to 12, 10.5. This is one inch bigger all around than an 11x8.5 “A” size drawing sheet.

**Zoom All** must be invoked before the new Limits will be seen.

3. Snap and Grid

From the top menu select **Tools**. Then, pick **Drawing Settings**. The dialog box shown will appear.

X and Y spacing may be different values. At times, Snap and Grid are set to the same value. Grid may be a different value or it may be a multiple of Snap.

- Values may be changed at any time.
- **F7** - function key 7 turns Grid on/off.
- **F9** - function key 9 turns Snap on/off.

**Note:** Grid only extends as far as Limits are set. If the grid does not fill the screen it is because limits and the screen are different ratios.

Click the commands in the prompt area to turn on/off.

Limits should be set to the maximum drawing size in real units so the drawing is created actual size. An Architectural layout might use limits of 200’ x 300’ for a building. An automobile fender might need limits of 50” x 40”.

For an Architectural drawing Snap might be set to 1” while Grid is set to 1’
4. Layers, Colors and Linetypes

4a. Command: Layer
Layers are a very important part of drawing management. On projects where a number of people work on sets of drawings, layer naming and use must be carefully monitored. Layers add structure to the design process. Each layer may be assigned a particular use. AutoCAD allows layers to be named which makes the intent of each layer a little more logical and apparent. There is no limit to the number of layers that may be used. Layers may be created at any time.

Click **Format .. Layer** to access the dialog box shown above. Or, pick the **LAYER** box at the upper left corner of the screen. Create layers by typing in the edit box as shown. Click **New** to load the new names. Names may be edited and deleted. Names may contain numbers, “-” and “_” characters.

- Layers display in alphabetical order the next time the layer command is invoked.
- Layer 0 (zero) is a special-purpose layer and should never be deleted or renamed.
- *AutoCAD at times creates other special layers which should not be changed in any way!*
- Each layer may be assigned a **Color** and a **Linetype**.
- Layers may be **On** or **Off**, **Frozen** or **Thawed**.

Layers used in the example above are named according to use: construction lines (it will be frozen before plotting), object lines, hidden lines, text, center lines, border lines, dimension lines, etc. Short layer names are easier to type when necessary (con, obj, txt). Create your own layer names based on drawing use or intent. Layer names may be set by standards. See AIA Architectural Standards at the end of the chapter.
4b. Layer colors
Colors help differentiate objects in a large complex drawing. Colors give more meaning to the display which makes design and analysis easier. Clearances or space conflicts can more easily be seen. Most drawings are eventually plotted black on white.

To set the line color for a layer, click on (highlight) the layer name. Click the Set Color box. A color chart will appear as seen below. Click on the desired color. More than one layer be set to the same

Setting the Current Layer.
Type Layer or click the Layers pop-down on the standard toolbar.

The Layer selection box will appear. Click the layer name to be current.

Newer versions of AutoCAD have a icon to set the layer to a selected object.
Layers - text screen method - older versions of AutoCAD

Text screen dialogs may be used to create layers, change line colors and change layers. Pick (or type) layer \( \downarrow \) to start. Type a letter for the option \( n=\text{new}, c=\text{color} \), etc.

Follow the prompts at the bottom of the screen. All layer functions may be handled with the text screen. Dialog boxes used on later versions are much easier to learn and set. Press F1 (older versions) or F2 (newer versions) to toggle back to the graphics screen.

4c. Linetypes - loading

Linetypes are not immediately available when starting a new drawing. \textit{Linetypes must first be loaded, then they can be assigned to layers.}

Click the \textbf{Linetype} column next to a layer name. This brings up the current linetypes box. Either select a currently loaded linetype or pick \textbf{Load}.

A dialog box displays the linetypes available from a library. Select the line type to load and click O.K.

Many linetypes are available. Before they can be set to a layer, they must be loaded into the current drawing.
Linetype scale

AutoCAD assumes a scale factor of 1 (one) for an “A” size 8 1/2" x 11" or 9" x 12" size drawing page. If a larger drawing is created, then the linetype scale must be adjusted. Otherwise dashes or other broken lines would be too close and would appear solid.

Type: \texttt{LTScale}. Key in the new setting. \{If the drawing were 48" wide try an LTScale of 4, if the drawing were 50’ (fifty feet), try a LTScale of 50\}.

\textit{LTScale is global and cannot be changed for only one line.}

Turning Layers On/Off

Layer controls are shown in the middle of the dialog box. Layers may be:

- On/Off
- Frozen/Thawed
- Frozen/Thawed in current viewport
- Frozen/Thawed in new viewports
- Locked/Unlocked

Freeze/Thaw is preferred to Off/On as it speeds up the redraw process.
5. Text Style setting

Pick **Format**
Select **Text Style …**

This brings up the Text Style dialog box.

- Create New style names.
- Select Font.
- Set fixed text height.*
- Set special effects.

* A text height of 0 (zero) means the text height may be adjusted. If a fixed height is set, the height cannot be changed. Zero text height is

Upside down, backward, vertical text styles are needed at times.

Obliquing angle allows the text to be slanted. Some Architectural firms prefer slanted text.

Text on all drawings in a set of drawings should be all the same style.

A text style "leroy" has been created. (Leroy refers to a mechanical lettering set widely used when drawings were made on paper). The Romans font was selected. Text height was set to "0" which allows the height to be set as the text is being placed.

Drawings may be made in the default "standard" text font then the text may be changed just before plotting to a more desirable appearance.
Placing Text - three commands

- **Text** - Single line of text. Legacy command from earlier versions of AutoCAD.
- **Dtext** - Dynamic text. Command from earlier versions of AutoCAD.
- **Mtext** - Multiple lines of text.

**Command: text**

Type `text` and press Enter.

- Click a start point.
- Type in text height.
- Type in rotation angle.
- Type in the text line at the bottom of the screen.
- Press Enter to set the text.

Newer versions of AutoCAD provide **dynamic text** which gives more control over input. Text may be seen as it is input directly on the screen.

**Command: mtext**

1. Draw a rectangular box to act as a window in which to place the text.
2. The `mtext` dialog box is used to type, edit, set text height, text color, etc.
3. When the text is correct, click O.K. To set the text.
**Importing Text Files**

Text files may be created on a word processor and imported to a drawing. Specifications and other long files can be created, edited, spell-checked, etc. on a word processor. This greatly simplifies the process of incorporating many lines of text into drawings.

Pick Draw Text Multiline Import Text …

The Open Text file dialog box will appear.

**Text options**

**Justify** allows text to be centered a number of ways. The *default* is **left justify**.

**Style** allows changes in the text font. The *default* style is **standard**. This font is quicker to generate and display but it is not pretty. Other styles are provided for more professional appearance.

**Text height should be adjusted to the drawing size.**

1/8" text on a 1’ drawing would have to be 1/8 x 50 = 6” on a 50’ drawing

**Special characters:**

- `%%p = ±`
- `%%c = Ø`
- `%%d = °`

These characters may not appear until return key is pressed twice.

**Text Editing**

1. Click the **Modify.. Object.. Text.. Edit** pop-down menu. Pick the text to be edited.

2. Type **DDEDIT**. Select the text line to be changed. The edit dialog box appears. If multiline text is selected, the multiline edit dialog will appear.
Other settings
Larger drawings require adjustments for items like dimensions and crosshatch patterns. All of these must be adjusted larger so they appear scaled properly when the drawing is plotted. Scaling will be covered during the instruction on those topics.

Layout techniques
Working with large, complex drawings requires some thought as to how to organize the design. Breaking the objects down into layers gives control for displaying certain types of information and plotting specific types of drawings. Also, certain types of information may be exported for machining or analysis. Some layers may be designated strictly for underlying construction or reference use. Those layers and the information on the layers would not be seen in the final plots. The construction layer set up in the pervious pages is intended for that task. The constr layer has a unique color (maybe red) which identifies any entities in that color as construction reference data.

Using Points and layout Lines on Drawings
Points are the smallest entities on a drawing. Technically they are just a single dot (pixel) of light. Points are extremely useful in layout work for designating key locations on drawings, especially in 3D space. layout Lines are special lines used for "blocking in" a drawing.

Command: Point
Since points are very small and may be impossible to see, AutoCAD has special icons which make points more visible.

Command: PDMODE
(Try pdmode= 34 ).

Or, from the Top Menu pop down Format and select Point Style ....

The point setting dialog box may be shown by typing ddptype.

Point drawing commands are under the Draw menus.
Once a point is set, object snap uses the term node to pick a point.

Set points at key locations — those places that must be referenced many times. For example, set a point at the center of an equilateral polygon. You can snap the center of a circle, but you cannot snap to the center of a polygon. That center is needed to draw the polygon and later it is needed several times to dimension the polygon.

Points may be set on the construction layer and then made invisible at plot time, or pdmode may be set to 0 or 1 at plot time to shrink point sizes.
Construction Lines

Command: xline

This is a new entity type in AutoCAD and AutoCAD LT. These lines are infinite in length. They are intended to act as layout guides when beginning a new drawing.

Starting a layout:

Begin a new drawing using a template or wizard.

Make a construction layer the current layer.

Draw a horizontal line - any length.

Draw an intersecting vertical line - any length.

Place points at key locations, especially the centers of polygons. (You can object snap to the center of a circle but, there is no object snap to the center of a polygon).

Before the drawing is plotted the construction layer is frozen so the construction/layout lines and points do not show.

Construction lines may be specified by giving one point and a direction or by specifying two points. Start the xline command. Pick one point. A line is displayed across the entire screen. Pick a second point or key in a second point.

@X,Y or @dist<angle
Exercise 1. Create a Model Space decimal-inch template file.

Start a new drawing. Use Acad.dwt as the seed file.
(File ... New)

a. Set Units to decimal.
   Set 3 decimal digits.
   (Format ... Units)

b. Set Limits: -1,-1 to 14,12
   Zoom All.
   (Format ... Drawing Limits)
   or type Limits

c. Set Snap and Grid
   (Tools ... Drafting Settings)
   Set Snap = .125, Grid = .250
   Turn Snap and Grid on.
d. Create Layers as shown. Set colors.
Load and set linetypes: cen layer = center2, hid layer = hidden2.

![Layer Properties Manager screenshot]

e. Create a leroy text style as shown. Be sure to click Apply.

![Text Style screenshot]

Save the template as decimal-inch-a.dwt. This will be a "A" size (8-1/2" x 11") startup template. Save this to your personal filespace. School computers may not allow or want saving to the tempalte directory.

End of exercise 1.
Exercise 2. Start a new drawing using `decimal-inch-a.dwt` as the seed file.

a. Type **Limits**. Set limits: -1,-1 to 18,12. **Zoom All** to see the new setting.

b. Save the new file to your personal file space as `decimal-inch-b.dwt`. This will be a "B" size (11" x 17") template file.

"A" size and "B" size Title Blocks.

Exercise 3. Draw a "A" size title block. This will be saved as a drawing for use in creating a template for plotting on a 8-1/2" x 11" printer.

a. Start a new drawing using the `decimal-inch-a.dwt`

b. Set the current layer to **con**

Several layout lines will be drawn to "block in" the title block. Later, the "con" layer will be frozen so the construction/layout lines do not show or print.
c. Draw layout lines:
(absolute coordinates)
Start 0,0 to 11,0
to 11,8.5
to 0,8.5 & c. to close.
This will define the outer sheet size.

d. Offset the outer lines
inside .375

Click a line then click the side to offset to.
See dialog below.
Offset the bottom line up twice as shown.

```
Command: offset
Current settings: Erase source=No Layer=Source OFFSETGAPTYP=0
Specify offset distance or [Through/Erase/Layer] <Through>: .375
Select object to offset or [Exit/Undo] <Exit>:
Specify point on side to offset or [Exit/Multiple/Undo] <Exit>:
Select object to offset or [Exit/Undo] <Exit>:
Specify point on side to offset or [Exit/Multiple/Undo] <Exit>:
Select object to offset or [Exit/Undo] <Exit>:
Select object to offset or [Exit/Undo] <Exit>: *Cancel*
```

e. Change to the border layer. Draw lines on top of layout lines as shown.
Be sure Snap is turned on.
f. Change the grid setting to .125.

g. Change layer to txt. Type in text as shown. Use leroy style. Height = .125" You can type your name in permanently.

Save the drawing as tblk-a.dwg in your personal filespace.

Later, the con layer will be frozen and layout lines will not show.

Exercise 4. Start a drawing using decimal-inch-b.dwt Create a "B" size 11" x 17" title block.

Save as tblk-b.dwg
Exercise 5. Create a plotter template (Paper Space Drawing) from tblk-a.dwg.


b. Click Layout1 at the bottom of the screen.

c. Pop down Insert...Block and select tblk-a.dwg from your personal filespace.

d. Insert tblk-a.dwg as shown.

e. Erase the inner rectangle. This rectangle is the border for a viewport. Viewports will be created later as needed.
f. Save the layout as tblk-a.dwt.
Print the drawing.

Note:
.dwt files saved from model space will only work in Model Space.
.dwt files saved from Layout (Paper) space will only work in Layout (Paper) space.
A .dwg file may be inserted as a block from Model Space to Layout Space.
Exercise 6.

Modify an existing title block.

a. Load an existing template as shown.

b. Add custom titles and information.

c. Save in your own filespace as std-tblk-b.dwt.

All these template drawings will be used in later chapters.