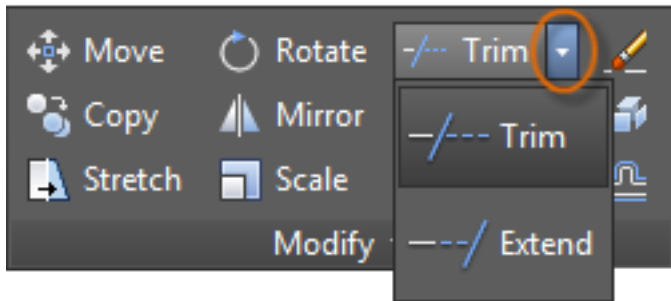# AUTOCAD®

LEARN ABOUT AUTOCAD

An Introduction to AutoCAD for Beginners

Trim and Extend

A popular technique is to use the **OFFSET** command in combination with the **TRIM** and **EXTEND** commands. In the Command window, you can enter **TR** for **TRIM** or **EX** for **EXTEND**. Trimming and extending are some of the most commonly used operations.

In the following illustration, you want to extend the lines that represent the steps for this deck. Enter the **EXTEND** command, select the boundary, and then press Enter or the Spacebar.
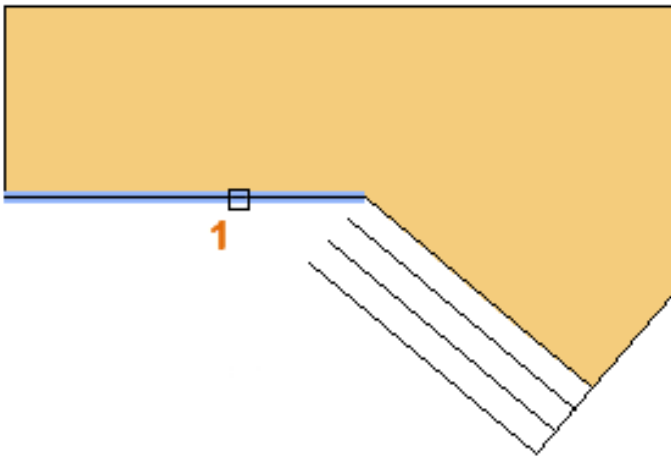

Pressing Enter or the Spacebar indicates that you've finished selecting the boundaries, and that you're now ready to select the objects to be extended.

Here's a Tip:

A faster method is to press Enter or the Spacebar right away instead of selecting any boundary objects. The result is that all objects are available as possible boundaries.

Trim and Extend (continued)

Next, select the objects to be extended (near the ends to be extended), and then press Enter or the Spacebar to end the command.
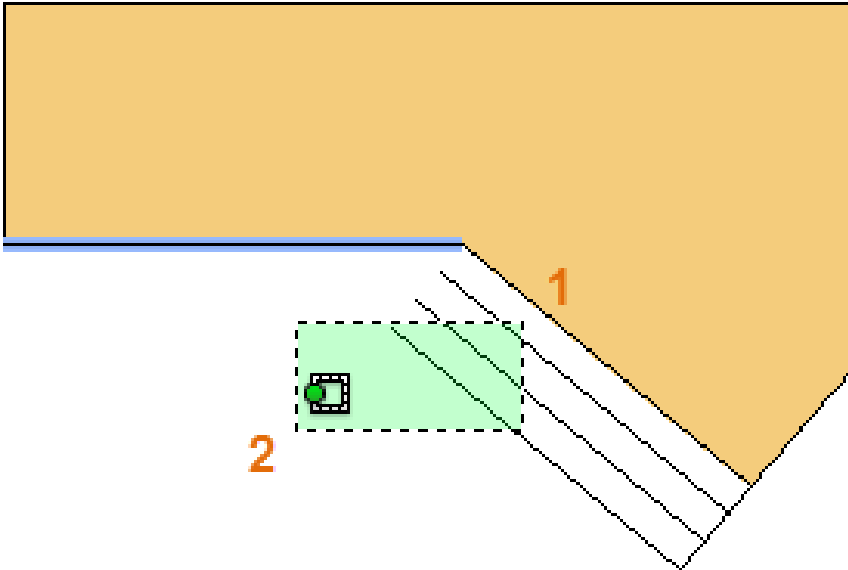
As you can see, the selected lines are extended to the boundary.
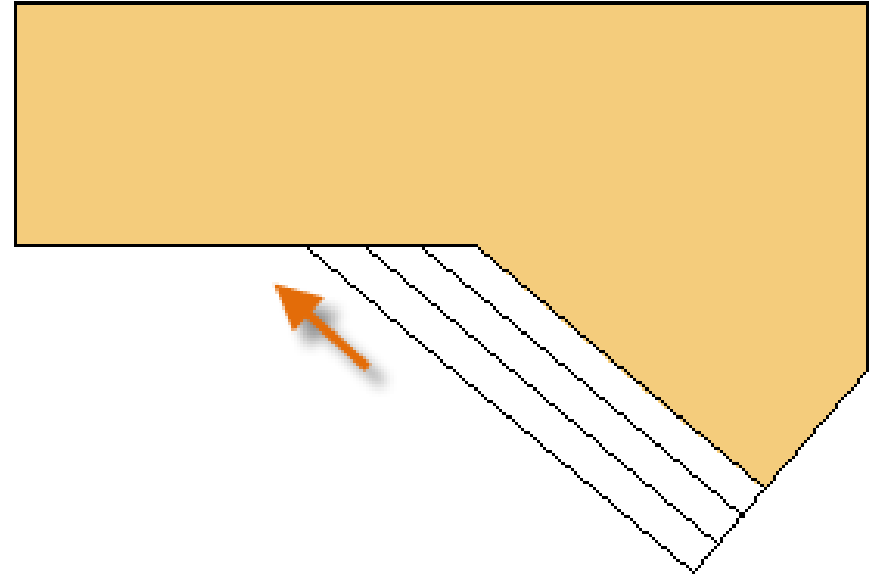
The **TRIM** command follows the same steps, except that when you select the objects to trim, you select the portions to trim away.

Mirror

The following illustration comes from a tile project. The walls in this residential bathroom are flattened out to be able to lay out the tile pattern and estimate the number of tiles needed.

BACK WALL

LEFT WALL   **RIGHT WALL**
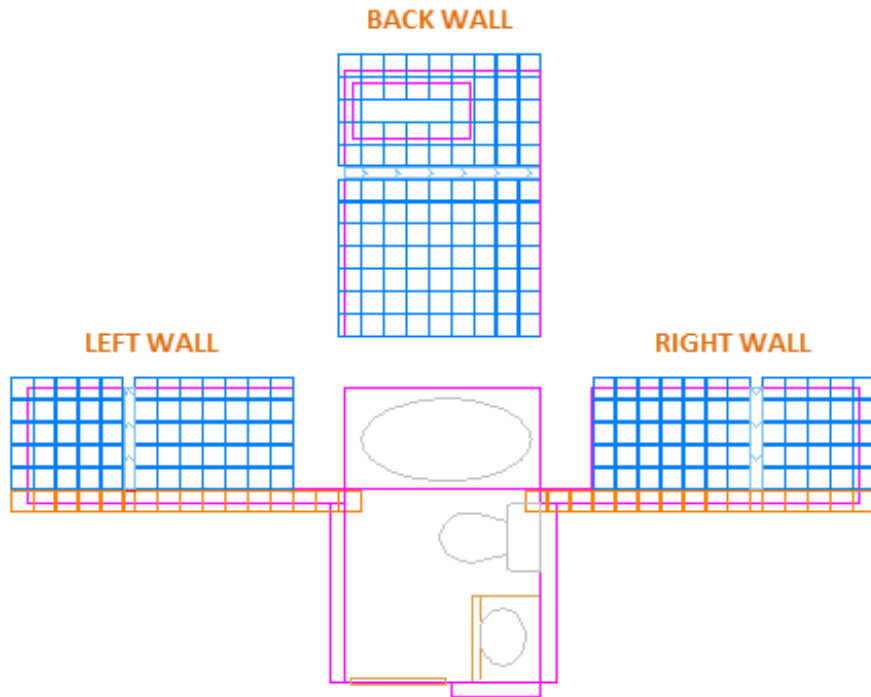
You can save a lot of work by taking advantage of the symmetry between the left and right walls. All you need to do is create the tiles on one wall and then mirror the wall across the center of the room.

In the example below, begin the **MIRROR** command (or enter **MI** in the Command window), use window selection (1 and 2) to select the geometry on the right wall, press Enter or the Spacebar, and then specify a mirror line (3 and 4) corresponding to the centerline of the bathroom.

mirror line

4

1 select objects

3
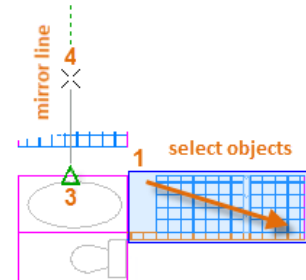
Finally, decline the option to "Erase source objects" by pressing Enter or the Spacebar.
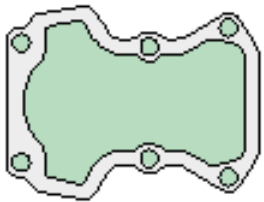
**mirrored result**
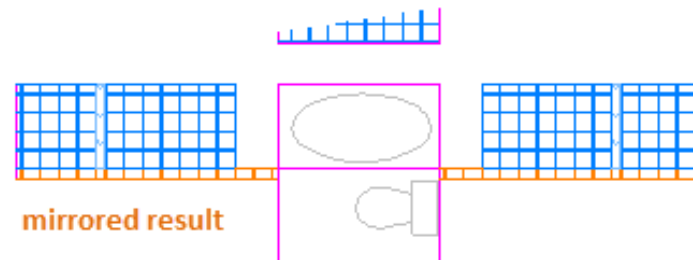
Here's a Tip:

Always look for symmetry to save yourself extra work, even if the symmetry is not 100% identical.

Stretch

You can stretch most geometric objects. This lets you lengthen and shorten parts of your model. For example, this model might be a gasket or the design for a public park.

Use the **STRETCH** command (or enter **S** in the Command window) and select the objects with a crossing selection as shown below (1 and 2). The crossing selection is mandatory— only the geometry that is crossed by the crossing selection is stretched. Then click anywhere in the drawing area (3), move the cursor to the right, and enter 50 as the distance. This distance might represent millimeters or feet.
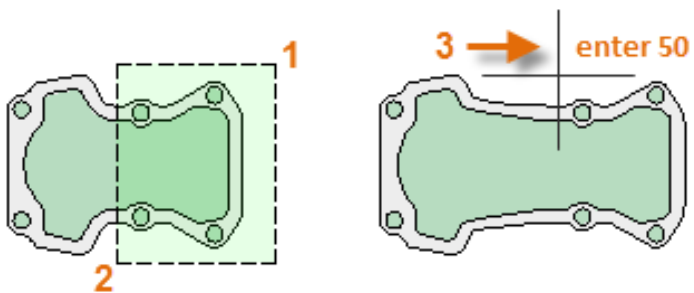
To shorten the model by a specified amount, you'd move your cursor to the left instead.

Fillet

The **FILLET** command (enter **F** in the Command window) creates a rounded corner by creating an arc that is tangent to two selected objects. Notice that the fillet is created relative to where you select the objects.

You can create a fillet between most types of geometric objects, including lines, arcs, and polyline segments.
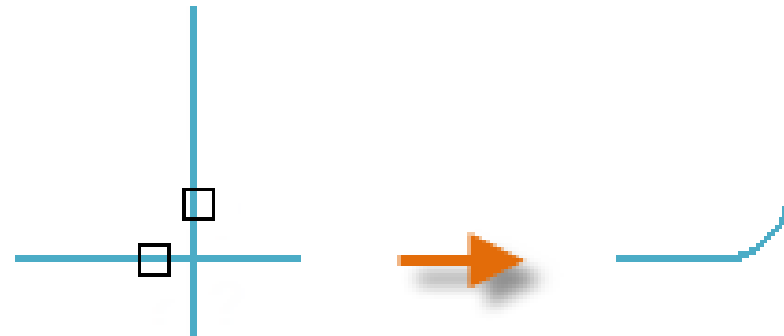
Here's a Tip:

If you specify 0 (zero) as the radius of the fillet (imagine a circle shrinking to a radius of 0), the result trims or extends the selected objects to a sharp corner.

Explode

The **EXPLODE** command (enter **X** in the Command window) disassociates a compound object into its component parts. You can explode objects such as polylines, hatches, and blocks (symbols).

After you explode a compound object, you can modify each resulting individual object.

Edit Polylines

You can choose from several useful options when you want to modify a polyline. The **PEDIT** command (enter **PE** in the Command window) is located on the drop-down list of the Modify panel.
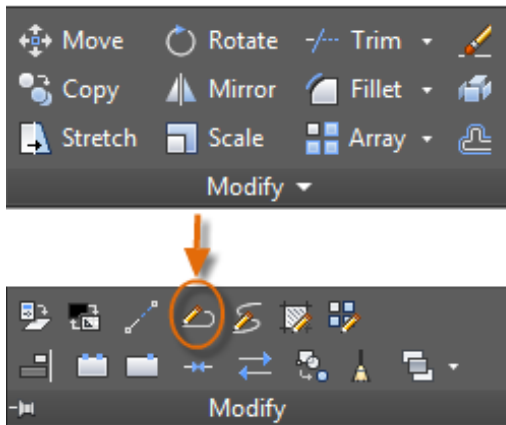

With this command, you can:

- Join two polylines into a single polyline if they share a common endpoint

- Convert lines and arcs into a polyline—simply enter **PEDIT** and select the line or arc

- Change the width of a polyline

Here's a Tip:

In some cases, the easiest method to modify a polyline is to explode it, make the modifications, and then turn the objects back into a polyline using the Join option of the **PEDIT** command.

Grips

Grips are displayed when you select an object without starting a command. Grips are often handy for light editing. For example, the line below accidentally snapped to the wrong endpoint. You can select the misaligned line, click on a grip and then click to specify the correct location.
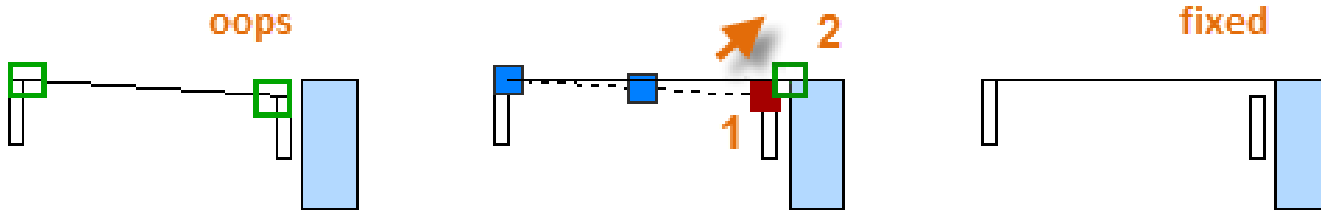
By default, when you click a grip, you automatically start in **STRETCH** mode as indicated in the Command window. If you want to explore other ways of editing objects with grips, press Enter or the Spacebar to cycle through several other editing modes. Some people perform most editing operations using grips.

Blocks

Insert symbols and details into your drawings from commercial online sources or from your own designs.

In AutoCAD, a *block* is a collection of objects that are combined into a single named object. The following are some sample blocks at various scales.
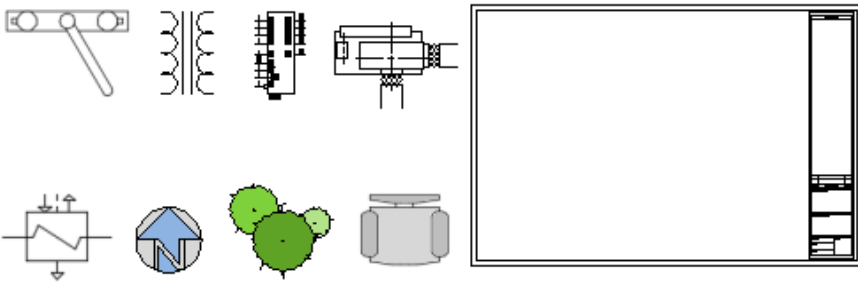
Some of these blocks are realistic representations of objects, some are symbols, and one of them is an architectural title block for a D-size drawing.

Insert a Block

Typically, each of these blocks is an individual drawing file, perhaps saved in a folder with similar drawing files. When you need to insert one into your current drawing file, use the **INSERT** command (or enter **I** in the Command window).
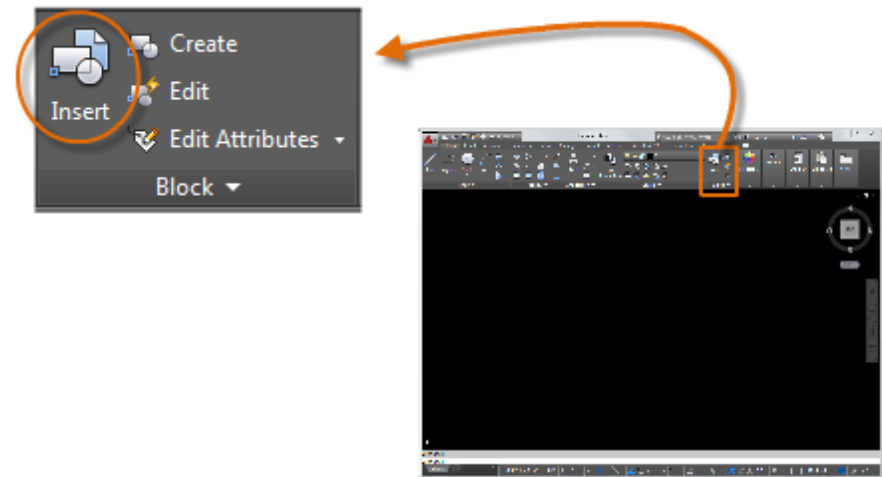
Insert a Block (continued)

The first time you insert the drawing as a block, you must click Browse to locate the drawing file. Make sure you organize your blocks into easy-to-find folders.
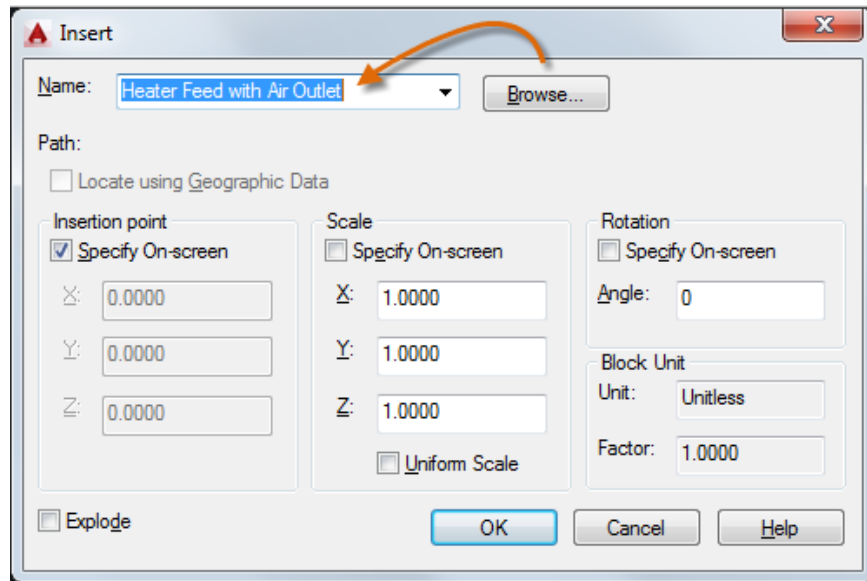

Once inserted, the *block definition* is stored in your current drawing. From then on, you can choose it from the Name drop-down list without needing to click the Browse button.

Here's a Tip:

The default settings in the Insert dialog box are usually acceptable. After you choose the block name, click OK, and then specify its location in your drawing. You can rotate it later, if necessary.

Notice that when you insert a block, it is attached to your cursor at the point indicated. This location is called the insertion point. By default, the insertion point is the origin point (0,0) of the original drawing.
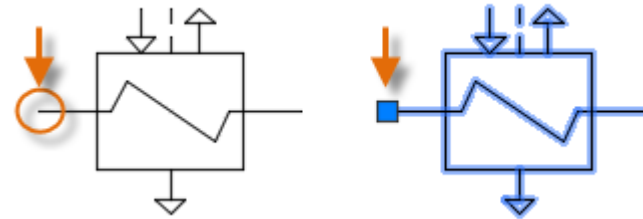
After inserting the block, you can select it and a grip appears. You can easily move and rotate this block using this grip.
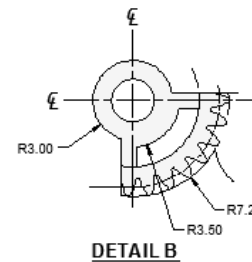
In the following example, a drawing file is inserted into the current drawing to provide a standard detail view.

*NOTE: Inserting a drawing file as a block provides a static reference to the specified drawing. For a reference that automatically updates, you can attach the drawing with the External References palette (**XREF** command) instead.*
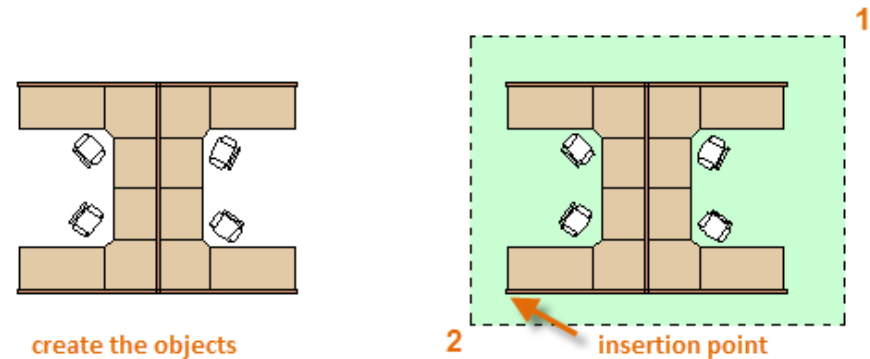
Create a Block Definition

Instead of creating a drawing file to be inserted as a block, you might want to create a block definition directly in your current drawing. Use this method if you do not plan to insert the block into any other drawing. In that case, use the **BLOCK** command to create the block definition.

For example, this is how you could create a module for a cubicle design.

1. Create the objects for the block.

2. Start the **BLOCK** command.

3. Enter a name for the block, in this case Quad-Cube.

4. Select the objects that you created for the block (click 1 and 2).

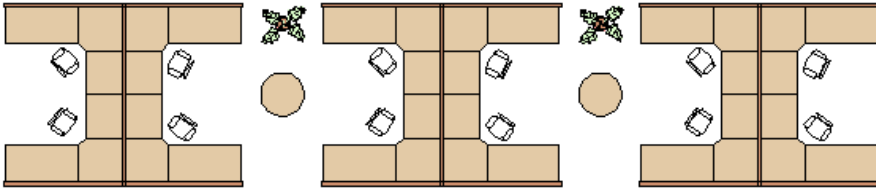5. Specify the block insertion point.

create the objects   2   insertion point

You can enter the information for steps 3, 4, and 5 into the Block Definition dialog box in any order.
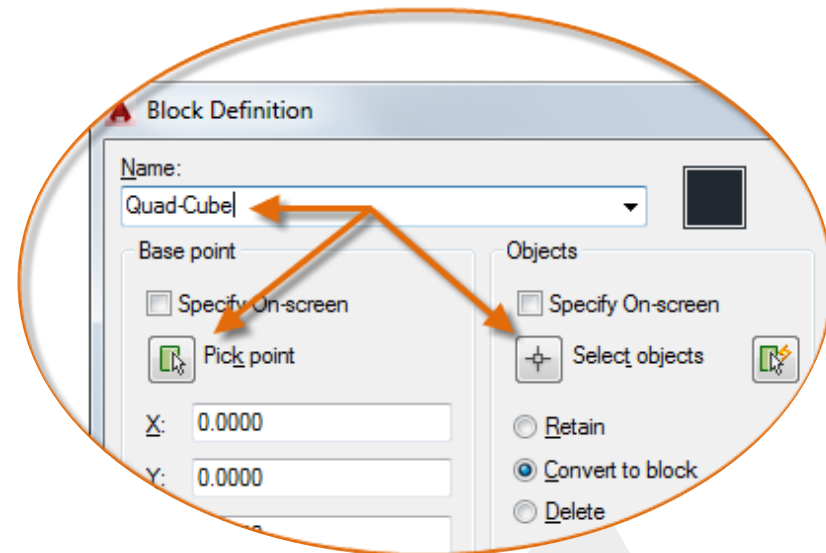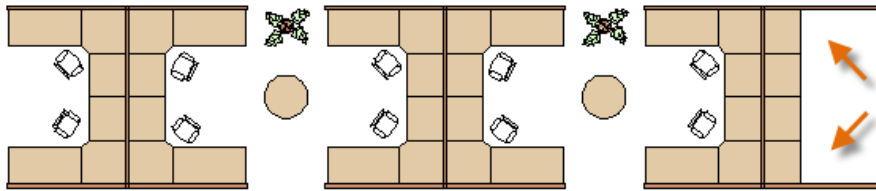
Create a Block Definition (continued)

After creating the block definition, you can insert, copy, and rotate the block as needed.

Using the **EXPLODE** command, you can explode a block into its component objects if you need to make changes. In the illustration below, the cubicle on the right was exploded and modified.

In this example, you would probably create a new block definition from the objects in the exploded block.

Recommendations

There are several different schemes for saving and retrieving block definitions.

→ You can create an individual drawing file for each block that you intend to use. You save these drawing files in folders, each of which would contain a family of related drawing files.

→ You can include the block definitions for title blocks and common symbols in your drawing template files to make them available immediately when starting a new drawing.

→ You can create several drawing files, which are sometimes called *block library drawings*. Each of these drawings contains a family of related block definitions. When you insert a block library drawing into your current drawing, all the blocks that are defined in that drawing become available.

Here's a Tip:

With online access, you can download AutoCAD drawing files from the web sites of commercial vendors and suppliers. This can save you time, but always check to make sure that they are drawn correctly and to scale. Autodesk Seek (**http:// seek.autodesk.com/**) is a convenient way of accessing BIM (building information modeling) libraries.

*NOTE: You can create block definitions that include one or more attributes that store and display information. The command that you would use is **ATTDEF**. Typically, attributes include data such as part number, name, cost, and date. You can export block attribute information to a table or to an external file.*
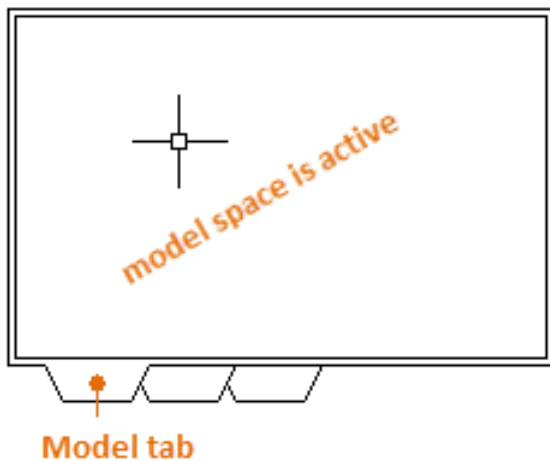
Layouts

Display one or more scaled views of your design on a standard-size drawing sheet called a layout.

After you finish creating a model at full size, you can switch to a *paper space layout* to create scaled views of the model, and to add notes, labels, and dimensions. You can also specify different linetypes and line widths for display in paper space.

Model Space and Paper Space

As you know, you create the geometry of your model in *model space*.

Model tab

Originally, this was the only space available in AutoCAD. All notes, labels, dimensions, and the drawing border and title block were also created and scaled in the model space.

After *paper space* was introduced, you could click a layout tab to access a space designed specifically for layouts and scaling. In the following illustration, paper space is active. There are currently only two objects in paper space: a drawing border block, and a single *layout viewport*, which displays a view of model space.
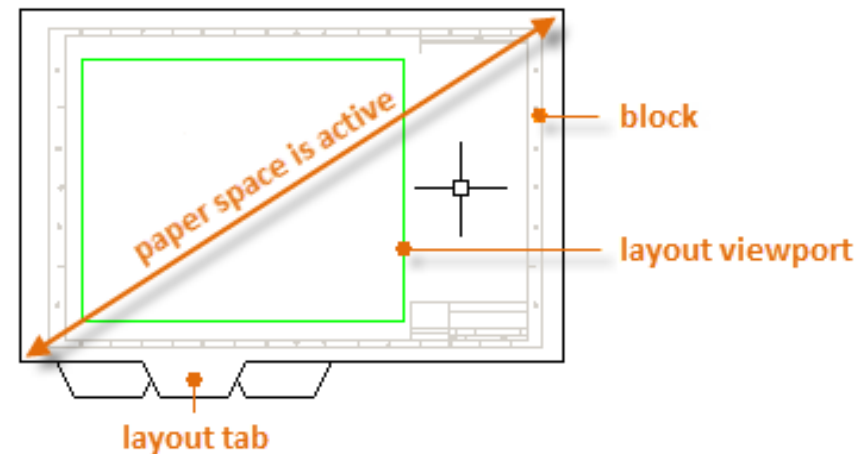
Working with layout viewports is described in more detail later in this topic.