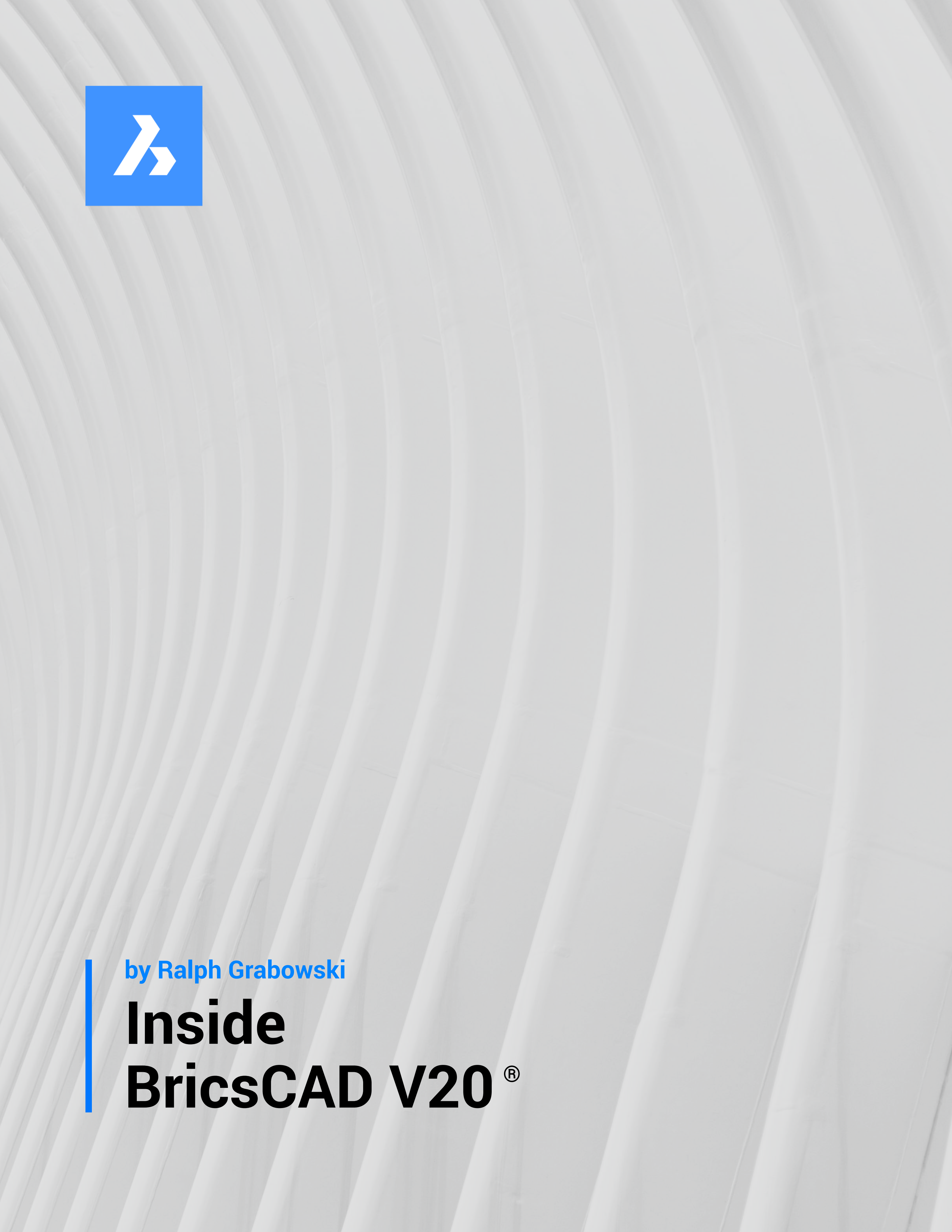

by Ralph Grabowski

Inside BricsCAD V20®

Copyright Information

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Technical Editing **Bricsys Staff**

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Visit the *Inside BricsCAD* Web site at www.upfrontezine.com/ib8. At this Web page, editions of this book are available for BricsCAD V8 through V19.

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CHAPTER 1

A Quick Tour Through BricsCAD

Welcome to BricsCAD V20!

You want to make drawings with BricsCAD, and this book shows you how — in as little as a day! Before doing any kind of drafting, you may want to take a tour of the BricsCAD user interface so that you know your way around the software.

Even when you know other CAD programs, you may find it useful to skim this chapter and take note of where BricsCAD operates differently from what you already know. In this chapter, you learn how to start this popular 2D/3D CAD program, take a tour through its user interface, and then get your feet wet by drawing a few lines.

IN THIS CHAPTER

- Starting BricsCAD V20
- Becoming familiar with parts of the user interface
- Understanding the crosshair cursor, command bar, auto-complete, and UCS icon
- Drawing lines
- Reversing errors
- Accessing online help

KEY TERMS IN THIS CHAPTER

Button — when clicked, executes the associated command

Cursor — provides you with feedback from the operating system and from BricsCAD

Flip screen — switches between the drawing window and a text window

Flyout — when clicked, reveals a secondary toolbar

Icon — represents commands pictorially

Layout — determines how drawings are plotted

Pickbox — specifies the area in which points being picked, entities are selected

Quick Access toolbar — miniature toolbar with often-used commands

Ribbon — collects buttons into tabs and panels

Right-click — involves pressing the right mouse button to display context-sensitive (shortcut) menus

Toolbar — collects buttons into a single, useful strip

USEFUL ABBREVIATIONS

Alt	Alternate key on PCs
Cmd	Command key on Macs
Ctrl	Control key on PCs
F	Function key
U	Undoes the last command or option
UCS	User-defined coordinate system

NEW COMMANDS

Command	Shortcut*	Menu Selection**	Ribbon
Help	? or F1	Help | Help	Home | Help | Help
Line	L	Draw | Line	Draw | Draw | Line
Quit	Alt+F4	File | Exit	...
TextScr	F2	View | Prompt History Window	...
Undo	Ctrl+Z	Edit | Undo	Quick Access toolbar: Undo
UcsIcon	...	...	...

* F1 means function key F1

Alt+F4 means hold down the Alt key, and then press function key F4.

** The vertical bar separates menu selections. **Draw | Line** means: from the Draw menu, select the Line item.

How to Start BricsCAD

If BricsCAD is not yet set up on your computer, do so. Your computer must be operating any release of Windows since Vista, or just about any recent dialect of Linux or MacOS.

The easiest way to start the program is to double-click the **BricsCAD** icon found on the computer desktop.

Icon for starting BricsCAD, found on the desktop

If you do not see the icon on the desktop, then start the program using traditional methods.

STARTING BRICSCAD ON WINDOWS

BricsCAD V20 works with Windows Vista and newer.

Windows Vista and 7

In Windows Vista or 7, you start the program from the taskbar:

1. Click the taskbar's **Start** icon.
2. Choose **All Programs**.
3. Select **Bricsys**, followed by the **BricsCAD V20** folder, and then click on **BricsCAD V20**.

The exact name you see depends on the language version you downloaded. For example, "BricsCAD V20 (x64) en_US" is the name of the 64-bit program for English speakers in the US dialect.

Windows 8

In Windows 8, you make these moves:

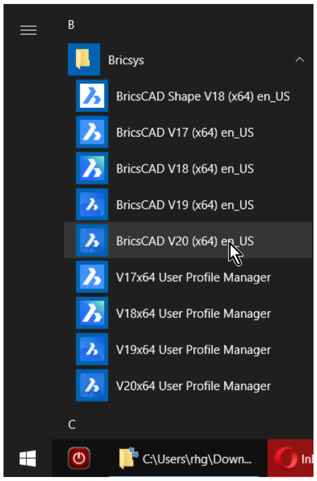
1. If necessary, switch to the **Start** screen.
2. In the Start screen, tap on the **BricsCAD V20** icon.

Windows 10

Under Windows 10, starting BricsCAD is like Windows 7:

1. Tap the **Start** button.
2. Choose **All Apps**.

3. In the **B** menu, tap **Bricsys** folder, and then tap the **BricsCAD V20** item.

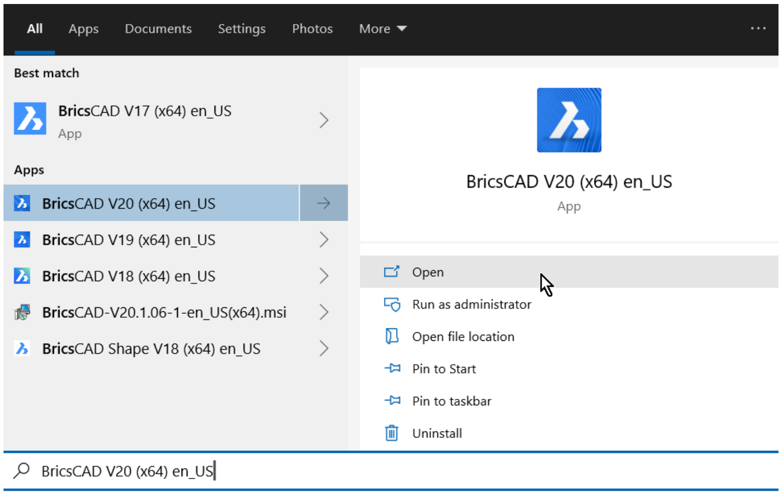

B
Bricsys
BricsCAD Shape V18 (x64) en_US
BricsCAD V17 (x64) en_US
BricsCAD V18 (x64) en_US
BricsCAD V19 (x64) en_US
BricsCAD V20 (x64) en_US
V17x64 User Profile Manager
V18x64 User Profile Manager
V19x64 User Profile Manager
V20x64 User Profile Manager
C
C:\Users\rhg\Down...
In

Selecting BricsCAD from the Start menu, alphabetically

TIP Your current licence runs all versions of BricsCAD going back to V14.

If the item is not visible in the menu, then follow these steps:

1. Tap the **Start** button.
2. Start typing "bricscad"
3. When you see **Bricsys V20**, tap it.



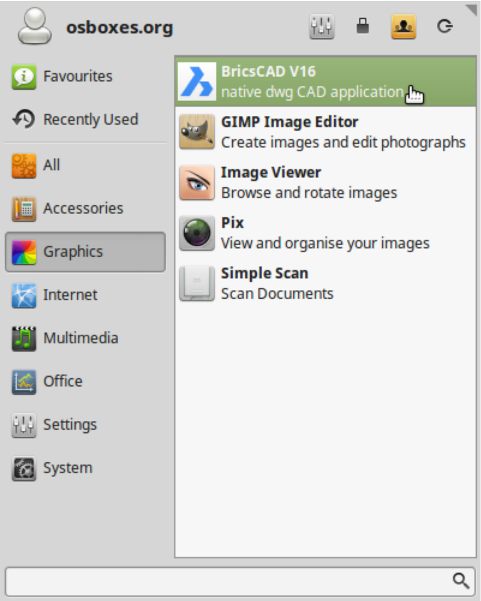
Searching for the BricsCAD program in Windows 10

STARTING BRICSCAD ON LINUX

BricsCAD works with just about any recent release of Linux, but is specifically supported on Fedora, OpenSuse, and Ubuntu. To start the program, follow these steps:

1. Click the task bar's **Main Menu** button.
2. Choose **Graphics**.

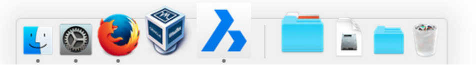
3. Click on **BricsCAD**.



Finding BricsCAD in the Main Menu of Linux

STARTING BRICSCAD ON MAC MACOS

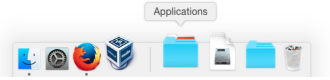
BricsCAD works with recent releases of MacOS on Mac computers, 10.8 or higher. (MacOS is the new name for OS X.) On the dock, click the **BricsCAD V20** icon:



BricsCAD on the MacOS dock

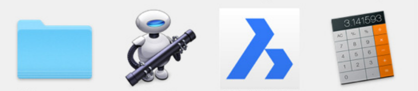
If you do not see the icon there, then follow these steps:

1. In the dock, open the **Application** folder.



Opening the Applications folder in the MacOS doc

2. Find the **BricsCAD V20** icon, and then click it.

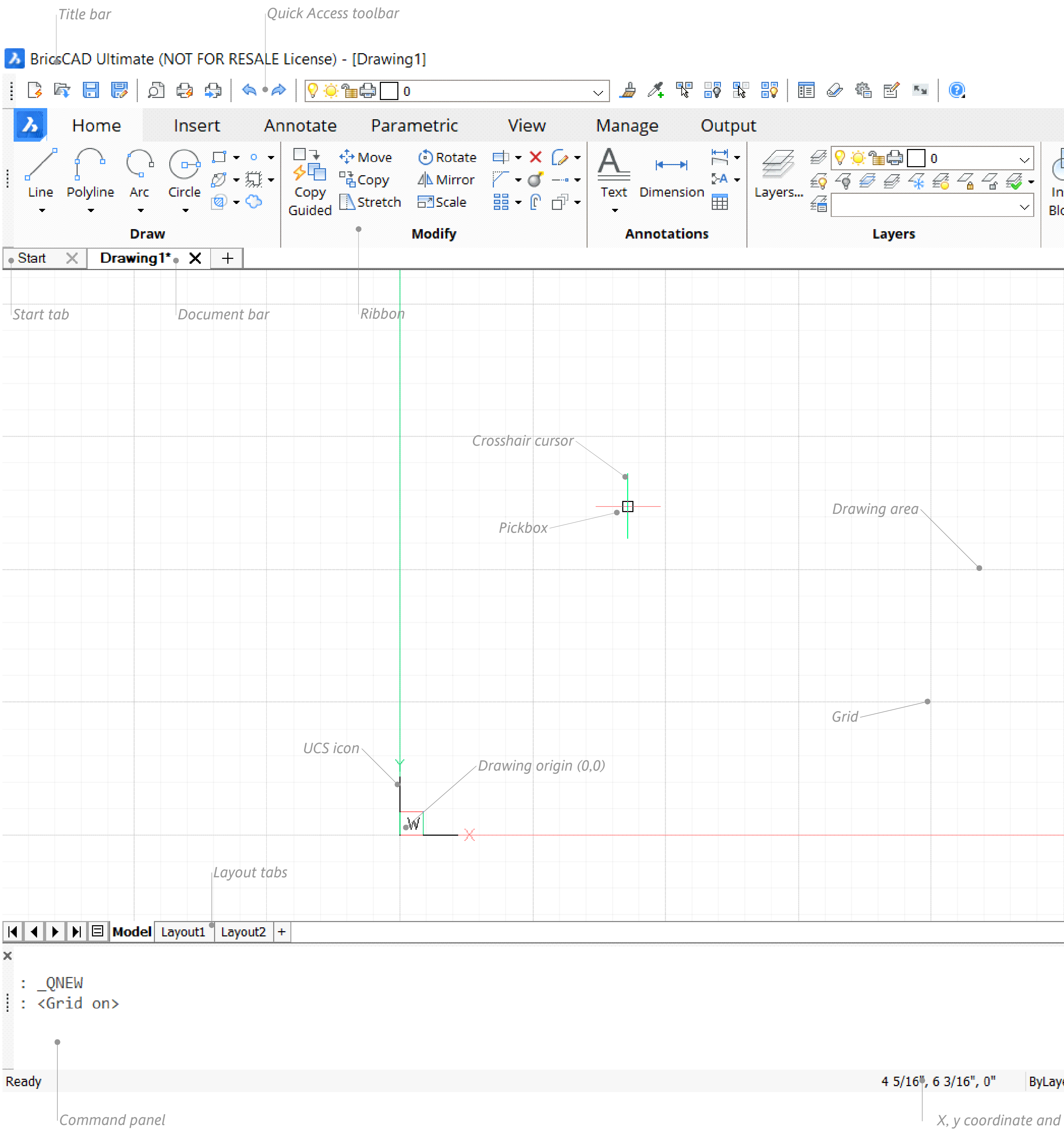


Finding BricsCAD in the Applications folder

To keep the icon in the dock, right-click the BricsCAD icon. From the shortcut menu, choose **Options**, and then choose **Keep in Doc**.

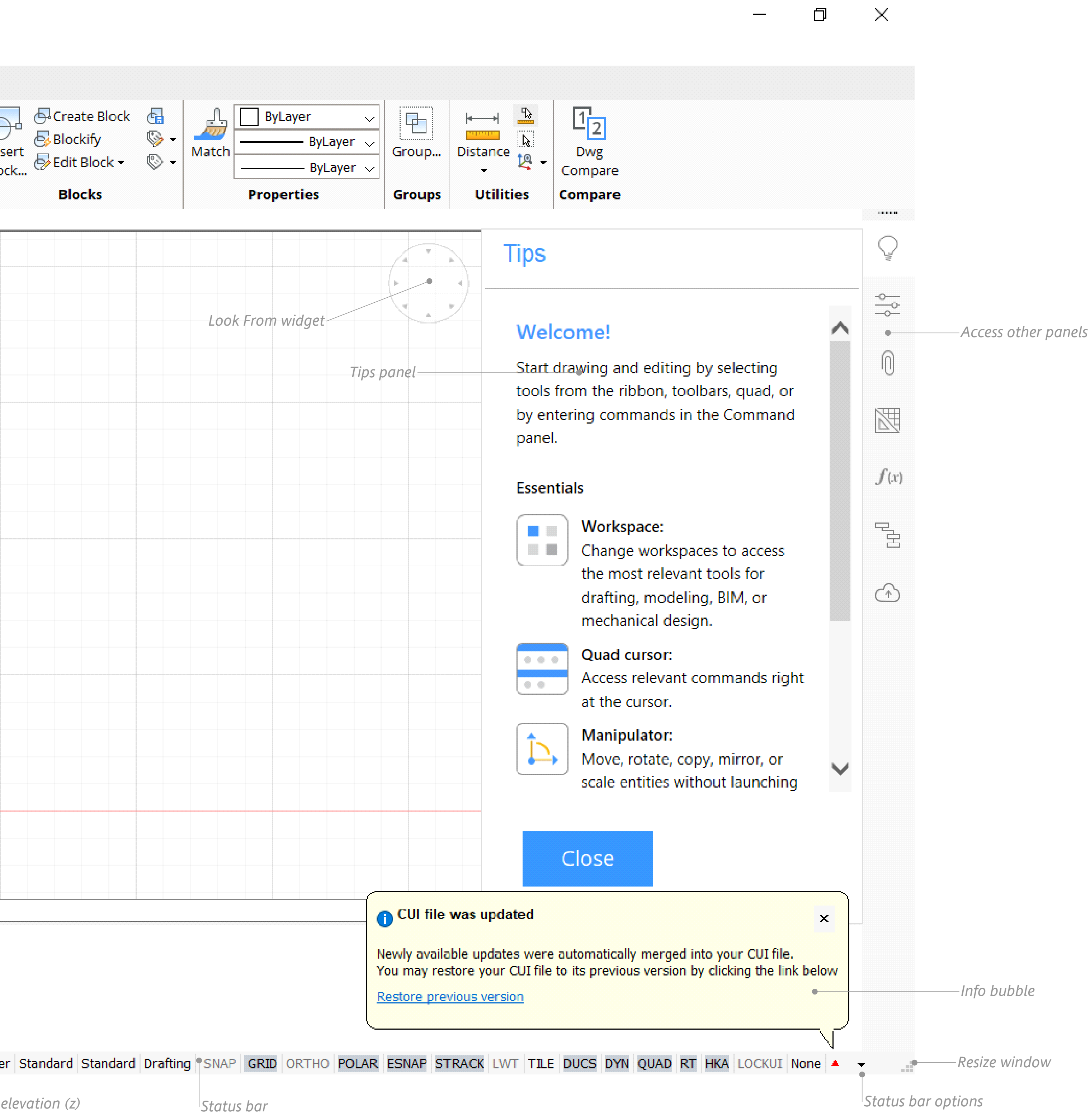
BRICSCAD V20 USER INTERFACE

Illustrated is BricsCAD running on Windows 10.



```
: _QNEW
: <Grid on>
```

No matter the operating system, BricsCAD looks pretty much the same in each one. Instructions in this book specific to Linux and Mac are shown in gray text.



Tips

Welcome!

Start drawing and editing by selecting tools from the ribbon, toolbars, quad, or by entering commands in the Command panel.

Essentials

Workspace:
Change workspaces to access the most relevant tools for drafting, modeling, BIM, or mechanical design.

Quad cursor:
Access relevant commands right at the cursor.

Manipulator:
Move, rotate, copy, mirror, or scale entities without launching

Close

CUI file was updated

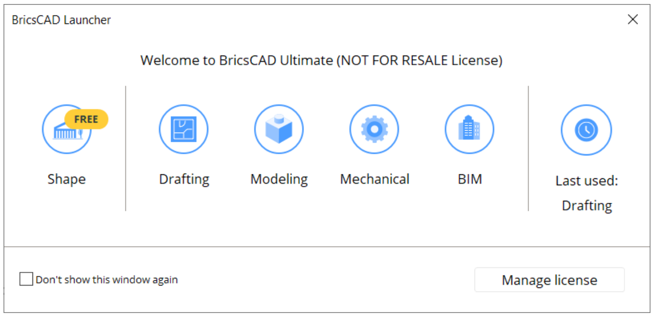
Newly available updates were automatically merged into your CUI file.
You may restore your CUI file to its previous version by clicking the link below

Restore previous version

Depending on the speed of your computer, it can take as little as a couple of seconds to load the program. First, the BricsCAD Launcher appears.

The Launcher and the Start Tab

(NEW IN V20) The first thing you see is the BricsCAD Launcher dialog box. (It replaces the Getting Started dialog box from earlier releases.)


BricsCAD Launcher

Welcome to BricsCAD Ultimate (NOT FOR RESALE License)

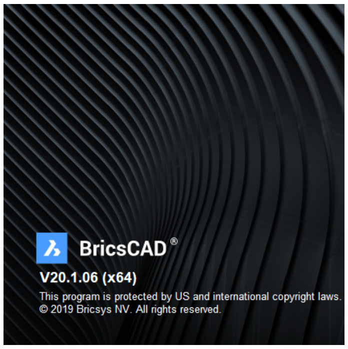
Launcher displayed by BricsCAD following start up

This dialog box has options:

- **Shape** — install and launch Shape, the free 3D modeler
- **Drafting** through **BIM** — start BricsCAD with one of these workspaces
- **Last used** — starts BricsCAD with the workspace you last used
- **Manage license** — opens the Licensing Information dialog box to tell you about your licensee, such as expiry date
- **Do not show this window again** — disables this dialog box the next time you start BricsCAD

TIP The workspaces available in the Launcher depend on the license level of BricsCAD you purchased. When you are using the 30-day free version, you have access to all workspaces.

For now, enter the program by clicking **Drafting**. You see the splash screen, whose job is to entertain you while the program continues to load. It clears itself from the screen automatically.


BricsCAD®
V20.1.06 (x64)
This program is protected by US and international copyright laws.
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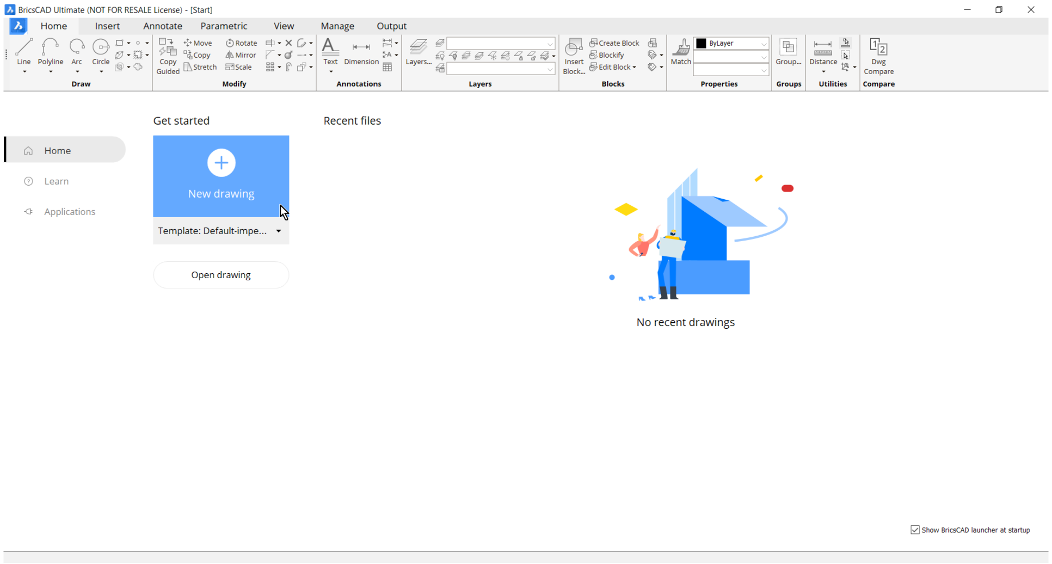
Splash screen for BricsCAD

THE START TAB

(NEW IN V20) Notice that BricsCAD opens up, displaying the Start tab. (The Start tab replaces the Welcome dialog box of V19.)



BricsCAD showing the Start tab

This tab has three ways of seeing it:

- **Home** — accesses drawing files
- **Learn** — accesses online resources for using BricsCAD
- **Applications** — access third-party applications that work with BricsCAD

The Home view lets you start a new drawing based on a specific template, chose a drawing you opened earlier (under Recent Drawings, empty on a new installation), or open any other DWG file through the operating system's file manager.

> **TIP** Most of the ribbon does not work when no drawing is open. Clicking the blue Bricsys icon does, however, access file and drawing management functions.

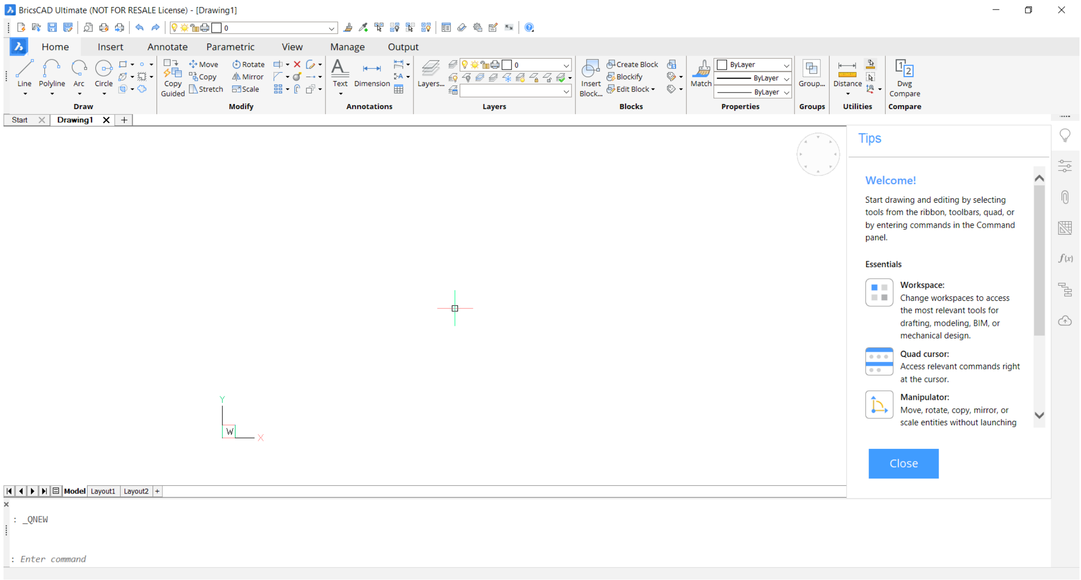
For now, click **New Drawing**.

THE BRICSCAD WINDOW

Take a look at the overview of the BricsCAD window by checking out the figure spread across the previous two pages.

BricsCAD's central area consists of a large graphical drawing region. Here you can see the red-green cursor with its pickbox, and the UCS icon with its red-green x,y axes. The drawing region is surrounded by several panels of information — toolbars or the ribbon, a status bar, and so on.

Along the very top of the BricsCAD window, you see the title bar with the name of the current drawing. Below it is the Quick Access toolbar with oft-used commands, below them the fat ribbon, and then the thin row of drawing tabs. (Depending on how BricsCAD is configured, you might see toolbars and the traditional dropdown menu bar.)



Welcome!

Start drawing and editing by selecting tools from the ribbon, toolbars, quad, or by entering commands in the Command panel.

Essentials

Workspace: Change workspaces to access the most relevant tools for drafting, modeling, BIM, or mechanical design.

Quad cursor: Access relevant commands right at the cursor.

Manipulator: Move, rotate, copy, mirror, or scale entities without launching

```
: _QNEW

: Enter command
```

The way BricsCAD initially looks

Near the bottom of BricsCAD are the layout tabs, the command prompt area, and then at the very bottom is the status bar.

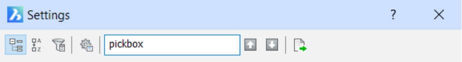
A BASIC TOUR OF THE USER INTERFACE

The user interface of any CAD program has many elements, and so it can be daunting to learn all of it at once. In this chapter, you look at just a few of BricsCAD's UI elements, the more important ones:

- Crosshair and arrow cursors
- Command bar
- UCS icon

Chapter 2 provides you with a detailed tour of BricsCAD and its UI (short for "user interface").

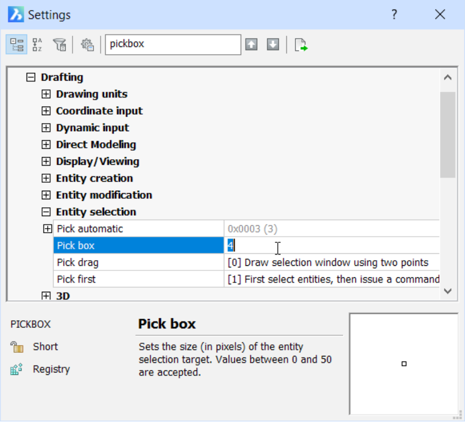
TIP Many user interface options are changed in BricsCAD with the **Settings** command. It provides you with over 1,000 settings that let you control how BricsCAD works and looks. The best way to find a specific setting quickly is to enter its name in its search field, as shown below.



Settings ? ×

pickbox

Changing the Pickbox Size. To change the size of the pickbox, enter "pickbox." The default size that you see on the BricsCAD screen is 3 pixels wide, but you can change the size from 0 (gone) to 50 pixels (huge); I suggest you change it to 5 to make it just a bit bigger.



Settings ? ×

pickbox

⊟ Drafting
- ⊞ Drawing units
- ⊞ Coordinate input
- ⊞ Dynamic input
- ⊞ Direct Modeling
- ⊞ Display/Viewing
- ⊞ Entity creation
- ⊞ Entity modification
- ⊟ Entity selection

⊞ Pick automatic	0x0003 (3)
Pick box	4
Pick drag	[0] Draw selection window using two points
Pick first	[1] First select entities, then issue a command

- ⊞ 3D

PICKBOX

Short

Registry

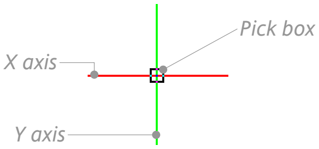
Pick box

Sets the size (in pixels) of the entity selection target. Values between 0 and 50 are accepted.

Changing the Crosshair Cursor Size. Should you find the crosshair cursor too small or too large, you change its size also through the Settings command: search for "crosshair." The default value is **5**, which means the length of the cross hairs is 5% of the screen's size. When set to 100, the cross hairs stretch across the entire drawing area.

Crosshair and Arrow Cursors

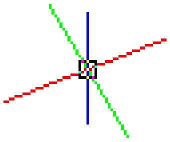
The cursor gives you feedback from BricsCAD, Windows, and other software. When the cursor is in the BricsCAD drawing area, it looks like a *crosshair* that shows you where "you" are in the drawing, precisely.



The elements of the cursor in the drawing area

Try moving the cursor now around the BricsCAD window: move your mouse.

Notice that the crosshair cursor has colors. They help you orient yourself, particularly in 3D (three dimensional) drafting. Here's what the cursor looks like in 3D, and what the colors mean:

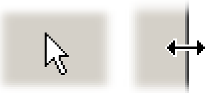


- **Red line** represents the x axis
- **Green line** represents the y axis
- **Blue line** represents the z axis; you don't see it when you draw in 2D mode
- **Black square** is the pick box, for selecting entities

The black square at the center of the crosshairs is called the "pickbox." It shows you exactly where you are picking entities to select and edit them. Entities outside the pickbox will not be picked. You use the pickbox during Chapter 5, "Adding Details to Drawings."

You can change the colors and the size of the crosshairs, as well as the size of the pick box, with the Settings command; see the tip coming up soon.

When you move the crosshair out of the drawing area, it changes its shape to an arrow — one with which you probably are familiar from other software. You use the arrow cursor to make menu selections, pick buttons on the toolbar or ribbon, and so on. The cursor can change to other shapes. For example, when the cursor becomes a double-ended cursor, you can resize windows and palettes.

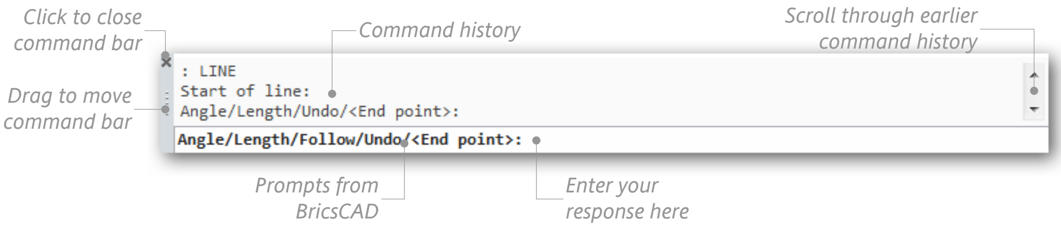


***Left:** Arrow cursor outside the drawing area, used to select UI elements;*
***right:** Double-headed cursor used to change size of UI elements*

COMMAND BAR

Commands	`CommandLine, CommandLineHide`
Ribbon	...
Shortcuts	`Ctrl+9, Shift+F2`
Alias	...

The command bar is near the bottom of the BricsCAD window. This is the one place where you enter the names of commands and specify their options. If you are a touch typist like me, then you'll find that you prefer specifying commands by typing them — instead of hunting through a menu or the ribbon. The command bar is also the place where BricsCAD prompts you for any additional information it needs to complete a command.



```
: LINE
Start of line:
Angle/Length/Undo/<End point>:
Angle/Length/Follow/Undo/<End point>:
```

The elements of the command bar

While you can turn off the Command bar with the **CommandLineHide** command, I don't recommend doing this; there is no good reason to do so! Here are the important parts of the command bar:

Let's take a look at how the command bar works.

All About Command Prompts

When you see the ' **:** ' (colon) symbol by itself in the Command bar, like this...

```
:
```

...it means that BricsCAD is ready for you to enter a command. The colon is called the "prompt." Should you wish to enter a command but there is text after the ' **:** ', press the **Esc** key once or twice to clear the command line.

Try drawing a few lines with the **Line** command now:

1. Enter the Line command, as follows:

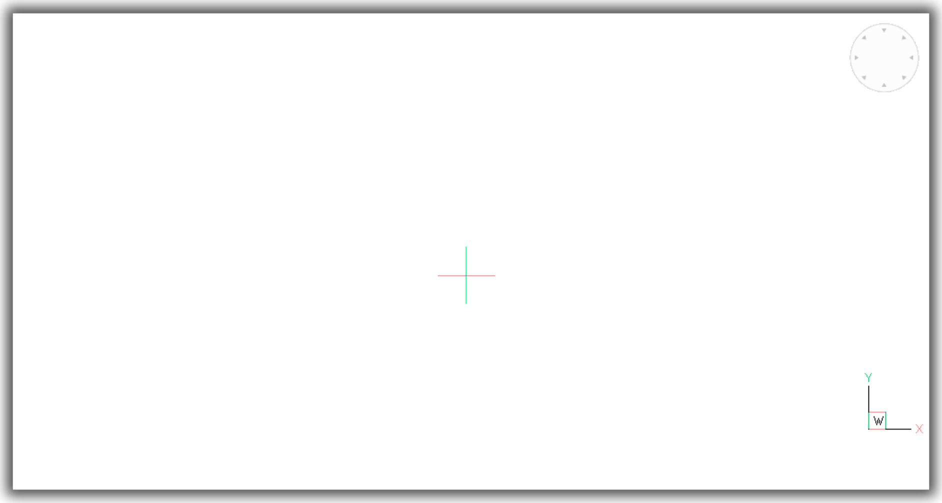
   `: line` *`(Press Enter)`*

   This means that you should type the word **line**, and then press the **Enter** key. Pressing Enter tells BricsCAD that you are finished typing the name of the command, and that it can now execute the command.

2. Notice that BricsCAD changes the prompt from ' : ' to ' Start of line: ' as follows:

   `Start of line:`

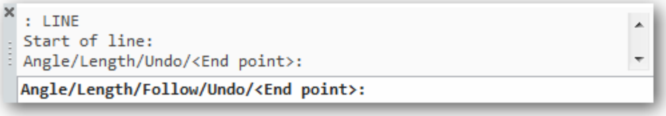
   BricsCAD is asking you where you want it to start the line. You move the cursor (to the spot at which you wish the line to begin) by moving the mouse. As you move the mouse, notice that the crosshair cursor moves in concert.



The BricsCAD drawing area

3. At any spot in the drawing area, pick a point on the screen by pressing the **first** button on your mouse. (The first button is the *left-most* one.) The left button is known universally in the CAD world as the "pick button." Notice that the Command bar changes the prompt wording by adding more options. I'll tell you their meanings later.

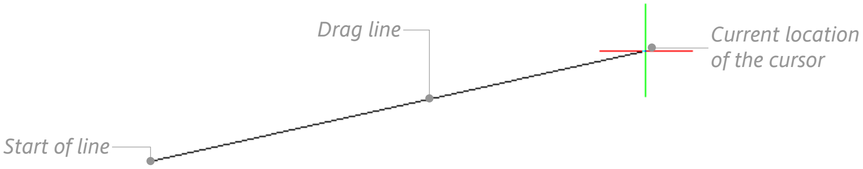
   `Set end point or [Angle/Length/Undo]:` *`(Pick another point)`*



```
: LINE
Start of line:
Angle/Length/Undo/<End point>:
Angle/Length/Follow/Undo/<End point>:
```

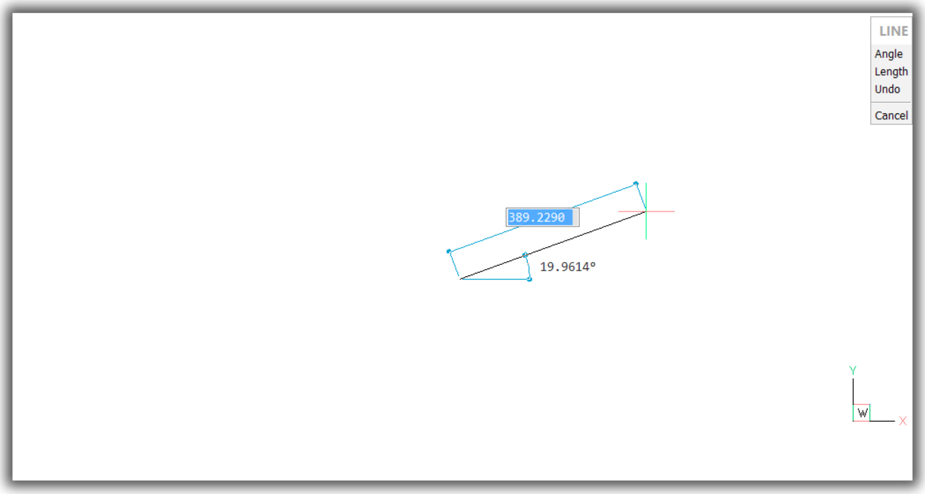
Following the instructions in the command bar

4. As you keep moving the mouse, notice the line that stretches like a "rubber band" from the point you picked. This rubber band is also known as the "drag line." It shows where the line would be located if you were to click now.



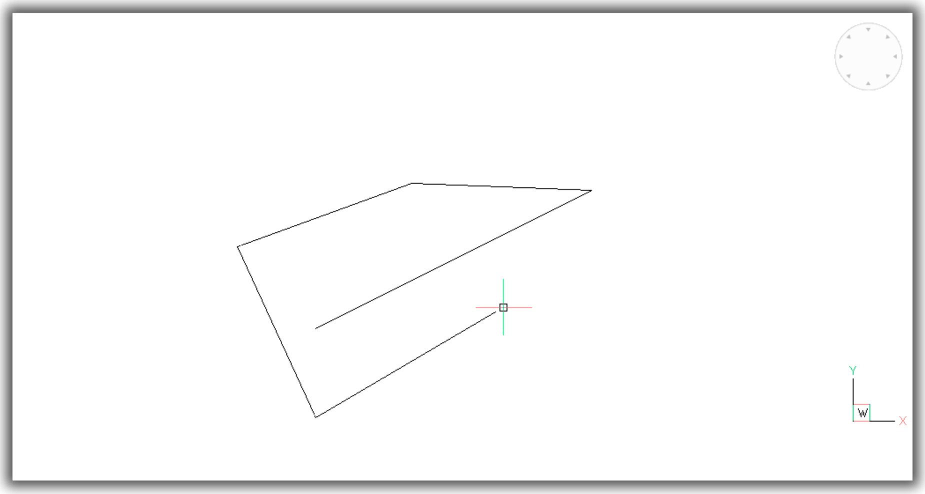
Rubberband drag line previewing the angle and length of the line being drawn

Notice that BricsCAD may be providing you with information about the state of the line, specifically its length and its angle. If this appears on-screen, then it is called "direct distance entry." You learn more about direct distance entry in a later chapter.



Using direct distance entry to specify the second point of a line

5. Move the mouse some more, and then press the pick button again. There: you've drawn your first line with BricsCAD!
6. Continue drawing some more lines by repeating the same steps: (a) move the mouse and then (b) press the pick button. Draw as many lines as you like.



Drawing more line segments

7. To end the Line command, press the **Esc** key. Pressing Esc stops just about any command, although in some commands you may need to press the key two or three times.

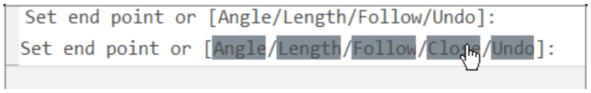
```
Set end point or [Angle/Length/Follow/Close/Undo]: (Press ESC)
```

Handling Options Efficiently

I want to point out that the prompt text — 'Set end point or [**A**ngle/**L**ength/**F**ollow/**C**lose/**U**ndo]' — has subtle aspects that are not immediately obvious.

You can specify an option by typing just the letter(s) displayed in uppercase, such typing 'a' to start the **Angle** option. Options are shown in the square brackets.

(NEW IN V20) Alternatively, you can use the mouse cursor to pick the name of the option.



```
Set end point or [Angle/Length/Follow/Undo]:
Set end point or [Angle/Length/Follow/Close/Undo]:
```

Picking an option with the cursor

Sometimes, there is a *default* option (or value) that is shown angle brackets, such as <Last point>. "Default" means that this is what BricsCAD will do when you just press **Enter**, without picking an option. This becomes a pretty fast way of working.

TIP Pressing **Enter** when you use the **Line** command has different effects, depending on the prompt that is currently active. (See the table below.) This is why it is important to always keep an eye on the prompts displayed by BricsCAD on the Command bar.

Prompt	Effect of pressing Enter at the prompt
Start of line:	Pressing **Enter** makes BricsCAD continue drawing from the last point, that was placed as a line or an arc. This is a great way to ensure that new lines are perfectly tangent to the ends of previous lines or arcs.
<End point>:	Pressing **Enter** terminates the Line command, just as **Esc** does.
:	Pressing **Enter** repeats the last command, which in this case is the Line command.

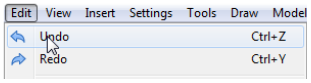
Undoing What You've Done: U

To erase the lines you drew, type **U** at the ':' prompt to *undo* the lines, as follows:

```
: u
```

Alternatively, you could access the U command in these other ways:

- Click the **Undo** icon on the Quick Access toolbar
- Select **Undo** from the **Edit** menu



- Press **Ctrl+Z** — the shortcut keystroke for undo, and one that your fingers should memorize!

Trust me, **Ctrl+Z** will become your best friend!

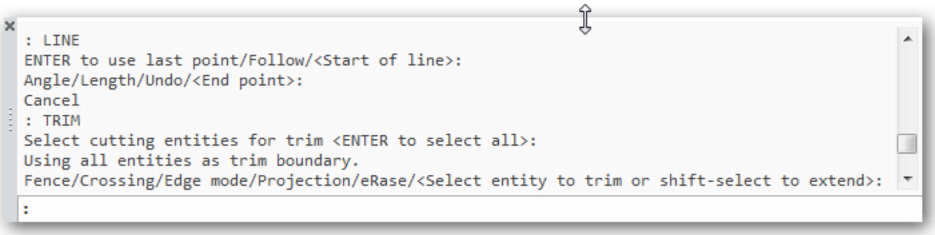
As you can see from the undo example, BricsCAD provides several ways to perform the same action. In the days to come, you will probably find yourself using a combination of keyboard typing, toolbar or ribbon icons, menu picks, keyboard shortcuts — whichever one you find the most convenient.

TIP At any time, right-click in the drawing area to display shortcut menus. (Press the mouse's right button.) These menus show commands that are relevant to the current action or the state of the drawing. Because these menus are *context-sensitive*, their content changes depending on what's going on at the time you right-clicked.

Seeing What You Did Before: Command History

The command bar typically displays three or four lines of *history*, which is the text of previously displayed prompts. When you need to see more lines of history, then you have a couple of choices:

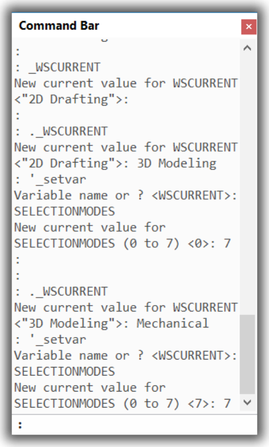
- Drag the command bar's top border to stretch it taller or shorter. For the exact point at which to do this, see the location of the double-ended arrow cursor in the figure below.



```
: LINE
ENTER to use last point/Follow/<Start of line>:
Angle/Length/Undo/<End point>:
Cancel
: TRIM
Select cutting entities for trim <ENTER to select all>:
Using all entities as trim boundary.
Fence/Crossing/Edge mode/Projection/eRase/<Select entity to trim or shift-select to extend>:
:
```

Changing the size of the command bar

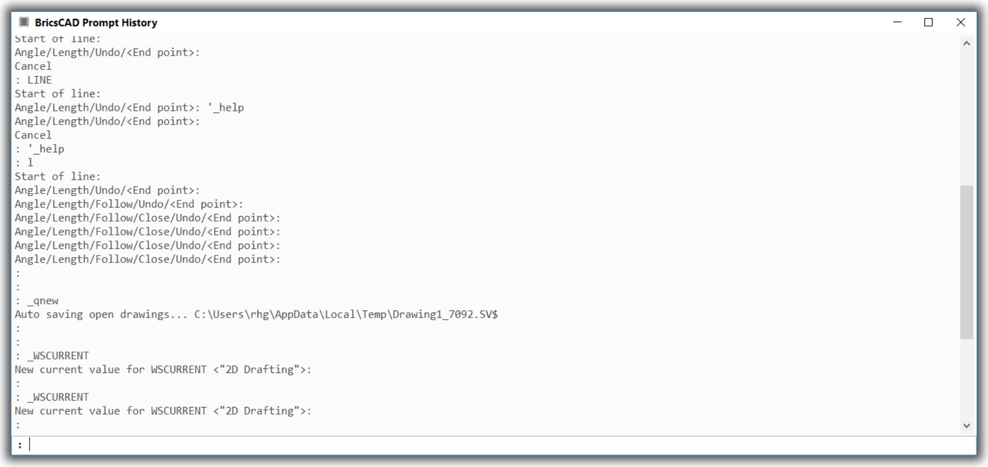
- Drag the bar away from its docked position, and then resize it, as shown below.



```
:
: _WSCURRENT
New current value for WSCURRENT
<"2D Drafting">:
:
: ._WSCURRENT
New current value for WSCURRENT
<"2D Drafting">: 3D Modeling
: '_setvar
Variable name or ? <WSCURRENT>:
SELECTIONMODES
New current value for
SELECTIONMODES (0 to 7) <0>: 7
:
:
: ._WSCURRENT
New current value for WSCURRENT
<"3D Modeling">: Mechanical
: '_setvar
Variable name or ? <WSCURRENT>:
SELECTIONMODES
New current value for
SELECTIONMODES (0 to 7) <7>: 7
:
```

Floating command bar

- Press **F2** to see the prompt window. BricsCAD display a second window on the computer screen. You can reposition it anywhere, such as on a second screen.



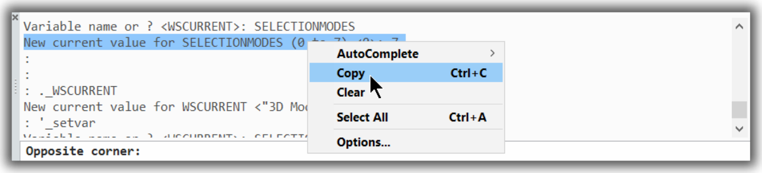
```
Start of line:
Angle/Length/Undo/<End point>:
Cancel
: LINE
Start of line:
Angle/Length/Undo/<End point>: '_help
Angle/Length/Undo/<End point>:
Cancel
: '_help
: l
Start of line:
Angle/Length/Undo/<End point>:
Angle/Length/Follow/Undo/<End point>:
Angle/Length/Follow/Close/Undo/<End point>:
Angle/Length/Follow/Close/Undo/<End point>:
Angle/Length/Follow/Close/Undo/<End point>:
Angle/Length/Follow/Close/Undo/<End point>:
:
:
: _qnew
Auto saving open drawings... C:\Users\rhg\AppData\Local\Temp\Drawing1_7092.SV$
:
:
: _WSCURRENT
New current value for WSCURRENT <"2D Drafting">:
:
: _WSCURRENT
New current value for WSCURRENT <"2D Drafting">:
:
: |
```

The text window

The Prompt History window displays the most recent 400 lines of command text. You can scroll back to earlier text by clicking on the vertical scroll bar along the right edge of the window. You can leave the window up, or else dismiss it by again pressing **F2**.

To keep a copy of the history, enter the **LogfileOn** command. Everything typed in the command bar is recorded to a *.log* file with the same name as the drawing in this folder: *C:\Users\userid\AppData\Local\Bricsys\BricsCAD\V20x64\en_US\*. Use the **LogFilePath** variable to specify a more convenient folder, and the **LogFileOff** command to turn off command logging.

Alternatively, you can copy the text to the clipboard: select the text, then press **Ctrl+C**. Once copied, you can paste the text in any word processor or text editor. (This is process we use to get command prompts and path names into this book!) Right-click the text window for more options in the shortcut menu.



```
Variable name or ? <WSCURRENT>: SELECTIONMODES
New current value for SELECTIONMODES (0
:
:
: ._WSCURRENT
New current value for WSCURRENT <"3D Mo
: '_setvar
Opposite corner:
```

Using Ctrl+C to copy command text

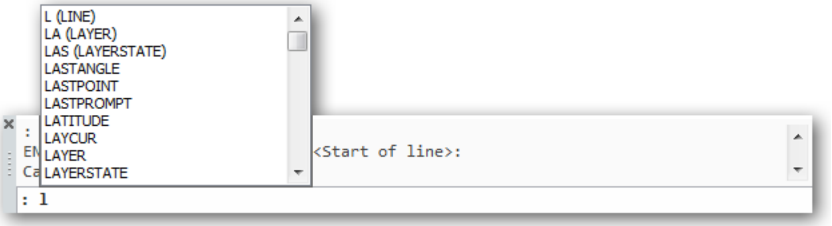
Typing Less: Aliases and AutoComplete

There are two ways to spend less time entering command names: by entering *aliases* and by taking advantage of the *auto-complete* function. I use both.

- **Aliases** — command abbreviations, such as 'l' for the Line command
- **Auto-complete** — lists the names of all commands that begin with the same letter(s) as you are typing

Aliases are described later in this book. Here is how auto-complete works with the Line command:

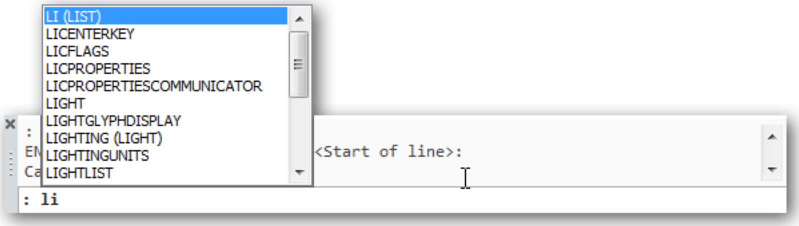
1. At the command prompt, type **L**. Notice that a box pops with listing the names of all commands and system variables that start with 'l'. To see them all, scroll down the list, all the way to where it ends with "lwunits."



```
<Start of line>:
: l
```

Auto-complete showing commands beginning with L

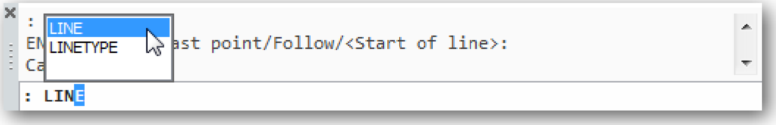
2. Now type **i**, the second letter of the Line command. Notice that the suggestion list shortens to just the names that begin with 'li.'



```
<Start of line>:
: li
```

Auto-complete showing commands beginning with LI

   a. You can keep typing letter of the command name to further reduce the suggestion list.



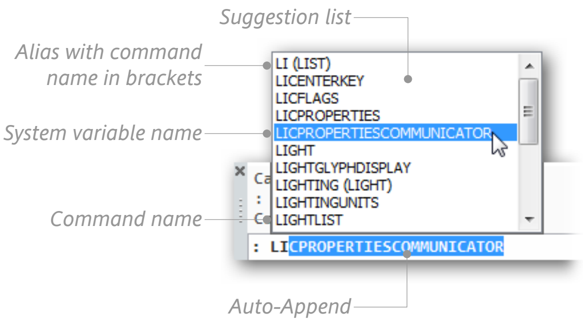
```
ast point/Follow/<Start of line>:
: LINE
```

Auto-complete showing commands beginning with LIN

   Notice that the last letter ('E') is highlighted in blue. This means that if you now press **Enter**, the entire command shown ('LINE') will be executed.

   b. Or you can use the cursor to select a name from the list. It's your choice.

Here are all the elements of auto-complete:



The auto-complete display

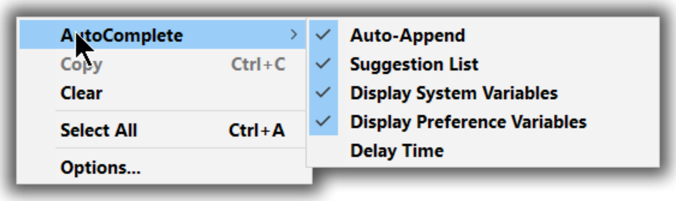
Suggestion list — lists the names commands and system variables; you can turn off the display of system variables to declutter the list, as described below.

Aliases — identified by appending the full command name in brackets, such as LI (LIST).

Systems variables — are settings that determine how BricsCAD looks and acts; often, they are the same as AutoCAD's. *Preference variables* are similar, but are unique to BricsCAD. See Appendix B for the list of all of them.

Auto-Append — highlights in blue letters of the command name BricsCAD automatically completes for you.

To change the auto-complete settings, right-click the command bar and then choose **AutoComplete** from the shortcut menu.



Changing auto-complete options

Here is what the options in the shortcut menu mean:

Auto-Append — turns on or off the command-completion characters, which are highlighted in blue

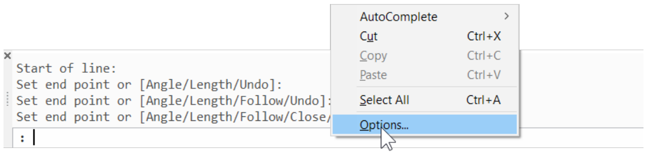
Suggestion List — toggles the display of the auto-complete list

Display System Variables — toggles the display of system variable names in the list

Display Preference Variables — toggles the display of preference variable names in the list

Delay Time — determines how long BricsCAD waits before displaying the auto-complete list

To change other things like the font or color used by the command line, right-click it and then choose **Options** from the shortcut menu.



```
Start of line:
Set end point or [Angle/Length/Undo]:
Set end point or [Angle/Length/Follow/Undo]:
Set end point or [Angle/Length/Follow/Close/
: |
```

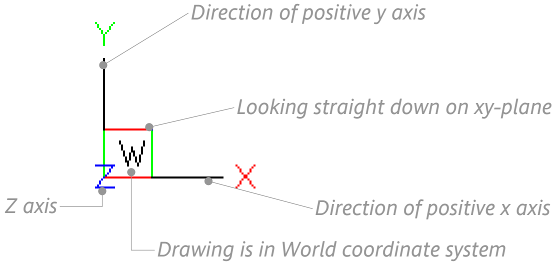
Accessing more options for the look of the command bar

This brings up the Settings dialog box at the Command Bar section, where you can make changes to your heart's delight.

UCS ICON

Command	UcsIcon
Ribbon	...
Menu	...
Alias	...

The *UCS* icon is usually found somewhere in the drawing area, and often in the lower-right corner. UCS is short for "user-defined coordinate system." Its job is to help you understand the drawing's orientation in 3D. In 2D mode, it looks like this, packing a whole bunch of information:


Direction of positive y axis
Y
Looking straight down on xy-plane
W
X
Z
Z axis
Direction of positive x axis
Drawing is in World coordinate system

The elements of the UCS icon

X,Y Axes — red X and green Y lines point in the direction of the positive x and y axes. At the intersection of the two lines is where the Z line points up, invisible now in 2D mode.

Drawing Origin — the UCS icon is usually (but not always) placed at the origin of the drawing. The origin is the intersection of the x and y axes, precisely where x = 0 and y = 0 are located.

SUPPORTED GRAPHICS BOARDS

BricsCAD works with whatever graphics board is built into your computer. When it comes to non-wireframe renderings, however, BricsCAD employs RedSDK GPU-acceleration technology provided by Redway3D. (GPUs are the processing chips on graphics boards.)

WINDOWS

On Windows, Redsdk hardware acceleration supports graphics boards made by AMD, Intel, and nVidia. See http://www.redway3d.com/downloads/public/documentation/bk_ba_gpu_chipset_reference.html for the comprehensive list.

MAC

Apple does not allow developers to have full access to the graphics hardware inside Mac computers, and so there are no third-party drivers available to speed up the display of CAD programs.

LINUX

On Linux, RedSDK support graphics chip sets for 3D graphics hardware acceleration from AMD and nVidia. Intel is not supported in Linux, nor are laptops with discrete graphics systems. You are advised to download the latest recommended drivers from NVIDIA and AMD.

World Coordinate System — when there is a 'W' at the origin, it means you are "looking straight down" the z axis, and straight onto the x,y-plane. The *W* means "world coordinate system," which is the normal Cartesian system you may have learned about in school.

User-defined Coordinate System — when the W is missing, it means the drawing is in a "user-defined coordinate system," or *UCS*, for short. A UCS is one that you or someone else created. User-defined coordinate systems are especially handy when drawing on the slopes of roofs or the undersides of boxes.

I find the UCS icon gets in the way of 2D drafting, and so I recommend that you turn it off, as follows:

1. Type the **UcsIcon** command at the ':' prompt in the command bar, as follows:

   ```
   : ucsicon (Press Enter)
   ```

   Press **Enter** to execute the command.
2. Notice the next prompt displayed by BricsCAD:

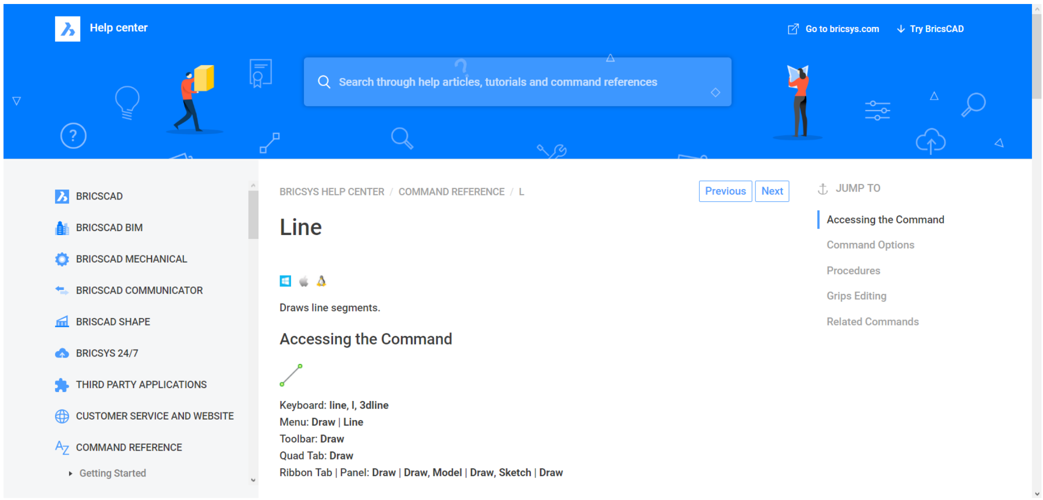
   ```
   [ucs icon ON/ucs icon OFf/display in All views/display at ORigin/display in Corner]
   <ON>: off (Press Enter)
   ```
3. Type 'Off,' and then press **Enter**. Notice that the UCS icon disappears from the drawing area.

You'll get to try out the UCS icon later during the 3D modeling portion of this book.

ONLINE HELP

Command	Help
Alias	?
Menu Bar	Help | Help
Ribbon	Home | Help | Help
Shortcut	F1

To peruse help during a command, press **F1** and BricsCAD displays Help in your computer's Web browser, such as is illustrated below for the Line command.



Line

Draws line segments.

Accessing the Command

Keyboard: **line, l, 3dline**
Menu: **Draw** | **Line**
Toolbar: **Draw**
Quad Tab: **Draw**
Ribbon Tab | Panel: **Draw** | **Draw, Model** | **Draw, Sketch** | **Draw**

Help being displayed in the Web browser

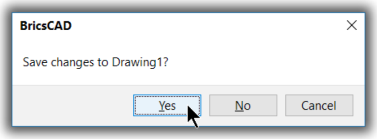
Icons indicate whether the command operates in each of the supported operating systems, Windows, MacOS, or Linux; most commands do. Other icons indicate which edition the command works with, such as Pro, Platinum, BIM, or Mechanical.

For the complete list of commands, see Appendix A.

Exiting BricsCAD

Command	Quit
Menu Bar	File | Exit
Shortcuts	Ctrl+Q, Alt+F4

To exit BricsCAD, use the **Quit** command. When BricsCAD asks if you want to save the drawing, click **No**.



Save changes to Drawing1?

Saving changes to the drawing before exiting BricsCAD

Alternatively, press **Ctrl+Q**, or else select **Exit** from the **File** menu.

CHAPTER 2

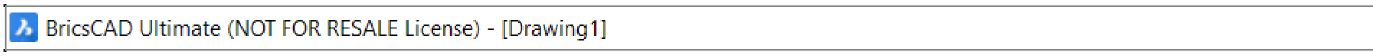
Navigating the BricsCAD Interface

In this chapter, we continue the tour of BricsCAD's user interface (UI) in great detail. We will work our way from the top of the screen to the bottom — from the title bar down to the status bar. In subsequent lessons, you'll learn some of the nuances to the UI, such as entering aliases and working with relative coordinates.

IN THIS CHAPTER

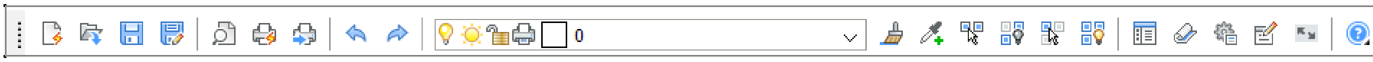
- Working with the upper half of BricsCAD: title bar, Quick Access toolbar, ribbon, and drawing tabs
- Trying out some user interface elements of the drawing area: Quad, LookFrom widget, and more
- Finding out about the lower half of BricsCAD: layout tabs, scroll bar, command bar, and status bar
- Understanding panels (palettes)

Title bar shows the name of the program and the current drawing, with program window controls at the far right:

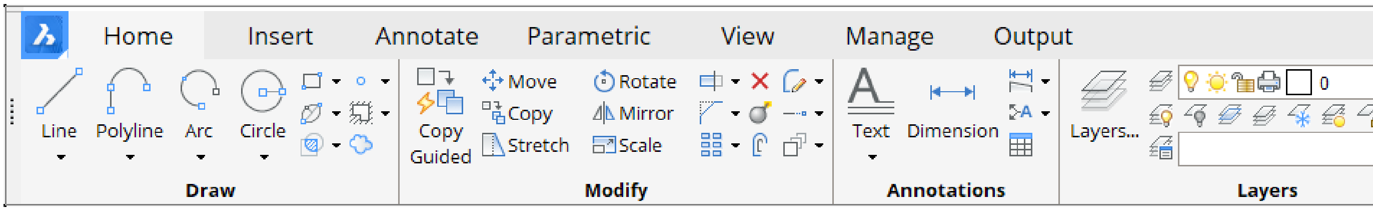


BricsCAD Ultimate (NOT FOR RESALE License) - [Drawing1]

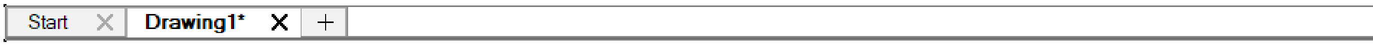
Quick Access toolbar lists commonly-used commands in a row of buttons:



Ribbon segregates commands into tabs and panels, with buttons, flyouts, and droplists:



Drawing tabs access all open drawings quickly:



Above the Drawing Area

The figure above explodes the top part of BricsCAD's interface into horizontal bars. Let's take a look at them, from top to bottom:

Title bar — reports the name of the program and current drawing file, as in "BricsCAD - [Drawing1.dwg]"

Quick Access toolbar — contains commands that are often used in BricsCAD

Ribbon — collects tabs and panels with icons that execute commands

Drawing tabs — switches between open drawings quickly

Let's take a look at each one to better understand their function in BricsCAD.

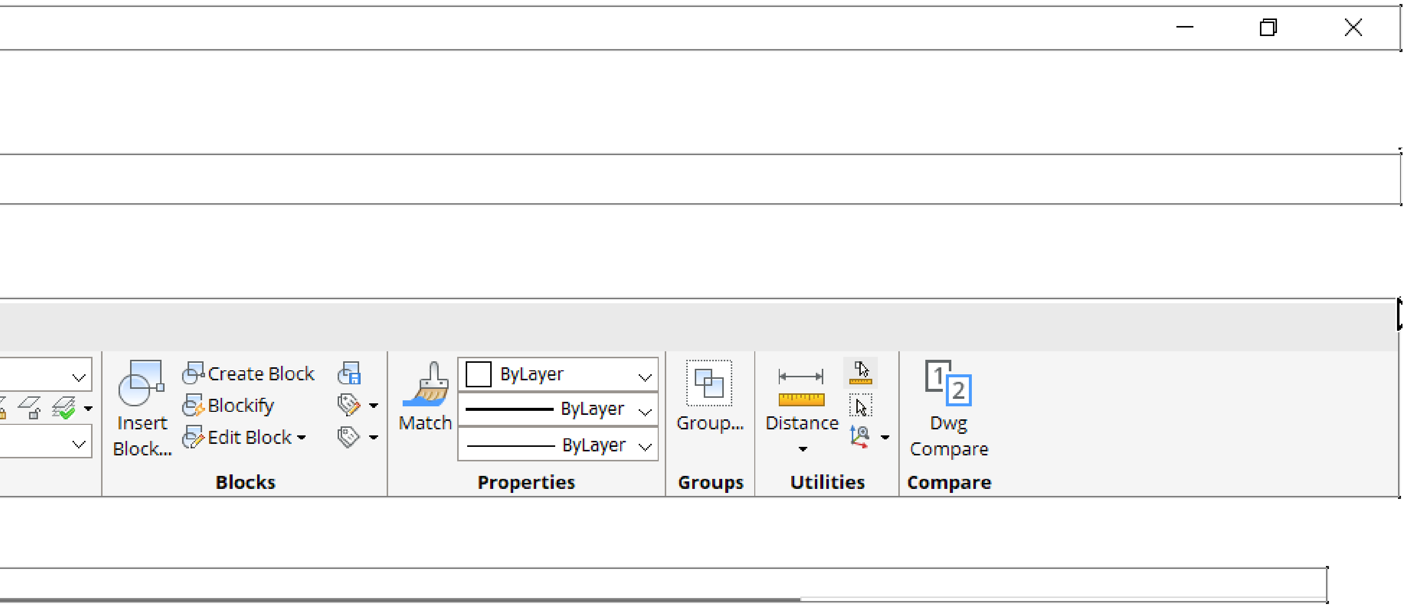
TITLE BAR

The title bar displays the name of the BricsCAD program, its edition (such as "Platinum"), and the drawing with which you are working, such as...

```
[Office Fixtures.dwg]
```

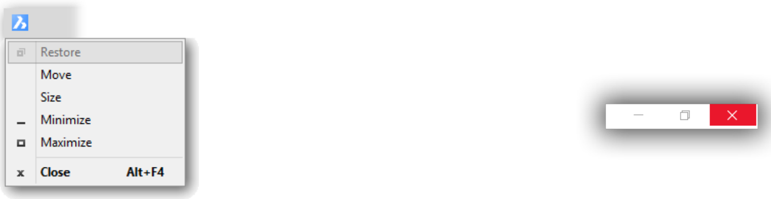
If the drawing is "read-only," it cannot be saved by the same name to its original file location, usually because another copy of the drawing is already open elsewhere. In this case, a note is added to the file name that looks like this:

```
[Office Fixtures.dwg (Read-Only)]
```



TIP The title bar has a couple of hidden tricks:
Maximize the BricsCAD window quickly by *double-clicking* the title bar
Restore the window by double-clicking the title bar a second time
Open a drawing quickly by dragging it from the file manager to BricsCAD's title bar

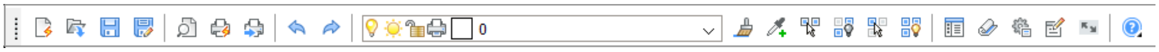
Both ends of the title bar provide buttons and menus that control the size of the BricsCAD window. At the left end is a rarely used-menu that performs many of the same functions as the trio of buttons over on the right end: to — minimize, maximize or ❐ restore the BricsCAD window, or else ☒ exit BricsCAD.



Left: *Control menu at left end of title bar;* ***right:*** *equivalent control buttons at right end of title bar.*

QUICK ACCESS TOOLBAR

(NEW IN V20) The Quick Access toolbar lists commonly used commands in a format that is always visible — unlike the ribbon, where tabs and drop panels hide commands from view.



The Quick Access toolbar for 2D drawings

The commands are grouped as follows, with a vertical bar separating groups:

Filing commands — start a new drawing, open a drawing, save the current drawing, and save the current drawing by another name

Printing commands — preview, print the drawing, and publish a group of drawings

Undo and redo — undo the last command, and then redo it

0 **Layer droplist** — control the display of layers

Selection commands — match properties from one entity to others, and isolate-unisolate entities

Panel toggles — turn the display of Drawing Explorer, Sheet Sets, Settings, and Properties panels on and off, as well as maximize the drawing area (clean screen)

Help — access online help

A slightly different toolbar is displayed in 3D workspaces; it offers 3D selection modes.

TIP The Quick Access toolbar can contain any commands that you like. To change the toolbar, enter the **Customize** command, choose the **Toolbars** tab, and then select the **Access** node.

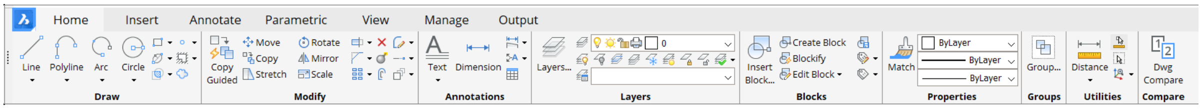
RIBBON TABS AND PANELS

The ribbon dominates the upper part of the user interface. It's kind of like a series of overlapping toolbars, where of *tabs* segregate the "toolbars" into groups of functions. Each tab has is further segregated by a series of *panels*, and each panel contains a group of buttons, flyouts, and droplists.

If you do not see the ribbon, enter the **Ribbon** command:

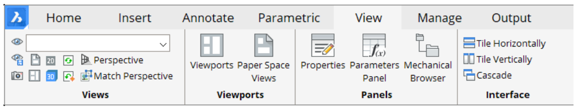
```
: ribbon
```

Shown below is the ribbon's standard Home tab for 2D drawings. The Home tab changes somewhat when you work with 3D workspaces.



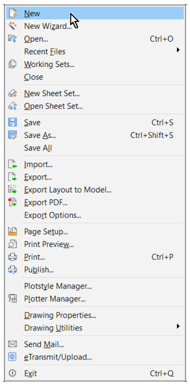
The Home tab of the ribbon for 2D drawings

Other tab names are Insert, View, and so on. Each tab in the ribbon shows a different group of panels. Click the name of a tab to switch to it. Here is the **View** tab.



The View tab of the ribbon

(NEW IN V20) The blue Bricsys logo is a button that accesses a menu of file-related commands. This replaces the Files tab from earlier releases of the program.

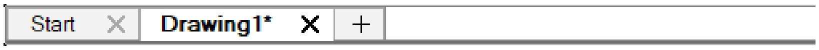


Menu of file-related commands

Because Bricsys wrote its own version of the ribbon interface, it is equally available on the Windows, Mac, and Linux versions, unlike all other CAD systems. You customize the ribbon through the **Customize** command.

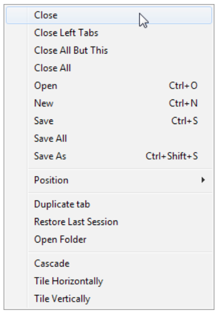
DRAWING (DOCUMENT) TABS

Drawing tabs let you switch quickly between open drawings, and provides a shortcut to file-related commands, such as Open and Close. BricsCAD calls this interface element "Document Tabs."



Drawing tabs show the drawing names that are currently open

To access the commands, right-click a drawing tab:



Context menu of document tab actions

Most of the commands are probably already familiar to you, but here are a couple that are unique to BricsCAD:

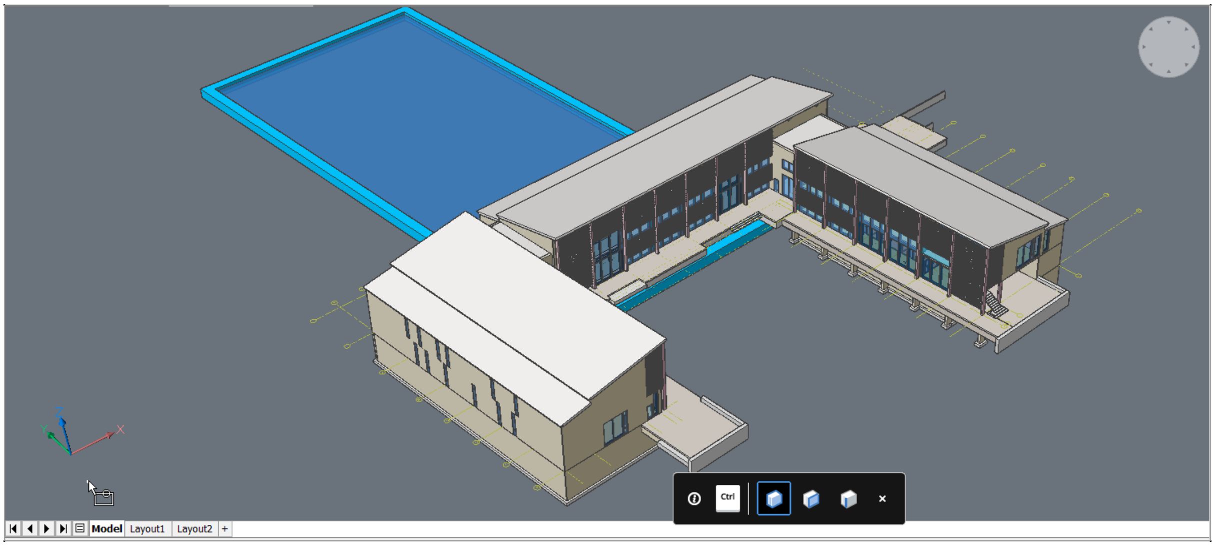
> **Close Left Tabs** — closes all drawings to the left of this tab. This is useful for closing older drawings, ones that were opened earlier
>
> **Close All But This** — closes all other drawings, except the current one. I could find this useful when I open an entire folder's worth of drawings, and then want to keep just one open
>
> **Save All** — saves all drawings at once
>
> **Duplicate Tab** — makes a copy of the current drawing, naming it *Copy_name.dwg*
>
> **Open Folder** — opens the folder from which the drawing was opened

(NEW IN V20) The first tab displays the Start window, in which you can also access recently-opened drawings, tutorials, and third-party add-ons. The tab is called "Start." See Chapter 1.

There are two variables that control the visibility of these drawing tabs. **DocTabPosition** places the tab at the top, bottom, left, or right of the drawing area. **ShowDocTabs** turns the tab row on and off.

In the Drawing Area

The drawing area is in the center of the BricsCAD window, as shown here.



The drawing area in BricsCAD

Here I will tell you about many of the user interface elements you find in the drawing area:

Look From widget — interactively changes the 3D viewpoint of drawings; turn on-off and adjust settings with the LookFrom command

UCS icon — reports the orientation of x,y- and x,y,z-axes in 2D and 3D space; turn on-off and adjust settings with the UCSicon command

Hot Key assistant — indicates options available with the Ctrl key; toggle on-off by clicking HKA on the status bar

Dynamic Input — reports distances and angles of entities as they are being drawn; toggle on-off by clicking DYN on the status bar

Shortcut menus — right-click menus that provide access to useful commands and options

Quad cursor — at-cursor access to context-sensitive drawing and editing commands; toggle on-off by clicking QUAD on the status bar

Scroll bars — pan the drawing horizontally and vertically; turn on and off with the ScrollBars command

Prompt menu — displays command options in a small menu

Manipulator widget — directly edits entities with actions like move, copy, and rotate; toggle on-off with the Manipulator system variable

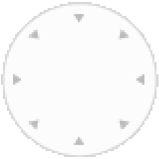
Clean screen — minimizes the user interface to maximize the drawing area

Panels — sources of information and control over drawings and entities

LOOK-FROM CONTROL

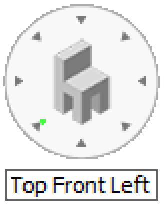
Command	LookFrom

The LookFrom widget lets you quickly change the viewpoint of 3D drawings. The widget is found in the upper right corner of the drawing area.



LookFrom control with no cursor interaction

When you pass the cursor over the widget, the preview image of a chair appears. Pausing the cursor over one of the triangles previews what the 3D view will look like as the chair changes its position. The small green dot indicates the cursor position, kind of like a laser pointer:



Cursor pausing over one of the direction triangles

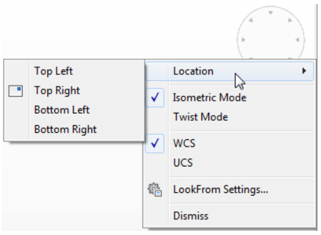
Clicking the triangle changes the 3D viewpoint.

Top Front Right

TIPS To see the bottom view, hold down the **Ctrl** (or **Cmd** in Mac) key.

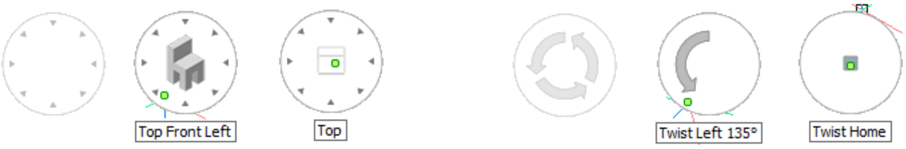
Click the center of the LookFrom control to return the view to its **home** view, or press the **Home** button on the keyboard. This is particularly helpful when running the widdget in Twist mode.

There are two ways you can change the way in which the LookFrom control operates. The easier one is right-click the control, and then choose an option from the shortcut menu.



Most of the options in the shortcut menu are straight-forward, but I do want to explain the difference between Isometric and Twist modes:

- **Isometric** mode is like using the Viewpoint or View commands
- **Twist** mode is like using the RtRotF (real time view rotation) command



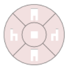
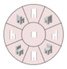
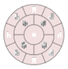
Left: *LookFrom in isometric mode;* ***right:*** *And in twist mode*

The other method is to enter the **LookFrom** command, from which you can turn off (and on) the control and access its settings:

```
: lookfrom
LookFrom [ON/OFF/Settings] <ON>:
```

The **Settings** option opens the Settings dialog box at the LookFrom section. Here you can adjust the properties of the widget, such as its translucency and position. Of particular interest is the number of isometric viewpoints it can display, which is set through "Direction Mode" or the **LookFromDirectionMode** variable.

The following table shows you the options:

LookFromDirectionMode		Number of Views
0		6 orthogonal views
1		14 views; no flat views of corners
2		18 views; top down corners
3		26 views; eight top down corners

TIP To quickly change the visual style of 3D models, such as wireframe or rendered, use the **Visual Style** setting in the Properties panel.

THE UCS ICON

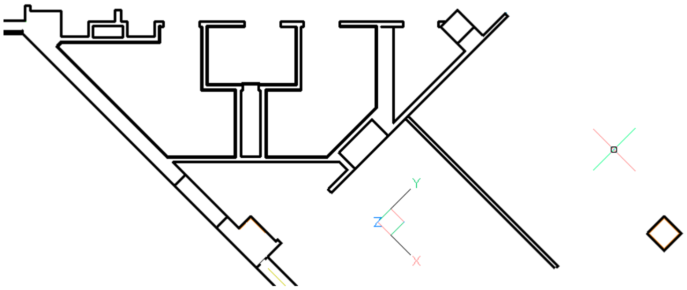
`Status bar`	DUCS
`Command`	UCS

I described the function of the UCS (user-defined coordinate system) icon in the last chapter. Here I want to talk a bit about how the UCS is used — rarely in 2D drafting, but a lot in 3D modeling.

In short, a UCS lets us to twist the x,y,z axes in any direction in space. This makes it easier to draw and edit at unusual angles and on slanted surfaces.

Static UCSs for 2D. When we draw things at an angle in 2D, such as the angled wing of a building, then we would use a UCS. Changing the x,y-coordinate system to match the angle of the building rotates everything else as well.

In the figure below, I rotated the UCS icon to match the angled walls; notice that the crosshair cursor matches the new angle.

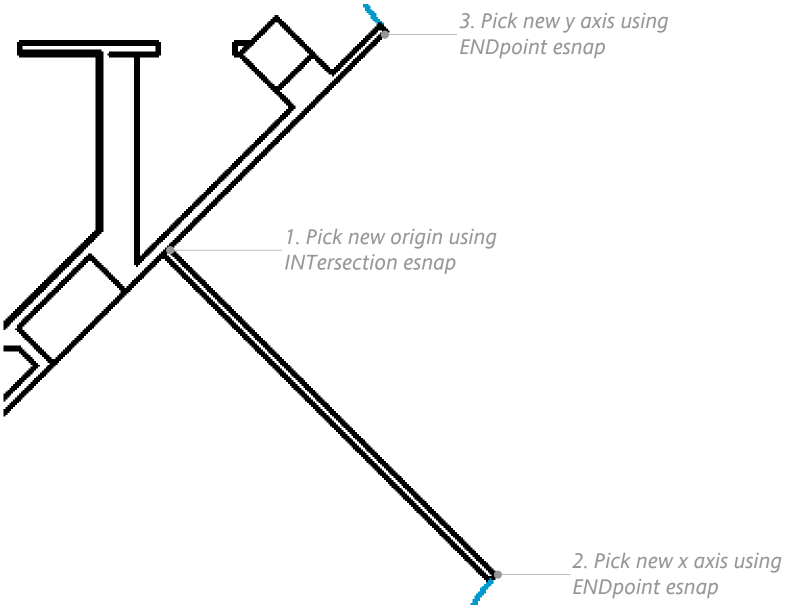


Reorienting the coordinate system

To change the angle of the UCS icon, you need to tell the **UCS** command three things:

1. The new origin point (0,0)
2. The new direction of the x axis (determines the angle)
3. The new direction of the y axis (determines the orientation)

From the three parameters, BricsCAD figures out the direction of the z axes using the right-hand rule.



3. Pick new y axis using ENDpoint esnap

1. Pick new origin using INTersection esnap

2. Pick new x axis using ENDpoint esnap

Determines a new orientation of the UCS

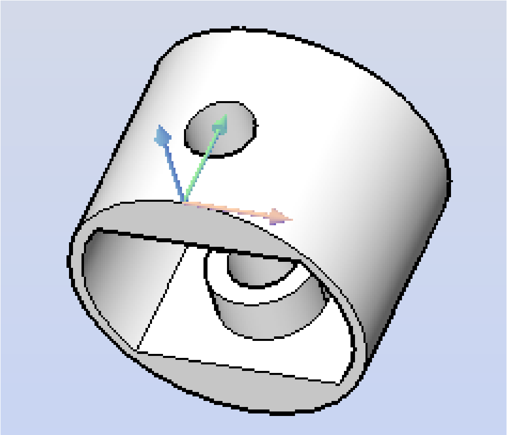
Here is how to use the **UCS** command to do this:

```
: ucs
Specify origin of UCS or [Face/NAmed/Entity/Previous/View/X/Y/Z/Z Axis/Move/World] <World>:
(Pick a point like the intersection of two lines; BricsCAD turns on INTersection entity snap
automatically)
Point on X-axis/<Accept>: (Pick a point along the new x axis; BricsCAD turns on ENDpoint
entity snap automatically)
Point on the XY-plane with positive Y value/<Accept>: (Pick a point along the new y axis;
BricsCAD turns on ENDpoint entity snap automatically)
```

If you use the same UCS often, then you can give it a name to let you switch to it quickly. Named UCSes are managed by the **ExpUcs** command.

Dynamic UCSs for 3D. The same system works for manually changing the UCS in 3D modeling. Why would you want to change the UCS in 3D? Because CAD is still fundamentally 2D, even when in 3D. To draw on a face, the UCS needs to match the orientation in space of the face.

BricsCAD, fortunately, does this for you automatically through *dynamic UCS*. Here is a brief example. In the figure below, the UCS icon is initially in the default "world" position:

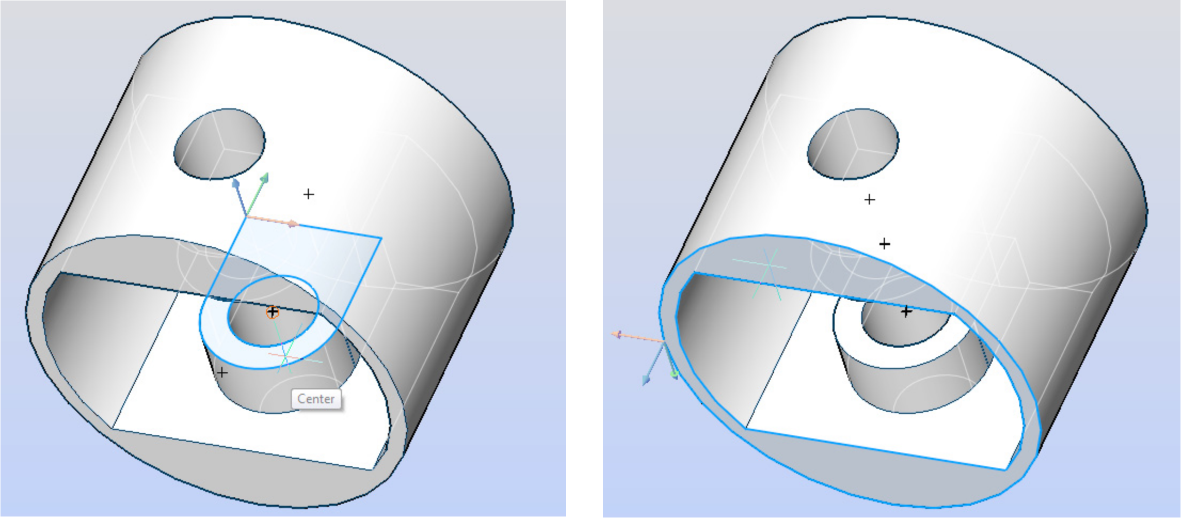


UCS in the world (default) position, aka world coordinate system

To write some text on the bottom of the piston, the UCS needs to be relocated to the flat face. To do so, first turn on dynamic UCS mode by clicking the **DUCS** button on the status bar (so that it looks black). Then start the **Text** command:

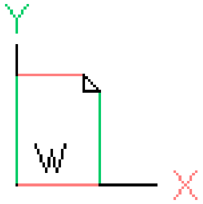
```
: text
Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>: (Pick a face on which to
write)
```

Now move the cursor around faces of the piston. Notice how the UCS jumps to match the orientation of the face. I show the two areas highlighted in blue in the figures below.



***Left:** UCS dynamically placed on one face...; **right:** ...and on another face*

UCSs in Layout Mode. In paper space of layout modes, the UCS icon turns into something that looks like a sheet of paper. It serves no purpose.



UCS icon in paper space

In the early days of CAD, back when layout mode was first introduced, the software interface was vague enough that it could be difficult for users to determine if they were in model or paper space. In some CAD programs, the UCS icon would switch to a triangle shape; in BricsCAD, it looks like a piece of paper.

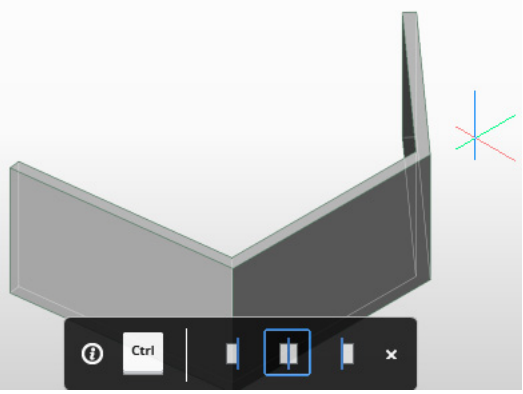
In CAD today, however, the user interface makes layout mode quite clear, and so the paper icon remains for sentimental reasons.

HOTKEY ASSISTANT

Status bar HKA

The hotkey assistant is an interactive widgets that reports options accessible by pressing the Ctrl key that might otherwise be unknown to users. (In earlier releases of the software, it was known as the "Tips widget.")

For example, the following widget appears during the Polysolid command.



Hotkey Assistance appears at the bottom of the drawing area

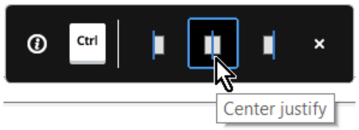
Notice that the widget shows several icons in a row:

i and **Ctrl** "buttons" — remind you to tap the **Ctrl** key to make the changes; the **i** and **Ctrl** icons do nothing

Two or more icons — display the change that would occur each time you tap the Ctrl key

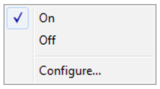
x — closes the widget

In the figure above, tapping the **Ctrl** key changes the justification of the polysolid changes between left, centered, and right. To get a brief description of the purpose, pause the cursor over the Tip.



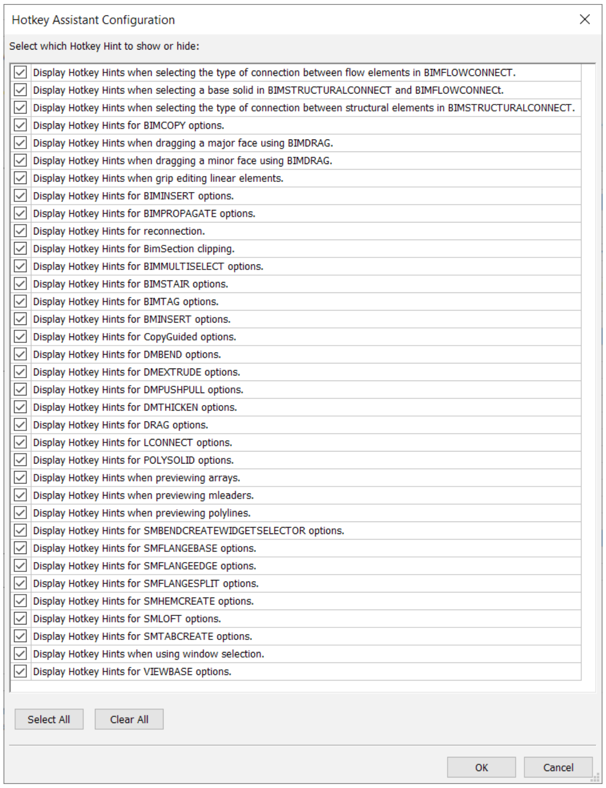
Tooltips reporting the purpose of the icon

To dismiss the widget, click the small **x** at the right end. The display of the widget is toggled through the **HKA** button on the status bar. Right-click the button for options.



Shortcut menu for the status bar's HKA button

The **Configure** button leads you to a dialog box that lets you determine when the widget appears.


Hotkey Assistant Configuration

Select which Hotkey Hint to show or hide:

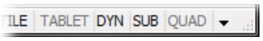
- ☑ Display Hotkey Hints when selecting the type of connection between flow elements in BIMFLOWCONNECT.
- ☑ Display Hotkey Hints when selecting a base solid in BIMSTRUCTURALCONNECT and BIMFLOWCONNECt.
- ☑ Display Hotkey Hints when selecting the type of connection between structural elements in BIMSTRUCTURALCONNECT.
- ☑ Display Hotkey Hints for BIMCOPY options.
- ☑ Display Hotkey Hints when dragging a major face using BIMDRAG.
- ☑ Display Hotkey Hints when dragging a minor face using BIMDRAG.
- ☑ Display Hotkey Hints when grip editing linear elements.
- ☑ Display Hotkey Hints for BIMINSERT options.
- ☑ Display Hotkey Hints for BIMPROPAGATE options.
- ☑ Display Hotkey Hints for reconnection.
- ☑ Display Hotkey Hints for BimSection clipping.
- ☑ Display Hotkey Hints for BIMMULTISELECT options.
- ☑ Display Hotkey Hints for BIMSTAIR options.
- ☑ Display Hotkey Hints for BIMTAG options.
- ☑ Display Hotkey Hints for BMINSERT options.
- ☑ Display Hotkey Hints for CopyGuided options.
- ☑ Display Hotkey Hints for DMBEND options.
- ☑ Display Hotkey Hints for DMEXTRUDE options.
- ☑ Display Hotkey Hints for DMPUSHPULL options.
- ☑ Display Hotkey Hints for DMTHICKEN options.
- ☑ Display Hotkey Hints for DRAG options.
- ☑ Display Hotkey Hints for LCONNECT options.
- ☑ Display Hotkey Hints for POLYSOLID options.
- ☑ Display Hotkey Hints when previewing arrays.
- ☑ Display Hotkey Hints when previewing mleaders.
- ☑ Display Hotkey Hints when previewing polylines.
- ☑ Display Hotkey Hints for SMBENDCREATEWIDGETSELECTOR options.
- ☑ Display Hotkey Hints for SMFLANGEBASE options.
- ☑ Display Hotkey Hints for SMFLANGEEDGE options.
- ☑ Display Hotkey Hints for SMFLANGESPLIT options.
- ☑ Display Hotkey Hints for SMHEMCREATE options.
- ☑ Display Hotkey Hints for SMLOFT options.
- ☑ Display Hotkey Hints for SMTABCREATE options.
- ☑ Display Hotkey Hints when using window selection.
- ☑ Display Hotkey Hints for VIEWBASE options.

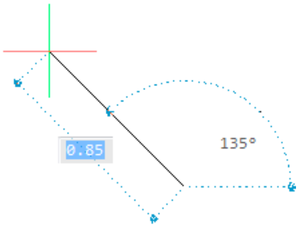
Determining when the Hotkey assistant appears

DYNAMIC INPUT

Status bar DYN

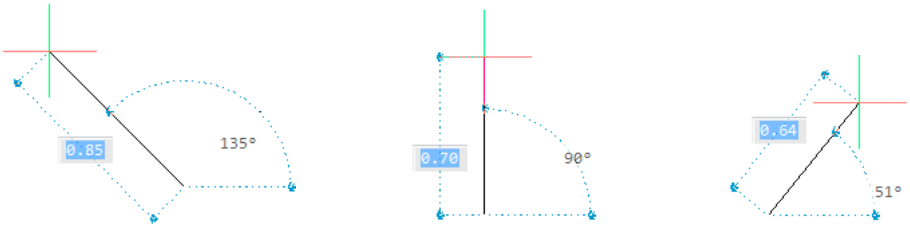
While traditionally we controlled drawing and editing commands by entering options, distances, and angles in the Command bar, another way of using dynamic input, because it is interactive within the drawing area. It is known as “dynamic input,” because it displays distances and angles dynamically — right at the cursor in the drawing.

(On the status bar, ensure **DYN** is turned on. When it is **black** (instead of gray), then dynamic input is on: .)



Dynamic input displaying the length and angle of the line being drawn

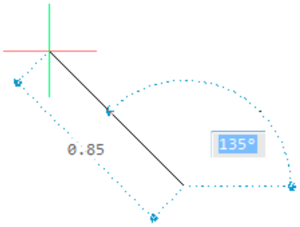
These are known as “dynamic dimensions,” because the length and angle values change as you move the cursor:



Dynamic dimension values changing as the cursor moves

Notice in the figures above that the length field is highlighted in blue. You can change the value at the keyboard, which is better for precision. As you enter a number at the keyboard, it replaces the value in the blue length field.

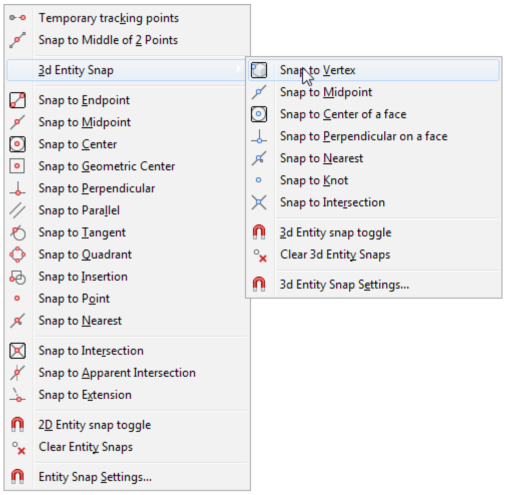
Then, to change the angle, press the **Tab** key. The angle field turns blue, and the **distance** field turns red (to indicate it is locked temporarily). Now you can enter an angle value at the keyboard. If necessary, press **Tab** to return to the length field, or else press **Enter** to finish data input.



Changing the value of the angle field

SHORTCUT MENUS

Scattered all about BricsCAD are accesses to shortcut menus. Nearly any place you right-click, a context menu will pop up with a list of commands. The commands usually are relevant to the spot you right-click. Holding down the **Shift** or **Ctrl** keys sometimes displays other shortcut menus.



*Holding down the **Shift** key while clicking the right mouse button accesses entity snap modes*

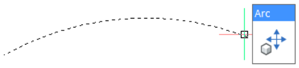
I don't detail them all here, because there are so many, but in later lessons, I'll point out shortcut menus that are useful to the work at hand. Short cut menus can be customized with the Customize command.

QUAD CURSOR

Status bar QUAD
Shortcut F12

The Quad is unique to BricsCAD in the way that it incorporates drawing and editing commands into a single interface that operates right at the cursor. This multifunction cursor takes its cue from the "heads-up" style of computer interface design, placing in the drawing area many useful commands.

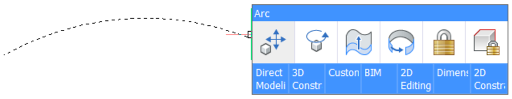
The Quad is normally not visible; most of the time, you see the standard tri-color crosshair cursor or arrowhead cursor. When you pass the cursor over an entity, however, the quad cursor appears, first as a single button; see figure below.



Initial display of the Quad

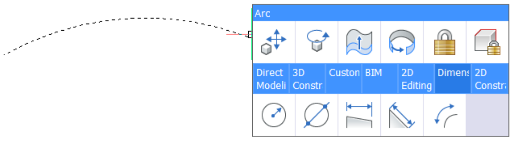
(If the Quad does not appear, then turn it on by clicking **QUAD** on status bar or pressing function key **F12**.)

When you move the arrow cursor onto the sole button, the quad cursor expands to show additional buttons, usually for commands most commonly used with the nearest entity.



Editing commands

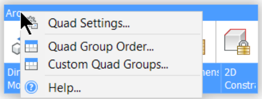
In addition, there are blue boxes for groups of additional buttons. Some groups are for common operations, while others are specific to the entity. To access the additional buttons, pass the cursor over a blue box. Click a button to execute its command.



More editing commands

The Quad changes its content, depending on the nearby entity and the workspace. BricsCAD comes with sets of predefined Quads for the various workspaces.

To customize the Quad, right-click it to access the following shortcut menu:



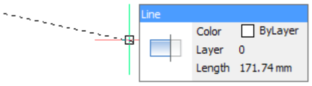
Changing the settings of the Quad

Quick Properties

Status bar RT

When Quick Properties are turned on, the properties of an entity are displayed by the Quad. To turn this on, click **RT** on the taskbar. (RT is short for “rollover tooltips” — I know, the name keeps changing. Sigh.)

Hover the cursor over an entity, and the Quad displays some of its properties:



Line

Color □ ByLayer
Layer 0
Length 171.74 mm

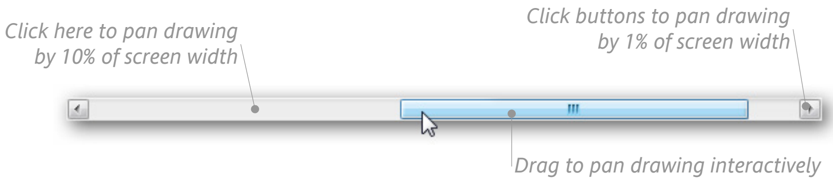
Properties displayed by the Quad

If you want the Quad to display other properties, then use the **Customize** command’s Properties tab to specify the properties for every entity.

SCROLL BARS

Command ScrollBars

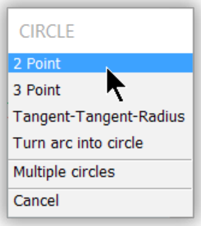
A pair of scroll bars rest at the right and bottom edges of the drawing area. They let you pan the drawing up and down, and left and right. Normally, they are turned off, but I find them handy, so I turn them on. To turn on scroll bars, enter the **ScrollBar** command.


Click here to pan drawing by 10% of screen width
Click buttons to pan drawing by 1% of screen width
Drag to pan drawing interactively

Scroll bars are limited to panning left-right and up-down. As a faster and more flexible alternative to scroll bars, hold down the mouse's middle button (or roller wheel), and then drag the mouse around. This action pans the drawing in any direction during any command.

PROMPT MENU

The Prompt menu shows the options that are available for the current command. This menu appears in the upper right corner (usually) of the drawing area. It is useful when the Command bar is turned off, because then you see what options are available for each command.



To select an option, just pick it from the list shown. When the command ends, the Prompt menu disappears.

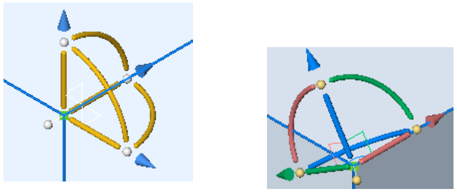
To change how the Prompt menu works, search for 'prompt menu' in the Settings dialog box.

MANIPULATOR WIDGET

Variable Manipulator

To manipulate entities interactively, BricsCAD provides the manipulator widget. When the widget is turned on, it attaches to the 2D or 3D entity that you select.

You drag arrowheads and bars of the widget to rotate, move, mirror, or scale 2D and 3D entities along the x, y, or z axes or xy, xz, or zy planes. An an alternative to dragging, you can entering values via dynamic dimensions.



Left: *Manipulator in default colors;* ***right:*** *...and in classic colors*

The **Manipulator** variable determines when the manipulator widget is available:

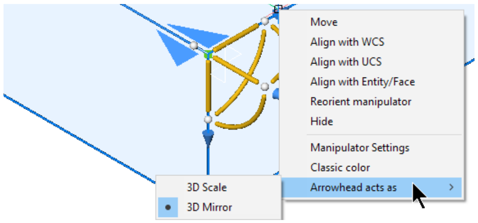
Manipulator	Meaning
0	Not displayed (default)
1	Display manipulator when entities are selected
2	Display manipulator when left mouse button is pressed longer than 250msec

Manipulator Action	Action
Drag a blue arrowhead	Scale (resize) or Mirror (default)
Drag a gold bar	Move
Hold **Ctrl** while dragging bar	Copy
Drag a yellow arcs	Rotate

You can move the widget about the entity to edit different parts of it:

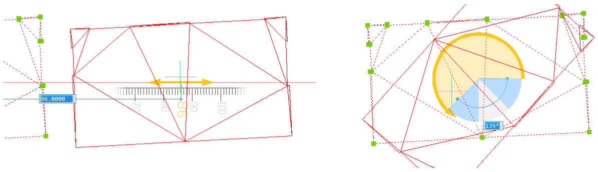
- **Relocate** the widget by dragging the white ball (found nearest to the origin); entity snaps are respected
- **Twist** the widget by dragging one of the three while balls adjacent to each arrowhead

Right-click the widget for the following options:



- **Move** moves the widget freely to another location
- **Align with WCS** or **UCS** or **Entity/Face** aligns the arms of the widget with the x,y,z-axes of the world, or a user-defined coordinate system, or to the nearest face of an entity.
- **Reorient manipulator** acts like reorienting the UCS icon.
- **Hide** hides the widget.
- **Manipulator Settings** opens the Settings dialog box at the Manipulator section.
- **Classic color** changes the arm colors so that the x arm is red, the y one is green, and z is blue.
- **Arrowhead Acts As** switches between 3D scaling (indicated by the double blue arrowhead) and 3D mirroring (indicated by the mirrored blue triangles).

As you drag the manipulator's arm, a ruler shows you the distance interactively. Or, if you are changing an angle, then a protractor illustrates increments of 45 degrees.

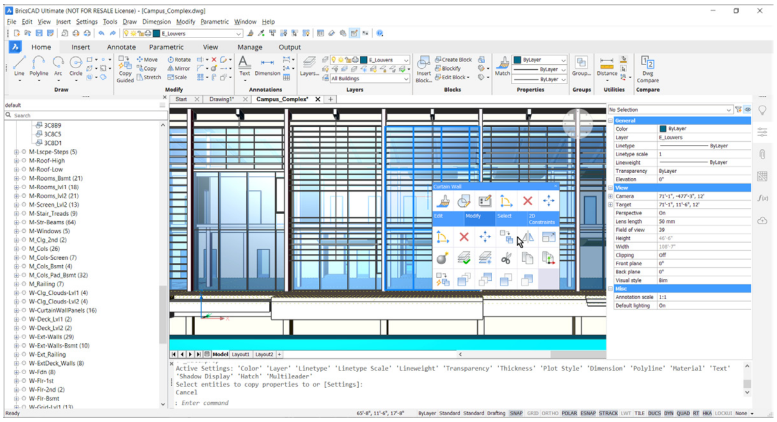


Left: *Ruler showing the drag distance;* ***right:*** *protractor showing the angle rotation*

CLEAN SCREEN

Shortcut `Ctrl+0`

Sometimes the user interface can get pretty cluttered, and so it's a good idea to clear most of it away quickly.

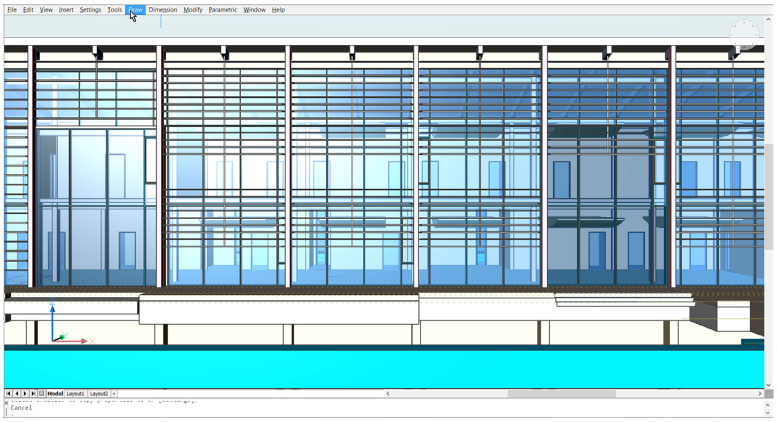

BricsCAD Ultimate (NOT FOR RESALE License) - [Campus_Complex.dwg]
File Edit View Insert Settings Tools Draw Dimension Modify Parametric Window Help
Home Insert Annotate Parametric View Manage Output
Draw Modify Annotations Layers Blocks Properties Groups Utilities Compare
Start Drawing1* Campus_Complex*
Model Layout1 Layout2

No Selection	
General	
Color	ByLayer
Layer	E_Louvers
Linetype	ByLayer
Linetype scale	1
Lineweight	ByLayer
Transparency	ByLayer
Elevation	0"
View	
Camera	71'-1", -477'-3", 12'
Target	71'-1", 11'-6", 12'
Perspective	On
Lens length	50 mm
Field of view	39
Height	46'-6"
Width	108'-7"
Clipping	Off
Front plane	0"
Back plane	0"
Visual style	Bim
Misc	
Annotation scale	1:1
Default lighting	On

```
Active Settings: 'Color' 'Layer' 'Linetype' 'Linetype Scale' 'Lineweight' 'Transparency' 'Thickness' 'Plot Style' 'Dimension' 'Polyline' 'Material' 'Text'
'Shadow Display' 'Hatch' 'Multileader'
Select entities to copy properties to or [Settings]:
Cancel
: Enter command
```

UI cluttered

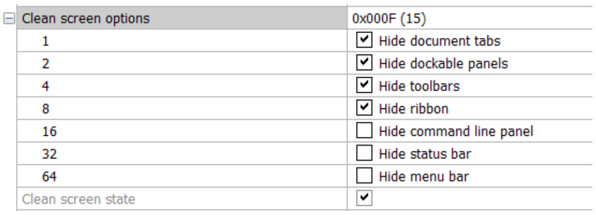
Press **Ctrl+0** to toggle clean screen on and off.


File Edit View Insert Settings Tools Draw Dimension Modify Parametric Window Help
Model Layout1 Layout2

```
Cancel
```

UI minimized

The Settings dialog box lets you choose which user interface elements you still want to see during a clean screen session: enter “cleanscreen” in the search field.



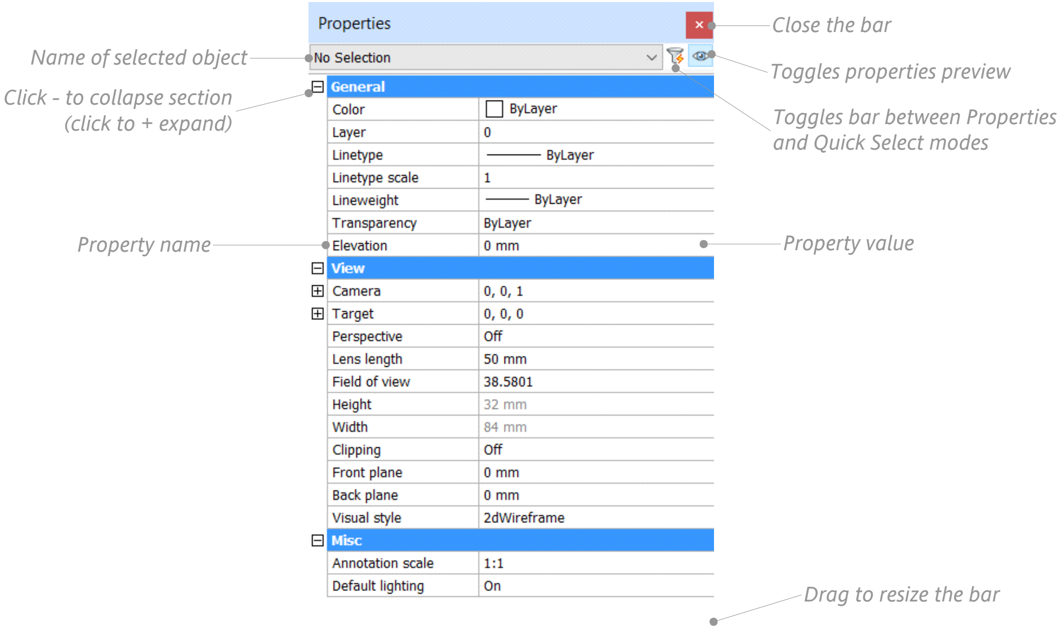
⊟ Clean screen options	0x000F (15)
1	☑ Hide document tabs
2	☑ Hide dockable panels
4	☑ Hide toolbars
8	☑ Hide ribbon
16	☐ Hide command line panel
32	☐ Hide status bar
64	☐ Hide menu bar
Clean screen state	☑

Options for clean screening

As an alternative to clean screening your display, consider adding a second monitor and placing panels, toolbars, and other UI item there.

PANELS

Panels are so important to BricsCAD that I am going to give you this quick tour of their user interface features. (These were known as “bars” or “palettes” in earlier releases.) Once you learn the UI elements for the Properties panel, you can use them with many other panels, as well. (See Chapter 6 for how to use the Properties panel.)



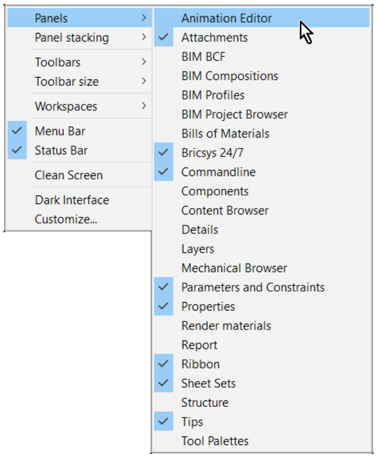
Properties ×

No Selection

⊟ General	
Color	☐ ByLayer
Layer	0
Linetype	——— ByLayer
Linetype scale	1
Lineweight	——— ByLayer
Transparency	ByLayer
Elevation	0 mm
⊟ View	
⊞ Camera	0, 0, 1
⊞ Target	0, 0, 0
Perspective	Off
Lens length	50 mm
Field of view	38.5801
Height	32 mm
Width	84 mm
Clipping	Off
Front plane	0 mm
Back plane	0 mm
Visual style	2dWireframe
⊟ Misc	
Annotation scale	1:1
Default lighting	On

Properties	Drag the title bar to relocate the panel on the screen
×	Click the x button to close the panel
Circle ⌄	Click the droplist to access other entities (if any)
⊞	Click the + node to expand a section
⊟	Click the - node to collapse a section
🟥 Red ⌄	Click on a property value to modify it
⤡	Drag the edge of the panel to change its size, smaller or larger

The easiest way to access panels is to right-click the ribbon or a toolbar, and then choose **Panels**:



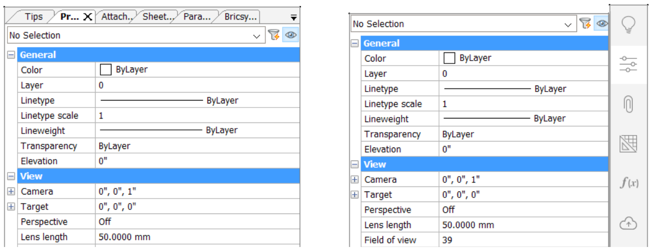
Turning panels on and off

From the list, select the panels you wish to open or close. The blue checkmark indicates the associated panel is open. Several panels and bars can be opened and closed using shortcut keystrokes. Here are some of them:

Action	Command	Shortcut Keystroke
Open or close Properties panel	Properties	Ctrl+1
Open the Drawing Explorer	Explorer	Ctrl+2
Open or close Command bar	CommandLine	Ctrl+9

It can become cumbersome when many panels are open, and so BricsCAD offers two ways to stack them:

- **Tabs** — overlapping panels are indicated by tabs along the top of the panels
- (NEW IN V20) **Icons** — overlapping panels are indicated by an icon along the edge of the panels



No Selection

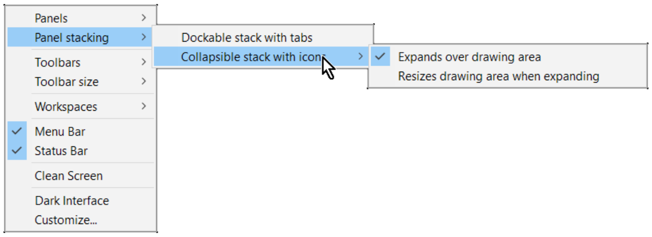
General	
Color	□ ByLayer
Layer	0
Linetype	ByLayer
Linetype scale	1
Lineweight	ByLayer
Transparency	ByLayer
Elevation	0"
View	
Camera	0", 0", 1"
Target	0", 0", 0"
Perspective	Off
Lens length	50.0000 mm

No Selection

General	
Color	□ ByLayer
Layer	0
Linetype	ByLayer
Linetype scale	1
Lineweight	ByLayer
Transparency	ByLayer
Elevation	0"
View	
Camera	0", 0", 1"
Target	0", 0", 0"
Perspective	Off
Lens length	50.0000 mm
Field of view	39

Left: *Tabbed panels;* ***right:*** *iconized panels*

To choose the method, right-click the ribbon or a toolbar, and then choose **Panel stacking.**

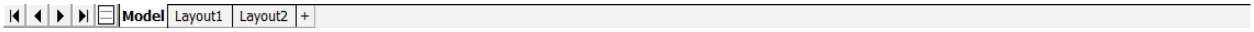


Choose the method of panel stacking

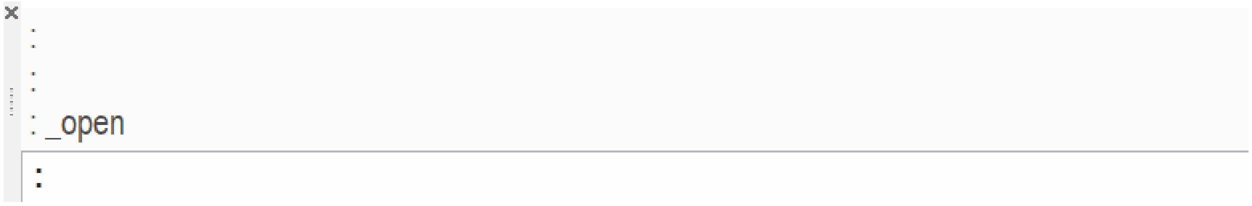
Below the Drawing Area

At the bottom of BricsCAD are several more lines of information:

Layouts bar controls the display of layouts; also holds the horizontal scroll bar:

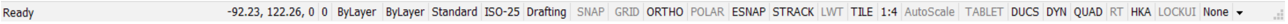


Command bar is where you enter commands and read prompts:



```
:
:
:
:
: _open
:
```

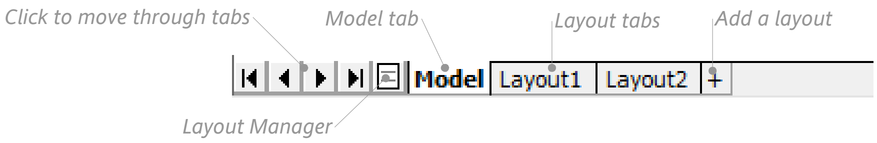
Status bar shows help text, reports the status of the drawing, and changes settings:



Let's examine the function of each area.

LAYOUT TABS

Under the drawing area are three tabs labeled **Model**, **Layout1**, and **Layout2**. You may have seen similar tabs in other Windows or Linux software, such as the sheet tabs in LibreOffice Calc or Excel.



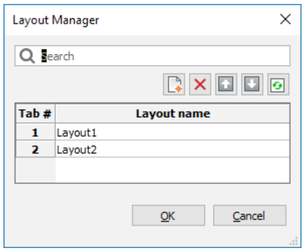
Elements of the layout tabs

In BricsCAD, these tabs switch the drawing view among *layouts*. Layouts allow you to define how the drawing will look as it is plotted. They let you position its location on the paper, adding drawing borders, specific views, and/or specifying shades of gray instead of colors.

Click the **+** plus sign to add more tabs.

The arrow buttons let you see more of them there are too many tabs to fit along the bottom of BricsCAD.

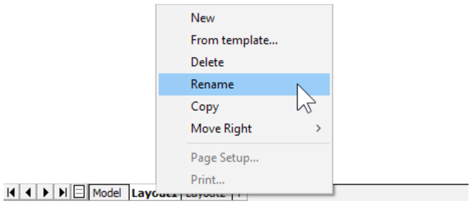
Click the **Layout Manager** button to display the Layout Manager dialog box, which lets you create, name, and organize more than layout at a time.



Tab #	Layout name
1	Layout1
2	Layout2

Layout Manager dialog box

To access a shortcut menu of layout options, right-click any layout tab. The options let you create more layouts,rename them, or specify settings specific to a variety of plotter models.



Right-clicking a layout tab

Layouts are discussed in greater detail later in this book.

So, there are drawing tabs to switch between drawings, and then layout tabs to switch between layouts inside each drawing.

Scroll Bar

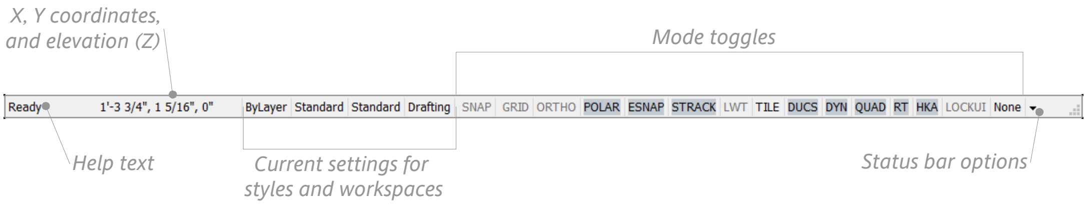
The scroll bar is described earlier in this chapter.

Command Bar

The Command bar is described in the previous chapter.

STATUS BAR

Below the command prompt area is the status bar, which reports the status of the drawing. From left to right, the status bar displays the following info by default:



"By default," because you can select which items are displayed on the status bar: click the Status Bar Options button to see the full list.

Help text — displays a line of helpful text when the cursor is paused on a toolbar button or a menu item.

X,Y Coordinates — 2D coordinates of the cursor's current location in the drawing.

Elevation (Z) — the current setting of the elevation.

Linestyle — reports the current line type (ByLayer)

Style — reports the current text style (Standard)

Dimension style — reports the current dimension style (Standard)

Workspace — reports the current workspace (Drafting).

Drawing Settings — reports the status of drawing settings; each word is an on/off button called a "toggle:"

- **Gray** text — the toggle is turned off, as for **SNAP** in the figure above.
- **Black** text — the toggle is turned on, as for **POLAR**.

Click a button to turn the mode on and off.

SNAP — toggles the distance snap

GRID — toggles the display of grid lines

ORTHO — toggles orthographic drawing mode to draw at right angles

POLAR — toggles snapping at common polar angles, such as 15 degrees

ESNAP — toggles all entity snaps (a.k.a. object snaps)

STRACK — toggles snap tracking

LWT — toggles lineweight display

TILE / P:Layout — switches between paper and model modes

DUCS — toggles dynamic UCS mode for drawing on the faces of 3D entities

DYN — toggles dynamic dimensions mode

QUAD — toggles the cursor between crosshair and quad modes

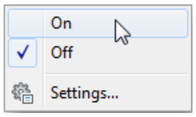
RT — toggles the display of rollover tooltips

HKA — toggles the hotkey assistant widget

LOCKUI — prevents the toolbars from being moved accidently

None — selects a GIS coordinate system

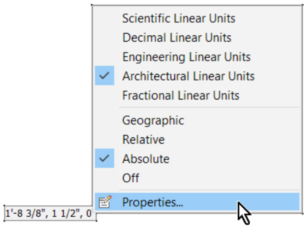
When you right-click any of status bar button, BricsCAD displays a shortcut menu. Many have the same set of options: On, Off, and Settings,



Typical context menu for status bar buttons

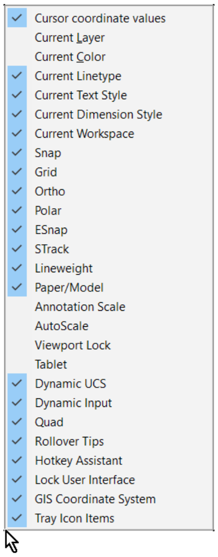
Settings is a shortcut to the Settings dialog box that regulates the mode. For example, to change the settings for the grid, right-click the GRID button and then select Settings; BricsCAD displays the Snap/Grid section of the Settings dialog box.

When you right-click other status bar buttons, they may present longer lists of options. For example, right-click the x,y,z coordinates to change how they are displayed:



Changing the coordinate display style

Clicking the tiny black triangle ▾ near the right end of the status bar displays a shortcut menu that lists all possible toggles.



This lefts you turn on and off toggles that you want (or don't want) to see.

You have learned all about BricsCAD's user interface, from the title bar at the top to the status bar along the bottom, and nearly everything in between

Next, you begin to start working with BricsCAD, beginning with starting a new drawing.

CHAPTER 3

Setting Up A New Drawing

Before creating new drawings with BricsCAD, you must prepare them. Here you learn how to set up new drawings with the following tutorials:

Step 1:	**Start** new, blank drawings
Step 2:	**Name** the drawing file
Step 3:	Specify the **units** of measurement
Step 4:	Set the **snap** and **grid** spacings
Step 5:	Indicate the drawing **limits**
Step 6:	Create **layers**
Step 7:	**Save** the drawing

In this chapter, you learn how to set up new drawings, save your work to disk, and then exit BricsCAD.

IN THIS CHAPTER

- Preparing a drawing for first-time use with a wizard
- Specifying units, angle formats, and drawing settings
- Understanding how layers organize drawings
- Saving drawings
- Recognizing the importance of automatic backups

KEY TERMS IN THIS CHAPTER

Default — value of a setting defined by the program and unchanged by the user

File name — uniquely identifies drawing files and other documents

Grid — displays a grid of lines or array of dots as visual guides

Layers — organize drawings by segregating common elements

Limit — specifies the nominal limits of drawings, and constrains the range of grid marks

Scratch — brand-new drawing with no preset parameters

Snap — constrains cursor movement to discrete distances

Template — a drawing file read by BricsCAD that sets default parameters for new drawings

Unit — specifies units of measurement, such as metric, architectural, and engineering

Wizard — series of dialog box that step users through a procedure

Working set — set of one of more drawings that were previously open in BricsCAD or were saved for future use

Workspace — predefined user interface specific to tasks, such as 2D drafting or 3D modeling

Zoom — enlarges and reduces the visual size of drawings

USEFUL ABBREVIATIONS

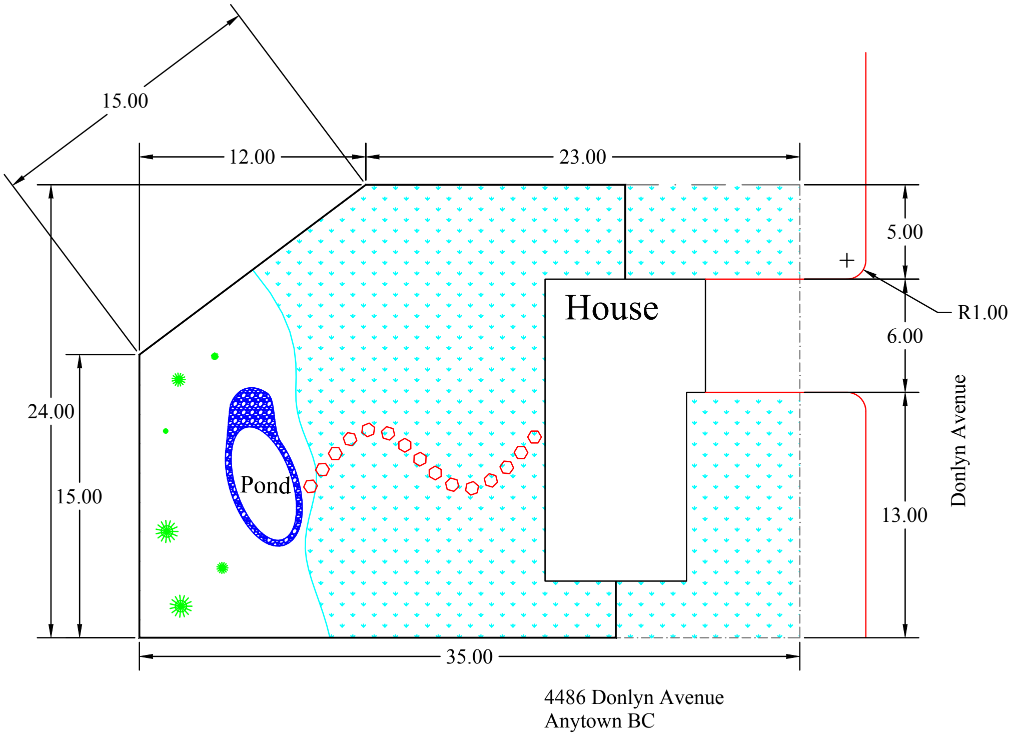
' or ft	Feet
" or in	Inches (12 inches per foot)
ANSI	American National Standards Institute
BAK	Backups of BricsCAD drawing files
DWT	Portion of file names that identify them as template files
DWG	Portion of file names that identify them as BricsCAD drawings
mm	Millimeters (1 000mm per meter)
m	Meters

NEW COMMANDS

Command	Alias	Menu Bar	Ribbon Tab
Layer	la	Settings | Layers	Home | Layers | Layers
Limits	...	Settings | Drawing Limits	...
NewWiz	ddnew	File | New Wizard	...
Settings	options	Settings | Settings	Settings | Settings | Settings
Save	Ctrl+S	FIle | Save	File | Save
SaveAs	...	File | Save As	...

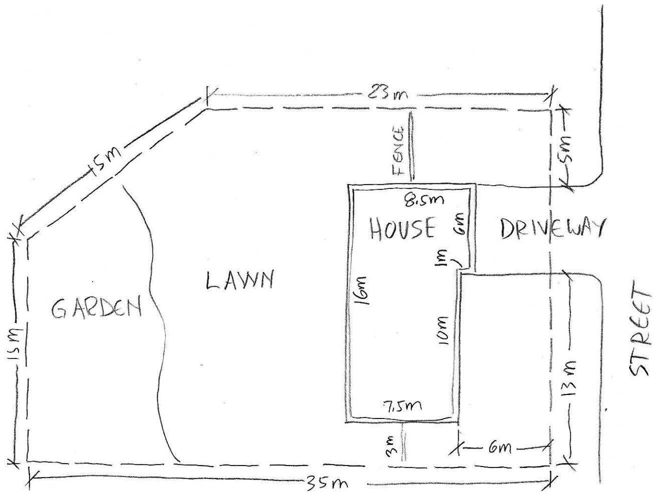
Before You Begin

To learn how to draw with BricsCAD, you will work on a landscape plan, creating and modifying the drawing of the yard around a house. By the end of chapter 7, the drawing should look like this:



Completed 2D tutorial drawing

Before beginning this series of tutorials, however, you may want to measure your own yard to locate its major features, such as the house, driveway, and garden areas. If you'd rather not measure the yard or you don't have access to one, then follow along with the following sketch, which is used for the lessons.



Sketch of proposed drawing

The tutorial in this book will be done in metric units (meters), but for North American readers I'll include imperial units (feet and inches).

The complete set of tutorial files for this book can be downloaded from https://my.pcloud.com/publink/show?code=XZn98mkZ7T7f1pynxJYb7vDyVl6DwyqK1Ysk. If you draw with imperial units, then use *Imp-Yard.dwg* found inside the ZIP file. The ZIP file also holds the drawing as it appears at the end of each chapter.

STARTING A NEW DRAWING

```
Command    NewWiz
Menu       File | New Wizard
Ribbon     Home | File | New Wizard
```

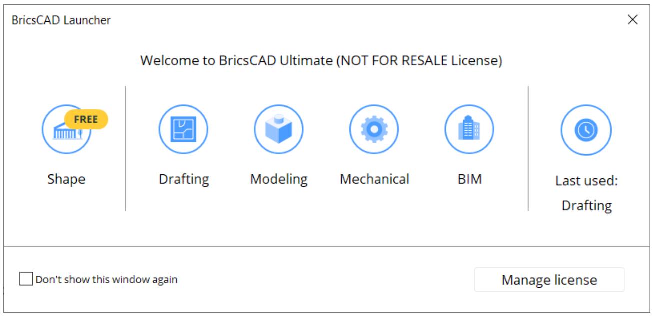
When you start a new drawing in BricsCAD, you can start in several different ways:

- From *scratch* — with a blank drawing
- Or, with a *template file* — that presets certain aspects of new drawings
- Or, enlist the services of a software wizard — this how you start the first tutorial

In this chapter, you create the new drawing with the assistance of a *wizard*. The wizard is meant to help you get through the initial few steps in preparing new drawings, which can be confusing to new users faced with a blank screen.

The "wizard" consists of a series of dialog boxes that take you through the steps needed to set up new drawings. They prompt you to select the drawing units (such as Imperial or metric), turn on the grid, and so on.

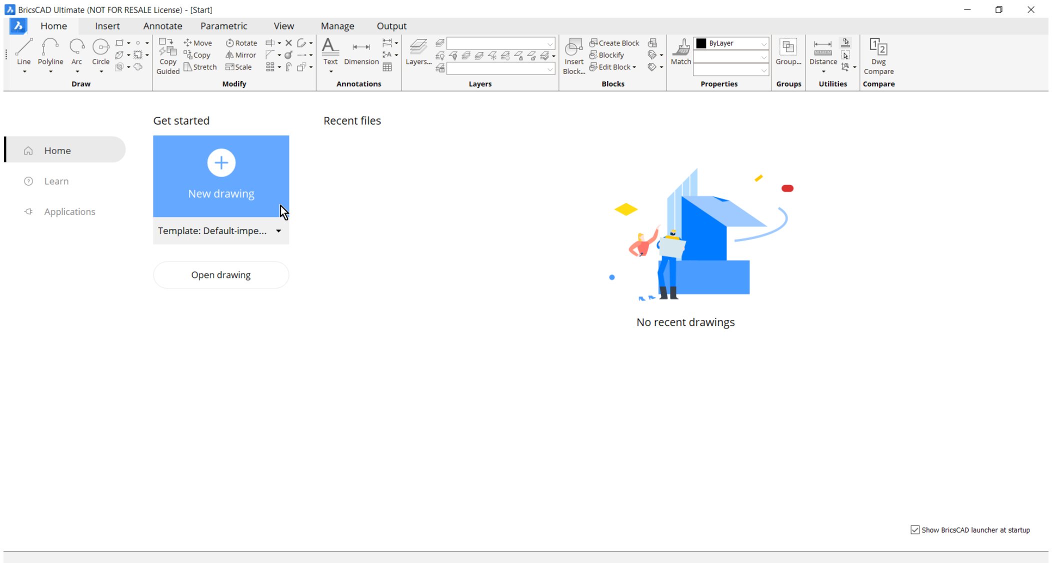
1. If BricsCAD is not running, start it now by double-clicking its icon on the desktop
2. Notice the BricsCAD Launcher dialog box. Choose "Drafting" as the workspace, as it is the one best suited for 2D drawings.



Welcome to BricsCAD Ultimate (NOT FOR RESALE License)

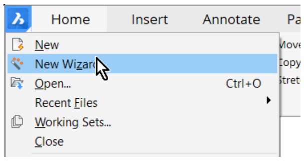
Choosing the Drafting workspace

3. When the Start page appears, click **New Drawing**.



4. Start the wizard by taking one of the following steps:

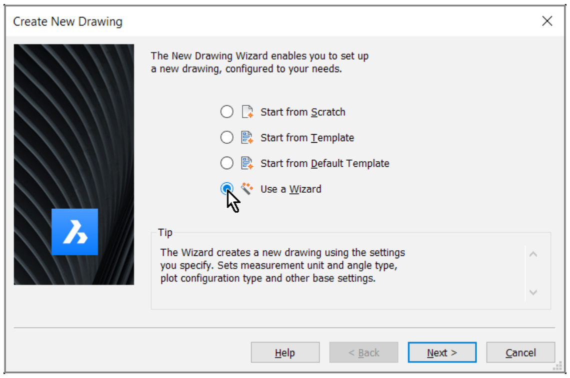
- In the ribbon, click the blue **B** and then choose **New Wizard**



- Or, from the menu bar, choose **File**, and then **New Wizard**
- Or, at the command line, enter the **NewWiz** command

```
: newwiz
```

Notice the Create New Drawing dialog box:



Create New Drawing

The New Drawing Wizard enables you to set up a new drawing, configured to your needs.

○ Start from Scratch
○ Start from Template
○ Start from Default Template
● Use a Wizard

Tip

The Wizard creates a new drawing using the settings you specify. Sets measurement unit and angle type, plot configuration type and other base settings.

The dialog box supplies you with options for starting new drawings:

Start from Scratch — starts empty new drawings, using either imperial or metric units

Start from Template — starts new drawings based on a template file that you select from a long list with names like *default-mm.dwt* and *Mechanical-imperial.dwt*

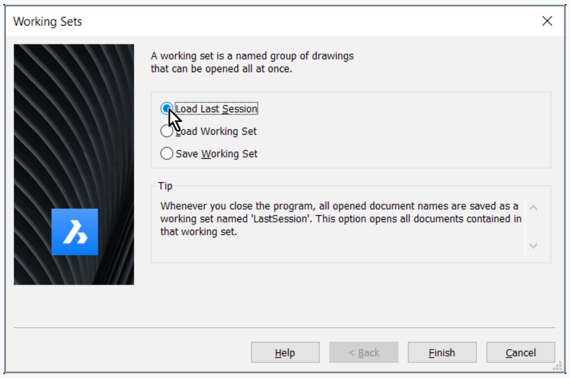
Start from Default Template — starts new drawings with the template file specified by the **BaseFile** system variable; the current default is *default-mm.dwt*

Use a Wizard — creates new drawings based on settings specified in a series of dialog boxes that follow

HOW TO START NEW DRAWINGS

OPENING WORKING SETS

A “working set” is a list of previously-opened drawings, or a saved sets of drawing file names. To save and open sets of drawings, use the **Workset** command.



Working Sets

A working set is a named group of drawings that can be opened all at once.

● Load Last Session
○ Load Working Set
○ Save Working Set

Tip

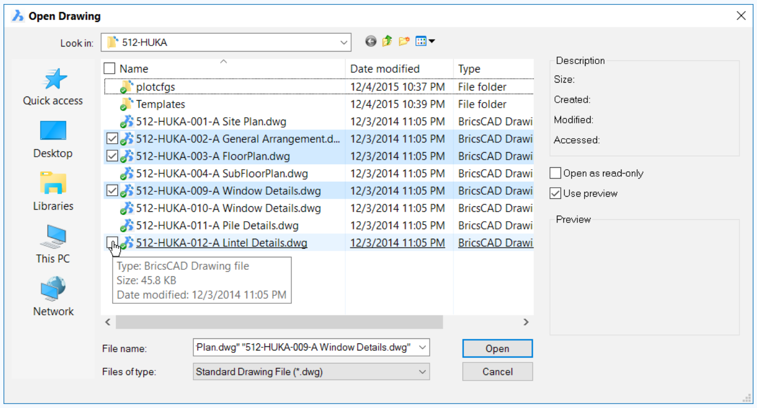
Whenever you close the program, all opened document names are saved as a working set named 'LastSession'. This option opens all documents contained in that working set.

OPENING MORE THAN ONE DRAWING

To select one than one drawing at once, hold down the following keys as you pick them in the Open dialog box:

Shift — select a contiguous range of .dwg files

Ctrl (Cmd on Macs) — to select non-contiguous files, as shown below



Open Drawing

Look in: 512-HUKA

□	Name	Date modified	Type
	plotcfgs	12/4/2015 10:37 PM	File folder
	Templates	12/4/2015 10:39 PM	File folder
	512-HUKA-001-A Site Plan.dwg	12/3/2014 11:05 PM	BricsCAD Drawi
☑	512-HUKA-002-A General Arrangement.d...	12/3/2014 11:05 PM	BricsCAD Drawi
☑	512-HUKA-003-A FloorPlan.dwg	12/3/2014 11:05 PM	BricsCAD Drawi
	512-HUKA-004-A SubFloorPlan.dwg	12/3/2014 11:05 PM	BricsCAD Drawi
☑	512-HUKA-009-A Window Details.dwg	12/3/2014 11:05 PM	BricsCAD Drawi
	512-HUKA-010-A Window Details.dwg	12/3/2014 11:05 PM	BricsCAD Drawi
	512-HUKA-011-A Pile Details.dwg	12/3/2014 11:05 PM	BricsCAD Drawi
□	512-HUKA-012-A Lintel Details.dwg	12/3/2014 11:05 PM	BricsCAD Drawi

Type: BricsCAD Drawing file
Size: 45.8 KB
Date modified: 12/3/2014 11:05 PM

File name: Plan.dwg" "512-HUKA-009-A Window Details.dwg"

Files of type: Standard Drawing File (*.dwg)

Description
Size:
Created:
Modified:
Accessed:

□ Open as read-only
☑ Use preview

STARTING FROM SCRATCH

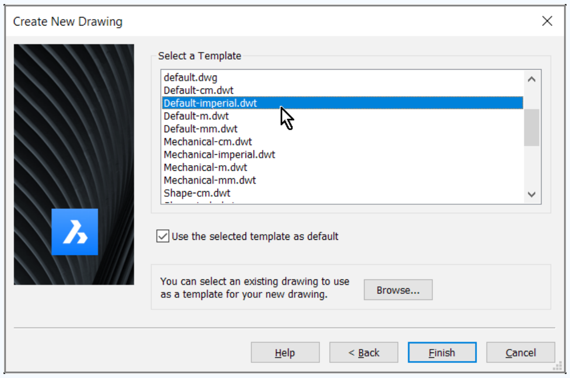
New “empty” drawings employ default settings for all variables, which are read from the following DWT template files:

default-mm.dwt — for drawings with metric units or unitless (millimeters)

default-imperial.dwt — for drawings with Imperial or British units (inches)

STARTING FROM TEMPLATES

New drawings are begun with a DWT template file that you select from the following list:



Create New Drawing

Select a Template

- default.dwg
- Default-cm.dwt
- Default-imperial.dwt
- Default-m.dwt
- Default-mm.dwt
- Mechanical-cm.dwt
- Mechanical-imperial.dwt
- Mechanical-m.dwt
- Mechanical-mm.dwt
- Shape-cm.dwt

☑ Use the selected template as default

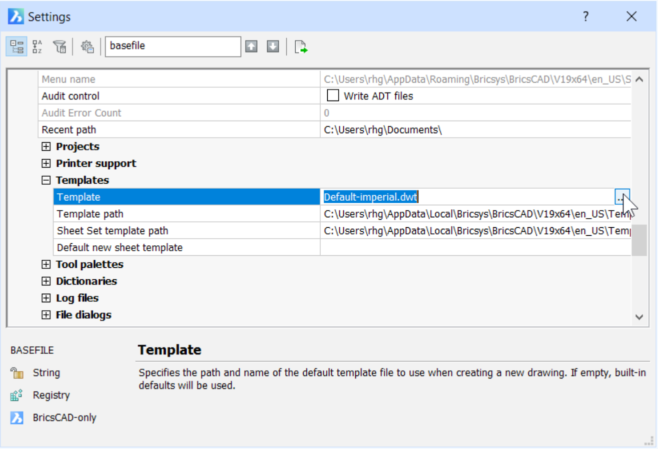
You can select an existing drawing to use as a template for your new drawing.

You can create your own templates: edit the current drawing, and then use the **SaveAs** command. Choose "Drawing Template (*.dwt)" from the **Save As Type** droplist.

STARTING FROM THE DEFAULT TEMPLATE

Start new drawings with the template file specified by the **BaseFile** system variable, usually *default-mm.dwt* if none other is specified. This option is useful when you usually want to start with the same template, such as one specified by your firm or a client.

To change the value of BaseFile, enter the **Settings** command, search for "basefile," and then click the ... button in the Template field. From the Choose a File dialog box, select the .dwt file you wish to employ as the new default template.



Settings

basefile

Menu name	C:\Users\rhg\AppData\Roaming\Bricsys\BricsCAD\V19x64\en_US\S
Audit control	□ Write ADT files
Audit Error Count	0
Recent path	C:\Users\rhg\Documents\
⊞ **Projects**	
⊞ **Printer support**	
⊟ **Templates**	
Template	Default-imperial.dwt
Template path	C:\Users\rhg\AppData\Local\Bricsys\BricsCAD\V19x64\en_US\Tem
Sheet Set template path	C:\Users\rhg\AppData\Local\Bricsys\BricsCAD\V19x64\en_US\Tem
Default new sheet template	
⊞ **Tool palettes**	
⊞ **Dictionaries**	
⊞ **Log files**	
⊞ **File dialogs**	

BASEFILE

String

Registry

BricsCAD-only

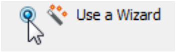
Template

Specifies the path and name of the default template file to use when creating a new drawing. If empty, built-in defaults will be used.

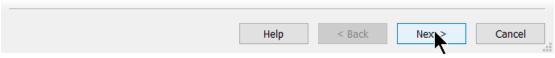
USING A WIZARD

Creates new drawings based on settings specified in a series of dialog boxes. See the tutorial in this chapter.

5. Select the **Use a Wizard** button by clicking the circle (called a "radio button") next to its label.


● Use a Wizard

6. Click **Next**.



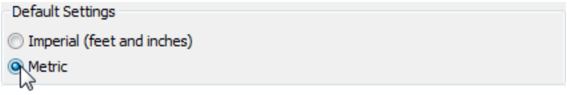
The wizard now takes you through a series of dialog boxes that ask you to specify the units, format of angles, and other drawing settings. I'll explain to you what each one means.

Default Settings

First, choose the units. Well, the wizard calls them the "default settings." This is because so many settings in drawings are affected by the type of units. Your choices are *metric* (decimal) or *Imperial* (feet and inches). Here is how to decide which units to choose:

- If you measured the yard in meters and centimeters, then click the radio button next to **Metric**
- If you measured the yard in feet and inches, then click the radio button next to **Imperial (feet and inches)**

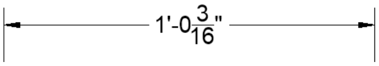
7. If you are following along with my tutorial drawing, then choose **Metric**, as shown below.


Default Settings
○ Imperial (feet and inches)
● Metric

8. Click **Next**.

SELECTING THE LINEAR UNITS

Drafting uses *linear measurements* for things like lengths and areas, as illustrated by the dimension below. The other kind of measurement is *angular measurements* for angles.



Linear measurements can be displayed in several different formats. Examine closely at the sample measurements shown in the table below to understand how BricsCAD displays each. Note that a *dash* (-) separates feet from inches, while a *space* separates inches from fractional inches.

Unit Name	Measures In	Example
Architectural	Feet, inches, fractional inches	4'-6 1/16"
Engineering	Feet, inches, decimal inches	4'-6.0625"
Fractional	Inches, fractional inches	54 1/6"
Decimal (default)	Units and decimal units	2128.4449
Scientific	Units, decimal units, and exponents	2.1284E+03

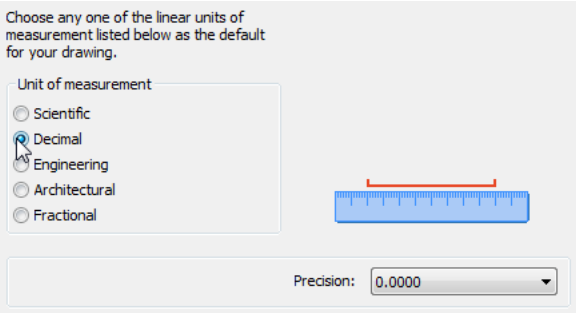
The default is called “Decimal,” which is often used for metric measurements, such as millimeters and meters. BricsCAD displays metric measurement as *unitless* decimals, such as 2128.4449 2128.4449; the “mm” for millimeter or “m” for meters is not displayed.

(How do you know if 2128.4449 displayed by BricsCAD is millimeters or meters? It’s whatever you start drawing with. If the first thing you draw is 2128.4449 meters long, then everything else is in meters, too.)

Once you select a style of unit, BricsCAD *displays* all measurements in that format. But don’t worry! You can always switch the measurement format to something else with the **Units** command.

(The reason you can switch between units is because internally BricsCAD actually ignores the type of units. It keeps track of measurements with unitless real numbers accurate to 14 decimal places, and then converts them on the fly to the units we want, for the sake of us humans.)

9. For this drawing tutorial, measurements were made in centimeters. Under **Unit of Measurement**, ensure “Decimal” is selected.

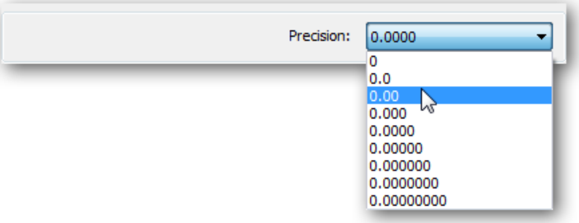


Choose any one of the linear units of measurement listed below as the default for your drawing.

Unit of measurement
- ○ Scientific
- ● Decimal
- ○ Engineering
- ○ Architectural
- ○ Fractional

Precision: 0.0000

10. This drawing is measured in meters, to the nearest centimeter (0.01m). Centimeters are represented by two decimal places. To set the number of decimal places, follow these steps: Next to **Precision**, click the down arrow and then select **0.00**.



Precision: 0.0000
- 0
- 0.0
- 0.00
- 0.000
- 0.0000
- 0.00000
- 0.000000
- 0.0000000
- 0.00000000

(For imperial drawings, measurements made in feet to the nearest 1/4-inch — accurate enough for this project. Select **Architectural** units, and then from the Precision droplist choose **0'-0 1/4"**.)

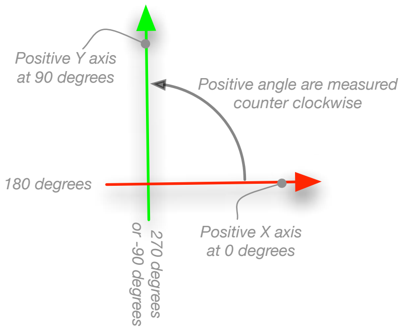
11. Click **Next**. As we see next, BricsCAD displays the set of options for measuring and displaying angles.

You are free to enter distances more accurately than a centimeter, such as 0.4441 because BricsCAD remembers distances to full accuracy. When BricsCAD displays coordinates, however, it will round them off to the nearest 0.01 units (“0.44”), because we told it to in step 9.

SELECTING THE ANGLE STYLE

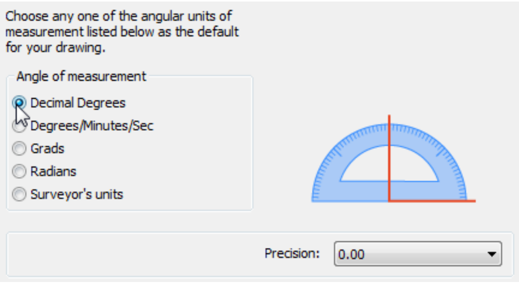
Before we choose the angle measurement system, I must mention some facts about angles:

- In CAD, angles are usually measured starting at the x-axis, which is at 0 degrees; see figure below
- Angles are usually measured counter-clockwise (backward from how a clock moves) from that positive x axis
- 90 degrees is at the positive y axis
- BricsCAD shows the x axis in red, and the y axis in green



The elements of axes and angles

The Angle dialog box is for choosing the formats of angular units that BricsCAD should display.



Choose any one of the angular units of measurement listed below as the default for your drawing.

Angle of measurement

● Decimal Degrees
○ Degrees/Minutes/Sec
○ Grads
○ Radians
○ Surveyor's units

Precision: 0.00

The options available are as listed below:

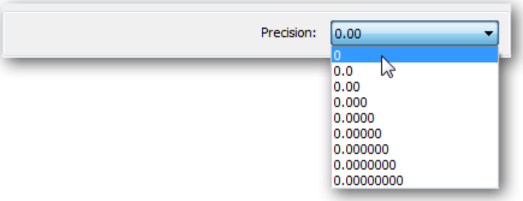
Angle Name	Measures In	Example
Decimal degrees	Degrees and decimals	12.3456
Degrees/Minutes/Seconds (d ' ")	Degrees, minutes, seconds, and decimal seconds	12d34'56"
Grads (g)	Grads and decimals	123.45g
Radians (r)	Radians and decimals	1.23r
Surveyor's Units (N d ' " E)	North/South degrees towards East/West	N12d34'56"E

There are 400 grads (used in Germany) and 2*pi radians (or 6.2832 radians) in a 360-degree circle.

If we were *real* land surveyors, we would use the Surveyor format. But we're not, and so we will stick with familiar decimal degrees. (Again, you can choose the change the angular format at anytime with the **Units** command without affecting the drawing.)

12. Ensure that the default, **Decimal Degrees**, is selected for the angle of measurement.

13. Change the default **Precision** to **0** degrees, as we are measuring angles to the nearest degree.



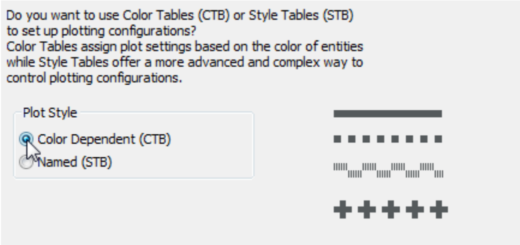
Precision: 0.00

14. Click **Next**.

CHOOSING THE PLOT STYLE

When it comes time to printing drawings, they can be prepared by BricsCAD for the printer using one of two methods. One method uses *color tables* (a.k.a. "color dependent" styles or CTB), while the other method uses *style tables* (a.k.a. "named" styles or STB). They are, unhappily, mutually exclusive, meaning that you can choose only one, ahead of time.

15. Right now, all you need to know is that neither color tables nor style tables really matter when plotting regular 2D drawings on printers. So, ensure that the **Color Dependent (CTB)** option is selected, as it is the easier of the two to work with.



Do you want to use Color Tables (CTB) or Style Tables (STB)
to set up plotting configurations?
Color Tables assign plot settings based on the color of entities
while Style Tables offer a more advanced and complex way to
control plotting configurations.

Plot Style

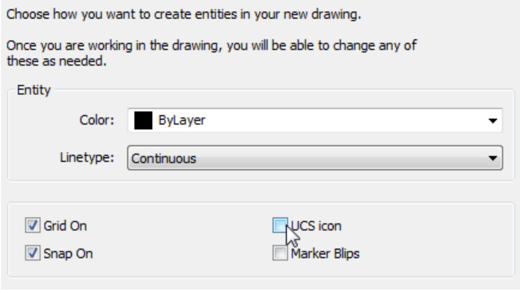
● Color Dependent (CTB)

○ Named (STB)

16. Click **Next**.

SETTING ENTITY PROPERTIES

You now come to the final dialog box of the wizard. Here you set the *defaults* of some entity properties (a.k.a. "objects"). Default means the "initial setting." Initially, all entities are drawn with a color named "ByLayer" and a linetype of the same name.



Choose how you want to create entities in your new drawing.

Once you are working in the drawing, you will be able to change any of
these as needed.

Entity

Color: ■ ByLayer

Linetype: Continuous

☑ Grid On

☑ Snap On

□ UCS icon

□ Marker Blips

ByLayer means that layers determine the color and linetype of entities. If the layer is set to red, for instance, then all entities on that layer appear red; change the layer's color to green and the entities follow suit.

In this dialog box, you could overrule the default properties. But you won't, because changing colors and linetypes is best left for later, and is done with the Layers dialog box.

17. The only change to make here is to turn off the UCS icon, because it is unhelpful (by being visually intrusive) to the tutorial drawing.
18. See figure above for how settings should look. Click **Finish** to close the dialog box.

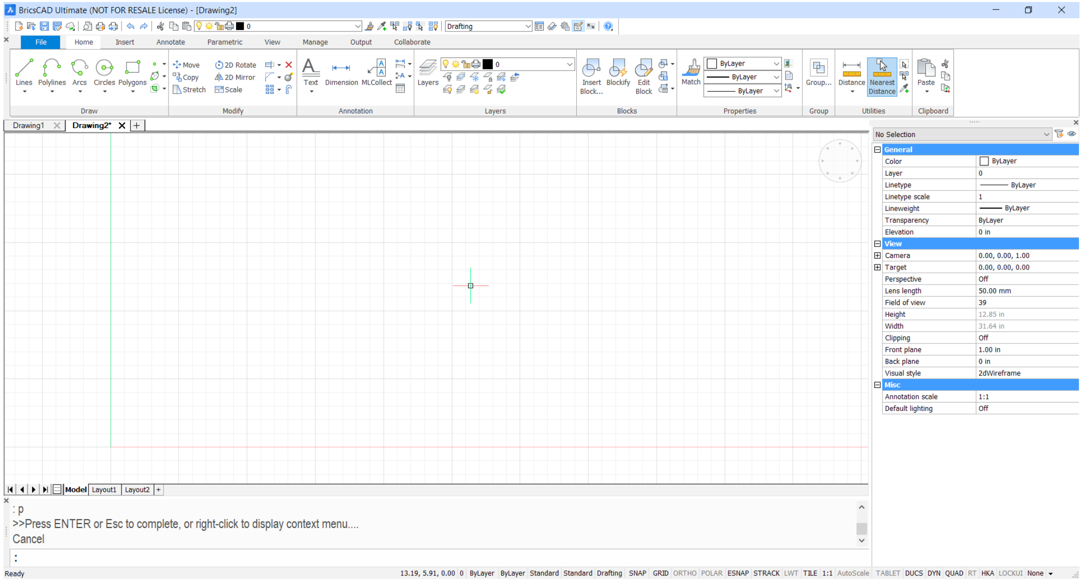
FINISHING THE WIZARD

When the wizard is done, the final dialog box closes, and you get to see BricsCAD's drawing area. It looks a little bit different from before: there is an array of lines in the drawing called "grid" lines. One grid line that goes horizontally through the origin (at 0,0) is red and a vertical one is green — the same colors as the x an y crosshair cursor lines:

Red grid line — x axis (horizontal)

Green grid line — y axis (vertical)

The UCS icon is gone, because you turned it off and because it is unnecessary for this tutorial. When you move the mouse, the coordinate display on the status bar changes in increments of 0.01 — this is due to setting the precision of units to 0.01.



No Selection

General	
Color	□ ByLayer
Layer	0
Linetype	ByLayer
Linetype scale	1
Lineweight	ByLayer
Transparency	ByLayer
Elevation	0 in
View	
Camera	0.00, 0.00, 1.00
Target	0.00, 0.00, 0.00
Perspective	Off
Lens length	50.00 mm
Field of view	39
Height	12.85 in
Width	31.64 in
Clipping	Off
Front plane	1.00 in
Back plane	0 in
Visual style	2dWireframe
Misc	
Annotation scale	1:1
Default lighting	Off

```
: p
>>Press ENTER or Esc to complete, or right-click to display context menu....
Cancel
:
```

```
Ready    13.19, 5.91, 0.00  0  ByLayer  ByLayer  Standard  Standard  Drafting  SNAP  GRID  ORTHO  POLAR  ESNAP  STRACK  LWT  TILE  1:1  AutoScale  TABLET  DUCS  DYN  QUAD  RT  HKA  LOCKUI  None
```

The wizard does not adjust all settings, and you have a few more to change, as described next.

Additional Important Settings

The wizard, unfortunately, does not set up everything you need in new drawings, and so in the rest of this lesson you use commands to set other aspects, such as the area of the drawing's limits, the spacing of the snap and grid, and the names of layers. Along the way, I'll explain what each aspect means.

SETTING DRAWING LIMITS

```
Command    Limits
Menu Bar   Settings | Drawing Limits
Toolbar    Settings
```

There is no limit to the size of drawings you can create with BricsCAD. If you wanted, you could draw the entire solar system — full size! Indeed, an early AutoCAD sample drawing showed this is possible: from the orbit of Pluto we can zoom all the way down to the individual letters on a plaque mounted on a lunar lander in a crater on the Earth's moon.

When it comes time to printing drawings, however, the size of a drawing is severely constrained by the size of the paper the printer handles, often just 210 x 297mm or 8-1/2 x 11" — a far sight smaller than the solar system! One way to control the size visually is by setting the limits. This is done with the **Limits** command.

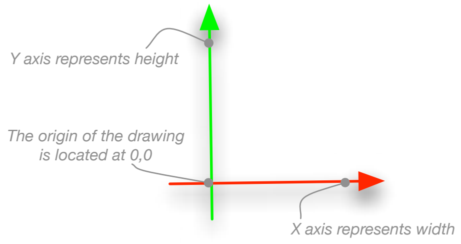
Limits show you the nominal area of a drawing, "nominal" because the drawing can be much larger, of course. Limits perform two more useful functions:

- Constrain the extents of the grid lines
- Determine the area shown by the Zoom command's All option

More on these later. In the meantime, here is how you determine what the limits should be, and then set them:

1. Examine the size of the yard in the sketch. After leaving a bit of "breathing room" around the plan, the drawing will need about 40m (130 feet) of width and 30m (100 feet) of height.
    - **Width** is represented by the **x** coordinate, which sometimes is usually in red
    - **Height** by the **y** coordinate, shown in green, usually

    The *origin* of the drawing is where the lines of the x and y axes meet, at 0,0.



2. At the ' : ' prompt, enter the **Limits** command, as follows:

```
: limits (Press Enter to execute the command)
```

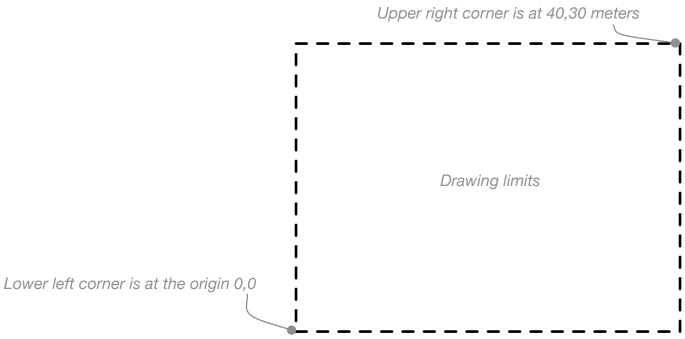
3. Notice that BricsCAD reports that the limits are turned off. Turn them on:

```
Limits are off: Set lower left corner or [turn limits ON] <0,0>: on
```

4. Restart the **Limits** command by pressing the spacebar. Pressing the spacebar is a handy shortcut to repeating the last-used command:

```
: (press the spacebar)
: LIMITS
```

Notice that BricsCAD reports the name of the command it is repeating, so you are sure it is the correct one.


Upper right corner is at 40,30 meters
Drawing limits
Lower left corner is at the origin 0,0

5. Now specify the x,y coordinates of the lower left corner. The default value of 0,0 is good for this drawing, so just press **Enter** to leave the value as is:

```
Limits are on: Set lower left corner or [turn limits ON] <0,0>: (Press Enter to accept
the default value of 0,0)
```

6. When BricsCAD asks for coordinates of the upper right corner, enter 40 and 30 (meters), as follows:

```
Upper right corner <12,9>: 40,30
```

Warning! If you work with imperial units, it is important that you include the **apostrophe** (') when you enter measurements as feet, like 130'.

When you leave out the apostrophe of 130, BricsCAD assumes you mean 130 inches, which is actually 10'-10.

7. I find it handy to restrict the lines of the grid to the extents of the limits. To do this, I change the value of the **GridDisplay** system variable to 2 like this:

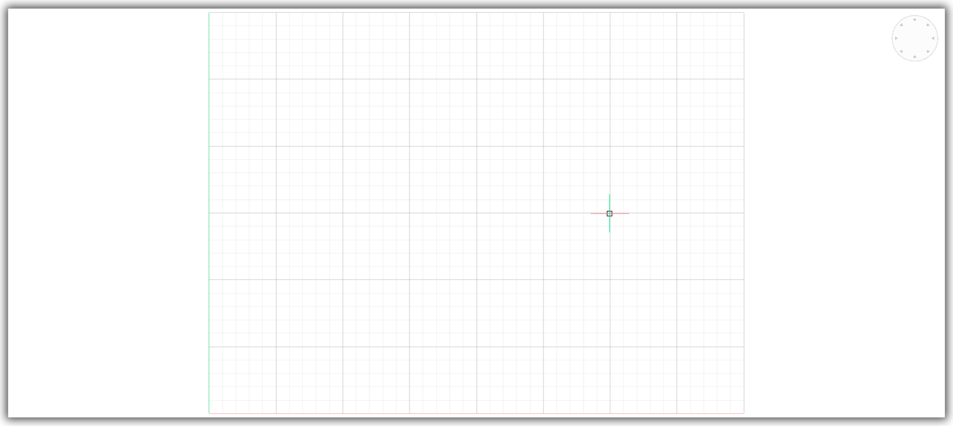
```
: griddisplay
New current value for GRIDDISPLAY (0 to 15) <3>: 2
```

The 2 tells BricsCAD to crop the grid at the limits. (A 0 tells BricsCAD to turn off the grid, while other values tell BricsCAD to treat the grid in other ways.)

8. Finally, use the **Zoom All** command to see all of the drawing. Enter the **Zoom** command, and then enter 'a' for the **All** option:

```
: zoom
Zoom [zoom In/zoom Out/All/Center/Dynamic/Extents/Left/Previous/Right/Scale (nx/nxp)/Window/
OBject] <Scale (nX/nXP)>:a
```

The Zoom command lets you see the "big picture," as well as zooming in for a detailed look.



The grid is now constrained to the area specified by the limits. Grids lines are shown faintly so that they do not obscure drawing elements.

When you began this tutorial, BricsCAD displayed an area of 12" by 9"; now BricsCAD displays an area of 40m by 30m (130' by 100'). When you move the cursor to the upper right corner of the drawing area, the status bar should report values near 40, 30 (meters).

ACCESSING AND CHANGING SETTINGS

```
Command    Settings
Aliases    SE, options, ddrmodes, rm, dsettings, ddsetvar
Menu Bar   Settings | Settings
Ribbon     Settings | Settings | Settings
Toolbar    Settings
```

Some of the values you entered at the command prompt can be changed through the Settings dialog box. This dialog box reports on the values of all *variables.* "Variables" hold settings that control and report on nearly all aspects of BricsCAD — and there are over 1,000 of them! I list them all in Appendix B. Variables are also known as "system variables," or "sysvars" for short.

For example, **GridDisplay** is the name of a variable that tells BricsCAD how to display the grid. This variable has four settings, each represented by a number (technically, an integer). Here is what the value of the GridDisplay variable controls:

GridDisplay	Meaning
0	Turns off the display of the grid
1 (default)	Display grid beyond the area of the limits
2 (default)	Display adaptive grid
4	Allow sub-divisons below grid spacing
8	Follow dynamic UCS

The numbers 1 through 8 can be added together to turn on two or more options. The default value is 3, which means that (1) the grid is displayed beyond the limits *and* (2) the adaptive grid is displayed. In the tutorial, you changed the value to 2 so that the grid no longer displayed beyond the limits.

(*Adaptive grid* means that as you zoom out, the grid shows fewer lines so that the drawing area is not overwhelmed with too many closely-spaced lines. *Allow subdivisions* means that additional lines are displayed between the specified grid spacing. *Follow DUCS* means that the grid rotates to match the x,y-plane specified by the current UCS.)

Here is another example: When you used the Limits command, for example, BricsCAD stored the values you entered in two system variables, LimMin and LimMax:

LimMin — stores the x, y coordinates of the lower left corner.

LimMax — stores coordinates of the upper right corner.

These two system variables can be accessed directly at the command bar by entering their names, just like they were commands:

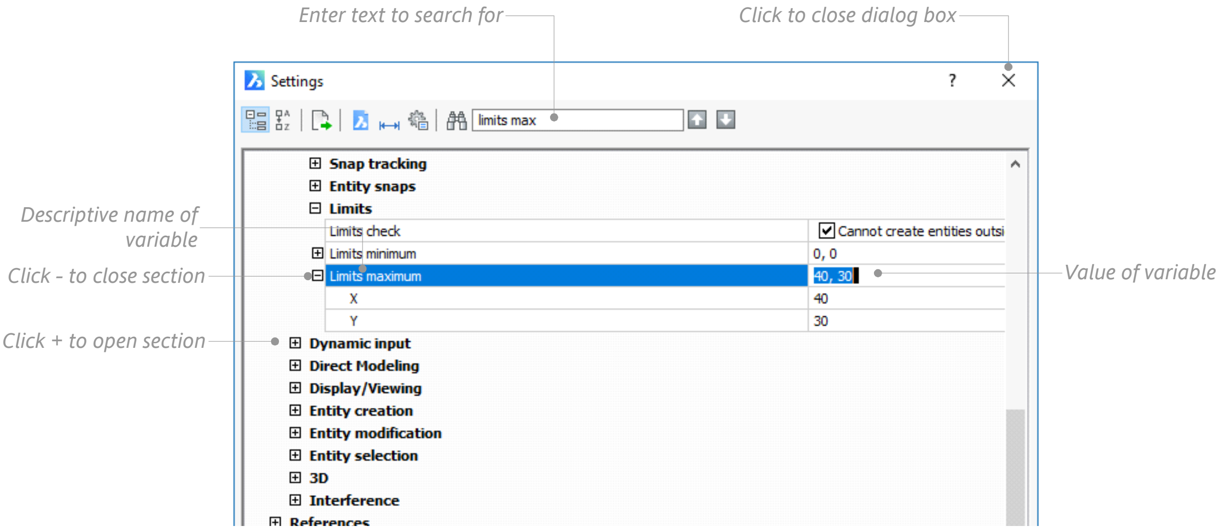
```
: limmin
New value for LIMMIN <0,0>: (Press Enter to keep values as they are)

: limmax
New value for LIMMAX <40,30>: (Press Enter)
```

A problem lies in remembering the names of hundreds of system variables. And so the other method is to use the Settings dialog box, as follows:

1. Enter the **Settings** command:

   ```
   : settings
   ```

   Notice the Settings dialog box.



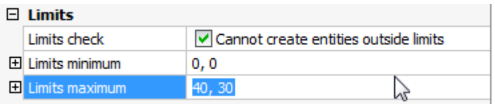
Limits check	☑ Cannot create entities outsi
Limits minimum	0, 0
Limits maximum	40, 30
X	40
Y	30

2. This dialog box contains hundreds of settings, and so the best way to access one of them is to use the **Find** function, as follows:

   a. In the **Find** field, enter "limits"

   b. Press **Enter**.

The Settings dialog box goes to the first setting with “limits” in its name. Click the ⬇ **Find Next** arrow until you arrive at the **Limits Minimum** and **Limits Maximum** settings:



⊟ **Limits**

Limits check	☑ Cannot create entities outside limits
⊞ Limits minimum	0, 0
⊞ Limits maximum	40, 30

3. Make changes to settings. When you change a setting, its text become **boldfaced**.
4. When you are done, just click the red **X**. There is no “Close” or “OK” button to dismiss this dialog box.

When the **Find** field turns orange, it means that there is no setting by that name. This occurs when you misspell the name, or when a variable of that name does not exist in BricsCAD.

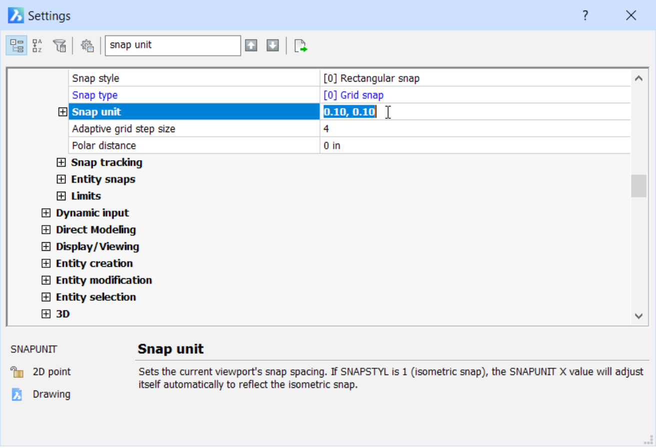
CHANGING THE SNAP AND GRID SPACING

One great advantage to drawing with CAD is that software permits you to draw *very* accurately. BricsCAD has several functions that help you draw with perfect accuracy. One you’ve already met, the grid. It helps you visualize distances.

Another helpful function is “snap mode.” Snap can be thought of as setting the *mouse resolution*. It determines the distance the cursor moves in the drawing. For instance, when the snap distance is set to 0.5, the cursor moves at precise 0.5-unit distances during drawing and editing commands. (You learn more about other accuracy aids in later lessons.)

Recall that during one of the wizard’s steps, you turned on snap mode. The default (initial) spacing of 0.5 meters is, however, too wide. Follow these steps to change the snap and grid spacing to 0.1 meters, which is more suitable for our drawing:

1. (If you are not in the Settings dialog box, then enter the **Settings** command.)
   In the Find field, enter “snap mode” and then press **Enter**. See figure below.
2. Notice that the listing jumps to the **Snap Unit** field.
3. Change the X and Y values of **Snap Unit** to **0.1, 0.1** to represent 1 decimeter in each direction of x and y (or **1",1"** in imperial units). This is the distance that the cursor moves. Leave other snap settings at their default values, such as Snap Angle = 0.



Settings

snap unit

Snap style	[0] Rectangular snap
Snap type	[0] Grid snap
⊞ **Snap unit**	0.10, 0.10
Adaptive grid step size	4
Polar distance	0 in

⊞ **Snap tracking**
⊞ **Entity snaps**
⊞ **Limits**
⊞ **Dynamic input**
⊞ **Direct Modeling**
⊞ **Display/Viewing**
⊞ **Entity creation**
⊞ **Entity modification**
⊞ **Entity selection**
⊞ **3D**

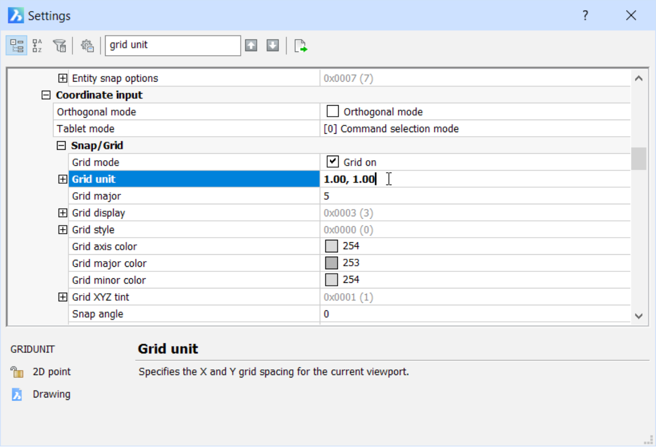
SNAPUNIT
2D point
Drawing

Snap unit

Sets the current viewport's snap spacing. If SNAPSTYL is 1 (isometric snap), the SNAPUNIT X value will adjust itself automatically to reflect the isometric snap.

4. Earlier, the wizard had turned on the grid as a visual guide. The grid is meant to guide you; the default spacing of 10 units is perhaps to far apart. (BricsCAD will in fact not display the grid when its lines or dots are too closely spaced.)

   Change the spacing to 1 meter by changing the value of **Grid Unit** to **1,1**.



Settings

grid unit

⊞ Entity snap options	0x0007 (7)
⊟ **Coordinate input**	
Orthogonal mode	□ Orthogonal mode
Tablet mode	[0] Command selection mode
⊟ **Snap/Grid**	
Grid mode	☑ Grid on
⊞ **Grid unit**	**1.00, 1.00**
Grid major	5
⊞ Grid display	0x0003 (3)
⊞ Grid style	0x0000 (0)
Grid axis color	254
Grid major color	253
Grid minor color	254
⊞ Grid XYZ tint	0x0001 (1)
Snap angle	0

GRIDUNIT

2D point

Drawing

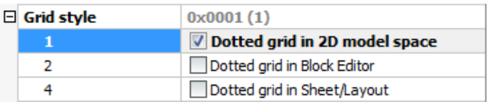
Grid unit

Specifies the X and Y grid spacing for the current viewport.

5. Finally, change the *type* of grid from lines to dots. Grid lines are useful in 3D modeling, but tend to cover up lines in 2D drafting, and so I prefer the more discrete dots. Here is how to do this:

   a. Click the ⊞ node next to **Grid Style** .

   b. Turn on the **Dotted Grid is 2D Model Space** option.

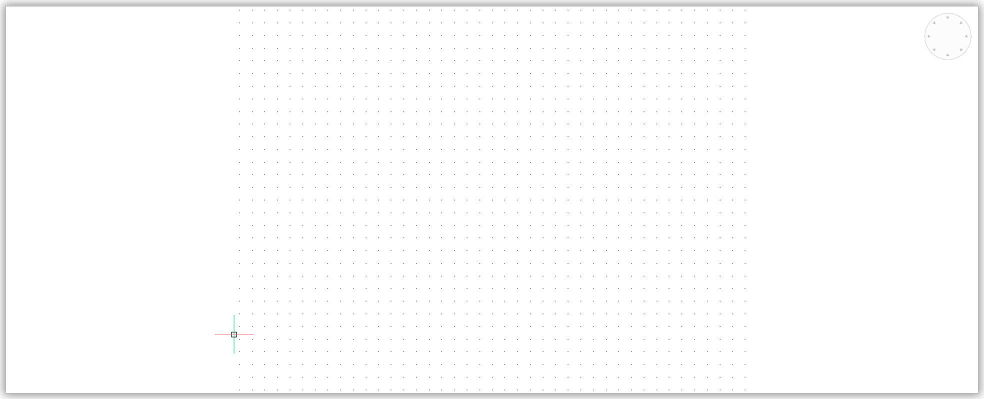


⊟ **Grid style**	0x0001 (1)
1	☑ **Dotted grid in 2D model space**
2	□ Dotted grid in Block Editor
4	□ Dotted grid in Sheet/Layout

6. To close the dialog box, click the **X** at the top right end.

A grid distance of 0 has a special meaning in BricsCAD. It means that the grid spacing matches the snap spacing, 1" in our case.

Notice that the drawing area is covered by a grid of dots.



Sometimes the snap function can get in the way of your drafting. You can turn the snap on and off at any time by pressing function key **F9**.

Similarly, the grid can be toggled with **F7**.

For many options in BricsCAD's commands, you need only type the first letter of the option. For example, type "a" as the abbreviation for the **All** option. When two options begin with the same letter, you need to type the first two characters of the option.

When entering text and numbers in a dialog box, pressing the **Tab** key is a quicker way to get to the next field. *Fields* are buttons, text entry boxes, list boxes, and other dialog box elements that you can change. To return to previous fields, press **Shift+Tab** (hold down the **Shift** key, and then press **Tab**).

When you change values in the Settings dialog box, they are shown in boldface text. And, the changes take effect immediately.

Creating Layers

Commands	Layer, layerpanelon
Alias	LA
Menu Bar	Settings | Layers
	Tools | Drawing Explorer | Layers
Ribbon	Home | Layers | Layers
Toolbar	Settings | Layers

If you ever worked with *overlay drafting*, then you may be familiar with the concept of layers. In overlay drafting, drafters draw the base plan on one clear sheet of Mylar, a common form of drawing medium made of plastic. The electrical plan would be drawn on another sheet, the structural on a third. Since the Mylar is transparent, drafters overlay two or three of the drawings to create a single blueprint — depending on how much detail was needed to be shown.

In CAD, *layers* operate in a similar manner. We draw different parts of drawings on different layers. The base is on one layer, electrical on another, and so on. Then we turn layers off and on to display the drawing with different amounts of detail. For example, the electrical contractor is usually interested in seeing the base plan layer with only the electrical layer.

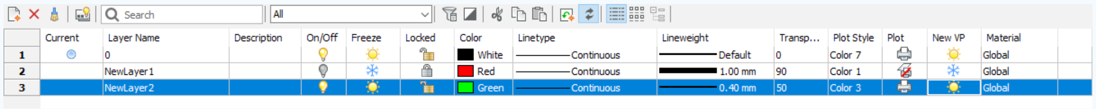
While it is possible to create hundreds or even thousands of layers in drawings, it is more common to work with a few dozen layers; in this book, we work with a half-dozen layers to segregate the text from the lines showing the yard, the roads, and so on.

A further advantage to CAD layers is that they apply global properties to entities. For example, when a layer is green, then all entities assigned to that layer are colored green. Changed the color of the layer, such as to brown, and the color of all entities assigned to that layer change to brown instantly.

Assigning entities to layers is this simple: (a) set the name of a layer as *current*, and then (b) start drawing! The most common way to set a name as current through the Layer droplist. BricsCAD has commands that let you move entities to other layers, should that be necessary.

QUICK SUMMARY OF LAYERS

BricsCAD provides great control over layers through the Drawing Explorer. It lists the names of all layers and their properties. The names of properties are indicated by the names on the header bar.



Search | All

	Current	Layer Name	Description	On/Off	Freeze	Locked	Color	Linetype	Lineweight	Transp...	Plot Style	Plot	New VP	Material
1	●	0		💡	☀	🔓	■ White	Continuous	Default	0	Color 7	🖨	☀	Global
2		NewLayer1		💡	❄	🔒	🟥 Red	Continuous	1.00 mm	90	Color 1	🖨	❄	Global
3		NewLayer2		💡	☀	🔓	🟩 Green	Continuous	0.40 mm	50	Color 3	🖨	☀	Global

(To sort layers alphabetically, click a header such as Name or Linetype. Click a second time to sort in reverse order, Z to A.)

Current — ● indicates the current layer.

Layer Name — specifies the name up to 255 characters long. You can use numbers, letters, and the following punctuation marks: dollar sign ($), hyphen (-), underline (_), and spaces. Two layers cannot have the same name in the same drawing. There is no limit to the number of layers in a drawing.

Description — describes the purpose or content of the layer; optional.

On/Off *(obsolete)* — when on, displays entities; when off, entities cannot be seen nor are they plotted.

Freeze — freezes entities so that they cannot be seen, edited, or plotted. Also, frozen layers are not included when BricsCAD performs drawing regenerations and hidden-line operations. When thawed, the layers behave normally. (*To thaw layers* means to turn off their frozen status.) It is better to freeze layers than turn them off.

Locked — locks layers, which means entities are seen but cannot be edited. Unlock layers to make entities available for editing.

Color — specifies the color of entities on the layer. Entities drawn on the layer are displayed in this color, but the entity color can be overridden with the Color command. The default color is 7, which is displayed as white or black, depending on the background color.

Linetype — specifies the line type by which entities are displayed. The default linetype is Continuous (solid line). Before any other linetype can be used, its definition must be loaded into the drawing; can be overridden by the Linetype command.

Lineweight — specifies the line widths for entities on the layer. The default is 0.00mm; the maximum is 2.11mm (about 0.08", or 6 points wide); can be overridden by the Lineweight command.

Plot Style — specifies the plot style with which entities plotted. Named plot styles defines colors, widths, and percentages of black during plotting. This option is not available for drawings created with color-dependent plot styles.

Plot — specifies whether the layer is plotted; when off, the layer is not plotted.

If some of the drawing does not plot, check the setting of the **Plot** property.

Transparency — specifies the translucency of the layer; the default of 0 means all entities are displayed opaque.

New VP — specifies whether the status of new viewports, visible or frozen.

Material — specifies the material used during renderings.

...continued

NAMING LAYERS

Layers are specified by name. BricsCAD lets you give layers names of up to 255 characters long. Start setting up new layers with the **Layer** command; it displays a dialog box. (Layers can also be displayed in a panel, which stays open always, with the **LayersPanelOpen** command.)

1. Enter the **Layer** command:

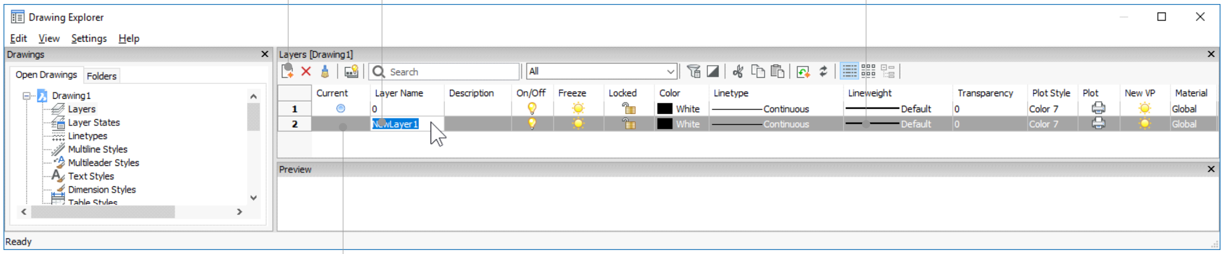
   : **layer**

   Notice the Drawing Explorer dialog box. This dialog box lets you control nearly all aspects of layers in drawings.

Step 1: Click to create a new layer

Step 2: Rename the generic name

Step 3: Set the properties



Drawing Explorer

Layers [Drawing1]

	Current	Layer Name	Description	On/Off	Freeze	Locked	Color	Linetype	Lineweight	Transparency	Plot Style	Plot	New VP	Material
1	●	0					White	Continuous	Default	0	Color 7			Global
2		NewLayer1					White	Continuous	Default	0	Color 7			Global

Step 4: Set the layer as current

The drawing already has one layer: 0. Every new BricsCAD drawing has that layer "0", which you can never erase. Layer 0 has special properties that affect the creation of blocks, as discussed later.

2. Follow these steps to create and name a layer:

   a. Click the **New Layer** button. Notice that BricsCAD creates a new layer called "NewLayer1." See figure below.

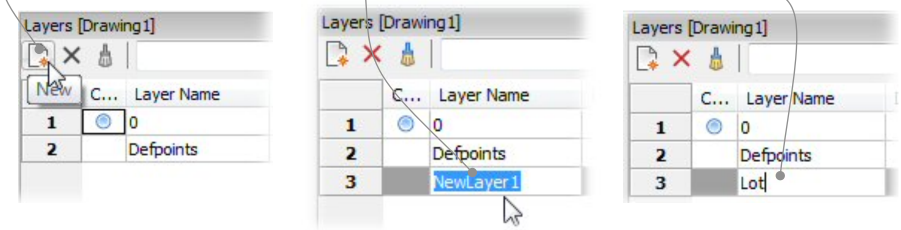
   b. Change the name by clicking "NewLayer1."

   c. Type **Lot**, and then press **Enter**.

Step 1: Click New button

Step 2: Click new layer name

Step 3: Enter name and press Enter



Layers [Drawing1]

New	C...	Layer Name
1	●	0
2		Defpoints

Layers [Drawing1]

	C...	Layer Name
1	●	0
2		Defpoints
3		NewLayer1

Layers [Drawing1]

	C...	Layer Name
1	●	0
2		Defpoints
3		Lot

...*continued*

LAYOUT MODE PROPERTIES

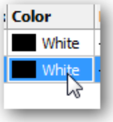
When drawings are in layout mode, this dialog box displays more columns. (VP is short for "viewport.")

VP Freeze — freezes the layers in new viewports created in paper space

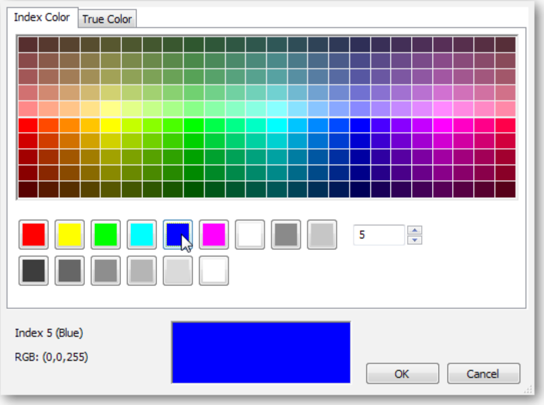
VP Color, VP Linetype, VP Lineweight, VP Transparency, and **VP Plot Style** — specifies the color, linetype, lineweight, transparency, and plot style for the layer in paper space viewports

3. Assigning a color to each layer makes it easier to determine which lines belong to which layers. Change the color of the Lot layer to blue, as follows:

   a. Click the black square under the **Color** column across from the layer name **Lot**.



   b. Notice that the Select Color dialog box appears. It displays 255 colors: which one to choose?



Index 5 (Blue)
RGB: (0,0,255)

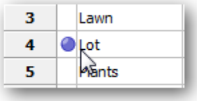
   Above the row of black and gray squares is the row of BricsCAD's "standard colors." Red, yellow, green, and so on are the ones used most-commonly. Select the dark blue square. Notice that the number "5" appears in the **Color** text box, because this is color number 5, according the color-numbering system used by many CAD systems.

   c. Click **OK** to exit the Select Color dialog box. Notice that the color of the square across from layer Lot changes to blue.

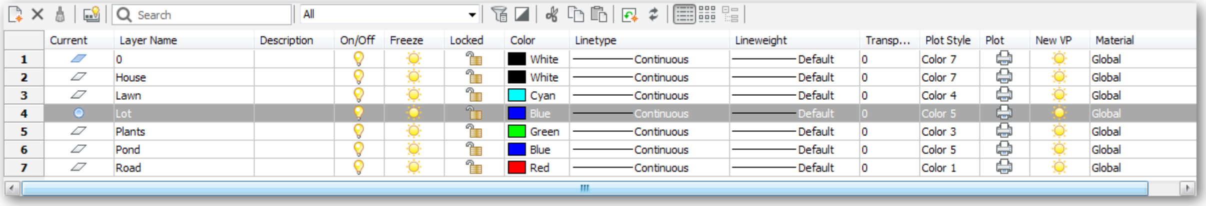
4. Add the remaining layer names and colors, using the table below as a guide. If you make a spelling mistake, just click the layer name and type the correction.

New Layer	Layer Color	Color Number
Lot	*Blue*	Color #5
House	White (black)	...
Road	Red	Color #1
Lawn	Cyan (light blue)	Color #4
Plants	Green	Color #3
Pond	Blue	Color #5

5. When you finish assigning colors to layer names, pick the **Lot** layer name, then click in the blank square between the 2 and Lot. Notice the **blue** ball that appears; it indicates the *current* layer.



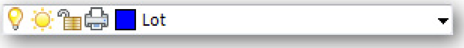
For now, drafting takes place on the **Lot** layer — until you select another layer name as the current layer.



	Current	Layer Name	Description	On/Off	Freeze	Locked	Color	Linetype	Lineweight	Transp...	Plot Style	Plot	New VP	Material
1	▰	0		💡	☀	🔓	⬛ White	Continuous	Default	0	Color 7	🖶	☀	Global
2	▱	House		💡	☀	🔓	⬛ White	Continuous	Default	0	Color 7	🖶	☀	Global
3	▱	Lawn		💡	☀	🔓	🟦 Cyan	Continuous	Default	0	Color 4	🖶	☀	Global
4	○	Lot		💡	☀	🔓	🟦 Blue	Continuous	Default	0	Color 5	🖶	☀	Global
5	▱	Plants		💡	☀	🔓	🟩 Green	Continuous	Default	0	Color 3	🖶	☀	Global
6	▱	Pond		💡	☀	🔓	🟦 Blue	Continuous	Default	0	Color 5	🖶	☀	Global
7	▱	Road		💡	☀	🔓	🟥 Red	Continuous	Default	0	Color 1	🖶	☀	Global

6. To exit the Drawing Explorer dialog box, click **X** at the top right of the dialog box.

Notice that the name of the layer in the Entity Properties toolbar or ribbon changes from **0** to **Lot**. The color has changed from black to blue.



When you click the layers droplist, you see all of the newly-created layer names and their properties. Later in this book, you make use of this droplist to control layer properties.

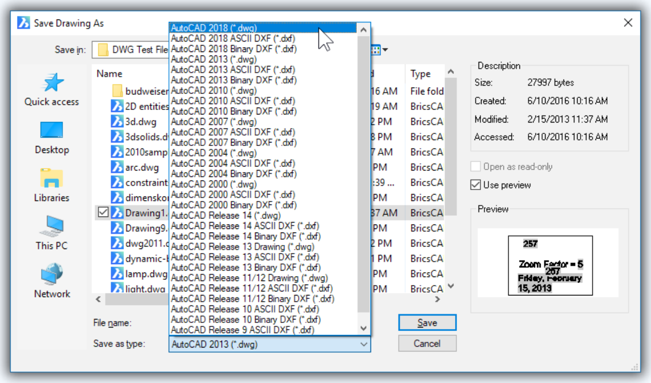
There can be some confusion over the color "white" — or is it black? BricsCAD switches white and black depending on the background color of the drawing area. When the background is black, BricsCAD displays white lines; when white, BricsCAD displays black lines. So, white can be black — at least in the world of CAD.

To set the background color in BricsCAD, from the **Settings** menu, select **Settings**. In the **Find** field, enter "background color," and then press **Enter**. Select the color.

QUICK SUMMARY OF THE SAVEAS COMMAND

The Save As dialog box lets you save drawings in several dialects of DWG. Newer versions of BricsCAD can always read drawings created by older versions of BricsCAD, Ares, DraftSight, AutoCAD, AutoCAD LT, IntelliCAD, and other DWG-based drafting programs.

Older versions of BricsCAD cannot, however, read drawing files created by newer versions, and this is true of all other CAD programs, including AutoCAD. For example, BricsCAD V8 cannot read drawings created in BricsCAD V20. When working with users of older versions of CAD programs, you must make BricsCAD explicitly save drawings in earlier formats compatible with older software, as described by this table:


Save Drawing As
Quick access
Desktop
Libraries
This PC
Network
Name
budweiser
2D entities
3d.dwg
3dsolids.d
2010samp
arc.dwg
constraint
dimensko
Drawing1.
Drawing9.
dwg2011.c
dynamic-l
lamp.dwg
light.dwg
Type
File fold
BricsCA
AutoCAD 2018 (*.dwg)
AutoCAD 2018 ASCII DXF (*.dxf)
AutoCAD 2018 Binary DXF (*.dxf)
AutoCAD 2013 (*.dwg)
AutoCAD 2013 ASCII DXF (*.dxf)
AutoCAD 2013 Binary DXF (*.dxf)
AutoCAD 2010 (*.dwg)
AutoCAD 2010 ASCII DXF (*.dxf)
AutoCAD 2010 Binary DXF (*.dxf)
AutoCAD 2007 (*.dwg)
AutoCAD 2007 ASCII DXF (*.dxf)
AutoCAD 2007 Binary DXF (*.dxf)
AutoCAD 2004 (*.dwg)
AutoCAD 2004 ASCII DXF (*.dxf)
AutoCAD 2004 Binary DXF (*.dxf)
AutoCAD 2000 (*.dwg)
AutoCAD 2000 ASCII DXF (*.dxf)
AutoCAD 2000 Binary DXF (*.dxf)
AutoCAD Release 14 (*.dwg)
AutoCAD Release 14 ASCII DXF (*.dxf)
AutoCAD Release 14 Binary DXF (*.dxf)
AutoCAD Release 13 Drawing (*.dwg)
AutoCAD Release 13 ASCII DXF (*.dxf)
AutoCAD Release 13 Binary DXF (*.dxf)
AutoCAD Release 11/12 Drawing (*.dwg)
AutoCAD Release 11/12 ASCII DXF (*.dxf)
AutoCAD Release 11/12 Binary DXF (*.dxf)
AutoCAD Release 10 ASCII DXF (*.dxf)
AutoCAD Release 10 Binary DXF (*.dxf)
AutoCAD Release 9 ASCII DXF (*.dxf)
Save
Cancel
Preview
257
Zoom Factor = 5
Friday, February 15, 2013

Save in: DWG Test File

File name:

Save as type: AutoCAD 2013 (*.dwg)

Description

Size: 27997 bytes

Created: 6/10/2016 10:16 AM

Modified: 2/15/2013 11:37 AM

Accessed: 6/10/2016 10:16 AM

□ Open as read-only

☑ Use preview

Save As File Format	Saves Drawings in File Formats Used By AutoCAD Version #				
AutoCAD 2018	2018	2019	2020		
AutoCAD 2013	2013	2014	2015	2016	2017
AutoCAD 2010	2010	2011	2012		
AutoCAD 2007	2007	2008	2009		
AutoCAD 2004	2004	2005	2006		
AutoCAD 2000	2000	2000i	2002		
AutoCAD Release 14	Release 14 (released in 1997)				
AutoCAD Release 13	Release 13 (released in 1994)				
AutoCAD Release 11/12	Releases 11 and 12 (released in 1990 and 1992, respectively)				

When you regularly save drawings in older formats, then you can make one of them the default. In the Settings dialog box, search for "save format," and then choose the file format you wish to make the default.

Should you need to go all the way back to AutoCAD 10 through 2.5, then use the DXF format:

AutoCAD Release 10	R10 (released in 1988)
AutoCAD Release 9	R9 (released in 1987)
AutoCAD Release 2.6	R2.6 (released in 1987)
AutoCAD Release 2.5	R2.5 (released in 1986)

DXF is short for "drawing interchange format," and is a (mostly) open file format read by many CAD and other programs. BricsCAD imports and exports drawings in DXF format.

Warning! BricsCAD may erase and alter some entities when it translates drawings to earlier formats of DWG and DXF. Check drawings after translation. This is not the fault of BricsCAD but due to the way that entities are represented in drawing files.

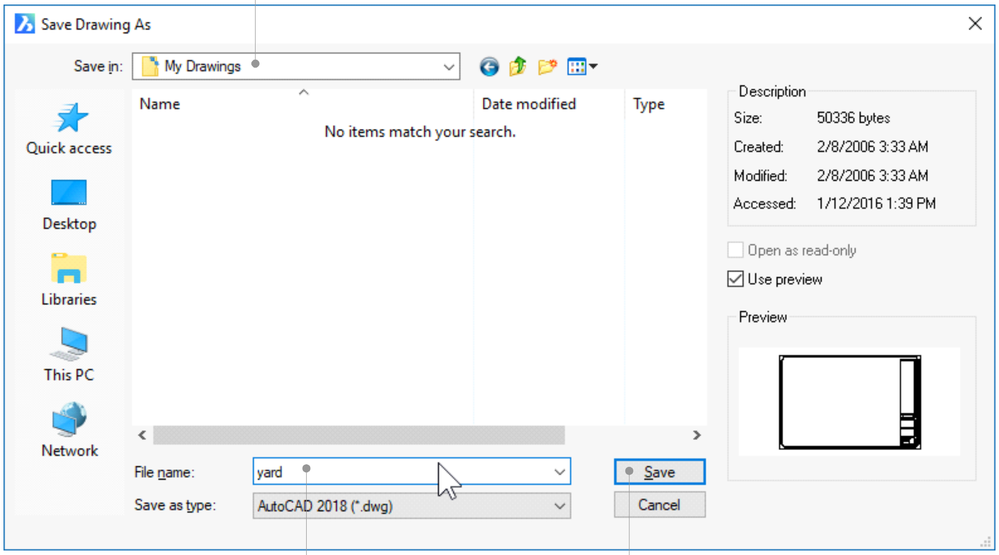
Saving Drawings

Command	Save
Menu Bar	File | Save
Ribbon	B | Save
Shortcut	Ctrl+S (Cmd+S on Macs)
Toolbar	Standard

The most important task in your CAD career is to save your work. Use the **Save** command:

1. In the Quick Access toolbar, click the **Save** button that looks like a diskette 💾 (tooltip: Save).
2. Because new drawings have a generic name, such as "Drawing1.Dwg," BricsCAD displays the Save Drawing As dialog box so that you can give it a better name. Type the name "Yard" in the **File name** text entry box.

Step 1: Choose drive and folder in to save the file



Save in: My Drawings

No items match your search.

Size: 50336 bytes
Created: 2/8/2006 3:33 AM
Modified: 2/8/2006 3:33 AM
Accessed: 1/12/2016 1:39 PM

☐ Open as read-only
☑ Use preview

File name: yard

Save as type: AutoCAD 2018 (*.dwg)

Step 2: Name the drawing file

Step 3: Click Save

QUICK SUMMARY OF ALTERNATIVE SAVE FORMATS

In addition to saving drawings in AutoCAD's *.dwg* file format, BricsCAD exports drawings with the **Export** command in these formats:

- ASCII and binary DXF (*.dxf*)
- Template (*.dwt*)
- Adobe Portable Document Format (*.pdf*)
- Scalable Vector Graphics (*.svg*)
- Industry Foundation Classes (*.ifc) for other BIM programs
- Mudbox in ASCII and binary formats (*.fbx) and Collada (*dae) for rendering software
- STL (stereolithography) for 3D printers (*.stl)
- Windows Bitmap (*.bmp*), Metaformat (*.wmf*), and Enhanced Metaformat (*.emf)
- And many versions of DWF, such as 2D, 3D, binary, ASCII, and compressed ASCII

COMMUNICATOR TRANSLATOR

When you need to translate drawings to and from other formats, then you may want enlist Communicator. This is an optional, extra-cost add-on for BricscAD Pro and Platinum that translates files in additional formats.

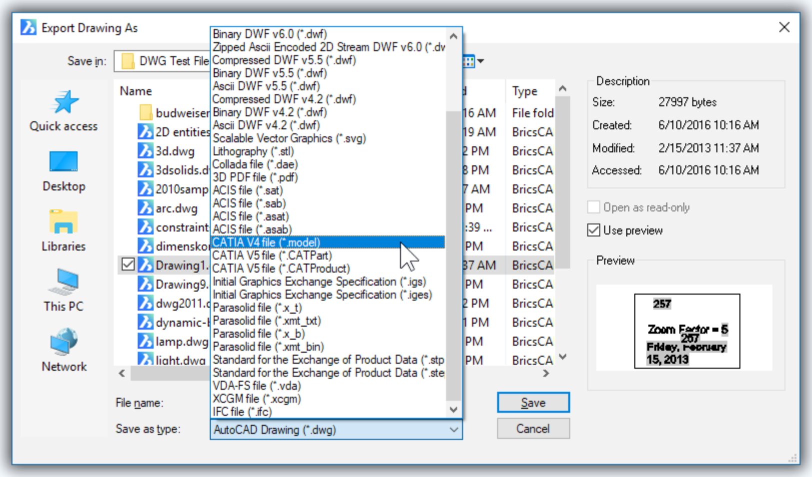
IMPORT

File format	Direct translator	Parasolid-based translator
ACIS	R1 – 2018 1.0	
CATIA V4	4.1.9 – 4.2.4	
CATIA V5	V5R8 – V5–6R2017	
3DExperience(CATIA V6)	up to V6 R2017x	
IGES	up to 5.3	
Inventor	V6 – V2017	
JT	8.x, 9.x and 10.x	8.x, 9.x and 10.x
NX	NX 1 – NX 11	11 – NX 11
Parasolid	9.0.x – 29.0.x	9.0 – 29.0.137
Pro/E / Creo	16 – Creo 4.0	
Solid Edge	V18 – ST9	V18 – ST9
SolidWorks	2003 – 2017	98 – 2017
STEP	AP203, AP214, AP242	
STL	All (Graphical data only)	
VDA-FS	1.0 – 2.0	
XCGM	R2012 – 2018 1.0	

EXPORT

File format	Direct translator	Parasolid-based translator
3D PDF	1.7	
ACIS	2018 1.0	
CATIA V4	4.2.4	
CATIA V5	V5–6R2017	
IGES	5.3	
STEP	AP203, AP214, AP242 (*)	
VDA-FS	2.0	
XCGM	2017 1.1	
Parasolid	N/A	29.0.137

You access the additional formats with the **Import** and **Export** commands, after the software in installed on your computer.



Save in: DWG Test File

File name:

Save as type: AutoCAD Drawing (*.dwg)

Description

Size: 27997 bytes

Created: 6/10/2016 10:16 AM

Modified: 2/15/2013 11:37 AM

Accessed: 6/10/2016 10:16 AM

□ Open as read-only

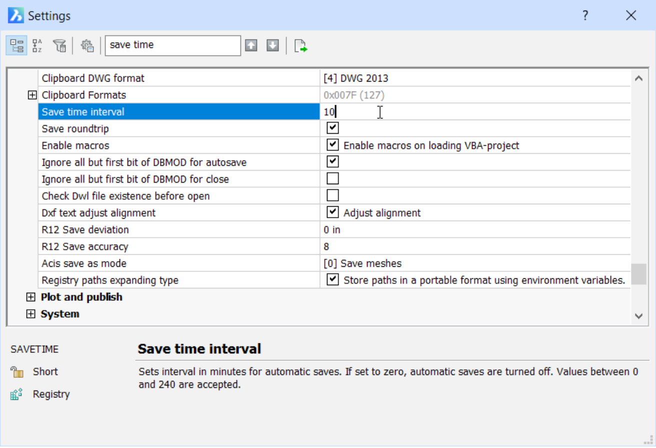
☑ Use preview

3. Click the **Save** button. BricsCAD saves the drawing with the name *yard.dwg*. From now on, you and BricsCAD refer to this drawing as “Yard.”

MAKING BACKUPS AUTOMATIC

For speed, BricsCAD keeps parts of the drawing in the computer’s memory (RAM). The drawback is that when the operating system crashes or when the power is cut to the computer, you may lose some or all of your work. Because your work is valuable, it is an excellent idea to save drawings every few minutes.

BricsCAD does this automatically, but also lets you specify the interval in which the drawings are saved — without you needing to repeatedly use the Save command. The default setting is **60** minutes, which is far too long.



Clipboard DWG format	[4] DWG 2013
⊞ Clipboard Formats	0x007F (127)
Save time interval	10
Save roundtrip	☑
Enable macros	☑ Enable macros on loading VBA-project
Ignore all but first bit of DBMOD for autosave	☑
Ignore all but first bit of DBMOD for close	☐
Check Dwl file existence before open	☐
Dxf text adjust alignment	☑ Adjust alignment
R12 Save deviation	0 in
R12 Save accuracy	8
Acis save as mode	[0] Save meshes
Registry paths expanding type	☑ Store paths in a portable format using environment variables.

⊞ **Plot and publish**

⊞ **System**

SAVETIME

Short

Registry

Save time interval

Sets interval in minutes for automatic saves. If set to zero, automatic saves are turned off. Values between 0 and 240 are accepted.

To enable automatic backups and to make automatic saves frequently, follow these steps:

1. Enter the **Settings** command.
2. In the **Find** field, enter “save time,” and then press **Enter**.
3. Change the value from **60** to **10**.

> The **Save Time Interval** setting specifies how much time elapses before BricsCAD saves drawings. Don’t set this number too low, such as 1 minute, otherwise the computer spends so much time saving to disk that other work cannot be done.

4. If it isn’t already, you might want to turn on **Incremental Save Backup**. This means that BricsCAD keeps the previous saved file as a backup copy — and that’s a good thing. Automatic saves and backups are not made to the original files. Instead, during automatic saves, BricsCAD saves drawings with the extension of *.sv$* and backed up files with *.bak*.
5. Click **X** to close the dialog box.
6. If you need to take a break at this point, use **File | Exit**. BricsCAD closes its window and you find yourself back at the desktop.

Although BricsCAD automatically saves drawings, it is still a good idea for you to save your work manually after finishing a significant amount of editing. Press **Ctrl+S** often!

Let’s review the drawing to this point. Although you haven’t drawn anything yet, the drawing file contains a fair amount of information.

- On the toolbar, you see the color of the current layer is blue and its name is **Lot**.
- On the status line, you see that the coordinates are displaying in decimal units, or meters; and that drafting modes, like snap, grid, and model, are turned on.
- The grid is made of dots.
- The automatic backup feature saves your drawing six times an hour.

In the next lesson, you begin to draw!

CHAPTER 4

Creating Your First Drawing

The point to 2D CAD is to draft drawings efficiently, and then print them, either on paper or electronically.

In this chapter, you learn how to draw lines, accurately, how to make changes to them, and then produce a copy of your first drawing on a printer.

IN THIS CHAPTER

- Drawing with lines and polylines
- Understanding absolute and relative distances
- Using polar coordinates
- Modifying entities
- Plotting (printing) drawings
- Exporting drawings in PDF format

KEY TERMS IN THIS CHAPTER

Absolute coordinate — refers to measurements made relative to the drawing's origin

Aperture — refers to the area around the cursor in which BricsCAD searches for entities to snap to

Direct distance entry — specifies points by moving the mouse in a direction, then entering the distance

Extent — refers to the invisible rectangle that encompasses all entities in drawings

Fillet — rounds corners

Mirror — mirrors copies of entities

Entity (entity) snap — snaps the cursor to geometric features, such as to the ends of lines

Origin — refers to the location of x=0, y=0, usually at the lower-left corner of drawings

Ortho — constrains cursor movement to the vertical and horizontal; short for "orthographic"

Pick cursor — refers to the square cursor in which BricsCAD searches for entities to select

Polar coordinate — describes measurements specified by distances and angles

Relative coordinate — describes measurements made relative to the last point

USEFUL ABBREVIATIONS

@	At symbol specifies relative coordinates, such as @2,3
#	Pound sign specifies absolute coordinates, such as #4,5
<	Angle bracket specifies angles, such as 10<45
-	Dash forces BricsCAD to use the command-line version of a command, such as -layer
[*option*]	Square brackets indicate command options, such as [Undo]
<*value*>	Angle brackets indicate the default (current) value, such as <LOT>
x	x specifies the X coordinate along the horizontal axis
y	y specifies the Y coordinate along the vertical axis

NEW COMMANDS

Command	Aliases	Menu Selection	Ribbon Tab
Cancel	Esc	...[1]	...
Fillet	f or fi	Modify | Fillet	Change | Modify | Fillet
Mirror	mi	Modify | 2D Mirror	Change | Arrange | 2D Mirror
Move	m	Modify | Move	Change | Arrange | Move
Open	op *or* Ctrl+O	File | Open	Home | File | Open
OSnap	os *or* F3	Settings | Settings	Settings | Settings
PLine	pl	Draw | Polyline	Draw | Draw | Polyline
Plot	Ctrl+P	File | Print	Home | Print/Plot | Print
QSave	Ctrl+S	File | Save	Home | File | Save
Zoom	z	View | Zoom	View | Zoom | Zoom

[1] The ellipsis (...) indicates no menu selection is available.

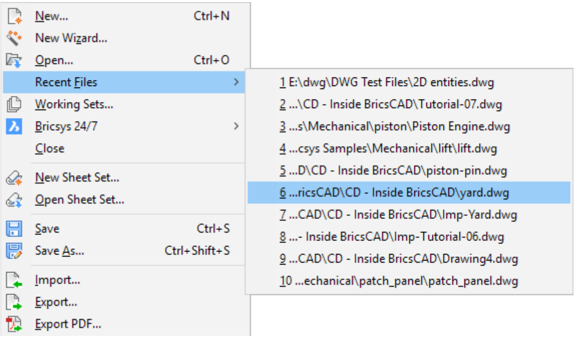
Reopening Drawings

Command	Open
Alias	op
Menu Bar	File | Open
Ribbon	B | Open
Shortcut	Ctrl+O
Toolbar	Standard

If you exited BricsCAD at the end of the last lesson, you need to restart the program and then load the *Yard* drawing.

TIP The **Recent Drawings** item always holds the names of the last ten drawings you opened with BricsCAD. Here's how to access them:

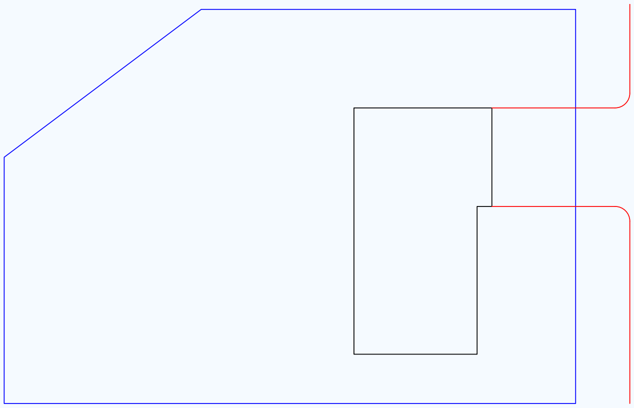
1. Click **B** on the ribbon.
2. Slide cursor down to **Recent Drawings**, and then click on **yard.dwg**. See figure below:


New... Ctrl+N
New Wizard...
Open... Ctrl+O
Recent Files
Working Sets...
Bricsys 24/7
Close
New Sheet Set...
Open Sheet Set...
Save Ctrl+S
Save As... Ctrl+Shift+S
Import...
Export...
Export PDF...
1 E:\dwg\DWG Test Files\2D entities.dwg
2 ...\CD - Inside BricsCAD\Tutorial-07.dwg
3 ...s\Mechanical\piston\Piston Engine.dwg
4 ...csys Samples\Mechanical\lift\lift.dwg
5 ...D\CD - Inside BricsCAD\piston-pin.dwg
6 ...ricsCAD\CD - Inside BricsCAD\yard.dwg
7 ...CAD\CD - Inside BricsCAD\Imp-Yard.dwg
8 ...- Inside BricsCAD\Imp-Tutorial-06.dwg
9 ...CAD\CD - Inside BricsCAD\Drawing4.dwg
10 ...echanical\patch_panel\patch_panel.dwg

Notice that when the *Yard* drawing opens in BricsCAD, it looks exactly the same as when you last saw it — that is to say blank, except for the grid. Let's get some lines on the screen!

THIS CHAPTER'S DRAWING

By the end of this lesson, your drawing should look like this one:



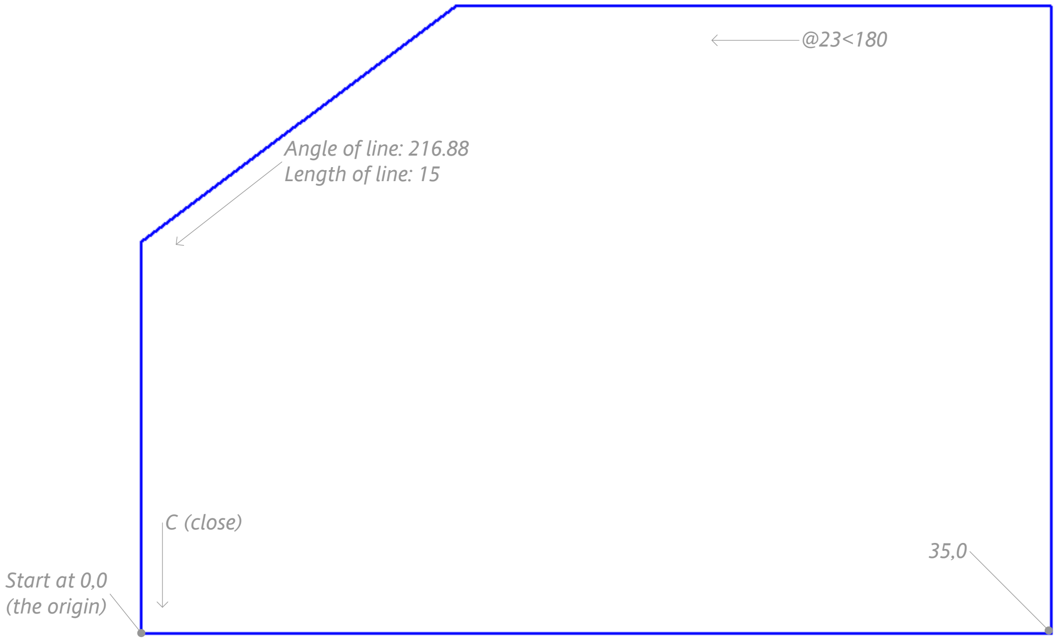
Tutorial-04 drawing file

Drawing the Lot's Boundary

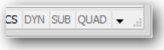
Command	Line
Alias	L
Menu Bar	Draw | Line
Ribbon	Draw | Draw | Line
Toolbar	Draw

The first thing to do is to draw the boundary of the yard, as this helps orient yourself on the screen and lets you see the extents of the drawing.

You draw the lot boundary with *lines*, and so you use the **Line** command to do that. You begin drawing at the lower-left corner, starting at the origin (0,0) of the drawing, and then working your way counterclockwise around the lot, as shown by the illustration below.



1. First, ensure **DYN** is turned off on the status bar; the word should look gray. This turns off dynamic input, so that all prompts appear in the Command bar.



2. Enter the **Line** command, as follows:

```
: line
```

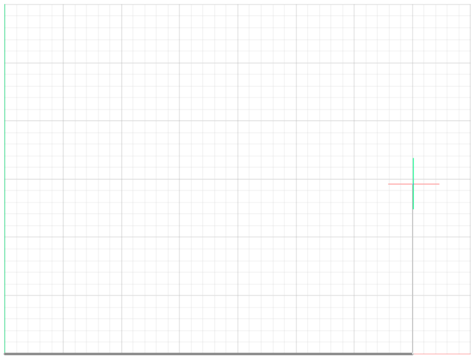
3. Respond to the 'Specify first point:' prompt by typing the coordinates of the origin:

```
Start of line: 0,0
```

4. To draw the lower boundary line 35m long, you need to tell BricsCAD that the far end of the line is located at the x,y coordinates of **35,0**. (In imperial units, this is **116',0"**.)

```
Set end point or [Angle/Length/Undo]: 35,0
```

See the figure below for what the first line should look like.



5. The next line is 24m north (up). Its endpoint is located at coordinates 35,24.

   (In imperial units, draw to 116',80'. Remember to include the apostrophe ' with each distance; the apostrophe indicates *feet*; if you were to leave it out, BricsCAD would interpret the numbers as inches, and you would end up with a very small yard!)

   ```
   Set end point or [Angle/Length/Follow/Undo]: 35,24
   ```

TIP Notice that the prompt line **Angle/Length/Follow/Undo** is separated by slashes. The *slashes* indicate that these words are *options* of the Line command. If you were now to type "undo"...

```
    Set end point or [Angle/Length/Follow/Undo]: undo
```

...BricsCAD would un-draw the last line segment.
"Angle" and "Length" are other options that specify the angle and length of line segments. Later, you encounter another option, "Close."

---

6. You drew the first two lines with *absolute coordinates*, where you calculated the coordinates based on measurements relative to the origin at 0,0.

   BricsCAD, however, can do these calculations for you when you use *polar coordinates*. In these, you specify the distance and angle. Continue drawing the lot boundary by combining relative and polar coordinates like this:

   ```
   Set end point or [Angle/Length/Follow/Close/Undo]: @23<180
   ```

   (In imperial units: End point: **@76'<180**)

   With all that punctuation, entering **@23<180** can feel like quite a typing chore. I'm getting you to type coordinates for now so that later you will appreciate other, less cumbersome input methods!

TIPS Using relative polar coordinates makes sense when you have many angled lines to draw.
When you tell BricsCAD to draw a line with the relative polar coordinates, you use a special notation with the following meaning:

Notation	Meaning
@	Use relative coordinates
23	Distance is 23m from the current point
<	Draw the line at the angle...
180	...of 180 degrees

Lines are drawn *relative* to the current point.
Angles are measured in *absolute* degrees, using the East-is-0-degrees convention.

If you make a mistake entering any coordinate notation, simply use the **Undo** option (type **U** at the prompt) and reenter the coordinates.

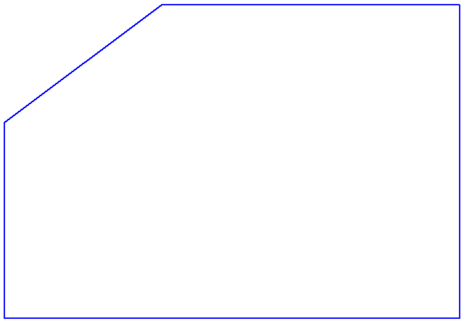
7. Enter the coordinates for the next endpoint using the **Angle** option, which then prompts for the length of the line. Notice that the **Angle** option removes the need to use the **@** and **<** characters:

   ```
   Set end point or [Angle/Length/Follow/Close/Undo]: a
   Angle of line: 216.88
   Length of line: 15
   ```

   (In imperial units, the angle is the same, but the length of the line is **50'**.) The **Length** option works the same way, but prompts first for the length, then the angle.

8. To finish the lot boundary, you use a shortcut. Type **c** (short for "Close") to close the boundary — instead of typing the final coordinates (0,0). BricsCAD automatically draws a line from the current endpoint to the beginning of the first line, without you having to work it out.

   ```
   Set end point or [Angle/Length/Follow/Close/Undo]: c
   ```



As I mentioned earlier, slashes separate option names. The options for the Line command are Angle/Length/Follow/Close/Undo:

Angle — asks you to enter the angle of the next line segment, followed by length

Length — asks you to enter the length of the next line segment, followed by angle

Follow — asked you to show the angle with the mouse, and then enter the length

Close — BricsCAD closes the polygon formed by the lines

Undo — BricsCAD undoes the drawing of the last line segment

9. To add some working space around what you've drawn, use the **Zoom** command to make the drawing 10% smaller, as follows:

   ```
   : zoom
   Zoom [zoom In/zoom Out/All/Center/Dynamic/Extents/Left/Previous/Right/Scale (nx/nxp)/
   Window/OBject] <Scale (nX/nXP)>: 0.9x
   ```

   This zooms the drawing 90% as large as it was before.

Instead of drafting on paper, you have created your first digital drawing!

More importantly, you have drawn it full size — even if it looks small on your computer screen. This is one of the most powerful aspects of CAD: everything is drawn full size. There is no need for a scale ruler or to divide distances by a scale factor as in manual drafting.

Planning the Next Steps

The next step is to draw an outline of the house. The lower-right corner of the house is located 3m up and 6m in from the lot corner.

There are several ways in BricsCAD to locate one entity relative to another, in this case the corner of the house relative to the property line. You could...

- Calculate the coordinates of the house's corner, or
- Use the **XLine** command to draw a pair of construction lines, then start drawing from their intersection, or
- Use the **From** option to start drawing from an offset from the lot corner, or
- Use *tracking* to offset the starting point (described in a later lesson),or
- Draw the house in the corner of the lot, then move the house into position

Please don't feel overwhelmed by this long list of options. I show them to you purely to make the point that in CAD there are many ways to skin a cat.

In this tutorial, you use the last method in the list: Draw, and move. It demonstrates two of BricsCAD's most powerful commands, **PLine** and **Move**. But first, make a layer change.

CHANGING LAYERS

```
Command     -Layer
```

Before you draw the house, you need to change the layer to "House." Continuing with the theme of many approaches to solving drafting problems, a significant number of BricsCAD's commands can be carried out in more than one way. Here you learn an alterative way to control layers.

In the previous chapter, you used the Drawing Explorer dialog box to create and color new layers and to set "Lot" as the current (or working) layer. Another method is to change the layer through a droplist on the ribbon, status bar, toolbar, or use the Layers panel. And then there is another method:

In this part of the tutorial, you change layers by typing the command and its options at the keyboard by entering the **-Layer** command. The hyphen (-) in front of Layer forces the command to display its prompts in the command bar.

1. To set the House layer with the **-Layer** command, enter the following:

   ```
   : -layer
   ```

2. The -Layer command has more than a dozen options, most of which you ignore for now:

   ```
   Layer [? to list/New layer/Make new current layer/Set layer as current/Color/Linetype/
   LineWeight/TRansparency/MATerial/Plot/stAte/turn layer ON/turn layer OFf/Freeze/Thaw/
   LOck/Unlock]: s
   ```

   Enter **s**, and BricsCAD invokes the Set option. It sets another layer as active, whose name you specify.

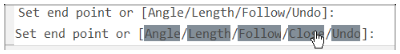
TIPS For all command options, BricsCAD capitalizes one or more letters. To specify an option, you need type only its capitalized character, such as **S** for the **Set** option.

When two (or more) options begin with the same first letter — such as **LType** and **LWeight** — then you need to enter the first two characters, as specified by the capitalization — **LT** and **LW** in this case.

There are a few times when the character that's capitalized is in the middle of the option name, such as **A** for the **stAte** option. This is because the all the other letters in stAte were already taken up by other option abbreviations.

By capitalizing them, BricsCAD shows you the fewest characters you need to enter for it to know what you're talking about. So you could enter **SE** for the **Set** option or **LTY** for **LType.**

(NEW IN V20) As an alternative to typing option abbreviation, you can use the mouse cursor to pick options at the command line:



```
Set end point or [Angle/Length/Follow/Undo]:
Set end point or [Angle/Length/Follow/Close/Undo]:
```

3. BricsCAD prompts you to enter the name of the layer to make active. To change the working layer to **House**, type "house" and then press **Enter**.

   ```
   Layer to set as current: house (Press Enter)
   ```

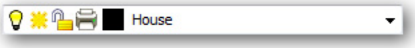
4. The *House* layer is now current.

   The -Layer command repeats its many-optioned prompt. Press **Esc** to cancel the command and return to the ':' prompt.

   ```
   Layer [? to list/New layer/Make new current layer/Set layer as current/Color/Linetype/
   LineWeight/TRansparency/MATerial/Plot/stAte/turn layer ON/turn layer OFf/Freeze/Thaw/
   LOck/Unlock]: (Press Esc)
   ```

TIP You can cancel commands at any time by pressing **Esc.** Sometimes, however, you may need to press **Esc** two or even three times in commands that have large numbers of sub-options, such as the PEdit command.

5. Look at the ribbon, status bar, or toolbar to confirm that BricsCAD has changed the working layer from layer Lot to layer House.



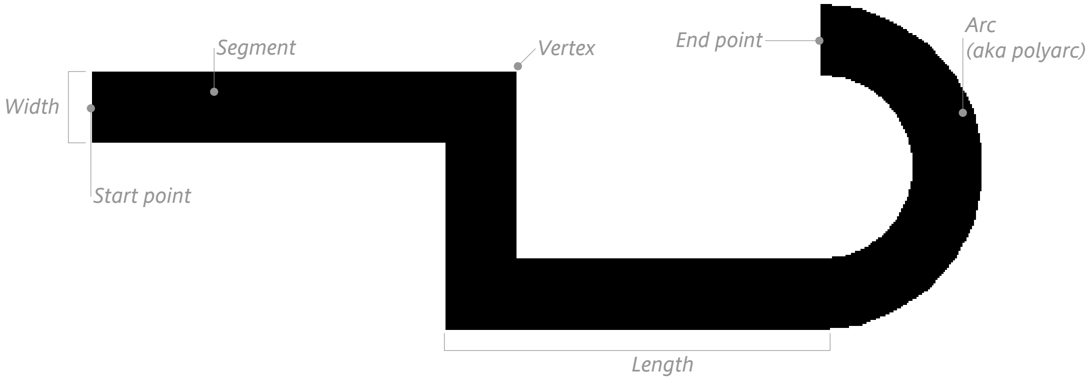
Drawing the Outline of the House

```
Command    Orthogonal
Aliases    ortho, or
Menu       Settings | Draw Orthogonal
Ribbon     Settings | Settings | Draw Orthogonal
Toolbar    Settings
Shortcuts  Ctrl+L, F8, Shift
Status Bar ORTHO
```

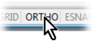
```
Command    Pline
Aliases    pl, polyline
Menu Bar   Draw | Polyline
Ribbon     Draw | Draw | Polyline
Toolbar    Draw | Polylines
```

Earlier, you drew the lot boundary as a polygon with the Line command. It looks like a continuous line that was made of several segments and *vertices* (corners). Each segment, however, is independent; they only *look* connected. This disconnected group of lines can be difficult to deal with when it comes to selecting something like the entire lot boundary at once.

To remedy this, BricsCAD offers a special kind of line called the "polyline." Polylines are lines made of many features, as suggested by the prefix *poly* (from the Greek word *polloi* for "many"). They can be made of lines and arcs, or of snaky-looking spline-like shapes; plines can have widths (which lines cannot), and even varying widths — all connected together as a single entity, as illustrated below.



1. Because the lines describing the house are all at right angles, I recommend using orthographic mode, known as "ortho" for short. The mode constrains cursor movement to the horizontal and vertical directions. Turn on ortho mode by clicking the **ORTHO** button on the status bar so that it turns black.

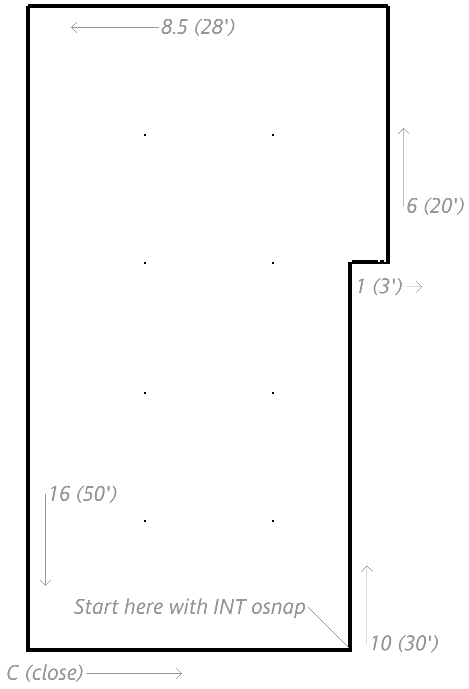


As confirmation, BricsCAD reports the following:

```
: <Ortho on>
```

2. Draw the house's outline as a polyline. (The figure further below gives the dimensions for the outline of the house.) To do so, enter the **PLine** command.

```
: pline
```



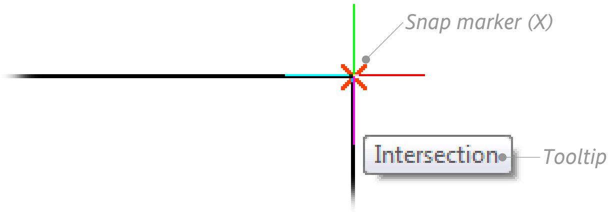
BricsCAD prompts you for the point from which to begin drawing the polyline. But instead of specifying coordinates, you this time ask BricsCAD to find a geometric feature by using *entity snap*:

```
Select start of polyline or [Follow] <Last point>: int
```

When you enter **int** (short for "intersection"), BricsCAD attempts to snap to the nearest intersection, rather than to the nearest 0.1m you specified earlier with the snap mode. *Entity* snaps override regular snap distance. Here, intersection snap overrides the 0.1m-snap. This is referred to as "entity snap override."

3. BricsCAD then prompts you to position the cursor near the intersection of two lines.

```
Snap to intersection of:  (Pick the lower-right corner of the lot.)
```



a. Move the cursor until the aperture is over the lower-right corner of the lot boundary. The tooltip reports the entity snap found by BricsCAD.

b. *Click* — press the left mouse button. BricsCAD "captures" the intersection as the starting point for the polyline.

DIRECT DISTANCE ENTRY

As CAD systems improve, they develop easier way to specify distances and angles. Let's now look at a shortcut called "direct distance entry" (DDE).

DDE combines cursor movement with keyboard entry — you show BricsCAD the angle directly, and then type in the distance. It's a lot quicker than typing values of angles — and more intuitive, too. It works best with large angles, such as 90 degrees.

Carrying on with the tutorial, here is how to use direct distance entry:

4. The second line of the prompt displays many options. Don't let them intimidate you! For now, you can ignore all of them, except for the default, 'Set next point.'

   ```
   Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width]: (Move cursor up)
   ```

   Here is how to do direct distance entry:

   a. First, move the cursor up towards the top of the drawing. The distance does not matter, only the direction. (Other ways of saying "move the cursor up" include "in the positive y direction," or "to the North," or "at 90 degrees").

   b. Then, on the keyboard, type **10** and press **Enter**. (In imperial units, **30'**)

Notice that the line is drawn upwards, 10m long. Direct distance entry is like another form of relative coordinates: BricsCAD measures the 10m relative to the last entered point.

QUICK SUMMARY OF THE PLINE COMMAND

A *polyline* consists of numerous lines and/or arcs connected together as a single entity. Polylines are drawn with the PLine command, and edited with the PEdit command. The PLine command has the following options for drawing polylines:

```
: pline
Start of polyline:
Set next point or [draw Arcs/Close/Distance/Follow/Halfwidth/Width/Undo]:
```

Next point — specifies the location of the polyline's next vertex

draw Arcs — draws polyarcs; you can switch between lines and arcs when drawing polylines

Close — joins the last endpoint with the starting point

Distance — specifies the length and angle of the next segment

Follow — draws a specific distance, in the direction of the cursor's angle

Halfwidth — specifies the width of polylines by the distance from their center line to their outside edge

Width — specifies the width of polyline segments; allows independent starting and ending widths for tapered polylines

Undo — undoes the last polyline drawing operation

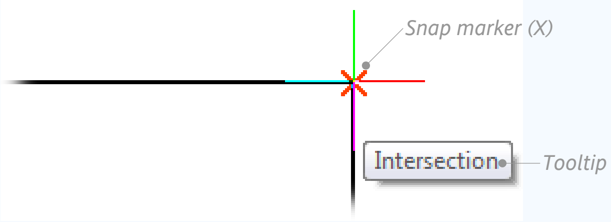
QUICK SUMMARY OF ENTITY SNAP MODES

BricsCAD has the following entity (object) snaps that look for specific geometric features on entities:

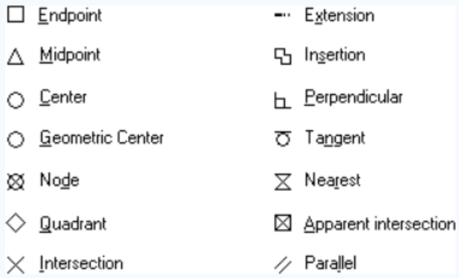
Mode	Command	Snaps to...
app	APParent	Apparent intersections of two entities
cen	CENter	Center of arcs, circles, and polyarcs
end	ENDpoint	Either end of lines, arcs, and other open entities
ext	EXTension	Extension of two entities
from	FROM	Offset distance from a specified point
gcen	GCENter	Geometric center of objects
ins	INSertion	Insertion point of blocks and text
int	INTersection	Intersection of lines, arcs, circles, and other entities
mid	MIDpoint	Middle of lines, arcs, and other open entities
nea	NEArest	Nearest point on the nearest entity
nod	NODe	Point entities
par	PARallel	Parallel to lines and other entities
per	PERpendicular	Perpendicular to lines, arcs, and other entities
qua	QUAdrant	0-, 90-, 180-, and 270-degree points on arcs, circles, and polyarcs
qui	QUIck	First geometric feature found by BricsCAD
tan	TANgent	The tangents of arcs and circles

The M2p command is kind of like an entity snap, in that it finds the middle between two points.

Because more than one entity snap can be active at a time, BricsCAD provides a number of visual cues to identify the current one. You can toggle the visual cues through the Program Options | Display section of the Settings dialog box.



The table below illustrates on-screen markers associated with each entity snap mode.



□	Endpoint	╍	Extension
△	Midpoint	⧉	Insertion
○	Center	⊥	Perpendicular
○	Geometric Center	○	Tangent
⊠	Node	⧗	Nearest
◇	Quadrant	☒	Apparent intersection
×	Intersection	//	Parallel

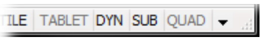
DYNAMIC INPUT

Let's now move to yet another way of using direct distance entry, one that is even more interactive in the drawing area. It is known as "dynamic input," because it displays distances and angles dynamically — right at the cursor in the drawing.

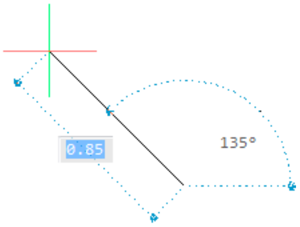
Let's see how it works:

5. To use dynamic input for the next segment (the 1m-long line), follow these steps:

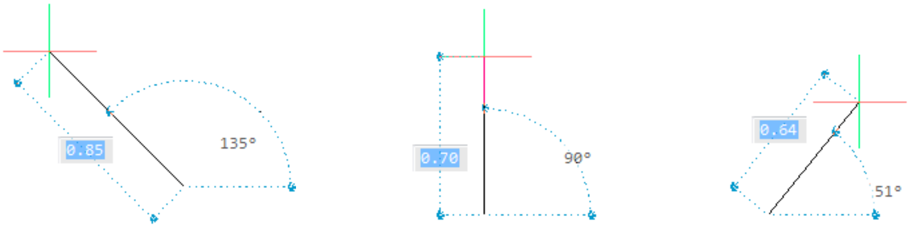
   a. On the status bar, ensure **DYN** is turned on. When **black** (instead of **gray**) then dynamic input is on.



   Notice that a couple of "dimensions" immediately appear in the drawing area:



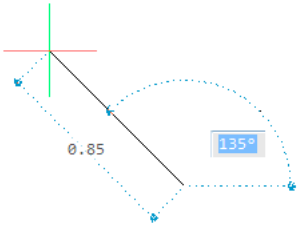
   b. These are known as "dynamic dimensions," because they change as you move the cursor. Try it now: Move the cursor and watch how the length and angle values change.



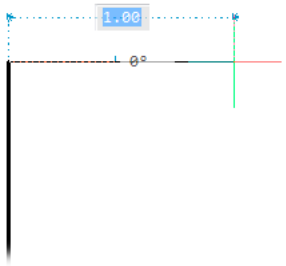
Left to right: *Dynamic dimension values changing as the cursor moves*

TIP When you enter numbers at the keyboard, they are entered into the blue field. Notice in the figures above that the **length** field is highlighted in blue.

To move between the length and angle fields, press the **Tab** key. In the figure below, the **angle** field (135 degrees) turns blue, as shown below.

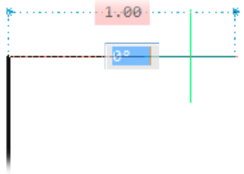


c. Enter the length of the wall, **1** and then press **Tab**. (In imperial units: **3'**.)



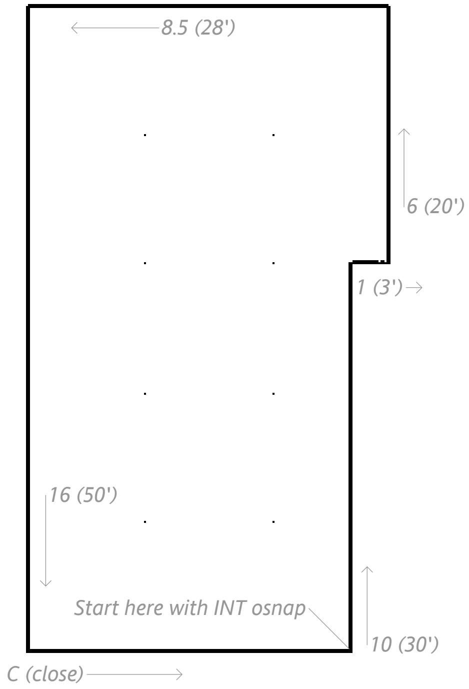
Notice that the length field (1 or 3) turns **red**. This tells you that BricsCAD has accepted the value, and has locked it temporarily.

d. When you pressed **Tab** in the last step, BricsCAD highlighted the angle field in blue. This means that it is ready for input: enter the angle of the wall, **0** (degrees)...



...and then press **Tab** again to lock the value. (To change the length and/or angle, press **Tab** to switch between the two fields.)

e. Accept the length and angle values by pressing **Enter**.



6. Draw the remainder of the house outline by moving the cursor in the appropriate direction, and then entering the distances:

Cursor Direction	Metric Distance	Imperial Distance
East	1	3'
North	6	20'
West	8.5	28'
South	16	50'

7. Complete the polyline with the **C** option, as you did with the Line command.

When Lot was the working layer, the lines you drew showed up in blue. You changed the layer to House, and so BricsCAD automatically drew the lines in black instead. This shows that lines take on the color specified by their layer. (You can, if you need to, change colors on-the-fly with the **Color** command.)

Moving the House into Position

```
Command    Move
Alias      m
Menu Bar   Modify | Move
Ribbon     Change | Arrange | Move
Toolbar    Modify | Move
```

Now that you've drawn the outline of the house, you need to move it into position. Entities are moved with the **Move** command.

1. Enter **Move** the 'Command:' prompt:

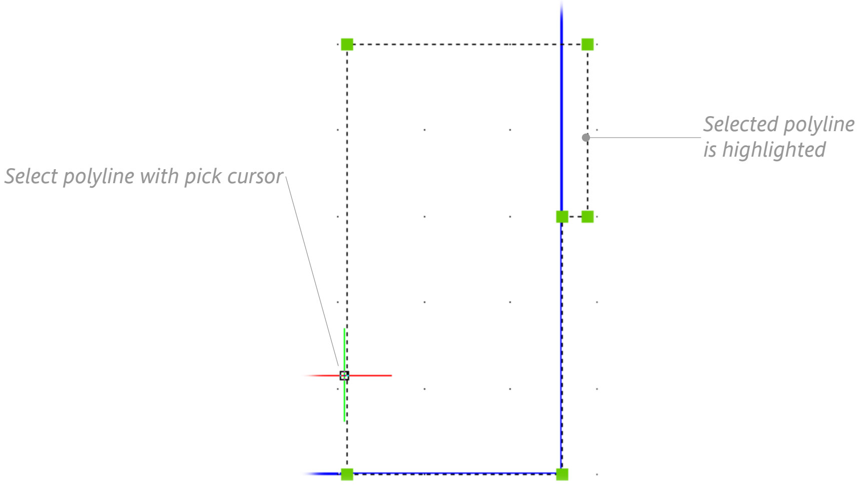
```
: move
```

2. Notice that BricsCAD asks what you want to move:

```
Select entities to move:
```

("Entities" is an older word that means the same as *objects*.)

At the same time, BricsCAD changes the crosshair cursor into a small square cursor, called the pick cursor.


Selected polyline is highlighted
Select polyline with pick cursor

3. Move the cursor to any part of the house's polyline, and then press the pick button.

   ```
   Select entities to move: (Pick the outline of the house)
   ```

   The entire house outline is highlighted. The highlighting shows as a dotted line, which is how BricsCAD lets you know it found the entity you picked.

4. BricsCAD reports the number of entity selected, and then lets you make additional selections, if need be. The prompt repeats:

   ```
   Select entities to move: (Press Enter to exit entity selection)
   ```

   Because you are moving just one polyline, press **Enter** to end the entity selection process.

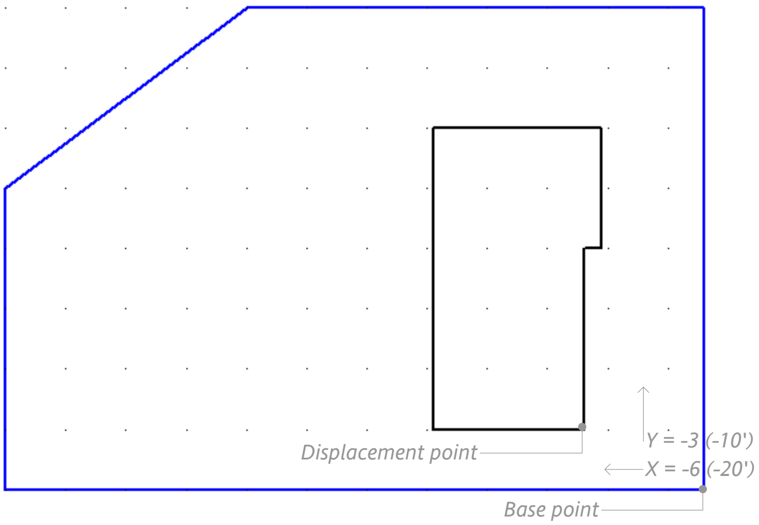
5. Just as when drawing lines, the Move command needs to know a *from*-point and a *to*-point. But here the from-point is named the "base point," as follows:

   ```
   Enter base point [Displacement] <Displacement>:  0,0
   ```

   (If you were to press Enter at the 'Enter base point <Displacement>' prompt, BricsCAD would ask for the displacement vector, which consists of a move in the x,y, and z directions.)

6. Now BricsCAD wants to know where you want to move the selected entities:

   ```
   Enter second point <Use base point as displacement>: -6,3
   ```



   The coordinates -6,3 (in imperial units: **-20',10'**) tell BricsCAD to move the house left by 6m (x direction = **–6** meters) and up by 3m (y direction = **3**). BricsCAD instantly relocates the house much faster than a house mover. The Move command shows you a powerful aspect of CAD: no eraser dust!

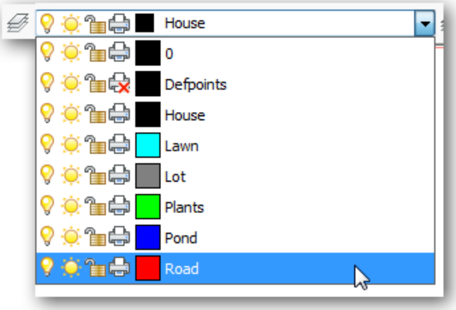
7. It's a good idea to regularly save your work. Press **Ctrl+S**, and BricsCAD silently saves the drawing to disk. The only indication is that ": _qsave" appears on the command line.

   Alternatively, you can click the diskette icon on the toolbar, enter the **QSave** command, or select **Save** from the **File** menu.

Starting on the Driveway

The final drafting for this project is to add the driveway and the street. Before drawing them, change the layer to **Road** using the technique described below. Once the layer is set correctly, you will draw the driveway and street outlines using a shortcut. Once the upper roadwork is drawn, you duplicate it with a single command to create the lower roadworks.

1. On the toolbar, status bar, or ribbon, click on the **House** layer name.
2. When the list box appears, select **Road**. BricsCAD changes the layer name from House to Road and the working color from black to red. Below, I show the toolbar version of the Layers droplist.



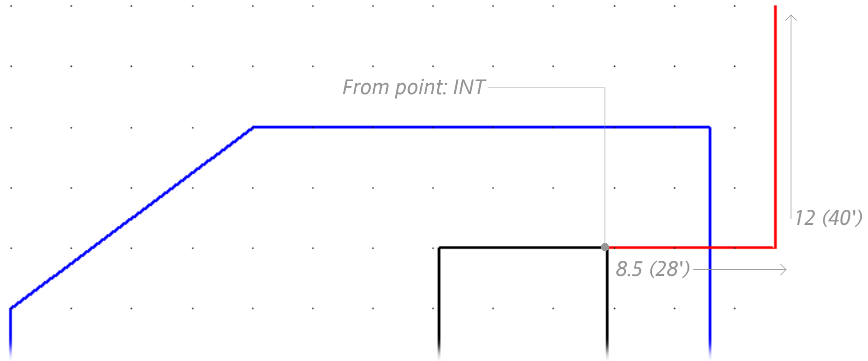
House
0
Defpoints
House
Lawn
Lot
Plants
Pond
Road

3. Ensure that ortho mode is still on by glancing at the status bar. The **ORTHO** button should be in black text.
4. Start the **Line** command:

```
: line
```

...and then draw the upper driveway and street line using direct distance entry, as follows:

```
Start of line or [Follow] <Last point>: int
Snap to intersection of: (Pick upper-right corner of house)
Set end point or [Angle/Length/Undo]: 8.5
Set end point or [Angle/Length/Undo]: 12
Set end point or [Angle/Length/Follow/Close/Undo]: (Press Enter to end the command)
```



Recall that you can “back up” to undraw incorrect lines with **u** (short for “Undo”), as follows:

```
Set end point or [Angle/Length/Follow/Close/Undo]: 21
Set end point or [Angle/Length/Follow/Close/Undo]: u
Set end point or [Angle/Length/Follow/Close/Undo]: 12
```

FINISHING THE DRIVEWAY

Command	Fillet
Alias	f
Menu Bar	Modify | Fillet
Ribbon	Change | Modify | Fillet
Toolbar	Modify | Fillet

To add the *curb return* — the arc joining the driveway and street — use BricsCAD's **Fillet** command. It draws arcs between intersecting lines. The lines don't have to physically meet; BricsCAD takes care of extending (or trimming) the lines so that the arc is drawn between them.

You use the Fillet command by specifying the radius of the arc, and then applying the fillet.

1. To start the command, enter the **Fillet** command, or else do as I do: just type **f** at the ' **:** ' prompt.

```
: f
```

2. Notice that BricsCAD reports the current fillet radius. To change it, enter the **r** option:

```
Fillet (radius=0.5): Select first entity or [Settings/Polyline/Radius/Trim/Undo/Multiple]: r
```

If you follow the official route by entering "s" for the **Settings** option, then BricsCAD opens the Settings dialog box at the section where the fillet radius is specified. Just another case of more than one way to skin cats.

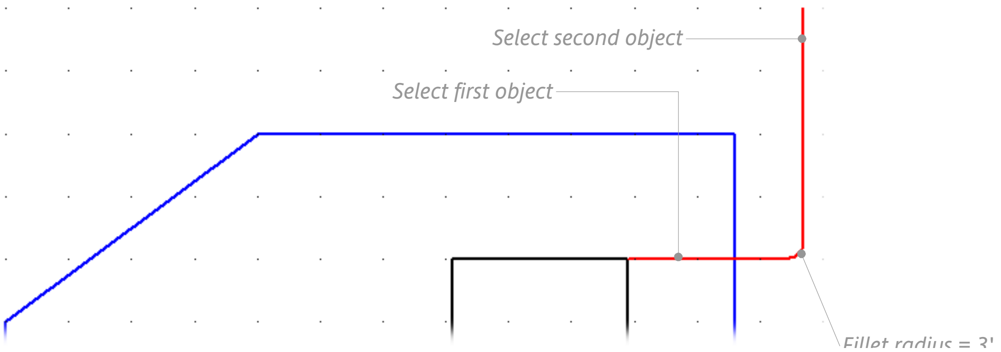
3. Enter the fillet radius of one meter (three feet in imperial units):

```
Fillet radius <radius=0.50>: 1
```

4. With the fillet radius set to 1m, perform the filleting, as follows:

```
Fillet (radius=1): Select first entity or [Settings/Polyline/Radius/Trim/Undo/Multiple]:
(Pick one line)
Select second entity (select with pressed SHIFT to make corner): (Pick the other line)
```

BricsCAD automatically adjusts the two lines to fit the 1m arc between them.



Mirroring Entities

Command	Mirror
Alias	mi
Menu Bar	Modify | 2D Mirror
Ribbon	Change | Modify | 2D Mirror
Toolbar	Modify | Mirror

Earlier, you used the Line and Fillet commands to create one part of the roadway. A very important concept behind computer-aided *anything* is that you should never have to draw the same line twice.

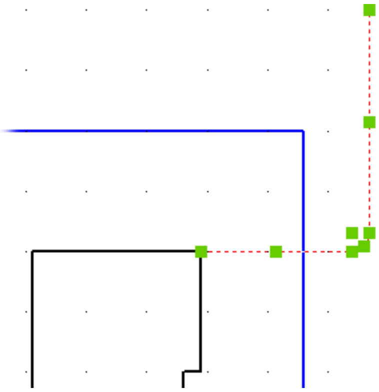
To illustrate the power of this concept, use the Mirror command to duplicate the lower driveway and street line without having to draw them! This command creates mirrored copies of entities, naturally: the copy is the reverse of the original.

1. Enter the **Mirror** command:

```
: _mirror
```

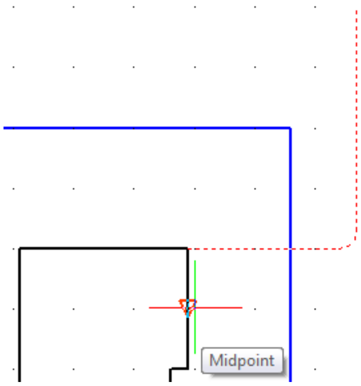
2. BricsCAD asks you to select the entities you want to mirror. Use the cursor to pick the line and arc segments, as follows:

```
Select entities to mirror: (Pick the driveway line)
Entities in set: 1

Select entities to mirror: (Pick the curb return)
Entities in set: 2

Select entities to mirror: (Pick the street line)
Entities in set: 3
Select entities to mirror: (Press Enter to end entity selection)
```

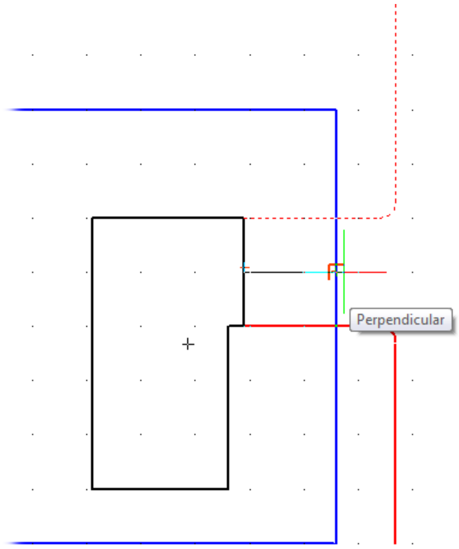


3. BricsCAD needs you to specify the mirror line, an imaginary line about which it mirrors the entities you just picked. Use **Mid**point and **Per**pendicular entity snaps to help you:

```
Start of mirror line: mid
Snap to midpoint of: (Pick center of garage entrance)
```



```
End of mirror line: per
Snap to perpendicular of: (Pick right-hand lot boundary)
```



The length of the mirror line is not important, but its angle is crucial. For this reason, you used two new entity snap modes: **mid** to find the midpoint of the garage entrance and **per** to ensure the mirror line is perpendicular to the lot boundary.

4. At this point, BricsCAD gives you the option of erasing the old entities — the two lines and arc you picked. In most cases, as in this case, you *don't* want them erased:

```
Delete the original entities? [Yes-delete entities/No-keep entities] <No-Keep entities>: n
```

BricsCAD draws the lower driveway and street outline as a perfect mirror image of the upper set.

You have now drawn the outline of the lot, house, and driveway. The work you have done is valuable and it is important that you save the drawing to disk. Use the **QSave** command to store the drawing on disk.

Putting Drawings to Paper

Command	Print
Menu Bar	File | Print
Ribbon	Home | Print/Plot | Print
Shortcut	Ctrl+P
Toolbar	Standard | Print

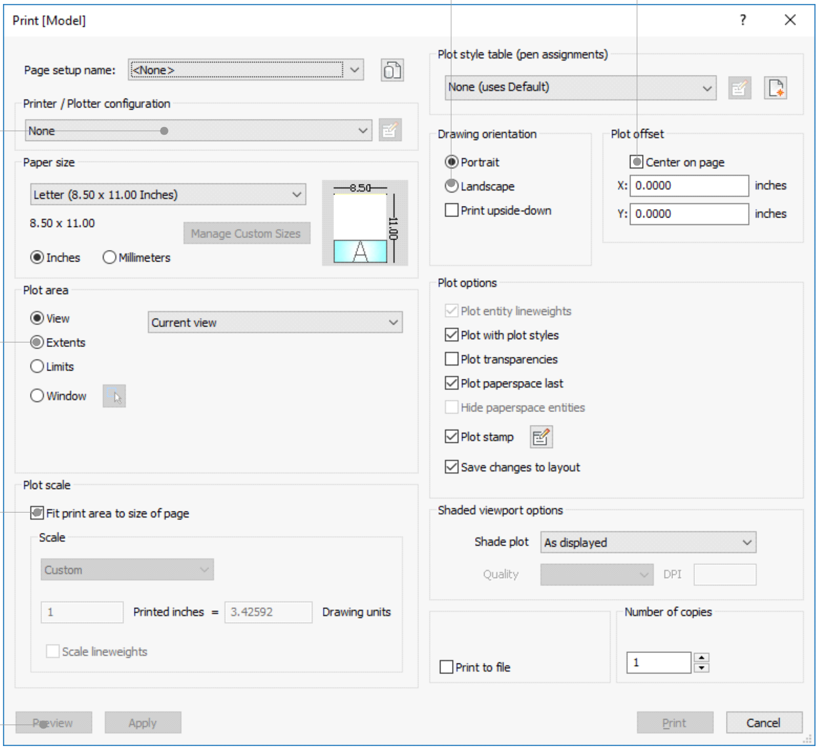
While it is efficient (and environmentally aware) to create and store drawings on computers and share them electronically via email or Web sites, you may want to print copies on paper. That lets you mark up the drawing with notes or show off your progress to friends and family!

The **Print** command sends the drawing to printers, plotters (oversize printers), and files. It is also known as the print command. Traditionally, "plotters" used motors to move the pen over the paper, while "printers" are what we mostly use today – laser printers and inkjet printers.

The following series of dialog boxes assumes you have a printer compatible with Windows or Linux attached to your computer. (The method for printing with Macs is roughly similar, but shows different dialog boxes.)

Drawing orientation: ***Landscape***

Plot offset: ***Center on Page***



Print [Model]

Page setup name: <None>

Printer / Plotter configuration: None

Paper size: Letter (8.50 x 11.00 Inches)

8.50 x 11.00

● Inches ○ Millimeters

Plot area: ● View ● Extents ○ Limits ○ Window — Current view

Plot scale: ☑ Fit print area to size of page

Scale: Custom

1 Printed inches = 3.42592 Drawing units

□ Scale lineweights

Plot style table (pen assignments): None (uses Default)

Drawing orientation: ● Portrait ● Landscape □ Print upside-down

Plot offset: ● Center on page; X: 0.0000 inches; Y: 0.0000 inches

Plot options: ☑ Plot entity lineweights; ☑ Plot with plot styles; □ Plot transparencies; ☑ Plot paperspace last; □ Hide paperspace entities; ☑ Plot stamp; ☑ Save changes to layout

Shaded viewport options: Shade plot: As displayed; Quality; DPI

□ Print to file

Number of copies: 1

Choose a printer

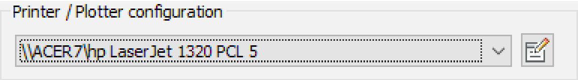
Choose ***Extents***

Choose ***Fit print area to size of page***

Click ***Preview***

1. Press **Ctrl+P** or enter the **Print** command. Notice that BricsCAD displays the **Plot [Model]** dialog box. (Old timers might enter the command as "plot," but this makes the command's options appear in the Command bar, instead of in the dialog box.)
2. The dialog box has many options. Fortunately you can ignore most options, except for these:

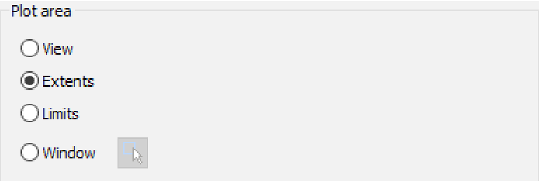
Printer/Plotter Configuration selects the printer you want to use:



Printer / Plotter configuration: \\ACER7\hp LaserJet 1320 PCL 5

Select a printer from the **Name** droplist. BricsCAD supports local and networked printers.

Plot Area determines which part of the drawing is printed:



Plot area

○ View

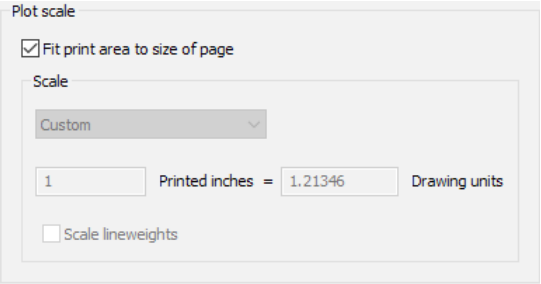
● Extents

○ Limits

○ Window

Select **Extents**. This option ensures *everything* in your drawing is plotted on the paper.

Plot Scale determines how large the drawing is printed on the paper:



Plot scale

☑ Fit print area to size of page

Scale

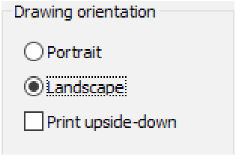
Custom

1 Printed inches = 1.21346 Drawing units

□ Scale lineweights

Check that **Fit Print Area to Size of Page** is selected. This ensures the entire drawing will be plotted, and that it won't be too large or too small.

Page Orientation turns the drawing by 90 degrees on the paper:



Drawing orientation

○ Portrait

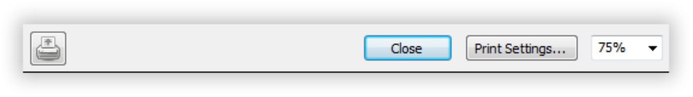
● Landscape

□ Print upside-down

If the orientation of the paper doesn't match that of the drawing, then you need to change it. The paper is tall but the drawing is wide. To change the orientation of the paper, select **Landscape**.

QUICK SUMMARY OF PRINT PREVIEW

The Print Preview window has the following controls:



Print button prints the drawing as shown.

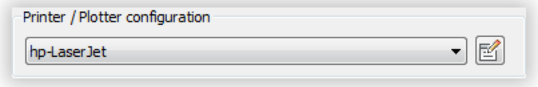
Close closes the window, and returns to the Print dialog box.

Print Settings also returns to the Print dialog box.

Zoom droplist changes the size of the preview image; scale ranges from 10% to 500%.

If the **Preview** button is grayed out in the Print dialog box, this means you have not yet selected a printer for this drawing.

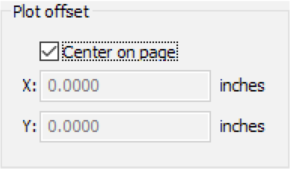
To make preview available, select a printer — any printer! — from the Printer/Plotter Configuration droplist:



Printer / Plotter configuration

hp-LaserJet

Plot Offset moves the drawing to one side of the paper or the other:



Plot offset

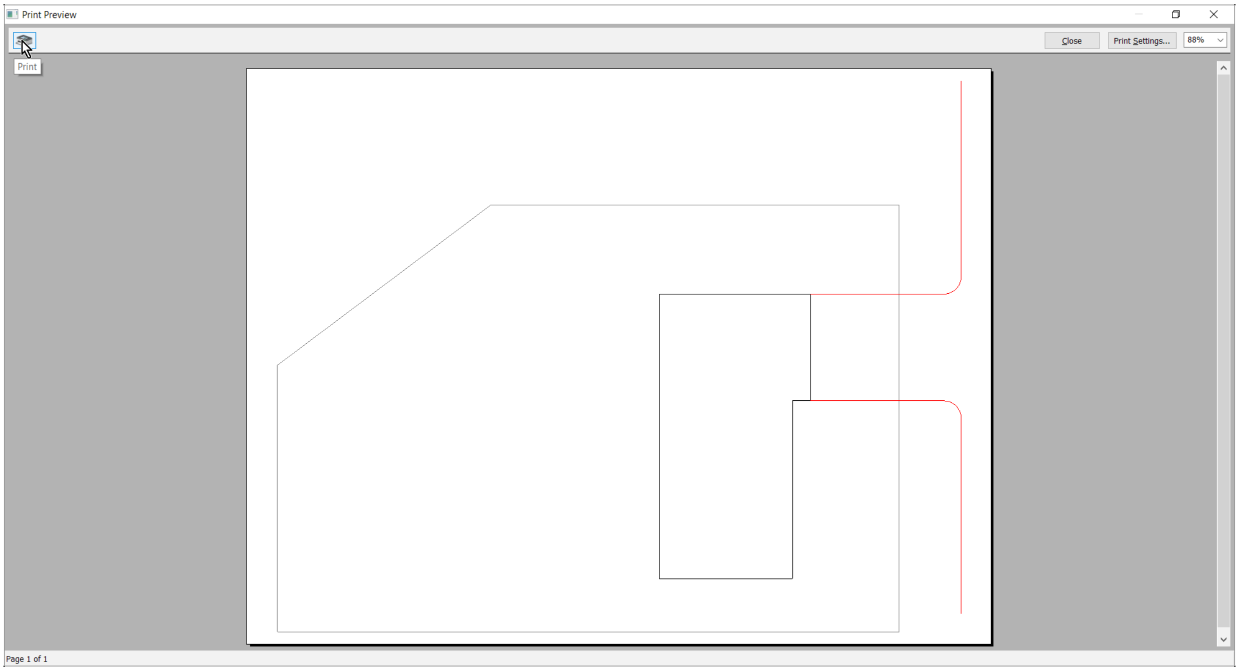
☑ Center on page

X: 0.0000 inches

Y: 0.0000 inches

Turn on the **Center on Page** option to have the drawing nicely centered on the paper.

3. To check that the drawing fits the paper, click the **Preview** button. The drawing appears simulated on a sheet of paper.



4. If the drawing looks alright to you, then click **Print** to print the drawing.
   (If, however, the preview shows a problem, click **Close** to exit, and return to the Print dialog box to adjust settings.)

BricsCAD sends the drawing to the printer, and it should appear a few seconds later looking exactly like the preview.

SOLVING PRINTING PROBLEMS

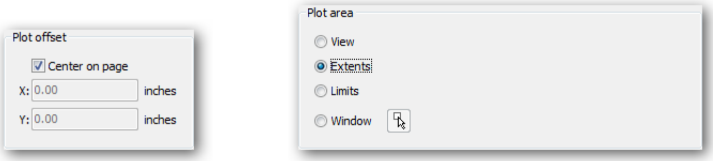
If the printer does not produce the drawing correctly, here are some items to check:

- Check in the Print dialog box that...

  ...drawings are centered on the page

  ...and check that the Extents of the drawings is printed.

  These two settings ensure that *all* of the drawing will be printed on the paper.



Plot offset

☑ Center on page

X: 0.00 inches

Y: 0.00 inches

Plot area

○ View

● Extents

○ Limits

○ Window

Left: *Print will be centered on page;* ***right:*** *Print will be made to drawing extents.*

- Check the printer:

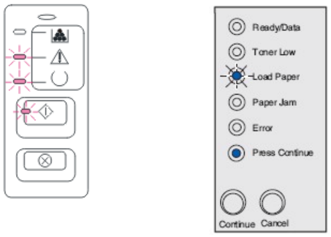
  Is it turned on?

  Does it have paper?

  Is the paper not jammed?

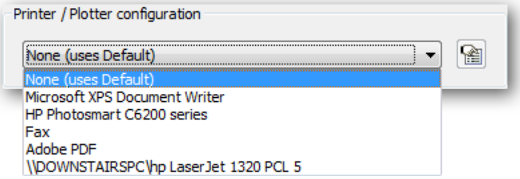
  Are the toner or ink cartridges not low”

  Are there any warning messages or lights?



Left and right: *Warning lights from various printers.*

- Check that the BricsCAD is printing to the correct printer. The operating system sometimes mistakenly lists names of printers that are not necessarily accessible by the computer.



Printer / Plotter configuration

None (uses Default)

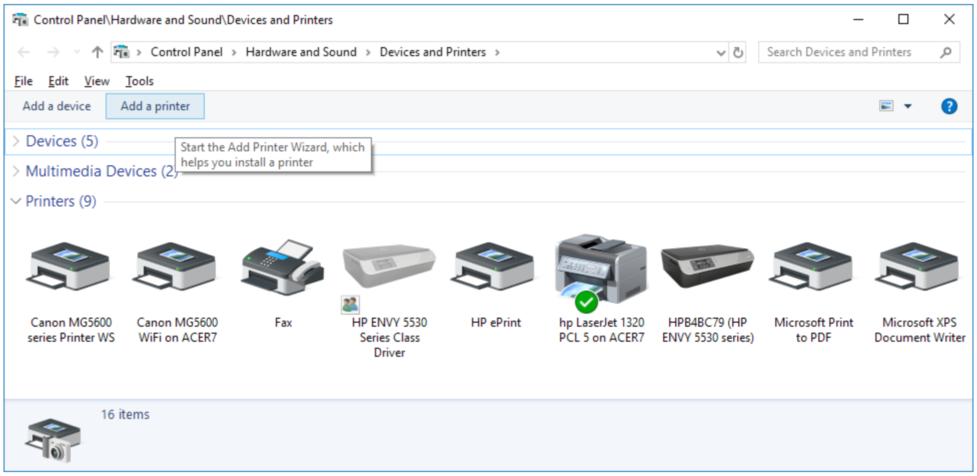
None (uses Default)
Microsoft XPS Document Writer
HP Photosmart C6200 series
Fax
Adobe PDF
\\DOWNSTAIRSPC\hp LaserJet 1320 PCL 5

- In some cases, the *spooler* acts up in Windows. The spooler is software that feeds data to the printer, allowing you to continue working with BricsCAD while the printer works in the background. Sometimes, the spooler plugs up, preventing the printer from receiving the data.

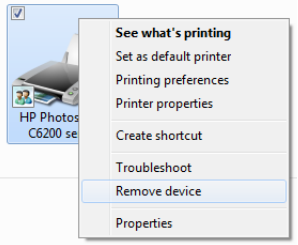
  To fix the problem, follow these steps:

  1. Click **Start**, and then **Devices and Printers** in Windows 7. (In Windows 8.x and 10, click the **Start** button, and then type “printers.” Choose **Devices and Printers**.)

2. In the window, click **Add a Printer**. Install the *same* printer as the one that is giving you problems.


Control Panel\Hardware and Sound\Devices and Printers
Control Panel › Hardware and Sound › Devices and Printers ›
Search Devices and Printers
File Edit View Tools
Add a device
Add a printer
Start the Add Printer Wizard, which helps you install a printer
Devices (5)
Multimedia Devices (2)
Printers (9)
Canon MG5600 series Printer WS
Canon MG5600 WiFi on ACER7
Fax
HP ENVY 5530 Series Class Driver
HP ePrint
hp LaserJet 1320 PCL 5 on ACER7
HPB4BC79 (HP ENVY 5530 series)
Microsoft Print to PDF
Microsoft XPS Document Writer
16 items

3. You now have two drivers for the same printer. Remove the first printer: right-click its icon, and then choose **Remove Device**.



4. Printing should now work.

Smudgy Prints with Bad Colors with Inkjets

Most printers use inkjet technology, which offers additional problems over laser printers. Here are solutions to some problems:

- When the prints come out smudgy, with ink that rubs off on your hand, the paper may have been put in the cartridge upside-down. The print side contains clay that absorbs ink; the underside does not.
- When prints of renderings or photographs have no detail in dark areas, this means that the black cartridge is low on ink. Cartridges low on ink do not squirt enough ink on the paper.
- When color prints are predominantly one color, this means that one or more colors of a multi-color cartridge are empty.

GENERATING PDFs OF DRAWINGS

The PDF format from Adobe is a popular way to share drawings *electronically.* You can send them by email or post them to Web sites, without anyone needing CAD software to view them.

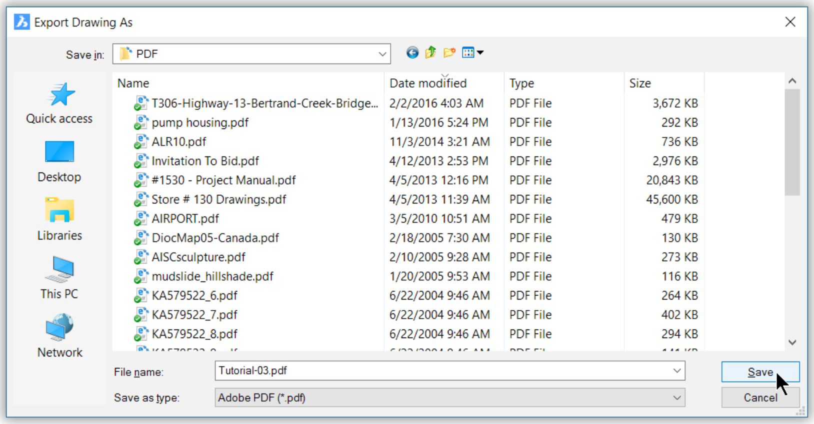
Another benefit: when drawings are saved in PDF format, they can be viewed but not altered. (Well, they can, but only with PDF editing software, which is not that common.)

To save drawings in PDF format in BricsCAD, follow these steps:

1. Enter the **ExportPDF** command.

```
: exportpdf
```

2. Notice the Export Drawing As dialog box. You can change the folder in which to save the file through the "Save In" field, as well as to change the file name ("File name"), such as *Yard-revision-1*.



Save in: PDF

Name	Date modified	Type	Size
T306-Highway-13-Bertrand-Creek-Bridge...	2/2/2016 4:03 AM	PDF File	3,672 KB
pump housing.pdf	1/13/2016 5:24 PM	PDF File	292 KB
ALR10.pdf	11/3/2014 3:21 AM	PDF File	736 KB
Invitation To Bid.pdf	4/12/2013 2:53 PM	PDF File	2,976 KB
#1530 - Project Manual.pdf	4/5/2013 12:16 PM	PDF File	20,843 KB
Store # 130 Drawings.pdf	4/5/2013 11:39 AM	PDF File	45,600 KB
AIRPORT.pdf	3/5/2010 10:51 AM	PDF File	479 KB
DiocMap05-Canada.pdf	2/18/2005 7:30 AM	PDF File	130 KB
AISCsculpture.pdf	2/10/2005 9:28 AM	PDF File	273 KB
mudslide_hillshade.pdf	1/20/2005 9:53 AM	PDF File	116 KB
KA579522_6.pdf	6/22/2004 9:46 AM	PDF File	264 KB
KA579522_7.pdf	6/22/2004 9:46 AM	PDF File	402 KB
KA579522_8.pdf	6/22/2004 9:46 AM	PDF File	294 KB

File name: Tutorial-03.pdf

Save as type: Adobe PDF (*.pdf)

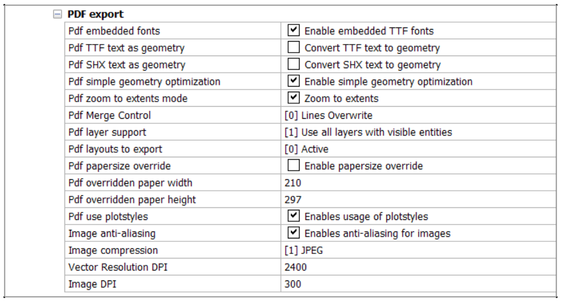
Click **Save**.

You can open the PDF file in Acrobat Reader or another program that displays *.pdf* files.

Specifying PDF Output Options

You control the way PDF files are produced by the Settings dialog box. The quick way to access PDF-specific options is with the **PdfOptions** command: it opens the dialog box at the section for PDF Export options, conveniently enough

Here is an explanation of the settings available, along with the default values in parentheses:



PDF export

Pdf embedded fonts	☑ Enable embedded TTF fonts
Pdf TTF text as geometry	☐ Convert TTF text to geometry
Pdf SHX text as geometry	☐ Convert SHX text to geometry
Pdf simple geometry optimization	☑ Enable simple geometry optimization
Pdf zoom to extents mode	☑ Zoom to extents
Pdf Merge Control	[0] Lines Overwrite
Pdf layer support	[1] Use all layers with visible entities
Pdf layouts to export	[0] Active
Pdf papersize override	☐ Enable papersize override
Pdf overridden paper width	210
Pdf overridden paper height	297
Pdf use plotstyles	☑ Enables usage of plotstyles
Image anti-aliasing	☑ Enables anti-aliasing for images
Image compression	[1] JPEG
Vector Resolution DPI	2400
Image DPI	300

What PDF Options Mean

Here is what the PDF output options mean (with default values shown in brackets):

PDF Embed Fonts (on) embeds TTF font files in the PDF file:

- **PDF TTF Text as Geometry** (off) converts text made with TTF fonts into geometry (lines and arcs)
- **PDF SHX Text as Geometry** (off) converts text made with SHX fonts into geometry

PDF Simple Geometry Optimization (on) reduces the file size by combining individual line segments into polylines, and uses Bezier control points to define splines. This option does not affect the drawing, only the resulting PDF file.

PDF Zoom to Extents Mode (on) exports the drawing unscaled, so that its extents fit the page

PDF Merge Control (0) determines the look of overlapping lines:

- **0** Overwrite; the topmost line obscures lines underneath it
- **1** Merge; overlapping lines are semi-transparent

PDF Layer Support (1) determines whether layers are included:

- **0** Exclude layers; export entities with no layer information (this is a form of security)
- **1** Export layers, except those turned off (default setting)
- **2** Export all layers, including those turned off and frozen

PDF Layouts to Export (1) specifies which layouts should be included in the PDF file:

- **0** Only the current "active" layout; this is like plotting a single layout (default setting)
- **1** All layouts; the PDF file becomes a multi-page PDF, with each layout on its own page

PDF Papersize Override — determines whether the PDF page should be of a custom size:

- **PDF Overridden Paper Width** (210) specifies the width of the PDF page in millimeters
- **PDF Overridden Paper Height** (297) specifies the height in millimeters

PDF Use Plotstyles (on) determines whether the drawing is exported to PDF format using named plot styles; named plot styles must exist in the drawing

Image Anti-Aliasing (on) applies anti-aliasing (smoothing) to images requiring upscaling for export to PDF format

Image Compression (JPEG) specifies whether to compress raster images in drawings; 0 = off.

Vector Resolution (2400) specifies the resolution of vector graphics.

(NEW IN V20) Options related to PRC export mode were removed from BricsCAD V20. PRC is short for "Product Representation Compact."

To export a 3D model as a 3D PDF file, you need to purchase and install the optional Communicator add-on, which reads and writes many 3D file formats. Use the Export command's "3D PDF File" option."

———

Congratulations! You've drafted your very first drawing using a computer. You can save the drawing as a memento of your introduction to computer-aided drafting.

In the next lesson, you learn how to add details to drawings, such as hatching and symbols (blocks).

Notes

CHAPTER 5

Adding Details to Drawings

So far, you've learned how to draw lines that represent the outlines of the lot, house, and driveway. Now it's time to add details, such as a lawn, some trees, and a pond.

In this chapter, you learn to use BricsCAD's intermediate commands like those that create ellipses, place hatch patterns, make arrays, and draw offset copies.

IN THIS CHAPTER

- Drawing circles and ellipses
- Editing with grips
- Applying hatch patterns to areas
- Arraying entities
- Making parallel offsets of entities
- Creating symbols (blocks)
- Performing real-time zooming and panning

KEY TERMS IN THIS CHAPTER

Drag — describes holding down the left mouse button while moving selected entities

Grip — describes the small square that indicates the editing points on a selected entity

Grips editing — refers to selecting entities, and then applying editing commands

Hatch pattern — displays a repeating pattern that indicates the material of entities

Palette or **bar** — describes a window that provides continuous information about BricsCAD

Real time — refers to an action in BricsCAD that occurs as the same time as you move the mouse

USEFUL ABBREVIATIONS

L	last (used by entity selection)
W	window (used by entity selection and zooming)

NEW COMMANDS

Command	Aliases	Menu Bar	Ribbon Tab
ArrayClassic	ar	Modify | 2D Array	Change | Arrange | 2D Array
Block	b	Tools | Create Block	Tools | Blocks | Create Block
Circle	c	Draw | Circle	Draw | Draw | Circle
Ellipse	el	Draw | Ellipse	Draw | Draw | Ellipse
Hatch	h	Draw | Hatch	Draw | Hatch | Hatch
Insert	i	Insert | Block	Insert | Blocks | Insert Block
Offset	o	Modify | Offset	Change | Modify | Offset
Pan	p, -p	View | Pan	View | Navigate | Real-time Pan
PEdit	pe	Modify | Edit Polyline	...
RtZoom	...	View | Realtime | Realtime Zoom	View | Navigate | Real-time Zoom
Zoom Window	z w	View | Zoom | Zoom Window	View | Zoom | Zoom Window

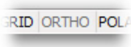
Dividing the Lot

The back of the yard has lawn and garden areas. For this tutorial, you draw the boundary between the two areas with a polyline using the **PLine** (short for "polyline) command, and then smooth it with the **PEdit** (polyline edit) command.

If BricsCAD is not running, start it now. If you did not complete the previous lesson, open the *tutorial-04.dwg* file. (If you're doing the tutorials in imperial units, open *Imp-tutorial-04.dwg*.) You can download them from my public cloud folder at https://my.pcloud.com/publink/show?code=XZn98mkZ7T7f1pynxJYb7vDyVl6DwyqK1Ysk.

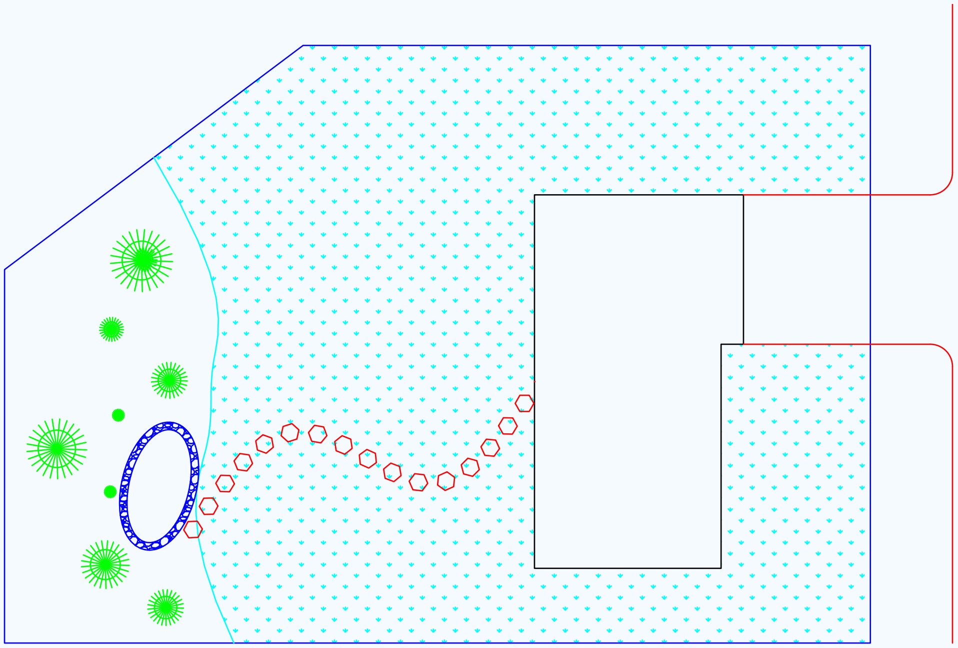
Before starting to draw, first take these steps:

1. You won't be drawing at right angles, so turn off ortho mode by clicking the **ORTHO** button on the status bar. It should look gray to indicate it is turned off.



THIS CHAPTER'S DRAWING

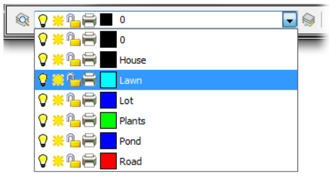
By the end of this lesson, your drawing should look similar to this one:



Tutorial-05 drawing file

2. Change the working layer to **Lawn**; its color is cyan (light blue). There are several ways to do this:

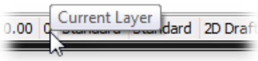
   - In the toolbar or ribbon interfaces, choose “Lawn” from the Layers droplist (shown below).



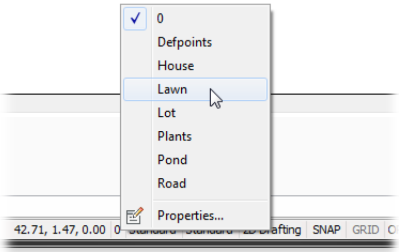
   - Or, from the Properties panel, choose “Lawn” from the Layers droplist

   - Or, open the Layers panel, and then click the **Current** field next to “Lawn”

   - Or in the status bar, right-click the current layer field...



   ... and then choose “Lawn” from the list



   - Or at the command prompt, use my favorite: enter the **CLayer** system variable (short for “current layer”). This is a quick way to change to another layer when you know the name. In fact, this is the method used by the status bar. Enter **clayer**, and then the name of the layer, **lawn**:

```
: clayer
New current value for CLAYER <"0">: lawn
```

   Now the drawing is ready for you to draw the boundary between the lawn and the garden .

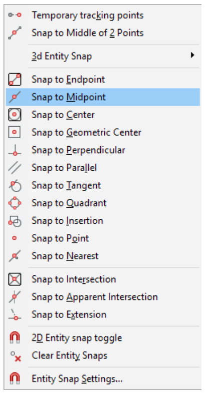
3. To draw the polyline, start the **PLine** command.

```
: pline
```

4. To determine where the polyline starts, you use an entity snap mode. But this time, you access entity snap in a different manner from last chapter:

   a. Hold down the **Shift** key on the keyboard.

   b. Press the right mouse button, and then let go of the **Shift** key. Notice the new menu that pops up on the screen. This is called a “shortcut menu.” The menu lists all of BricsCAD’s entity snap modes.

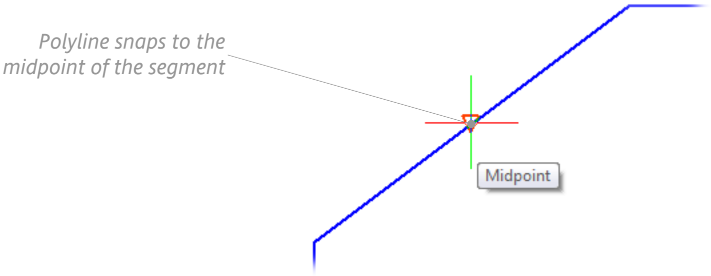


c. Move the cursor to **Snap to Midpoint**, and then click. On the command line, notice that BricsCAD reports that MIDpoint esnap is indeed activated:

```
Start of polyline: _midpoint
```

5. Move the cursor to *any* point on the diagonal portion of the lot line, and then select the line.

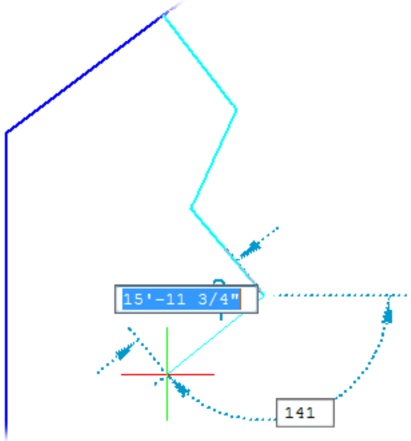
```
Snap to midpoint of: (Pick a point on the diagonal lot line)
```



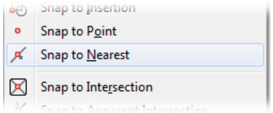
The MIDpoint esnap forces to BricsCAD snap to the precise midpoint of the lot's diagonal line.

6. Continue with the PLine command by moving your way down toward the bottom of the yard line, picking a few points along the way. Space them at roughly 3m to 6m intervals. (In imperial units, that's 10- to 20-foot intervals.)

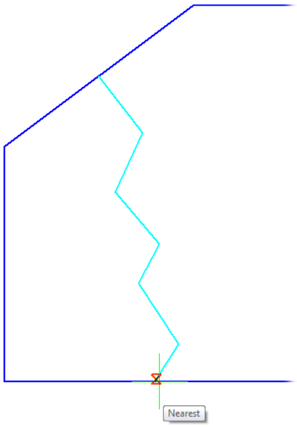
TIP If you are not sure how far 3m or 6m is (10ft or 20ft), turn on **DYN** on the status bar, and then watch the distance dynamically, as shown below:



7. When you get to the bottom lot line, you need to bring the polyline to a precise ending. Press **Shift**+right mouse button to again display the shortcut menu. This time, select **Nearest** entity snap.



8. Pick anywhere the line crosses the cursor's aperture box. BricsCAD snaps the polyline precisely to the lower lot line.



9. Press **Esc** to end the PLine command.

Here is the reason you created the boundary as a polyline: you now use the **PEdit** command to easily modify the crooked segments into a smooth flowing curve.

SMOOTHING POLYLINES

Command	PEdit
Alias	PE
Menu Bar	Modify | Edit Polyline
Mouse	Double-click a polyline

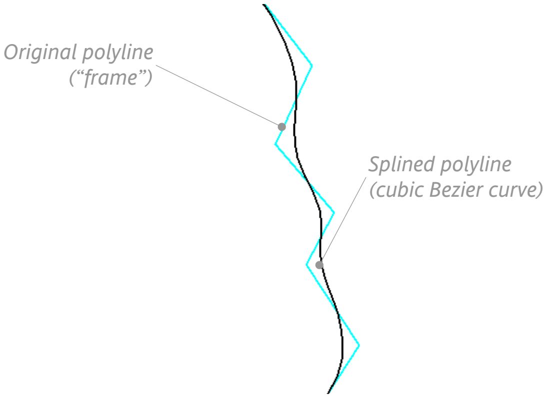
The PEdit command's purpose is to change the look of polylines. It has many, many options; don't worry, this time around you use just one of them.

1. To edit the polyline, begin the **PEdit** command (short for "polyline edit"):

```
: pedit
```

2. Pick the polyline.

`Select polyline to edit` [Multiple]: *(Pick the polyline, or type* **L** *to choose the last-drawn entity)*


Original polyline ("frame")
Splined polyline (cubic Bezier curve)

QUICK SUMMARY OF THE PEDIT COMMAND

The PEdit command edits polylines:

```
: pedit
Select polyline to edit [Multiple]:
Edit polyline [Edit vertices/Close/Decurve/Fit/Join/Linetype mode/Reverse direction/
Spline/Taper/Width/Undo] <eXit>:
```

Multiple — turns lines, arcs, circles, and/or splines into a single polyline.

Edit vertices — edits width and position of individual segments; inserts and removes vertices.

Close / Open — closes (or opens) an open (or closed) polyline by drawing (or removing) a segment between the starting and ending vertices; this Close prompt is displayed when an open polyline is being edited, the Open prompt when closed polyline is edited.

Decurve — reverts a curve-fit or splined polyline to its original form.

Fit — applies a curve fit to the polyline.

Join — joins this polyline with another polyline, line, or arc; entities must be connected to form a single polyline (best done with entity snaps); this option fails when gaps exist between the entities to be joined.

Linetype mode — determines if linetypes fit over the entire length of the polyline, or between vertices.

Reverse direction — reverses the direction of the polyline.

Spline — applies a Bezier spline to the polyline.

Taper – applies a varying width to segments.

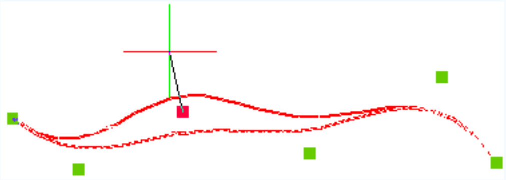
Width — applies a uniform width to all segments making up the polyline.

Undo — undoes the last polyline editing operation.

eXit — exits the command.

While you can use the **PEdit** command's **Edit vertices** option to change the shape of polylines, it is much easier to edit with grips, like this:

1. Select a polyline.



2. Notice the green squares, called "grips." Pick one; it turns red.
3. Drag the red grip. Notice that the polyline changes its curvature.
4. When done editing the polyline, press **Esc**.

3. Use the **Spline** option to smooth the jagged lines into a flowing curve, as follows:

   ```
   Edit polyline [Edit vertices/Close/Decurve/Fit/Join/Linetype mode/Reverse direction/
   Spline/Taper/Width/Undo] <eXit>: s
   ```

   The straight lines disappear and are replaced by a smooth curve. (Technically, BricsCAD redrew the line segments as a *cubic Bezier curve* based on the polyline as the frame.)

4. Exit the PEdit command by pressing **Enter**:

   ```
   Edit polyline [Edit vertices/Close/Decurve/Fit/Join/Linetype mode/Reverse direction/
   Spline/Taper/Width/Undo] <eXit>: (Press Enter)
   ```

Grips Editing

To edit the polyline, you took two steps in this order: (1) you started the PEdit command, then (2) you selected the polyline to edit. BricsCAD can do this in reverse — select an entity, and then edit it — which sometimes is a more convenient way to approach editing.

There are two ways to do this:

- Grips editing — click the entity, and then move grips found on it
- Double-click editing — double-click the entity, which accesses a command best suited for editing it

Grips editing is usually the fastest and most direct way to edit entities in CAD drawings — sometimes. I say "sometimes," because not all editing commands lend themselves to it. In the case of entities like polylines, it does.

Here you use grips editing to change the shape of the splined polyline that you drew to separate the garden from the lawn.

TIP If you cannot see the entity you want to edit, hold down the roller or middle button on the mouse and then move it. This pans the drawing. Alternatively, use the **Pan** command or a scroll bar to move the entity into view.

1. First, enlarge the view of the splined polyline with the **Zoom Window** command:

   a. Enter the **Zoom** command:

   ```
   : zoom
   ```

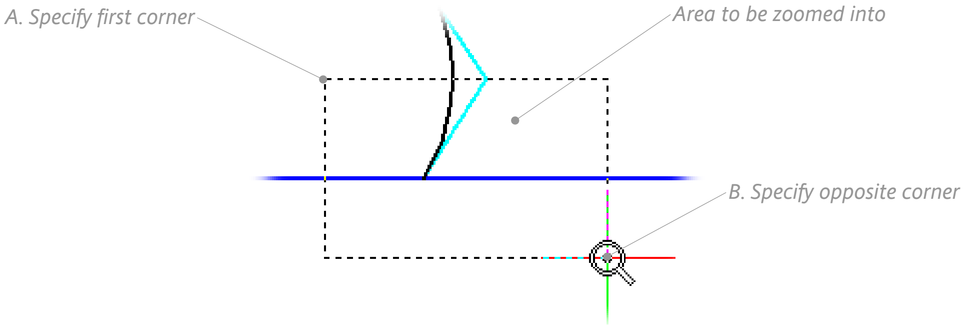
   b. Specify the **Window** option by entering 'w':

   ```
   Zoom [zoom In/zoom Out/All/Center/Dynamic/Extents/Left/Previous/Right/Scale
   (nx/nxp)/Window/OBject] <Scale (nX/nXP)>: w
   ```

   c. The Window option requires you to pick a rectangular area on the screen to magnify.

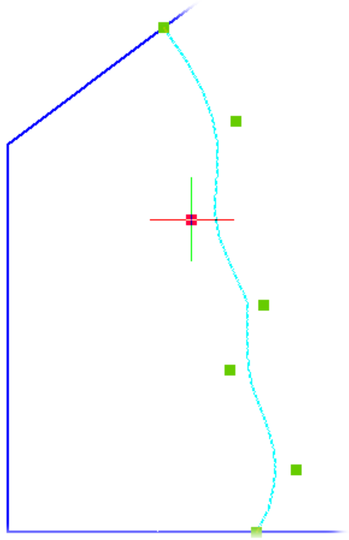
   ```
   Specify first corner: (Pick a point)
   Specify opposite corner: (Pick another point)
   ```

When you pick points for the first and other corners, you specify the two opposite corners of a rectangle, as shown in the following figure.



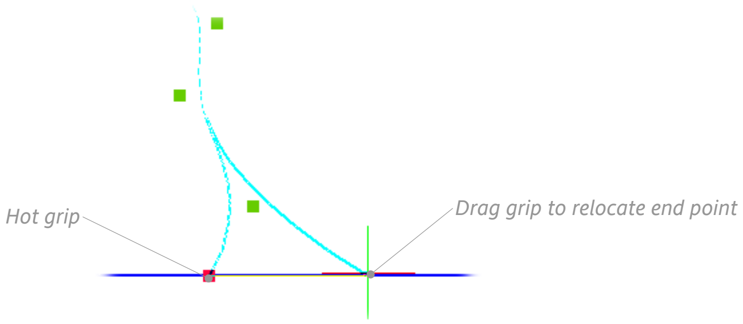
Now that the working area is larger, you can edit the polyline with grips. In an earlier lesson, I pointed out the small square at the center of the crosshair cursor. It is called the "pickbox." When BricsCAD displays the pickbox, you can select entities without any command being active.

2. Use the pickbox to pick the polyline. Notice that the polyline changes its look from solid to dashed, and that small green squares appear along the splined polyline. These green squares are called "grips," because they let you *grip* entities. (The green grips that are not on the spline are at the *vertices* of the straight polyline segments you originally drew, before you splined it.)



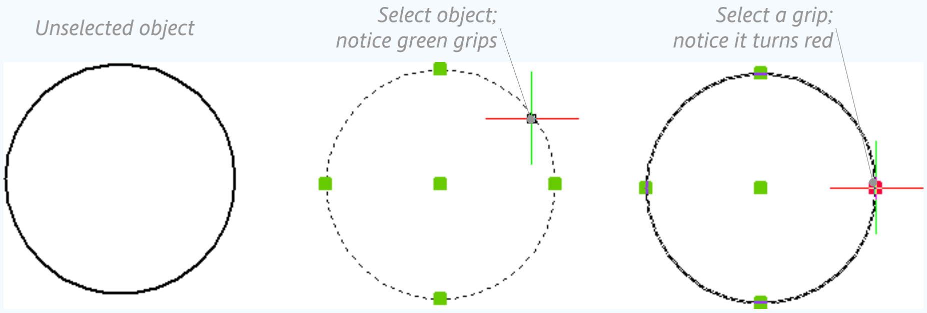
3. Pick the green grip at the very bottom end of the polyline. BricsCAD prompts:

```
New location for control point:
```

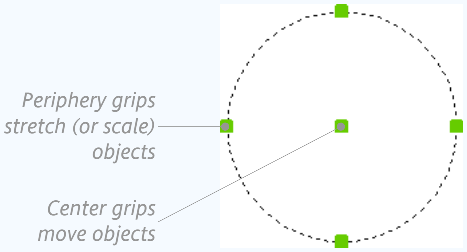


QUICK SUMMARY OF EDITING WITH GRIPS

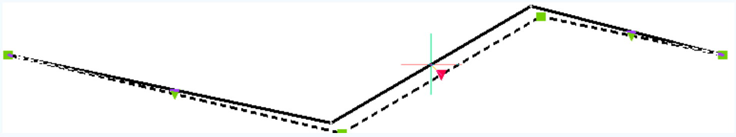
Grips editing is interactive: you select an entity (green grips appear), select one of the grips (grip turns red), and then manipulate the red grip to modify the entity.


Unselected object
Select object; notice green grips
Select a grip; notice it turns red

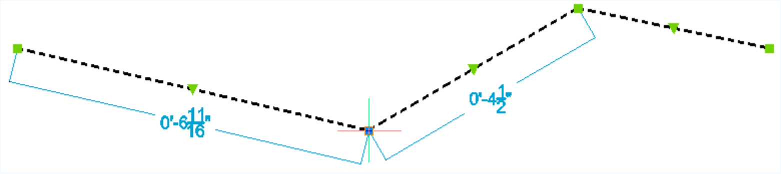
Most grips look the same (squares), and so it can be hard to determine their function — until you start to drag one of them. Typically, an interior grip moves an entity, while grips on the periphery stretch or scale the entity.


Periphery grips stretch (or scale) objects
Center grips move objects

Some entities sport triangular entities, such as polylines: this grip moves the segment.



When **DYN** is turned on, then dynamic dimensions appear during grips editing:

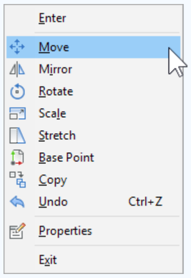


...continued

...continued

GRIPS EDITING COMMANDS

During grips editing, BricsCAD makes available these editing commands: Stretch, Move, Rotate, Scale, Mirror, and Copy. (The six operate identically to their non-grips equivalents.) To see the commands, you need to right-click for a shortcut menu, or eles press the **Spacebar** repeatedly.



Each time you press the Spacebar, BricsCAD displays another set of prompts:

```
** STRETCH **
<Stretch to point>/Base point/Copy/Undo/eXit: (Press spacebar)
** MOVE **
<Move to point>/Base point/Copy/Undo/eXit: (Press spacebar)
** ROTATE **
<Rotation angle>/Base point/Copy/Undo/Reference/eXit: (Press spacebar)
** SCALE **
<Scale factor>/Base point/Copy/Undo/Reference/eXit: (Press spacebar)
** MIRROR **
<Second point>/Base point/Copy/Undo/eXit: (Press spacebar)
```

After ** MIRROR **, BricsCAD repeats the cycle. The other editing options are:

Base point — specifies a base point other than the hot grip.

Copy — copies the entity, using the current grip as the base point.

Reference — changes the reference point from the current hot grip.

Undo — undoes the last editing operation.

eXit — exits non-modal editing mode (or press ESC).

4. Enter the **NEAr** entity snap mode, and then move the cursor. As you do, the last segment of the polyline curves and arcs to follow you. (The NEAr entity snap ensures the polyline ends precisely at the lot line.)

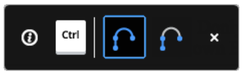
```
New location for control point: nea
```

5. Click along the lot line where you want the end of the polyline to move to. If you wish, feel free to interactively reshape the rest of the polyline, segment by segment.

6. When you are finished reshaping the polyline, press **Esc** twice to exit grips editing.

7. Enter the **Zoom Previous** command to see the entire drawing again.

When you move the cursor over a polyline, the Hotkey widget appears. It gives you the option to edit the entire polyline or just the segment over which the cursor is currently hovering — using grips editing. Press the **Ctrl** key to switch the editing type.



The Hotkey widget also appears for other entities.

Editing by Double-clicking

BricsCAD also lets you edit entities by simply double-clicking them. This is a fast way to do it, because you don't need to enter a command — or even remember the correct name of the command! The table below lists the entities that react to double-clicking.

Entity	Command Executed	Notes
	by Double-clicking	
Array	ArrayEdit	Edits only associative arrays
Attribute Definition	DdEdit	
Attribute Block	EAttEdit	
Block	BEdit	Opens block in Block Editor environment
Camera	-View	(NEW IN V20)
Dimension	DdEdit	Edits only text of dimensions
Hatch	HatchEdit	
Image	ImageAdjust	Adjusts brightness, contrast, fade of images
Polyline and Lwpolyline	PEdit	
Multileader	DdEdit	Edits only text of mleaders
Section	ClipDisplay	Adjusts clipping planes of sections
Spline	SplinEdit	
Text	DdEdit	
Tolerance	DdEdit	Edits only text tolerances
Viewport	VpMax	(NEW IN V20)
XRef	RefEdit	Opens xref in RefEditor environment

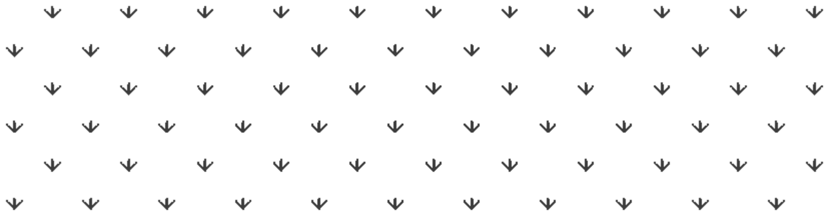
For all entities not listed in the table, BricsCAD opens the Properties panel when you double-click them. You use the Double Click Actions section of the **Customize** command's Mouse tab to change which command is activated when specific entities are double-clicked; see our book *Customizing BricsCAD*.

HATCHING THE LAWN

Command	Hatch
Aliases	H, BH, BHATCH
Menu Bar	Draw | Hatch
Ribbon	Draw | Hatch | Hatch
Toolbar	Draw | Hatch

You created a boundary between the lawn and garden, but how do you show the difference between what's grass and what's dirt? One way is to add a repeating pattern that identifies areas; the repeating pattern is called *cross-hatching* or a *hatch pattern*. It is a shorthand used by drafters to indicate material, just like the purpose of linetypes is to identify lines.

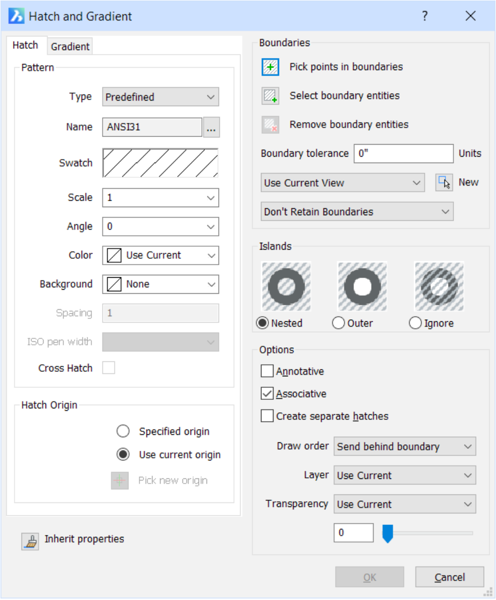
In BricsCAD, hatching is done with the **Hatch** command. In this tutorial you hatch the lawn with the symbol used by drafters for grass, shown below.



1. First, turn off the grid, since its lines can interfere with seeing the hatch pattern: on the status bar, click **GRID** so that the word turns gray.
2. Enter the **Hatch** command to start the Hatch command:

```
: hatch
```

Notice the Hatch and Gradient dialog box:



Hatch and Gradient

Hatch | Gradient

Pattern

Type: Predefined

Name: ANSI31

Scale: 1

Angle: 0

Color: Use Current

Background: None

Spacing: 1

ISO pen width

□ Cross Hatch

Hatch Origin

○ Specified origin

● Use current origin

Pick new origin

Inherit properties

Boundaries

Pick points in boundaries

Select boundary entities

Remove boundary entities

Boundary tolerance: 0" Units

Use Current View | New

Don't Retain Boundaries

Islands

● Nested ○ Outer ○ Ignore

Options

□ Annotative

☑ Associative

□ Create separate hatches

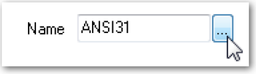
Draw order: Send behind boundary

Layer: Use Current

Transparency: Use Current

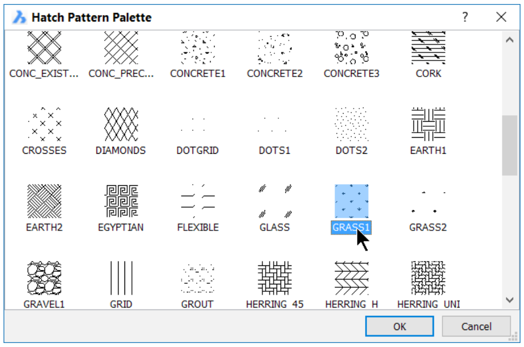
0

3. To find the grass hatch pattern, click the … button to the right of "Name."



Name ANSI31

4. BricsCAD displays the Hatch Pattern Palette dialog box. The patterns are listed alphabetically; look for the pattern named "Grass1."

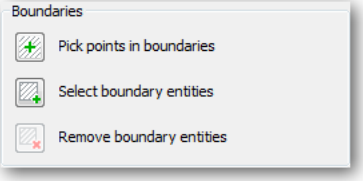


5. Select **Grass1**, and then click **OK** to return to the Hatch and Gradient dialog box. Notice that **Name** changes to "Grass1."

6. Ensure the **Scale** is set to **1** for metric drawings. (For imperial drawings, change the **Scale** to **25**.) If the scale factor were too small, BricsCAD would refuse to draw the hatch pattern and complain:

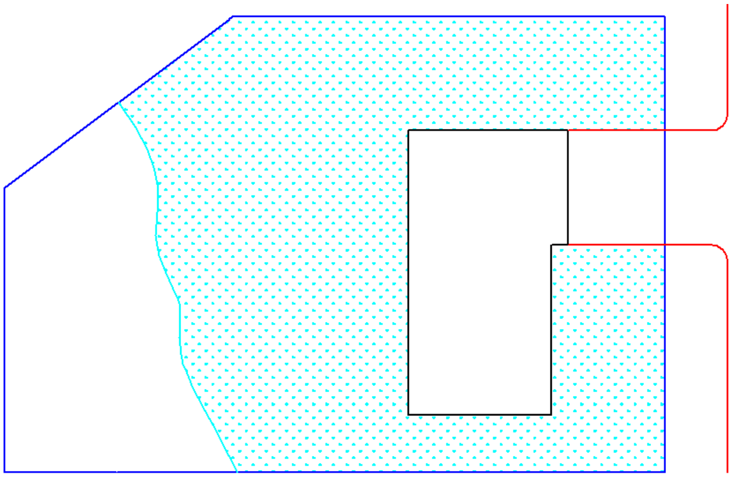
```
Hatch spacing too dense, or dash size too small.
Please increase pattern scale or MAXHATCH value.
```

7. BricsCAD has a useful feature that searches a contiguous area, no matter how many different borders the area has. To employ this feature, click the **Pick points in Boundaries** button.



8. The dialog box disappears and BricsCAD prompts you to pick a point. Do so anywhere in the lawn area.

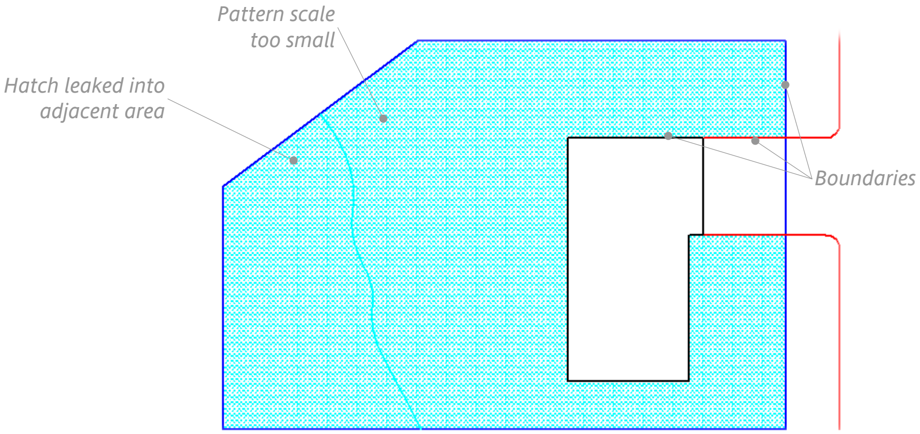
`Select a point to define a boundary or hatch area: (Pick a point inside the lawn area)`



Notice that BricsCAD immediately previews the hatch pattern for you, showing you that the scale factor is good enough. Note how precisely the hatch pattern is applied: it is clipped along boundaries automatically. Try doing a hatch pattern that neatly and that quickly by hand!

The hatch preview gives you a chance to correct errors, such as these two possible ones:

- The pattern floods incorrect areas — look out for patterns that leak into unwanted areas, or don't appear at all
- The pattern is applied at the wrong scale factor — look for one that is too large or too tightly spaced. When the scale is much too large, the hatch may seem invisible; when too small, the hatch pattern looks like a solid and can take a very time to display.



There is also a **-Hatch** command that operates at the command line. It is meant for use by macros and programming routines.

---

9. Press **Enter** to return to the Hatch and Gradient dialog box:

   ```
   Select a point to define a boundary or hatch area: (Press Enter)
   ```

   After you press **Enter**, the Hatch and Gradient dialog box reappears.
10. Click **OK** to exit it. Notice that the hatching is applied in the color of the layer.

With all this hard work on your drawing, it's a good idea to save the drawing to the computer's hard disk with the **Save** command... right now!

Modifying Hatching

Once hatch patterns are in place, they are not cast in stone. You can change them in several ways:

- Move **entities** that make up the boundary of a pattern, and the hatching automatically updates itself.
- Move, copy, and erase **hatch patterns**, because it acts like a block; it does not need to stay in one place.
- Click the hatch entity to bring up the **Properties** panel, and then change the color, pattern, scale factor, rotation angle, and so on.
- Double-click the hatch pattern to bring up the **Hatch Edit** dialog box, which looks identical to the Hatch and Gradient dialog box, but displays the settings of the current hatch pattern.

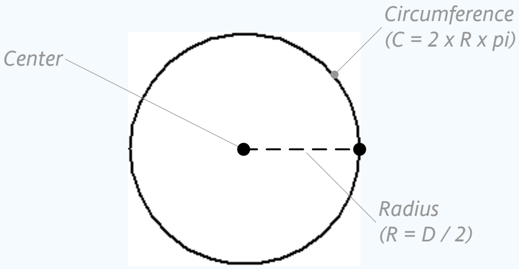
Creating Symbols (Blocks)

You've given the lawn its grass. Now it's time to add trees and shrubs to the garden. Instead of drawing complex things like trees with branches and leaves, landscape architects typically draw simple representations of them, such as a circle with radiating lines. This is known as a *symbol*, called a "block" by BricsCAD.

QUICK SUMMARY OF DRAWING CIRCLES

BricsCAD provides several methods for drawing circles. Use the one that suits your need the best.

```
: circle
Select center of circle or [2Point/3Point/TTR/Arc/Multiple]:
```

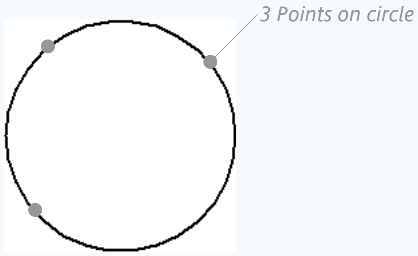


Center, Radius — pick the center point, and then specify the radius.

Center, Diameter — pick the center point, and then specify the diameter.

2-Point — pick two points to define the diameter.

3-Point — pick three points to define the circumference.



TTF — (Tangent, Tangent, Radius) pick two points of tangency to other entities, and then specify radius.

Arc — closes an arc to form a circle.

Multiple — repeats the command to draw more circles, until you press **Esc**.

And instead of drawing lines and circles over and over again, designers typically draw just one, and then repeatedly make copies of them. While you could use the Copy command, the most efficient method is to use the Block and Insert commands. **Block** creates a stencil-like object, of which you place multiple copies with the **Insert** command.

DRAWING CIRCLES

Command	Circle
Alias	C
Menu Bar	Draw | Circle
Ribbon	Draw | Draw | Circle
Toolbar	Draw | Circle

In this tutorial, you learn how to draw a single tree as a simplified symbol, and then turn it into *blocks* to be used over and over again.

1. Before starting to draw first tree, make sure the working layer is set to **Plants**. From a Layer droplist, select layer "Plants."
2. Next, draw a **0.15**-meter radius circle. Enter the **Circle** command, and then specify a radius of 0.15 units (in imperial units, **6"**):

```
: circle
Select center of circle or [2Point/3Point/TTR/Arc/Multiple]: (Pick a point anywhere in
garden area.)
Set Radius or [Diameter]: 0.15
```

When the radius is 0.15, the diameter is 0.3, so don't be surprised when BricsCAD draws the circle larger than you expect. (Recall that a radius is half the diameter.)

Zooming in Real Time

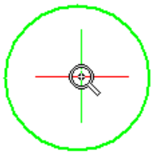
Command	RtZoom
Menu Bar	View | Realtime | Realtime Zoom
Ribbon	View | Navigate | Real-time Zoom
Shortcut	*roll the mouse wheel*
Toolbar	View | Real-Time Motion

Command	Regen
Menu Bar	View | Regen
Ribbon	View | View | Regen
Toolbar	View | Regen

The program lets you zoom in and out without needing to specify a zoom size or ratio. This is done with the mouse and the **RtZoom** command:

3. The 0.3m circle looks very small on the screen. The RtZoom command lets you see your work more clearly. The command name is short for "real time zoom."

```
: rtzoom
>>Press ENTER or Esc to complete, or right-click to display context menu....
```

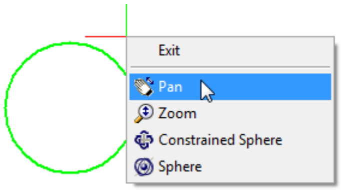


a. The cursor changes to a magnifying glass. Drag the cursor over the circle.

b. As you move the mouse forward and backward, BricsCAD dynamically increases and decreases the zoom. (As an alternative, if you have a mouse with a wheel, roll the wheel forward to zoom in.)

TIP To *drag* means to (a) hold down the left mouse button, (b) move the mouse, and then (c) let go of the mouse button.

4. If you find the tree going off the edge of the screen, right-click the drawing, and then choose **Realtime Pan** from the shortcut menu. This pans (shifts) the drawing in real time: as you drag the cursor, the tree-circle is moved back to the center of the drawing area.



5. When the tree-circle is at a satisfactory size, press **Esc** to get out of real-time pan-and-zoom mode:

```
>>Press ENTER or Esc to complete, or right-click to display context menu... (Press ESC)
```

If the circle looks like an octagon or diamond, select **View | Regen** to clean it up, as follows:

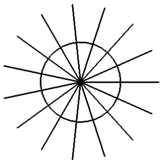
```
: _regen
```

Now that the circle looks rounder and larger, it is easier to work with.

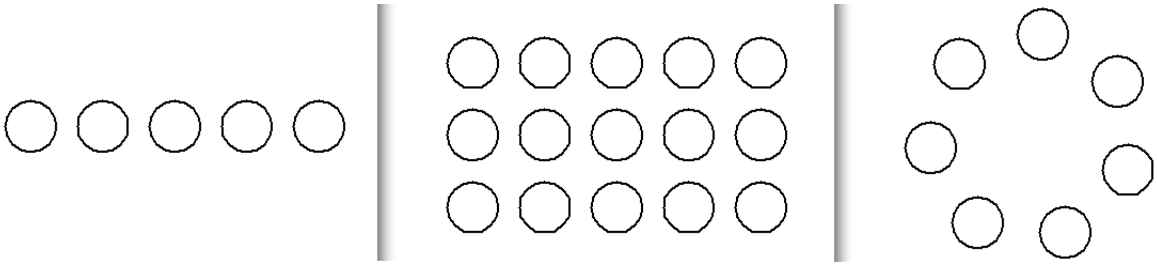
CREATING ARRAYS

```
Command    ArrayClassic
Menu Bar   Modify | Array | Array Dialog Box
```

With the circle in place representing the trunk, the tree branches can be added by an *array* of lines.



To create the array, use the **ArrayClassic** command to create the radiating lines. This command creates several types of arrays: linear, rectangular, and polar (circular). They are illustrated below.



Left to right: *Linear, rectangular, and polar arrays.*

(The Array command operates at the command line, but the ArrayClassic command displays a dialog box, which is easier to use!)

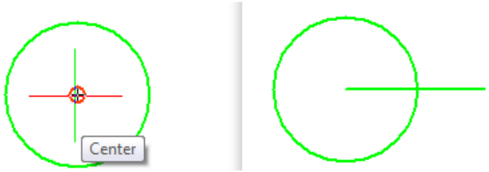
For this tutorial, you draw one line and then array it to create... more.

1. Enter the **Line** command:

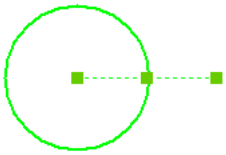
```
: line
Start of line or [Follow] <Last point>: cen
of (Pick center of circle)

Set end point or [Angle/Length/Undo]: (Pick point anywhere outside of circle)
Set end point or [Angle/Length/Follow/Close/Undo]: (Press Enter to end the command)
```

TIP To draw a perfectly horizontal or vertical line, hold down the **Shift** key. This action turns on ortho mode temporarily, for as long as you hold down the Shift key.

The **CENter** entity snap begins the line precisely at the center of the circle. The other end of the line extends beyond the edge of the circle.



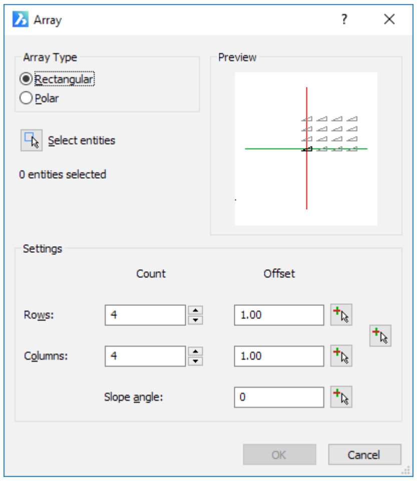
2. Select the line you just drew. Notice that it is highlighted.



3. Enter the ArrayClassic command:

: **arrayclassic**

Notice the Array dialog box.



Array

Array Type
● Rectangular
○ Polar

Select entities

0 entities selected

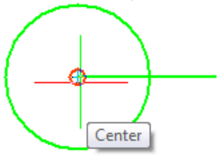
Preview

Settings

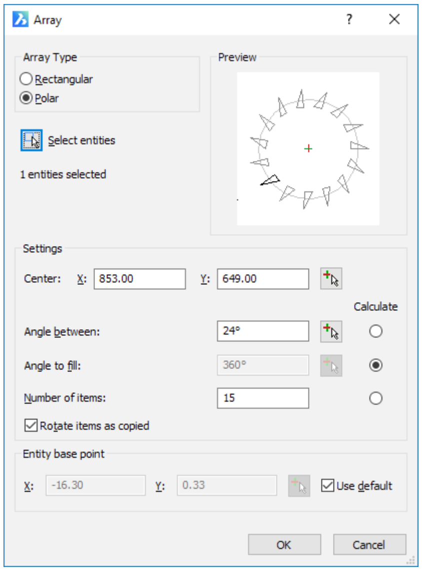
	Count	Offset
Rows:	4	1.00
Columns:	4	1.00
Slope angle:		0

4. Entering array parameters in this dialog box is just like filling out a form:

Array Parameter	Value
Array Type	**Polar**
Select Entities	Click **Select Entities,** and then choose the line

Center	Click **Pick Center Point,** and then choose the center of the circle like this:
	`Base/Center of polar array: cen`
	`Snap to center of: (Choose the circle)`
	

Number of items	**15**
Rotate items as copied	**Yes**



Array

Array Type
○ Rectangular
● Polar

Select entities

1 entities selected

Preview

Settings

Center: X: 853.00 Y: 649.00

Calculate

Angle between: 24° ○

Angle to fill: 360° ●

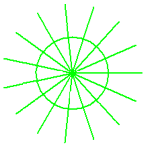
Number of items: 15 ○

☑ Rotate items as copied

Entity base point

X: -16.30 Y: 0.33 ☑ Use default

5. Ignore the other settings (leave them at their default values). Notice that the preview window gives you can idea of what the array will look like. Click **OK**.



Making Blocks

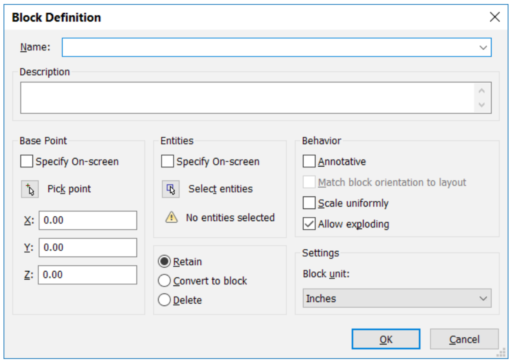
```
Command    Block
Alias      B
Menu Bar   Tools | Create Block
Ribbon     Insert | Blocks | Create Block
Toolbar    Tools | Block
```

CAD draws symbols more quickly and accurately than you can by hand. The key is to turn symbols into blocks, and then insert the blocks into drawings. In this section, you do this by adding tree blocks to the garden area.

1. To turn the tree symbol into a block, use the **Block** command:

   ```
   : block
   ```

   Notice that BricsCAD displays the Block Definition dialog box.



Block Definition

Name:

Description

Base Point

☐ Specify On-screen

Pick point

X: 0.00

Y: 0.00

Z: 0.00

Entities

☐ Specify On-screen

Select entities

No entities selected

● Retain

○ Convert to block

○ Delete

Behavior

☐ Annotative

☐ Match block orientation to layout

☐ Scale uniformly

☑ Allow exploding

Settings

Block unit: Inches

OK Cancel

2. Enter the following parameters:

Block Option	Value	Notes
Name	**Tree**	You can give the block any name you like, up to 255 characters long
Base point	⌖	Click **Pick Point** button, and then choose the center of the circle
Entities	▭	Click **Select Entities**, and then choose all 16
Convert to block	**On**	Converts the entities to a block when you click **OK**
Scale uniformly	**On**	Forces the circle to remain round

a. The *base point* is used later by the Insert command; this point is also called the “insertion point” — thia is the precise point where the block is inserted in the drawing.

The center of the tree symbol is a logical spot base point, so use CENter entity snap:

```
Insertion point for new block: cen
Snap to centerpoint of: (Pick the circle)
```

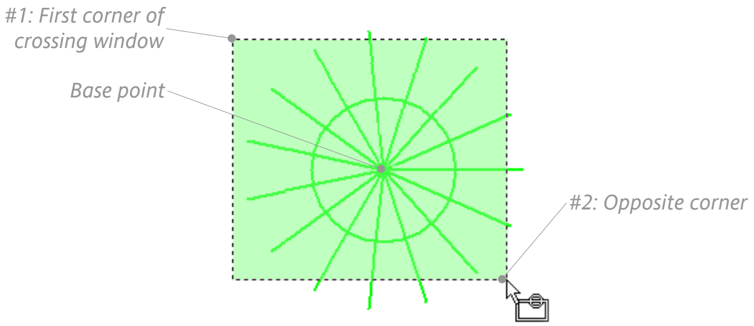
b. You need to tell BricsCAD which entities to turn into a block. BricsCAD lets you select entities by several different ways. So far, you have picked them one at a time with your mouse or with the L (last) option.

Just as you windowed the zoomed-in view, you can window the entities you want to select with the **W** option (short for “Window”).

```
Select entities for block: w
```

c. You pick the two corners of a rectangle that encompasses the circle and lines making up the tree symbol:

```
First corner of crossing window: (Pick point 1, shown in figure below)
Opposite Corner: (Pick point 2, shown in figure)

Entities in set: 30
Select entities for block: (Press Enter to end entity selection)
```



3. Click **OK**. BricsCAD records the tree symbol as a block in the drawing file. In the next section, you place many trees as blocks.
4. Use the **Zoom Extents** command to see the entire yard.

ADDING MANY MORE TREES (INSERT)

Command	Insert
Alias	I
Menu Bar	Insert | Block
Ribbon	Insert | Blocks | Insert Block
Toolbar	Insert | Block

With the bigger view in place, insert tree blocks in the drawing.

1. Start the **Insert** command:

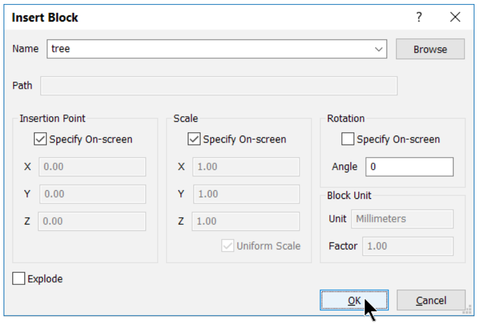
```
: insert
```

Notice that BricsCAD displays the Insert dialog box.

2. The **Tree** block name should be shown in the **Name** field.

You can ignore most of the dialog box, other than to ensure that the options are set as follows:

Insert Options	Value	
Insertion Point	Specify On-screen	✓(on)
Scale	Specify On-screen	✓(on)
Rotation	Specify On-screen Angle: **0**	(off)



Insert Block

Name: tree

Path:

Insertion Point
☑ Specify On-screen
X 0.00
Y 0.00
Z 0.00

Scale
☑ Specify On-screen
X 1.00
Y 1.00
Z 1.00
☑ Uniform Scale

Rotation
□ Specify On-screen
Angle 0

Block Unit
Unit Millimeters
Factor 1.00

□ Explode

3. Click **OK**. BricsCAD now prompts you in the command bar, because of the Specify On-Screen options that were turned on in the dialog box.

   The first item of business is to pick a spot for the block, a suitable spot in the garden area.

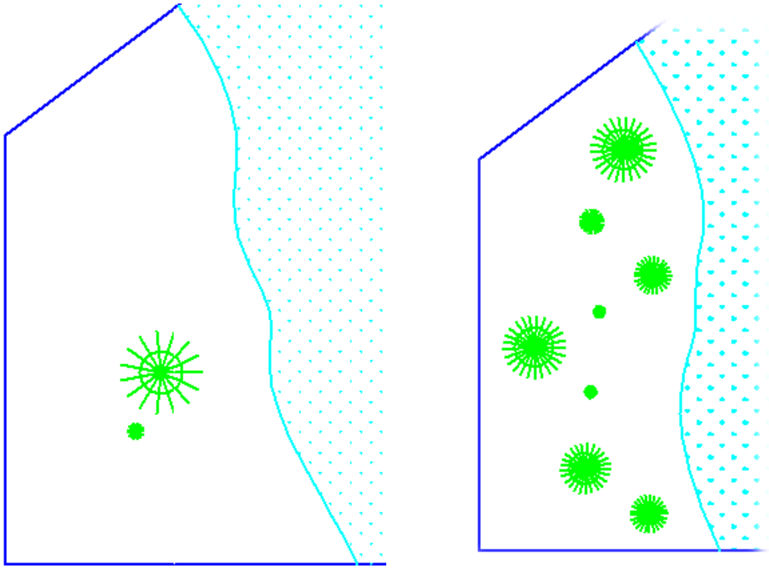
   ```
   Insertion point for block: (Pick a point anywhere in the garden area)
   ```

4. The other item is the scale, which draws the block larger or smaller than the symbol original size:

   ```
   Corner/XYZ/X scale factor <1.000000>: 2
   ```

   The x scale factor of 2 draws the block twice as large as the original. Because you drew the original symbol one meter in diameter, the newly inserted tree has a diameter of two meters. If you enter a scale factor smaller than 1, such as 0.5, then the block is drawn half-size.

   From this exercise, you can see that it makes sense to draw a symbol to *unit size* (to the nearest meter or foot), because that makes it easy to scale the block during insertion.



Left: *Two trees inserted at different scale factors*
Right: *Many more tree blocks inserted into the drawing*

5. Try a different way of repeating the **Insert** command. This time, right-click the drawing. Notice that the Insert command repeats, again displays the dialog box and prompts of the Insert command.
6. Add several more trees around the garden area using the **Insert** command and different scale factors, such as 2, 1.2, 0.6, and 0.4 (or 6.0, 4.0, 2.0, and 1.5 in imperial units).

   Use the **Mirror** command to double the number of trees.

Drawing the Pond

```
Command   Ellipse
Alias     EL
Menu Bar  Draw | Ellipse
Ribbon    Draw | Draw | Ellipse
Toolbar   Draw | Ellipse

Command   Offset
Alias     O
Menu Bar  Modify | Offset
Ribbon    Change | Modify | Offset
Toolbar   Modify | Offset
```

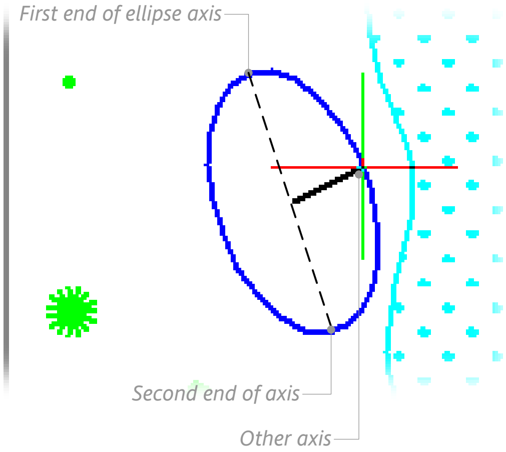
Drawing the garden pond illustrates another pair of useful commands: **Ellipse** draws oval shapes and **Offset** adds parallel copies. The pond will be an oval shape, while the pond's edging will be made from a parallel offset.

1. Switch to the **Pond** layer via the layer droplists.
2. The pond is to be an oval 4.5m tall by 1.5m wide (15 by 5 feet in imperial units). To draw the pond with the Ellipse command, follow these steps:

```
: ellipse
```

Pick the starting point of the ellipse anywhere in the garden area.

```
First end of ellipse axis or [Arc/Center]: c
Center of ellipse: (Pick a point)
```

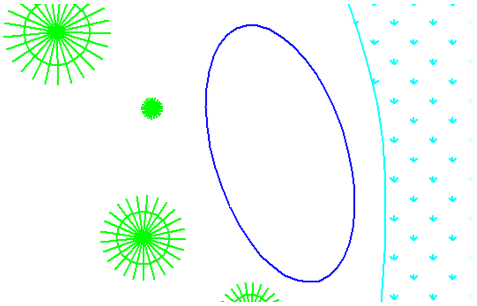


3. The other end of the pond is 4.5m away. Use direct distance entry by moving the cursor down and right:

```
Endpoint of axis: (Move the cursor) 4.5
```

4. And the pond is five feet wide:

```
Rotation/<Other axis>: 1.5
```



5. The pond is edged with rock. You could draw the edging of by repeating this command to draw a second, larger ellipse. The easier alternative is to use the **Offset** command, which creates precise parallel copies. Here, you use the command to create a concentric ellipse:

   a. The offset distance is 0.3m (or one foot in imperial units):

   ```
   : offset
   Enter offset distance or [Through point/Erase/Layer]  0.3
   ```

   b. Choose the ellipse; notice that only one entity can be offset at a time:

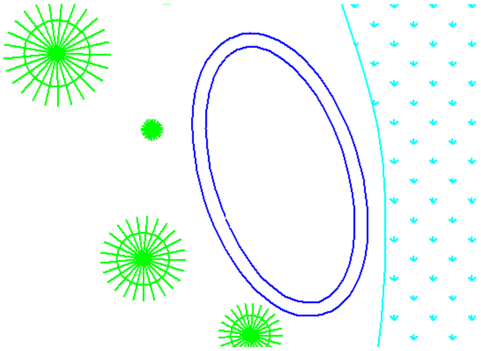
   ```
   Select entity/subentity or [Exit] <Exit>: (Pick the ellipse)
   ```

   c. Place the offset copy on the outside of the ellipse:

   ```
   Select side for parallel copy or [Both sides/Multiple]: (Pick outside ellipse)
   ```

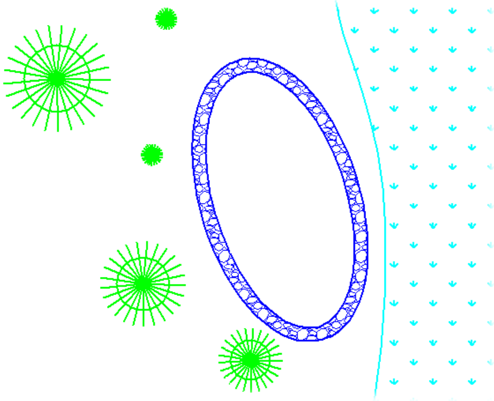
   d. The command repeats itself to offset other entities. But to exit the command, press **Enter**.

   ```
   Select entity/subentity or [Undo/Exit] <Exit>: (Press Enter to exit)
   ```



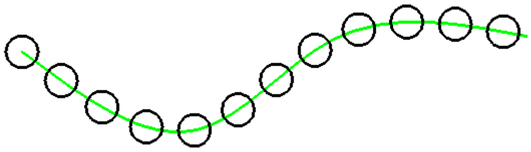
The Offset command also creates parallel lines, parallel polylines, and concentric circles and arcs.

If you wish, add the Gravel hatch pattern to the pond edging.



ARRAY PATHS

Another type of array repeatedly places objects along a *path*. This is called a "path array," appropriately enough. In the figure below, the path is in green while the arrayed entities are the circles.



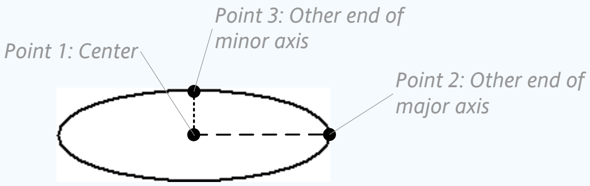
You use this construction technique to lay a path of paving stones from the house to the pond.

The path array needs a *guide path*, an entity along which to place the arrayed entities, and so in the first part of the tutorial, you draw the guide path as a spline (**Spline** command). The paving stones will be made of polygons (**Polygon** command), and then arrayed along the spline (**ArrayPath** command).

QUICK SUMMARY OF DRAWING ELLIPSES

BricsCAD provides three methods for drawing ellipses (ovals):

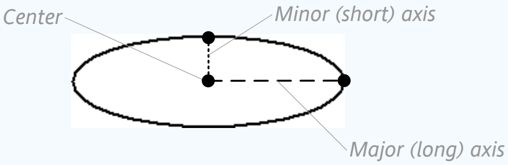
```
: ellipse
First end of ellipse axis or [Arc/Center]:
```



Point 3: Other end of minor axis
Point 1: Center
Point 2: Other end of major axis

Center — specifies the ellipse's center point, and then the endpoints of the major and minor axes.

Axis, End — specifies the endpoints of the ellipse's major axis, and then the minor one.



Arc — draws elliptical arcs.

This command also draws isocircles, which are isometric circles. This option is available only when isometric drawing mode is turned on through the Snap or Settings commands.

Drawing Splines

Command	Spline
Alias	spl
Menu Bar	Draw | Spline
Ribbon	Draw | Polylines | Spline
Toolbar	Draw | Spline

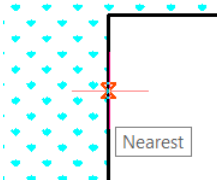
Splines were first used in ship building to make hulls from smooth flowing strips of wood. Hundreds of years later, we still use splines to draw smooth curves, but now with the **Spline** command. Follow along with these steps:

1. First, change the layer to "Road" using one of the methods you learned earlier. (My preferences is to use the "clayer road" shortcut shown below.) This means the paving stones are placed on the same layer as the driveway.

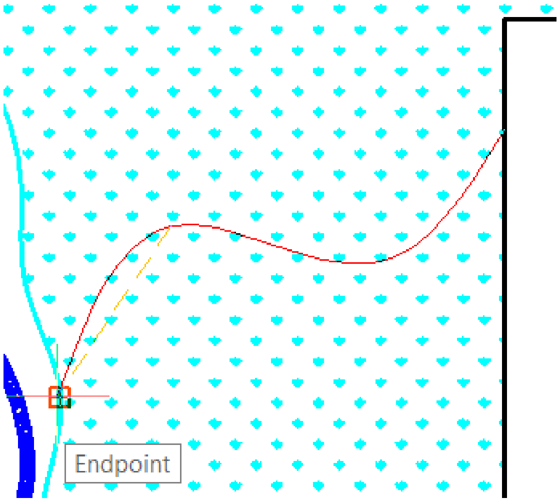
```
: clayer
New current value for CLAYER: road
```

2. Draw the guide path using a spline. If you ignore the command's many options, then drawing a spline is as just about as easy as drawing a line:

```
: spline
First point for spline: (Pick a point on the house outline, as show below)
```



```
Second point: (Pick a point along the lawn)
Set next point or [Close/Fit tolerance]: (Pick another point along the lawn)
Set next point or [Close/Fit tolerance]: (Pick a point at the edge of the lawn near the
pond, as shown below)
```



3. To end the command, press **Enter** three times, once for each prompt:

```
Set next point or [Close/Fit tolerance]: (Press Enter)
Starting point tangent point: (Press Enter)
Enter tangent for ending point: (Press Enter)
```

TIP Another way to draw splines is with polylines. Use the **PLine** command to draw the polyline, and then use the **PEdit** command's **Spline** command to convert it to a spline. This can in some cases be an easier approach than using the Spine command directly.

Zooming to Objects

The **Zoom** command has a handy option that lets you zoom to a single entity. Here is how it works:

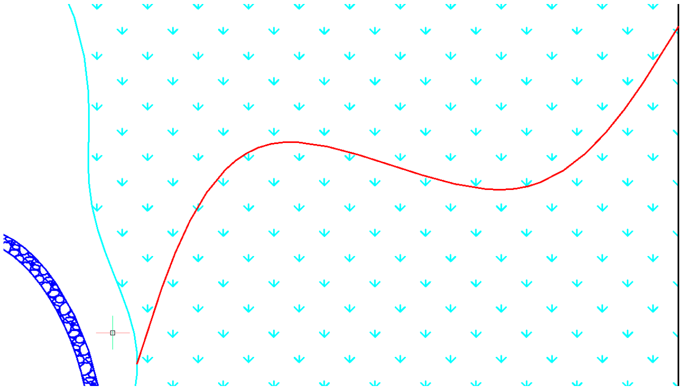
4. To zoom into the spline, use the **OBject** option of the **Zoom** command, as follows:

```
: zoom
Zoom [zoom In/zoom Out/All/Center/Dynamic/Extents/Left/Previous/Right/Scale (nx/nxp)/
Window/OBject] <Scale (nX/nXP)>: ob
Select an entity: l
```

By entering ' L ' at the select prompt, you ask the program to select the *last*-drawn entity, the spline. Using **OBject** with **Last** makes it easy for you to pick the spline out of the grass pattern.

5. Notice that the program zooms into the spline, so that it fills the screen. Zoom back out by 10% (i.e., 0.9x) so that you have some working space (see figure below):

```
:  zoom .9x
```



Paving Stones from Polygons

```
Command    Polygon
Alias      pol
Menu Bar   Draw | Polygon
Ribbon     Draw | Polygons
Toolbar    Draw | Polygon
```

6. The paving stones are *hexagonal* (six-sided) in shape, and so you draw them best with the **Polygon** command. This command, unfortunately, is an unintuitive one and so tricky to execute; so follow along:

a. The first thing you do is specify the number of sides. For a hexagon, you specify six sides:

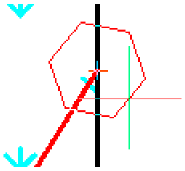
```
: polygon
Polygon: Set number of sides or [Width of line/Multiple polygons] <4>: 6
```

b. Then you pick a center point for the polygon. For this tutorial, pick one end of the spline using ENDpoint entity snap:

```
Specify by: Set center of polygon or [specify by Edge]: end
Snap to endpoint of: (Pick end of the spline near the house)
```

c. Now you specify the size of the hexagon, which is sometimes not easy to do. For this tutorial, just drag the cursor away from the center point, as shown by the figure.

```
Specify by: Select midpoint of side or [specify by Vertex]: (Move cursor away
from center of polygon)
```



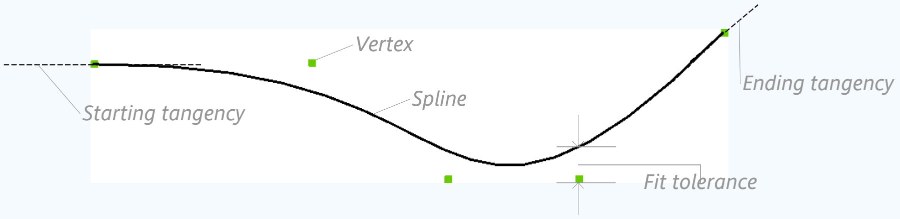
Arraying Along a Path

Command	ArrayPath
Menu Bar	Modify | Array | Path Array
Ribbon	Change | Array | Path Array
Toolbar	Modify | Arrays

With one paving stone (polygon) in place, you use the **ArrayPath** command to add all of the others along the path (spline). But first, a word of explanation.

QUICK SUMMARY OF SPLINE

```
: spline
First point for spline:
Second point:
Set next point or [Close/Fit tolerance]:
Select starting tangent point:
Enter tangent for ending point:
```



First point for spline — specifies the start of the spline's tangency

Second point — locates the first vertex of the spline

Close — closes the spline, joining the start and points

Fit — specifies how closely the spline matches its vertices (also known as "fit points"); a value of 0 forces the spline to pass through the points you pick, and then the higher the number the further the spline is from those points.

Select starting tangent point — locates a tangency for the starting point; press Enter to ignore the option

Enter tangent for ending point — locates a tangency for the ending point; press Enter to ignore the option

ArrayPath is one of a trio of commands that create *associative* array. (The other two are ArrayRect for associative rectangular arrays and ArrayPolar for polar ones.) *Associative* means that all the elements of the array act as a single entity, yet can be individually edited — like an associative dimension (see chapter 7). Here's the benefit: when you change the path, the array changes to suit.

1. Start the **ArrayPath** command, and then select the polygon, as follows:

```
: arraypath
Select entities to array: (Pick the polygon)
```

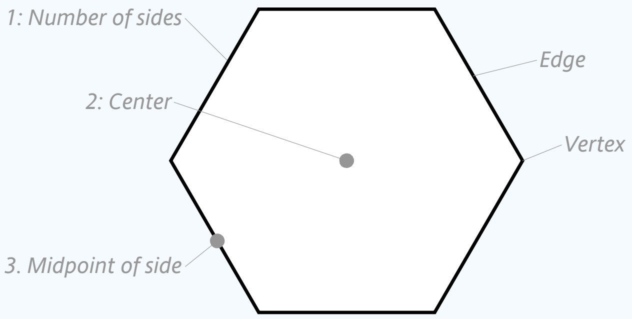
2. Press **Enter** to exit the selection process, and then pick the spline:

```
Entities in set: 1
Select entities to array: (Press Enter)

Type = Path, Associative = Yes
Select the path curve: (Pick the spline)
```

QUICK SUMMARY OF POLYGON

This command draws regular polygons, which means that every side has the same length. (To draw irregular ones, use another command such as Rectang or PLine.) The minimum and maximum sides is 3 and 1024.

```
: polygon
Polygon: Set number of sides or [Width of line/Multiple polygons] <4>:
Specify by: Set center of polygon or [specify by Edge]:
Specify by: Select midpoint of side or [specify by Vertex]:
```



Set number of sides — specifies the number of sides between 3 and 1024

Multiple polygons — repeats the command to draw additional polygons

Width of line — specifies the width of the lines, because polygons are made of polylines

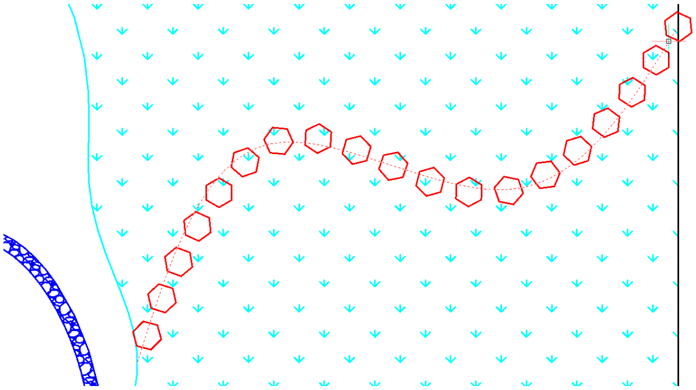
Specify by edge — specifies the length of one side

Set center of polygon — specifies the center point of the polygon

Specify by vertex — specifies the location of a vertex (corner of the polygon)

Select midpoint of side — specifies the midpoint of one of the sides

3. Now you see many paving stones along the path.



QUICK SUMMARY OF ARRAYPATH

```
: arraypath
Select entities to array:
Select path curve:
[ASsociative/Method/Base point/Tangent direction/Items/Rows/Levels/Align items/
Z direction/eXit]<eXit>:
```

Select path curve — you can use any linear (open) entity for the path, such as a line, polyline, arc, circle, ellipse, elliptical arc, spline, helix, or 3D polyline.

ASsociative — toggles the array between associative and non-associative

Method — determines how the entity is arrayed along the path:

Measure	Places entities at a specified distance apart
Divide	Places a specified number of entities along the path

Base point — specifies a new base point from which the entity is arrayed; useful for when you don't want the array on top of the path

Tangent direction — determines how the entity is aligned along the path in 3D, tangent or normal.

Items — specifies the (Measure mode) distance between items, or (Divide mode) number of items to place along the path; a maximum number of items can be specified to make the array shorter than the path.

Rows — specifies the number of rows for stacked arrays

Levels — specifies the number of levels and their spacing for 3D arrays

Align items — determines whether the entity is aligned along the path

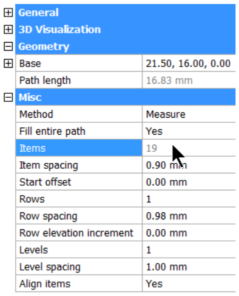
Z direction — toggles the z orientation between upright or banked along path

eXit — ends the command

Press **Enter** to exit the command.

4. Use the **Erase** command to remove the spline from the drawing.

To count the number of paving stones you need to order from the lumber store, select the array and then look at the **Items** field in the Properties pane: 19. Your number may differ, depending on how you drew the spline and the polygon stepping stones.



⊞ **General**	
⊞ **3D Visualization**	
⊟ **Geometry**	
⊞ Base	21.50, 16.00, 0.00
Path length	16.83 mm
⊟ **Misc**	
Method	Measure
Fill entire path	Yes
Items	19
Item spacing	0.90 mm
Start offset	0.00 mm
Rows	1
Row spacing	0.98 mm
Row elevation increment	0.00 mm
Levels	1
Level spacing	1.00 mm
Align items	Yes

QUICK SUMMARY OF ARRAYEDIT

```
: arrayedit
Select associative array:
```

Select associative array — select any entity in the associative array; after you do so, the next prompt depends on the command used to make the array.

- ArrayPath command:

```
Source/REPlace/Method/Base point/Items/Rows/Levels/Align items/Z direction/
RESet/<eXit>:
```

- ArrayPolar command:

```
Source/REPlace/Base point/Items/Angle between/Fill angle/Rows/Levels/ROTate items/
RESet/<eXit>: (Select an option.)
```

- ArrayRect command:

```
Source/REPlace/Base point/Rows/Columns/Levels/RESet/<eXit>: (Select an option.)
```

Many of the options are the same as with each array-creation command; those that are unique are described below.

Source — edits the source entity, following which all other entities in the array are modified; use the **ArrayClose** command when finished editing.

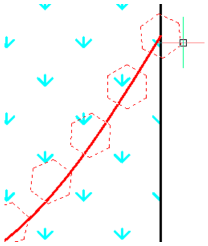
Replace — replaces some or all entities in the array with another entitiy.

Reset — restores erased entities and item overrides.

Editing Associative Arrays

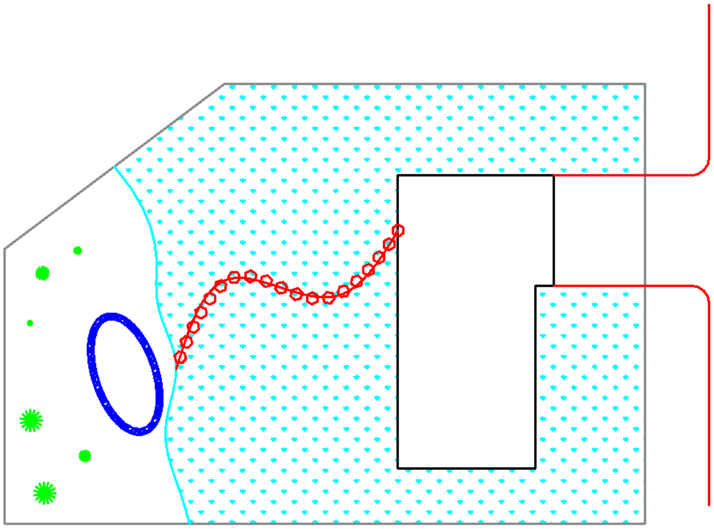
Command	ArrayEdit
Menu Bar	Modify | Array | Edit Array

One of the stepping stones sits on the edge of the house. When you select it to erase it, notice that the entire array of polygons is selected. (See figure below.)



This shows you that associative arrays act like a single entity, kind of like trying to edit part of a block. (The entities in arrays made with the regular Array command are individual.) Just as you can edit blocks with the RefEdit command, you edit associative arrays with the ArrayEdit command.

To see the progress you are making in learning BricsCAD, plot your drawing with the **Print** command. At this point, your drawing should look similar to the following:



Remember to save your work with the **Save** command.

You've added a number of details to the drawing. In the next lesson, you learn how to make changes to them — how to edit entities.

Notes

CHAPTER 6

Making Changes to Drawings

Previously, you added details to the drawing, such as the lawn, trees, and a pond. But CAD software shows itself to be most powerful at effortlessly making changes, and flushing out the data held by drawings.

In this chapter, you learn how to change parts of the drawing and how to extract information.

IN THIS CHAPTER

- Modifying properties of entities
- Applying linetypes
- Changing the length of open and closed entities
- Understanding selection sets
- Finding information about entities in drawings
- Nudging entities

KEY TERMS IN THIS CHAPTER

Extend — extends open entities to boundaries that are defined by other entities

Filter — creates specified subsets of entities based on their properties

Linetype — displays line patterns made of dashes, dots, gaps, and symbols

Properties — lists all the characteristics of an entity

Trim — cuts entities at cutting edges that are defined by other entities

USEFUL ABBREVIATIONS

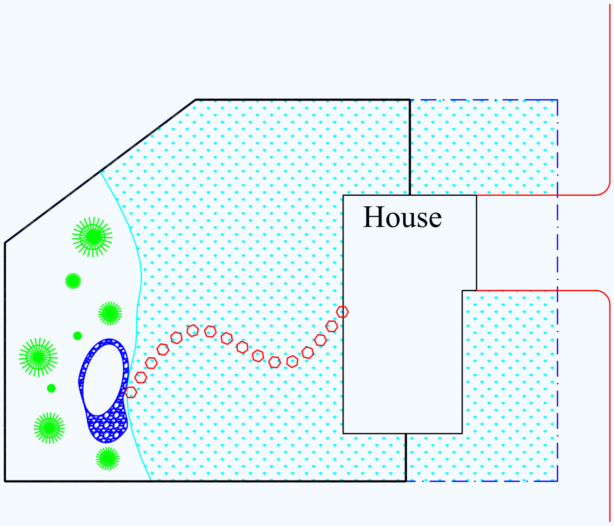
ISO	International Organization for Standardization
.lin	Linetype definition file

NEW COMMANDS

Command	Aliases	Menu Selection	Ribbon Tab
Dist	di	Tools | Inquiry | Distance	Tools | Inquiry | Distance
Lengthen	len	Modify | Lengthen	...
Linetype	lt	Settings | Linetypes	...
List	li	Tools | Inquiry | List Entity Info	Tools | Inquiry | List
Properties	pr	Modify | Properties	...
Stretch	s	Modify | Stretch	Change | Modify | Stretch

THIS CHAPTER'S DRAWING

By the end of this lesson, your drawing will look like this one:

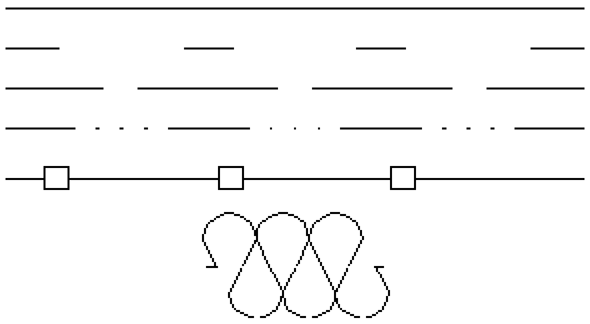


Tutorial-06 drawing file

Changing the Look of Lines

When you drew the lot lines in an earlier chapter, they were solid lines. Lot lines are, however, usually shown by a dashed pattern. Drafters use patterned lines to identify the meaning of lines, such as for hot water piping, insulation, or property lines, as in this case.

Just as BricsCAD comes with several hatch patterns, it also includes a number of line patterns made of dashes, dots, and more complex patterns. They are called "linetypes," even though they apply to nearly all 2D entities, such as also to circles and polylines. Some of them look like this:



To change a line from solid (called "Continuous" by BricsCAD) to dashed, you load a linetype definition into the drawing, and then select the entitity(ies) you want to have the new look.

LOADING LINETYPES

Command	Linetype
Alias	lt
Menu Bar	Tools | Linetypes
Status Bar	Linetype

Start BricsCAD, and then open your copy of *yard.dwg*. (Or open the *tutorial-05.dwg* file or *Imp-tutorial-05.dwg* in imperial units.)

1. If necessary, use the **Zoom** command's **All** option so that you see the entire drawing on the screen"

   ```
   : zoom
   Zoom [zoom In/zoom Out/All/Center/Dynamic/Extents/Left/Previous/Right/Scale (nx/nxp)/
   Window/OBject] <Scale (nX/nXP)>: all
   ```

   Follow this by an 80% zoom that adds breathing space around the drawing (80% is the same as 0.8x):

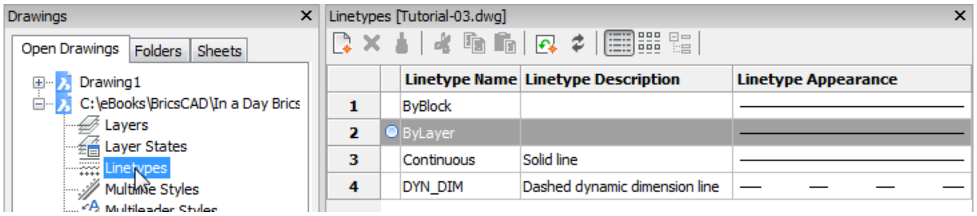
   ```
   : zoom
   Zoom [zoom In/zoom Out/All/Center/Dynamic/Extents/Left/Previous/Right/Scale (nx/nxp)/
   Window/OBject] <Scale (nX/nXP)>: 0.8x
   ```

2. Before you can apply linetypes, you must *load* their definitions from a file into the drawing. Linetype definitions are stored in files separately from BricsCAD and drawings. You recognize the file by the *.lin* found at the end of its name. Here is how to load linetypes into the drawing.

Enter the linetype command:

```
 : linetype
```

Notice that BricsCAD displays the Drawing Explorer. Drawing Explorer is a master dialog box that references many aspects of drawings, such as the way text and dimensions look.



	Linetype Name	Linetype Description	Linetype Appearance
1	ByBlock		
2	ByLayer		
3	Continuous	Solid line	
4	DYN_DIM	Dashed dynamic dimension line	

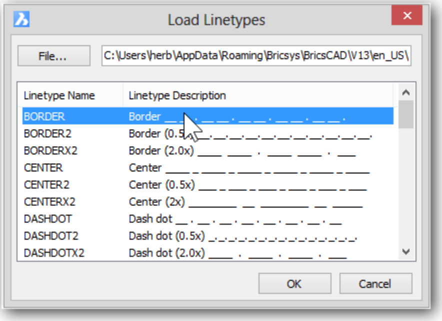
The drawing already contain some linetypes; all new drawings do. Here is what the names mean:

- **ByLayer** — entities take the linetype assigned by the layer. If a layer is assigned the Dashed linetype,then all entities on that layer are displayed with dashed linetypes.
- **ByBlock** — entities take the linetype defined by the block of which they are part
- **Continuous** — lines are solid, with no gaps
- **DYN_DIM** — lines with dashes; meant for use by dynamic dimensions

TIP Although BricsCAD uses terms like *line*types and *line* weights, these apply to nearly any kind of entity, such as circles and arcs — not just lines.
Linetypes and line weights are not, however, applied to text or 3D entities, because linetypes assign meaning to lines, which are not needed by text and 3D entities.

3. To add a linetype to this drawing, click the **New** button. BricsCAD doesn't actually create new linetypes, as suggested by the name of this button. Instead, the button leads you to a dialog box from which you load pre-defined linetypes.

Notice the Load Linetypes dialog box.



C:\Users\herb\AppData\Roaming\Bricsys\BricsCAD\V13\en_US\

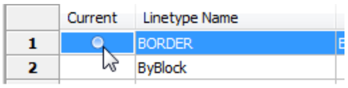
Linetype Name	Linetype Description
BORDER	Border
BORDER2	Border (0.5
BORDERX2	Border (2.0x)
CENTER	Center
CENTER2	Center (0.5x)
CENTERX2	Center (2x)
DASHDOT	Dash dot
DASHDOT2	Dash dot (0.5x)
DASHDOTX2	Dash dot (2.0x)

The dialog box lists in alphabetical order the names of all linetype definitions available to you. (They are stored in the *default.lin* file.) Scroll through the list to see what is available. Some linetypes have names that include a code number:

- Linetype names postfixed with **X2** are double scale
- Linetype names postfixed with **2** or **4** or **8** are half or quarter or eighth-size (0.5x, 0.25x, 0.125x scale)
- Linetype names prefixed with **ISO** match the ISO (International Standards Organization) standard

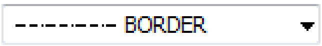
4. Load the Border linetype like this:
   a. Select **BORDER** from the Linetype Name list.
   b. Click **OK** to close the dialog box.

5. You are back in the Drawing Explorer dialog box. The next step is to set the Border linetype as *current*. "Current" means that it is the working linetype: if you were to draw a line, it would now take on the Border linetype. Lines already in the drawing are unaffected.
   a. Under Linetype Name, click **BORDER** to select it.
   b. Under the **Current** column, click the blank square so that the ● blue dot appears. This means the linetype is now current.
   c. Click **X** (in the upper right corner) to dismiss the Drawing Explorer dialog box.



	Current	Linetype Name
1	●	BORDER
2		ByBlock

When the Properties panel or the Entity Properties toolbar is open, notice that the default linetype changes to Border.



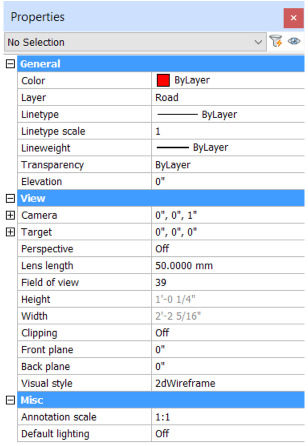
BORDER

CHANGING PROPERTIES

Command	Properties
Alias	pr
Menu Bar	Modify | Properties
Right-click	Properties
Toolbar	Standard | Properties

The Properties panel (or bar or palette) is excellent for changing the properties of one or more entities. It is one of the commands I probably use the most, because it provides three kinds of information:

- **When No Entities are Selected** — Properties panel reports the current default color, layer, linetype, and so on. The next entity you draw will take on these properties. For instance, the figure below shows that entities will be drawn with color ByLayer (red), on layer "Road," and so on.

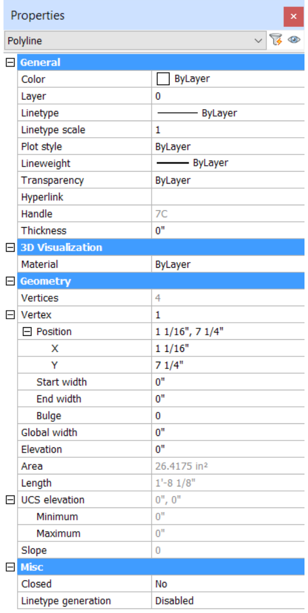


Properties

No Selection

General	
Color	🟥 ByLayer
Layer	Road
Linetype	ByLayer
Linetype scale	1
Lineweight	ByLayer
Transparency	ByLayer
Elevation	0"
View	
Camera	0", 0", 1"
Target	0", 0", 0"
Perspective	Off
Lens length	50.0000 mm
Field of view	39
Height	1'-0 1/4"
Width	2'-2 5/16"
Clipping	Off
Front plane	0"
Back plane	0"
Visual style	2dWireframe
Misc	
Annotation scale	1:1
Default lighting	Off

- **One Entity is Selected** — Properties panel describes the properties of the entity that you selected. The General section is common to all entities; the Geometry section changes depending on the type of entity selected. In the figure below (at left), you see that a line is selected and that the Properties panel shows the properties of the line.



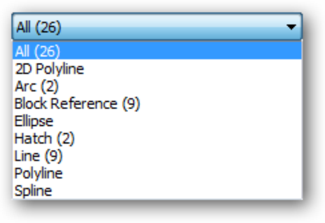
Properties

Polyline

General	
Color	□ ByLayer
Layer	0
Linetype	——— ByLayer
Linetype scale	1
Plot style	ByLayer
Lineweight	——— ByLayer
Transparency	ByLayer
Hyperlink	
Handle	7C
Thickness	0"
3D Visualization	
Material	ByLayer
Geometry	
Vertices	4
Vertex	1
Position	1 1/16", 7 1/4"
X	1 1/16"
Y	7 1/4"
Start width	0"
End width	0"
Bulge	0
Global width	0"
Elevation	0"
Area	26.4175 in²
Length	1'-8 1/8"
UCS elevation	0", 0"
Minimum	0"
Maximum	0"
Slope	0
Misc	
Closed	No
Linetype generation	Disabled

QUICK SUMMARY OF PROPERTIES PANEL

Properties describe the look and position of entities, such as color, linetype, starting and ending coordinates, and thickness. When two or more entities are selected, then the Properties panel (bar or palette) can displays a subset of entities. Click the droplist, and then choose a subset, such as "Arc (2)."



All (26)

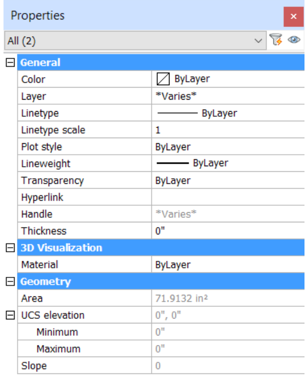
All (26)
2D Polyline
Arc (2)
Block Reference (9)
Ellipse
Hatch (2)
Line (9)
Polyline
Spline

The Properties palette does more than display properties; it also lets you to *change* values of properties — usually. Properties marked ***Varies*** can also be changed; in this case, all selected entities take on the same property. If you do this by accident, use the U command to undo the change.

There are three ways to change property values:

- Enter new values by typing them in
- Select preset values from droplists (when available)
- Click buttons (when available) to select values from drawings or dialog boxes

- **Two or More Entities Selected** — When two or more entities are selected, the Properties panel reports only those properties that the entities have in common. When properties are different between entities, the panel reports "*Varies*", as shown below.



Properties

All (2)

General	
Color	ByLayer
Layer	*Varies*
Linetype	ByLayer
Linetype scale	1
Plot style	ByLayer
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	
Handle	*Varies*
Thickness	0"
3D Visualization	
Material	ByLayer
Geometry	
Area	71.9132 in²
UCS elevation	0", 0"
Minimum	0"
Maximum	0"
Slope	0

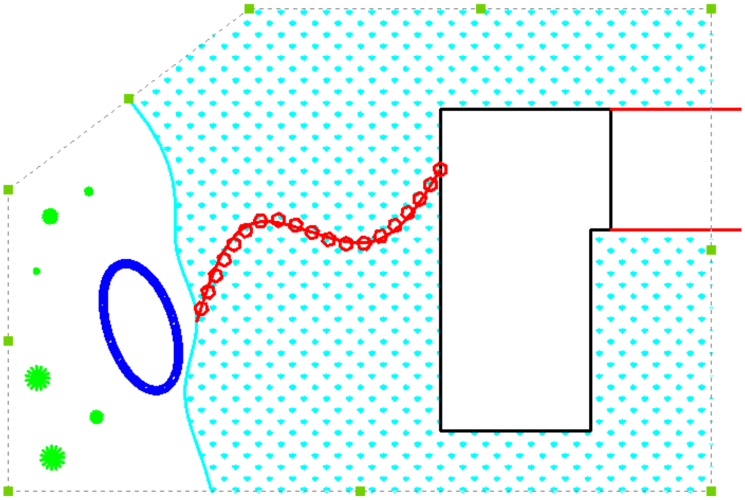
Use the Properties panel to now change the lot lines from 'Continuous' to 'Border', like this:

1. If the Properties panel is not displayed, enter the **Properties** command to open it now:

```
: properties
```

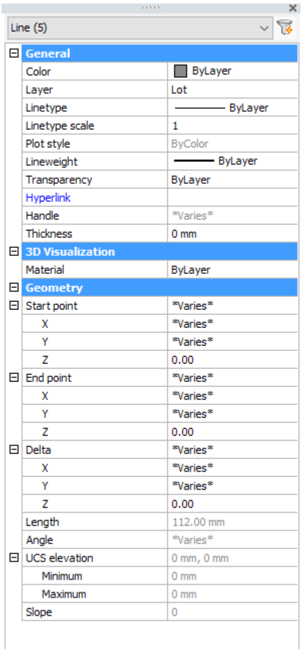
   Notice that the Properties panel appears.

2. In the drawing, pick the five blue lot lines directly. As you pick them, notice that BricsCAD highlights them: they are shown as dashed lines, with green grip squares appearing on each line.



   At the top of the Properties panel, notice that the list box reports **Line (5)**. This confirms that you selected five lines. The properties that you see listed belong to the five, such as layer "Lot" and linetype "Border."

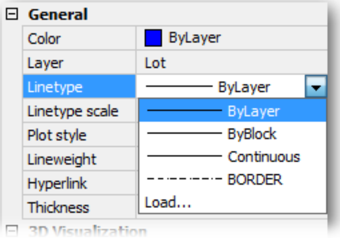
Because more than one line is selected, many properties are reported as *Varies*, because these properties, such as the x and y coordinates, differ for each line.



Line (5)

General	
Color	ByLayer
Layer	Lot
Linetype	ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	
Handle	*Varies*
Thickness	0 mm
3D Visualization	
Material	ByLayer
Geometry	
Start point	*Varies*
X	*Varies*
Y	*Varies*
Z	0.00
End point	*Varies*
X	*Varies*
Y	*Varies*
Z	0.00
Delta	*Varies*
X	*Varies*
Y	*Varies*
Z	0.00
Length	112.00 mm
Angle	*Varies*
UCS elevation	0 mm, 0 mm
Minimum	0 mm
Maximum	0 mm
Slope	0

3. Next to **Linetype** in the Properties panel, click **Bylayer**. Notice that BricsCAD lists the names of the linetypes in the drawing, ByLayer, ByBlock, Continuous, Dyn_Dim, and Border.



General	
Color	ByLayer
Layer	Lot
Linetype	ByLayer
Linetype scale	
Plot style	
Lineweight	
Hyperlink	
Thickness	

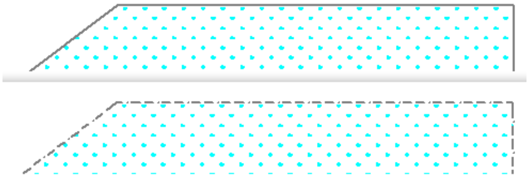
4. Select the **Border** linetype by clicking its name.
5. Press **Esc** to clear the grips.

Notice that the lines change their look and take on the dashed linetype. If, however, you see no change to the lines, then read the next section to learn how to fix this problem.

CHANGING THE LINETYPE SCALE

Command	LtScale
Alias	lts

The reason linetypes sometimes look continuous is that they are sensitive to scale, just like hatch patterns. Sometimes the lines should show gaps, but don't, as shown below. The solution is to change the scale (size) of the linetype.



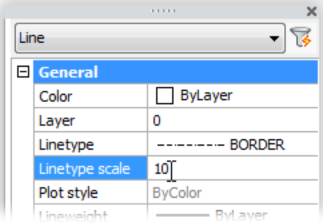
Top: *Incorrect linetype scale;* ***above:*** *Correct linetype scale.*

All linetypes start with a scale factor of 1.0, which usually is too small to display the pattern of dots and dashes correctly.

Warning! Setting linetype scale is one of the trickiest aspects of BricsCAD. Linetypes look continuous when the scale is too large — and when too small!

One method to change the scale of linetypes is to use **LtScale**; another is to use the Properties panel and its **Linetype Scale** field. I find using the Properties panel is the easier of the two methods:

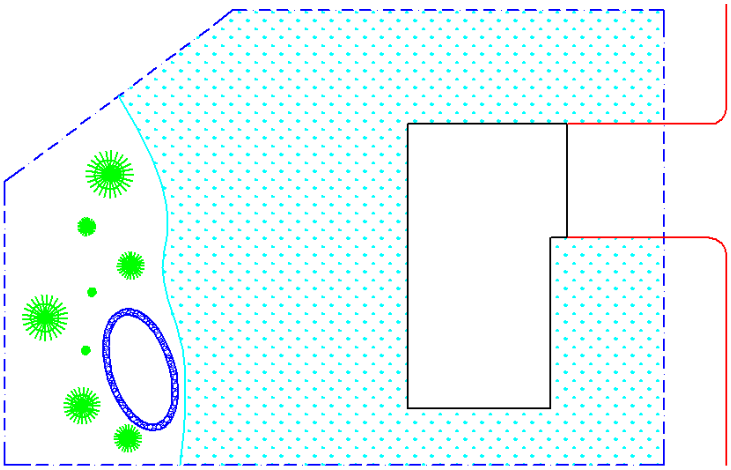
1. Continuing in the Properties panel, click the field next to **Linetype Scale**.
2. Change 1.0 to something like **10**.



Line

General	
Color	□ ByLayer
Layer	0
Linetype	‒‒‒‒‒‒‒ BORDER
Linetype scale	10
Plot style	ByColor
Lineweight	——— ByLayer

3. Check whether the linetype pattern becomes visible. If not, try another scale value, such as **0.01**.



QUICK SUMMARY OF SELECTION SET OPTIONS

Selection has many options for selecting entities, as shown in the following table. I find that I typically use just some of them, such as pick, W, C, L, P and Enter. To see the full list in BricsCAD, enter the **Select** command, and then press **?**, like this:

```
: select
Select entities to include in set: ?
Select entities: ALL/Add/+/Remove/-/Previous/Last/Window/Crossing/Outside/WPolygon/
CPolygon/OPolygon/WCircle/CCircle/OCircle/Box/POint/Fence/AUto/Multiple/Single/PROp-
erties/Dialog/Undo:
```

Mode	Abbreviation	Selects
Entity	*[pick]*	A single entity
All	ALL	All non-frozen entities; can also press Ctrl+A
Previous	P	Most recently selected entity
Last	L	Most-recently created entity visible on the screen
Window	W	All entities inside a rectangular window
Crossing	C	All entities crossing and within a rectangular window
Outside	O	All entities outside the rectangular window
Window Polygon	WP	All entities within a polygonal window
Crossing Polygon	CP	All entities crossing and within a polygon
Outside Polygon	OP	All entities outside a polygonal window
Window Circle	WC	All entities inside a circular window
Crossing Circle	CC	All entities crossing and inside a circular window
Outside Circle	OC	All entities outside a circular window
Box	B	All entities depending on how direction selection window is drawn: • Right to left: selects by Crossing mode • Left to right: selects by Window mode
Fence	F	All entities along a fence polyline
Auto	AU	Single pick selects one entity; otherwise acts like Box option
Single	S	Selects the first entity encountered, and ends the command

SELECTION MODIFICATIONS

Add or +	A or +	Enters add-entities mode
Remove or -	R or -	Enters remove-entities mode
Multiple	M	Select entities without highlighting; faster in complex drawings
Undo	U	Removes most recent selection group
Dialog	D	Displays the Entity Selection section of the Settings dialog box
End	*press* [Enter]	Ends entity selection
Cancel	*press* [Esc]	Cancels entity selection
Location	LO	Switches back to original select prompt
\		

...continued

4. It may be necessary for you to use the **Regen** command to fix the display, following a change in linetype scale.

SELECTING ENTITIES BY OTHER METHODS

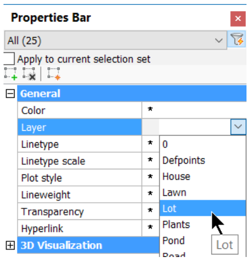
I find that it can be tricky trying to select entities in crowded drawings. No matter how tiny the pick box, sometimes I repeatedly pick the wrong entity. The program provides two workarounds that are useful to know.

Selecting Entities by Their Properties

In the previous tutorial I had you pick the five lot lines one by one. I did that on purpose so that I could show you easier ways! You can select entities by their properties. For this, BricsCAD offers the Quick Select mode, which isn't so much about selecting things quickly as selecting them by their properties.

When using Quick Select mode, you modify filters that apply to the current selection set. The current selection set starts off as all entities in the drawing. A *filter* reduces the selection set. For instance, when you select "Red" as the **Color**, you filter out all entities that are not red. When you select "Lot" as the Layer, you remove all entities that are not on layer Lot.

1. Press **Esc** to ensure no entities are selected.
2. In the Properties pane, click the **Quick Select** button. Notice that the panel changes its look to Quick Select mode:



Properties Bar

All (25)

□ Apply to current selection set

⊟ General

Color	*
Layer	
Linetype	*
Linetype scale	*
Plot style	*
Lineweight	*
Transparency	*
Hyperlink	*

⊞ 3D Visualization

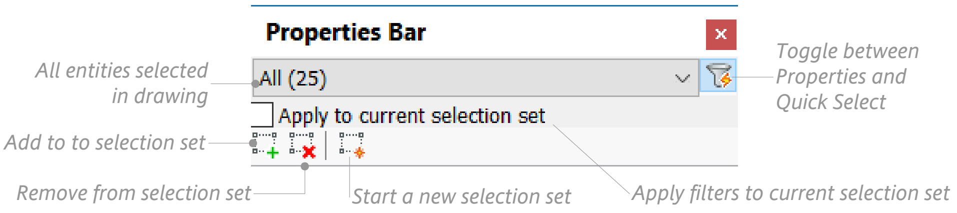
...*continued*

SELECTION BY PROPERTIES

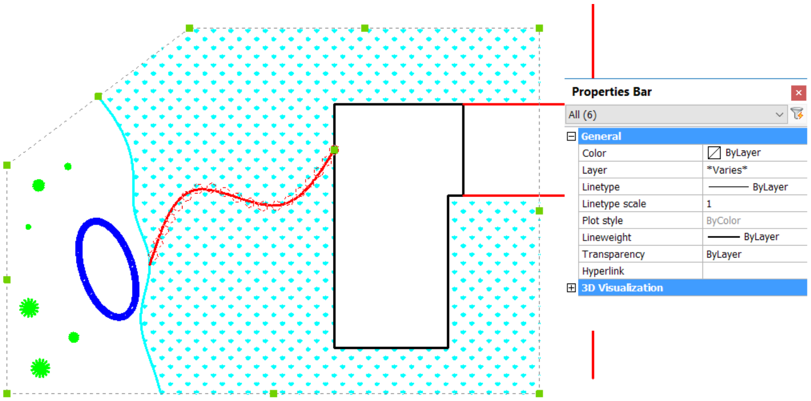
The **PROperties** option selects entities by common properties:

Properties	Abbreviation	Selects
Color	C	All entities of the same color name or number such as "red" or "122"
Layer	LA	All entities with the same layer name
Ltype	LT	All entities of the same linetype
Name	N	All entities of the same name
Thickness	TH	All entities of the same thickness
Type	TY	All entities of the same entity type, such as "circle"
Value	V	All entities of the specified handle
Width	W	All entities of the specified width

- All entities are selected, as shown by "All (25)"
- Thee buttons populate the toolbar
- Basic properties show ' * ' (asterisk), which means that all properties are selected.

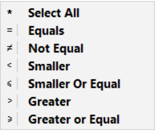

Properties Bar
All (25)
Apply to current selection set
All entities selected in drawing
Add to to selection set
Remove from selection set
Start a new selection set
Apply filters to current selection set
Toggle between Properties and Quick Select

3. In the **Layer** droplist, choose "Lot."
4. Click the **Add to Selection Set** button. Notice that only the lot lines are highlighted. You can now change the properties of just the lot lines, such as the linetype or color.



Color	ByLayer
Layer	*Varies*
Linetype	ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	

The * is not the only wildcard used by Quick Select. To see the full list, click the asterisk to uncover this shortcut menu. (That's left click, not right click.) These let you, for example, select all lines that have a Length property of less than 10m.



Selecting an Entity by Tabbing

When two or more entities overlap, you can press the **Tab** key to force the program to cycle through them. This trick works only when the **SelectionPreview** variable is turned on.

CHANGING LINE LENGTHS

```
Command    Lengthen
Aliases    len, editlen
Menu Bar   Modify | Lengthen
```

Once you've drawn stuff in a drawing, the entities are not static; you can change their size and position. When it comes to changing size, it matters if the entities are open or closed. *Open* entities are like lines, arcs, and open polylines; *closed* ones are like circles, ellipses, and polygons. Some commands work only with open entities, some only with closed ones, and some with both kinds.

One way to change lengths is using grips, as described earlier. A more precise method is with the **Lengthen** command. To see how it works, in this tutorial you extend the edge of the street line to the bottom of the screen. If necessary, first zoom in on the area around the pond with the Zoom Window command.

1. Start the **Lengthen** command:

```
: lengthen
```

2. At the 'Select entity to list length' prompt, pick the lower road line:

```
Edit length:  DYnamic/Increment/Percent/Total/<Select entity to list length>: (Pick
the line.)
```

Notice that BricsCAD reports its length in the command bar:

```
Current length: 11.00
```

This means that the line is 11m long (about 37', imperial units).

3. Lengthen the road to 12m by specify the **Total** option, and then entering **12** as the new length, as follows:

```
Edit length:  DYnamic/Increment/Percent/Total/<Select entity to list length>: t
Angle/<Enter total length (11.00)>: 12
```

QUICK SUMMARY OF THE LENGTHEN COMMAND

The **Lengthen** command provides you with the following options for changing the length (longer or shorter) of open entities, such as lines, arcs, and polylines:

```
: lengthen
Edit length: DYnamic/Increment/Percent/Total/<Select entity to list length>:
```

DElta — changes the length to an absolute amount,; you show the delta by picking a point the required distance from
the endpoint; default = 0. (*Delta* is the Greek word used by mathematicians to indicate change.)

DYnamic — changes the length by interactive dragging.

Percent — changes the length relative to 100%, as follows:

- Less than 100%, such as 50%, shortens the entity.
- More than 100%, such as 200%, lengthens the entity.

Total — changes the length to an absolute amount; default = 1 unit.

Undo — undoes the last change.

As is common in BricsCAD, there is more than one way to change the length of lines and other open entities. You can also use grips and these commands:

Change command — changes the length of a group of lines to a common endpoint.

Extend command — extends a line to a boundary entity.

Trim command — cuts back a line to a cutting edge entity.

4. Curiously, BricsCAD asks you a second time to select the entity; for some reason, it does not automatically change the length of the line you choose first:

```
Mode/<Select entity to change>: (Pick the same line)
```

5. Notice that the line lengthens. Press **Enter** to end the command.

```
Mode/Undo/<Select entity to change>: (Press ENTER)
```

Stretching the Pond

```
Command    Stretch
Alias      s
Menu Bar   Modify | Stretch
Ribbon     Change | Modify | Stretch
Toolbar    Modify | Stretch
```

You've used several editing commands to change entities, such as PEdit to modify polylines, grips editing, Properties to change linetype and scale, and Lengthen to alter the length of lines.

One of BricsCAD's most powerful editing commands is **Stretch**. This command lets you take parts of entities and stretch them wider and thinner, longer and shorter. Here you apply it to change the shape of the pond. You may find it helpful to first zoom in to the area around the pond.

1. Start the **Stretch** command.

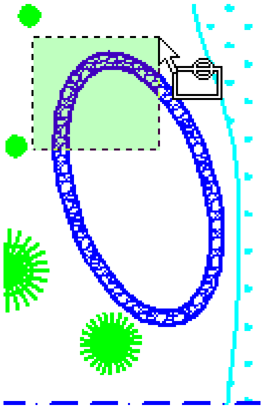
```
: stretch
Select entities to stretch by crossing-window or crossing-polygon: C
```

The Stretch command has an odd limitation, in that entities are initially selected only by two methods, both of them *crossing* selection modes. You can specify either Crossing (which forms a rectangular selection window) or CPolygon, which forms a polygonal selection window. The reason for the limitation is that it allows Stretch to know which entities to stretch — those crossing the selection window — and which to not stretch (those outside the selection window). Entities fully inside the selection window are moved.

Some entities cannot be stretched, like text and 3D solids. When they cross the selection window, the Stretch command moves them.

2. Select entities by picking two points to specify a crossing window, as follows:

```
First corner of crossing window: (Pick a point)
Opposite Corner: (Pick another point)
```



C is short for "crossing," an entity selection mode similar to the window mode you used earlier with the Zoom command. In this case, BricsCAD selects all entities within the selection rectangle and all entities crossing or touching the rectangle.

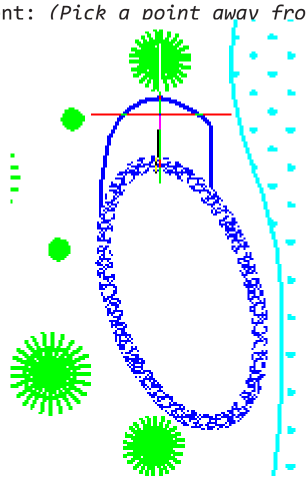
3. After picking two corners of a rectangle that covers part of the pond, press **Enter**:

```
Select entities to stretch by crossing-window or crossing-polygon: (Press ENTER to
end entity selection)
```

If the pond were entirely inside the entity selection rectangle, the Stretch command would **move** the pond, not stretch it. If you accidentally select entities other than the pond, type the R option (short for "remove from selection set"), and then select the entities to remove.

4. To tell BricsCAD how much you want the pond stretched, pick two points that indicate the distance:

```
Base point of displacement: (Pick a point near the pond)
Second point of displacement: (Pick a point away from the pond)
```



You have now created a whole new look to your pond! (Your pond will probably look different from the one illustrated in this book.) Note that you cannot create this effect using grips editing.

5. If you don't like it, you can undo the stretch with the U command, as follows:

```
: u
U: STRETCH
```

... and try stretching the pond again.

MOVING ENTITIES

```
Command    Move
Alias      m
Menu Bar   Modify | Move
Ribbon     Change | Modify | Move
Toolbar    Modify | Move
```

If the pond isn't exactly where you want it, you can relocate it with the **Move** command.

1. Start the Move command by entering the letter 'm', as follows:

```
: m
```

Here you used another of BricsCAD's shortcuts. **M** is the *alias* (abbreviation) for the Move command.

2. Continue the Move command by specifying Crossing selection mode to capture all three parts of the pond: two ellipses and hatch pattern.

```
Select entities to move: c
First corner of crossing window: (Pick a point near the pond)
```

```
Opposite Corner: (Pick another point to encompass the pond)
Select entities: (Press ENTER to end entity selection)
```

3. Just as with the Stretch command, you need to specify two points. They indicate the direction and distance to move the entities:

```
Vector/<Base point>: (Pick edge of pond.)
Displacement point: (Pick new location for pond.)
```

4. Notice that the pond has moved. If necessary, clean up the screen with the **Redraw** command, using its alias, as follows:

```
: r
```

5. Save your work.

BricsCAD lets you specify commands by typing just one or two letters at the '**:**' prompt. Shortcut keystrokes are listed at the start of each chapter. The complete list of command name abbreviations (called "aliases") is stored in the *default.pgp*.

NUDGING ENTITIES

Some parts of CAD call for precision, such as the legal definition of the property line and the location of the house. Other parts, however, call for artistry, such as the border between the lawn and garden areas. The only thing that mattered about the polyline curve was that it look attractive.

Similarly, the trees placed in the garden were placed arbitrarily. Now, a landscape designer may want to move them around a bit to create attractive groupings. For this, BricsCAD has the nudge function.

Nudge is a keyboard function that works like this:

1. Select one or more entities.
2. Hold down the Ctrl key.
3. Use the cursor keys on the keyboard to nudge the entity(ies) around — up, down, left, and right.

Each time you tap the cursor key, BricsCAD reports the distance moved:

```
NUDGE: offset along the Y axis of the UCS: 0.20000000
NUDGE: offset along the X axis of the UCS: 0.20000000
NUDGE: offset along the X axis of the UCS: -0.20000000
NUDGE: offset along the Y axis of the UCS: -0.20000000
```

The distance is a function of the zoom level. The closer in you are to the drawing, the shorter the zoom distance.

Adding the Fence

Say you've decided to add a fence to the backyard. You can use BricsCAD to help plan the materials needed: after drawing the fence (as a polyline), you can find out from the drawing the length of the fence.

The fence is drawn as a polyline, 0.1m-wide (4"-wide). Along the way, you'll use a variety of entity snap modes and tracking mode.

1. First, switch the working layer to **House** by selecting its name from the toolbar.

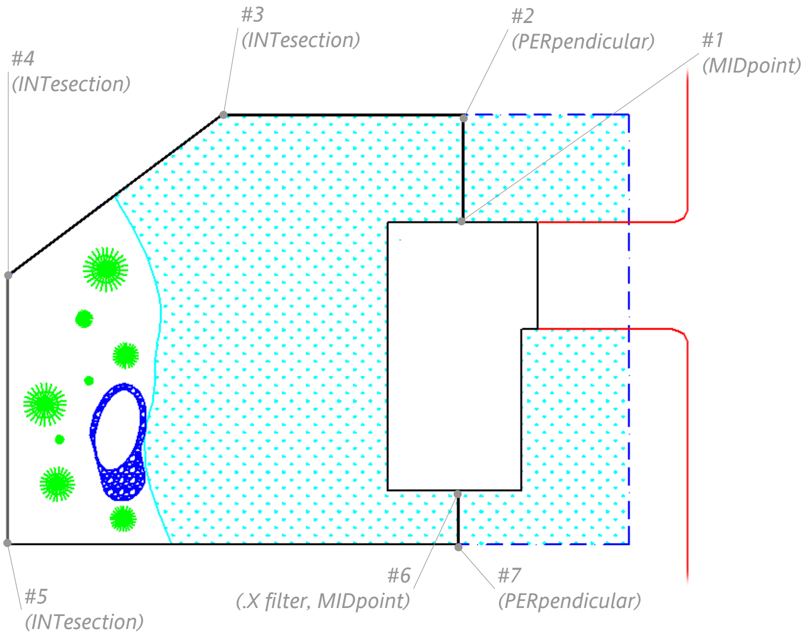
   Use the **Zoom Previous** command to see the full drawing.
2. If you turned off the **INTersection** entity snap, turn it back on using this method: at the command prompt, enter **Intersection** like a command.

   ```
   : intersection
   ```
3. Begin by entering the PLine command:

   ```
   : pline
   ```
4. Start the polyline at the middle of the upper house line. This point is shown as #1 in the figure below). Use the **MIDdle** entity snap to precisely locate the polyline's starting point at the middle of the house line. "Mid" is the abbreviation for MIDdle entity snap.

   ```
   : _pline
   Start of polyline: mid
   Snap to midpoint of: (Pick upper house line)
   Current line-width is 0
   ```
5. To change the width of the polyline from zero to 10cm (4"), use the PLine command's **w** (Width) option, as follows:

   ```
   Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width]: w
   Starting width <0>: 0.1
   Ending width <0.1>: (Press Enter to accept the new default)
   ```



   Note that you can specify different starting and ending widths, which would produce tapered polylines.

6. Now that the starting point and width are set, continue drawing the fence. Follow the path shown by numbers 2 through 5 in the figure above.

7. Pause the picking action at point #5. When you get to the bottom of the house, you get into a bit of tricky geometry. You want the fence to end at the same relative location as its starting point. You're not sure of the x coordinate, which is located *somewhere* along the bottom line of the lot. Fortunately, you can find that point with *point filters*.

   Normally, when you pick a point, you are providing BricsCAD with the x and y coordinate. Point filters capture a single coordinate, such as just the x or the y.

   a. Here is how to get just the x coordinate:

   ```
   Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width]: .x
   ```

   b. BricsCAD asks you to pick a point ('Select X of' prompt), from which it obtains that x coordinate. Tell it that you want the middleof the line:

   ```
   Select X of: mid
   Snap to midpoint of: (Pick the lower line of the house at #6)
   ```

   Notice that you can combine point filters and entity snaps to get very specific coordinates!

   c. And then BricsCAD asks you for the y coordinate. When it asks 'Still need YX,' BricsCAD is asking for the y and z coordinates. Well, the z you don't care about, but here is how to get the y with the perpendicular entity snap:

   ```
   Still need YZ of: per
   Snap to perpendicular of: (Pick the property line at #7)
   ```

   You completed the fence by drawing the last polyline segment using **PERpendicular** entity snap at #7.

8. Exit the **PLine** command:

   ```
   Set next point or [draw Arcs/Distance/Follow/Halfwidth/Width]: (Press ENTER to exit
   the command)
   ```

LISTING DATA

```
Command    List
Alias      li
Menu Bar   Tools | Inquiry | List Entity Info
Ribbon     Tools | Inquiry | List
Toolbar    Inquiry | List

Command    Dist
Alias      di
Menu Bar   Tools | Inquiry | Distance
Ribbon     Tools | Inquiry | Distance
Toolbar    Inquiry | Distance
```

Now that you've drawn the fence, you can use the **List** command to tell you its length.

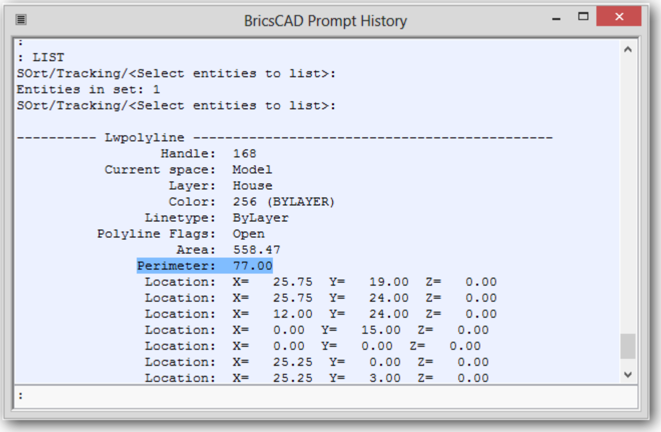
1. Enter the List command:

   ```
   : list
   SOrt/Tracking/<Select entities to list>: (Pick the fence's polyline)
   SOrt/Tracking/<Select entities to list>: (Press Enter to end entity selection)
   ```

   BricsCAD flips to the Prompt History and lists lines of information. The List command tells you every piece of information about the polyline (technically known as a "lwpolyline") that BricsCAD has stored in its database. Most of the information is about its vertices ("at point").

The total length of the polyline is shown in the middle of the listing:

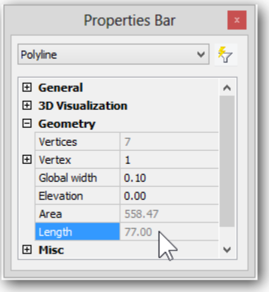
```
Perimeter  77.0
```



```
:
: LIST
SOrt/Tracking/<Select entities to list>:
Entities in set: 1
SOrt/Tracking/<Select entities to list>:

---------- Lwpolyline ---------------------------------------------
                  Handle:  168
           Current space:  Model
                   Layer:  House
                   Color:  256 (BYLAYER)
                Linetype:  ByLayer
          Polyline Flags:  Open
                    Area:  558.47
               Perimeter:  77.00
                Location:  X=   25.75  Y=   19.00  Z=   0.00
                Location:  X=   25.75  Y=   24.00  Z=   0.00
                Location:  X=   12.00  Y=   24.00  Z=   0.00
                Location:  X=   0.00  Y=   15.00  Z=   0.00
                Location:  X=   0.00  Y=   0.00  Z=   0.00
                Location:  X=   25.25  Y=   0.00  Z=   0.00
                Location:  X=   25.25  Y=   3.00  Z=   0.00
:
```

So, 77 meters (roughly 260 feet). Now you know how much fencing you would need.

2. Press function key **F2** to flip back to the graphics window. The Properties panel also reports area and length information:



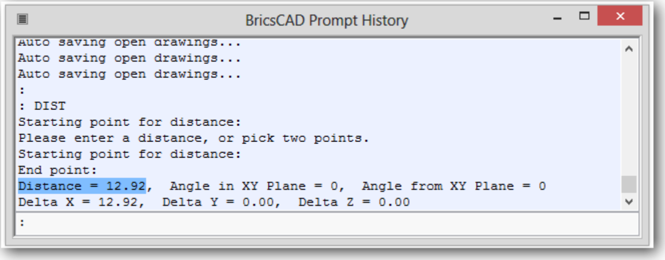
⊞ General
⊞ 3D Visualization
⊟ Geometry

Vertices	7
⊞ Vertex	1
Global width	0.10
Elevation	0.00
Area	558.47
Length	77.00

⊞ Misc

3. You can measure distances directly on the drawing with the **Dist** command (short for "distance"). To find the shortest distance from the house to the pond with the Dist command, from the **Tools** menu bar, select **Inquiry | Distance**:

```
: _dist
Starting point for distance: nea
Snap to nearest of: (Pick inside edge of pond.)
End point: per
Snap to perpendicular of: (Pick house wall.)
```



```
Auto saving open drawings...
Auto saving open drawings...
Auto saving open drawings...
:
: DIST
Starting point for distance:
Please enter a distance, or pick two points.
Starting point for distance:
End point:
Distance = 12.92,  Angle in XY Plane = 0,  Angle from XY Plane = 0
Delta X = 12.92,  Delta Y = 0.00,  Delta Z = 0.00
:
```

The beeline distance from house to pond is 13m (around 43 feet). The value on your drawing may differ, depending on where you located the pond.

You’ve done a lot of editing, so remember to save the work you have done on the drawing. You may also want to plot out the drawing.

Next, you learn how to add text and dimensions to the drawing.

CHAPTER 7

Adding Notes and Dimensions

With the yard plan nearly finished, now is the time to add *callouts* (or text) to describe the parts of the yard and dimensions to show its size. In this chapter, you learn how to add the finishing touches to the drawing.

IN THIS CHAPTER

- Learning about drawing scales
- Placing text in drawings
- Defining styles and justification modes for text
- Understanding annotative scaling
- Editing text
- Attaching dimensions to entities
- Placing leaders and multileaders

KEY TERMS IN THIS CHAPTER

Annotative — refers to entities that scale themselves to match the layout's scale factor

Associative — refers to entities, such as dimensions and hatches, that update automatically when associated entities are updated

Bounding box — describes the invisible rectangle that determines the word wrap of paragraph text

Callouts — describes the text attached to lines that describe significant parts of drawings

Dimension — indicates a measured distance, usually consisting of lines, arrowheads, and text

Justification — determines the relative positioning of text, such as right, left, and centered

Layout — presents editable print previews

USEFUL ABBREVIATIONS

dims	Dimensions
dimvars	Dimension variables
dimstyle	Dimension style

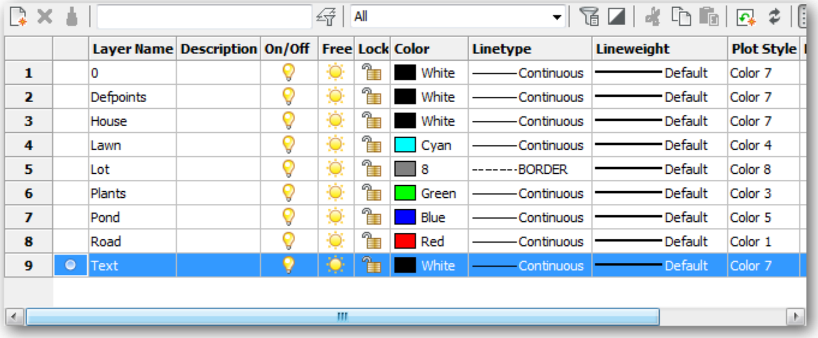
NEW COMMANDS

Command	Shortcuts	Menu Bar	Ribbon Tab
DdEdit	ed	...	Draw | Dimensions | Edit
DimAligned	dal, dimali	Dimension | Aligned	Draw | Dimensions | Aligned
DimBaseline	dba, dimbase	Dimension | Baseline	Draw | Dimensions | Baseline
DimContinue	dco, dimcont	Dimension | Continue	Draw | Dimensions | Continue
DimEdit	ded, dimed	Modify | Edit Dimension Text	...
DimLinear	dli, dimlin	Dimension | Linear	Draw | Dimensions | Linear
DimRadius	dra, dimrad	Dimension | Radius	Draw | Dimensions | Circle
MText	t	Draw | Multiline Text	Draw | Text | Multiline Text
QText	qt	...	...
Rotate	ro	Modify | Rotate	Change | Arrange | 2D Rotate
Style	st	Settings | Text Style	Home | Drawing Explorer
Text	tx	Draw | Text	Draw | Text | Text

Before starting this exercise, take the following steps:

1. Start BricsCAD, and then open the *yard.dwg* file.

   If you were unable to complete the exercises of previous lessons, open the *tutorial-06.dwg* file; for imperial units, open the *Imp-tutorial-06.dwg* file.
2. Do a **Zoom All** to make the full drawing visible.
3. With the **Layer** command, create a new layer, and then name it **Text**.
4. Assign color "White" to the layer, and then make the layer current by clicking in the **Current** column, so that the blue dot appears next to the "Text" name.

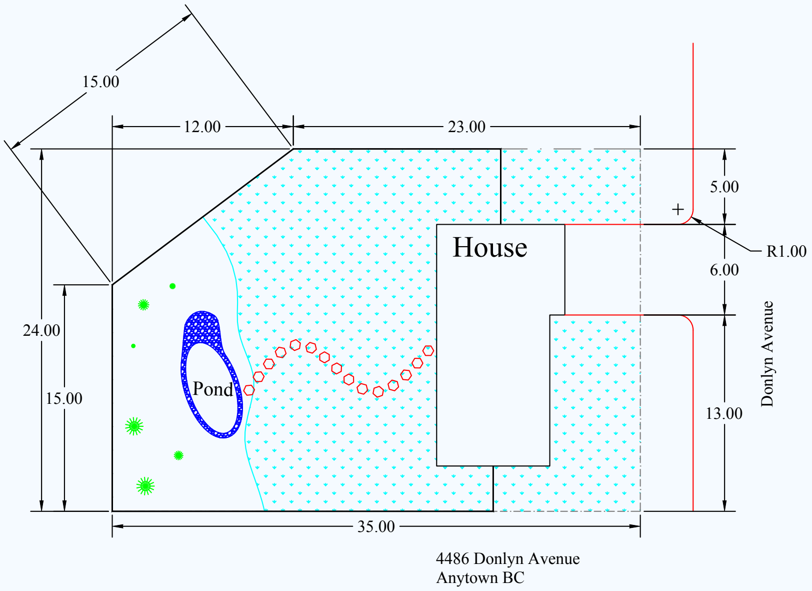


		Layer Name	Description	On/Off	Free	Lock	Color	Linetype	Lineweight	Plot Style
1		0		💡	☀	🔓	■ White	——— Continuous	——— Default	Color 7
2		Defpoints		💡	☀	🔓	■ White	——— Continuous	——— Default	Color 7
3		House		💡	☀	🔓	■ White	——— Continuous	——— Default	Color 7
4		Lawn		💡	☀	🔓	■ Cyan	——— Continuous	——— Default	Color 4
5		Lot		💡	☀	🔓	■ 8	------- BORDER	——— Default	Color 8
6		Plants		💡	☀	🔓	■ Green	——— Continuous	——— Default	Color 3
7		Pond		💡	☀	🔓	■ Blue	——— Continuous	——— Default	Color 5
8		Road		💡	☀	🔓	■ Red	——— Continuous	——— Default	Color 1
9	●	Text		💡	☀	🔓	■ White	——— Continuous	——— Default	Color 7

You are now ready to tackle text in drawings.

THIS CHAPTER'S DRAWING

By the end of this lesson, your drawing will look like this one:



Adding Notes to Drawings

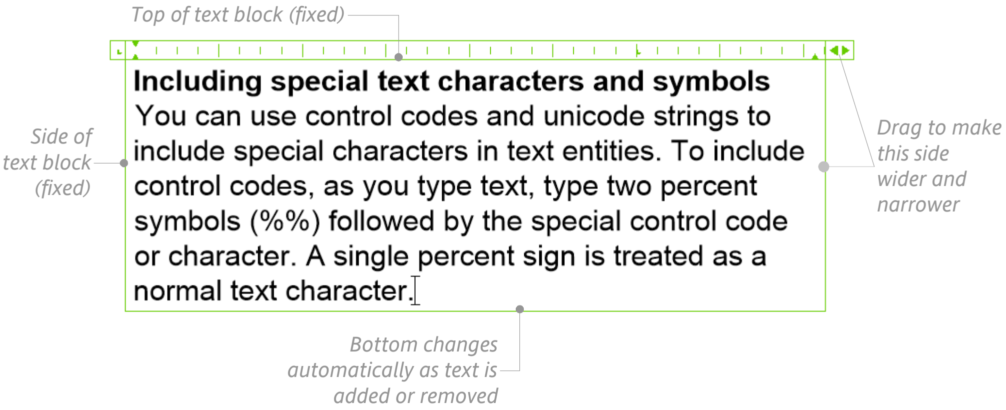
Command	MText
Aliases	MT, T
Menu Bar	Draw | Multiline Text
Ribbon	Draw | Text | Multiline Text
Toolbar	Tools | Multiline Text

You add callouts with the **MText** command, short for "multiple-line text" and known also as *paragraph text*.

Just as in a word processor, BricsCAD lets you format individual words and letters with **boldfacing** and *italics*, apply different fonts and colors, and include special symbols, such as the center line and the degree symbols. You can define indents for the margin (such as for the starts of paragraphs) and where tabs should be.

All this make the MText command flexible, but more complex than the simpler and less capable Text command you learn about later.

BricsCAD places multiline text in a rectangle, much like page design software does. In the figure below, the rectangle is shown in green.

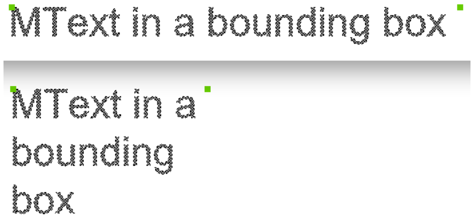


Including special text characters and symbols
You can use control codes and unicode strings to include special characters in text entities. To include control codes, as you type text, type two percent symbols (%%) followed by the special control code or character. A single percent sign is treated as a normal text character.

Top of text block (fixed)
Side of text block (fixed)
Drag to make this side wider and narrower
Bottom changes automatically as text is added or removed

TIP You change the width of the mtext boundary box using grips. Here is how to do this:

1. Select the text. Notice the two handles (green grips) , one at each corner of the top of the boundary box.
2. Select a grip; notice it turns red.
3. Drag a grip wider and narrower. Notice that the word wrap (paragraph lengths) change.



MText in a bounding box

MText in a
bounding
box

The paragraphs of text are constrained by the rectangle's top and two sides. This allows you to make the text fit specific areas in drawings, such as in title blocks and note blocks. As you add more text to the block, it flows down.

You define the size of the rectangle with just two picks at opposite corners, and later you can resize the rectangle at any time — narrower or wider, taller or shallower. You can move that entire block of mtext to any where in the drawing.

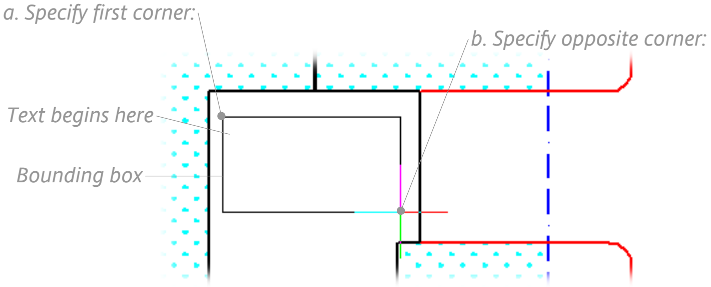
Place some mtext in the drawing now:

1. Start the **MTtext** command:

```
: mtext
```

2. Notice that BricsCAD wants from you a starting point. This is one of the two picks that define the initial size of the text rectangle. Pick the first point inside the house, as shown below:

```
Multiline Text: First corner for block of text: (Pick a point within the house outline)
```



3. Drag the cursor. As you do, notice that a rectangle is formed.
4. Pick a second point to define the width of the text:

```
Select Opposite corner for block of text or [Justification/Rotation angle/text Style/text
Height/Direction/Width]: (Pick a second point)
```

The rectangle forms a *boundary box* that constrains the text. The box is visible only when the MText command is active; it is otherwise neither displayed nor printed.

When you start typing the text you want in the boundary box, BricsCAD starts at the top of the bounding box. The right edge of the box will determine where the lines of text *wrap* — where the text flows to the next line. (For right-to-left text like Hebraic, it is the *left* edge that determines the line wrap location.) Adjusting the bounding box is like setting the margins in word processing software.

The bottom of the boundary is ignored, usually. When there is too much text for the box, BricsCAD automatically extends the bounding box downwards.

The bottom is used in only one case: when you want text to flow upwards, such as with revision notes. You change the direction of the flow with the command's **Direction** option.

QUICK SUMMARY ON CALCULATING TEXT HEIGHTS

Here are the steps to calculating the height of text appropriate for any size of drawing.

IN METRIC UNITS

Step 1: Work out the related drawing and paper widths, which will specify the scale factor:

$$\frac{\texttt{Width of drawing}}{\texttt{Width of paper}} = \frac{\texttt{40m}}{\texttt{0.297m}}$$

Step 2: Determine the scale factor by dividing the two values:

$$\frac{\texttt{40}}{\texttt{0.297}} = \texttt{134}$$

The scale factor is 134. That means text in the drawing should be about 140 times taller to plot big enough to be legible on the page. The standard for text heights in drawing is 3mm (0.003m), and for title text is 10mm.

Step 3: Multiply the standard text size by the scale factor:

```
0.003m  x  132  =  0.4m
```

In BricsCAD's MText and Text commands, specify a text height of 0.4m.

IN IMPERIAL UNITS

Step 1: Work out the related drawing and paper widths, which will specify the scale factor:

$$\frac{\texttt{Width of drawing}}{\texttt{Width of paper}} = \frac{\texttt{130 ft.}}{\texttt{11 in.}}$$

Step 2: To make the units consistent, convert the feet to inches:

$$\frac{\texttt{130 ft x 12 in/ft}}{\texttt{11 in.}} = \frac{\texttt{1,560 in.}}{\texttt{11 in.}}$$

Step 3: Determine the scale factor by dividing:

$$\frac{\texttt{1,560 in.}}{\texttt{11 in.}} = \texttt{142}$$

The scale factor is 142. That means text in the drawing should be about 140 times taller to plot big enough to be legible on the page. The standard for text heights in drawing is 1/8" (0.125"), and for title text 3/8" (0.375").

Step 4: Multiply the standard text size by the scale factor:

```
0.125"  x  142  =  18"
```

In BricsCAD's MText and Text commands, specify a text height of 18".

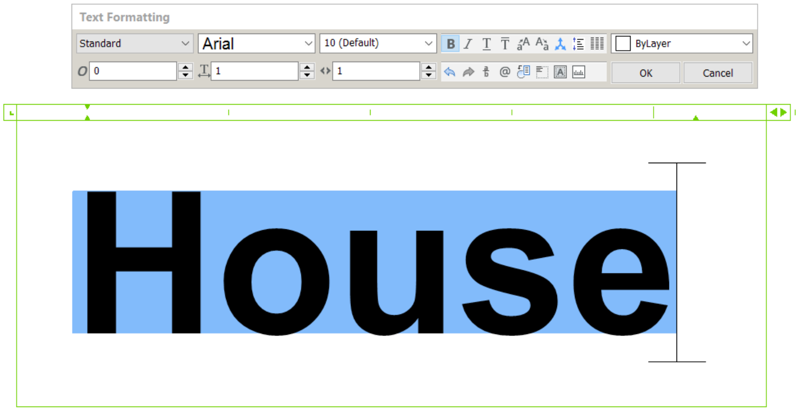
The reason for the difference between the metric scale factor of 134 and the imperial scale factor of 142 is that metric A4 paper is 18mm (0.7") wider than imperial A-size paper. See http://www.papersizes.org/a-paper-sizes.htm.

HOW TO DETERMINE THE SIZE OF TEXT

Now you need to pause to do some multiplication and division, because you have to work out the height of the text. The standard for the height of text is 10mm (or 3/8" in imperial units) tall in drawings. But if text were placed at that size, it would be nearly invisible when printed; after all, a few millimeters is tiny next to a 15-meter house! You need the text to be big enough to be legible, and so in this particular drawing it has to be larger, much larger.

To figure out how tall to make the text, read the boxed text on the next page, *Quick Summary on Calculating Text Heights*. Based on the size of this drawing, the text height needs to of 400mm tall (or 18" in imperial units).

4. After you pick the second point, BricsCAD displays the Text Formatting dialog box, in which you specify font settings. First, change the height, as follows:
   a. In the Font droplist, ensure "Arial" is specified.
   b. In the **Height** droplist, highlight 0.2, and then enter **0.4** (meters or 400mm).
   c. Press **Enter**. The height is set.



5. Type **House**, and then press **Enter**. Notice that the text appears in the drawing.
6. Click **OK** to exit the MText command.

CREATING TEXT STYLES

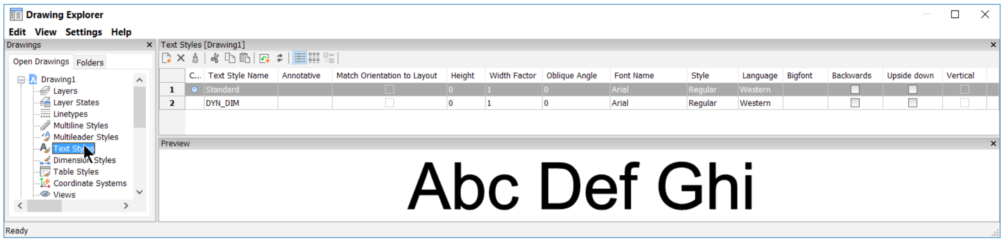
Command	Style
Aliases	st, ddstyle, expstyle, expstyles, expfonts
Menu Bar	Tools | Drawing Explorer | Text Styles
Ribbon	Home | Settings | Drawing Explorer | Text Styles

You can change the font used with text, but in CAD this is done indirectly with *styles*. "Styles" determine the look of the text, including which font should be used — just like in a word processor. The difference is that in CAD, styles *must* be used, where as in word processors they are optional.

Just as linetypes must be loaded into BricsCAD drawings, text fonts must be loaded before they can be assigned to styles, and this is done with the **Style** command.

1. Enter the **Style** command. Notice that the Drawing Explorer dialog box appears on the screen.

```
: style
```



	C...	Text Style Name	Annotative	Match Orientation to Layout	Height	Width Factor	Oblique Angle	Font Name	Style	Language	Bigfont	Backwards	Upside down	Vertical
1	●	Standard		□	0	1	0	Arial	Regular	Western		□	□	□
2		DYN_DIM		□	0	1	0	Arial	Regular	Western		□	□	□

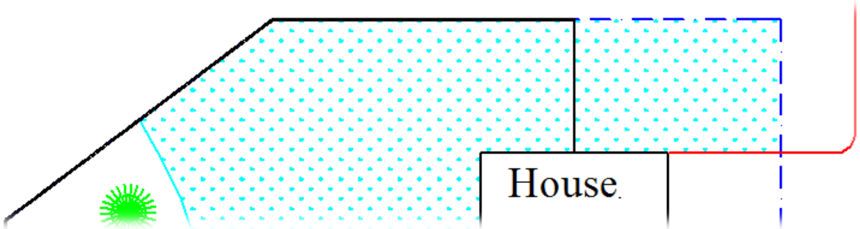
2. Make the following changes to the Standard style:

Property	Old Value	New Value
Font Name	Arial	Times New Roman
Height	0	0.75

(For imperial drawings, enter **26** as the height.) Ignore the other settings, as you don't need the text to be backwards, upside-down, or vertical.

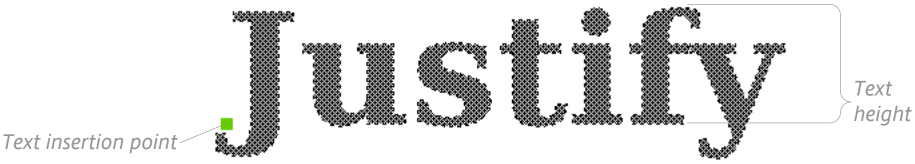
TIP A height of **0** has a special meaning in BricsCAD: it means that the text height is not predefined and so must be specified during the Text command.

3. Click **X** to close the dialog box. Notice that the House text changes immediately to the new font:



From now on, all text you place uses the Times New Roman font and is 0.75m tall (imperial units: 26 inches tall) — text that is half as tall as before.

Here is how BricsCAD measures the height of text: it starts at the *baseline* and then goes to the top of the *ascenders*.



SIMPLE TEXT

Command	Text
Alias	TX
Menu Bar	Draw | Text
Ribbon	Draw | Text | Text
Toolbar	Draw | Text

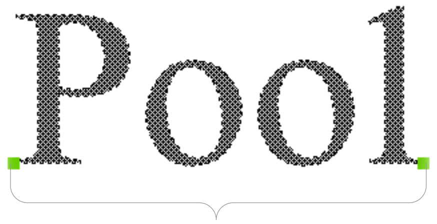
BricsCAD has another command for placing text that I find more efficient than MText. The **Text** command places a single line of text at a time. This makes it ideal for placing lots of bits of text all over the drawing, as needed.

To try out the Text command, first zoom in on the pond with **Zoom Window**.

1. Now that you see the pond area more clearly, start the **Text** command:

```
: text
```

2. Select the **Align** justification mode, which places the text fitted between two points that you pick:



Align between two points

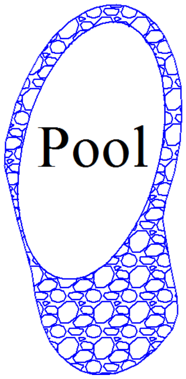
```
Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>: a
Text start point: (Pick one end of the pond)
Text end point: (Pick other end of the pond)
```

Because the Align option's pick points define the width and the angle of the text, Text doesn't ask you for a rotation angle, as the command normally would with most other justification options.

3. The Text command goes straight to the prompt:

```
Text: Pool
```

Notice that the word "Pool" is drawn with the Times New Roman text font, as specified by the Standard style.



4. Press **Enter** at the 'Text:' prompt to exit the command.

```
Text: (Press ENTER)
```

Changing Text

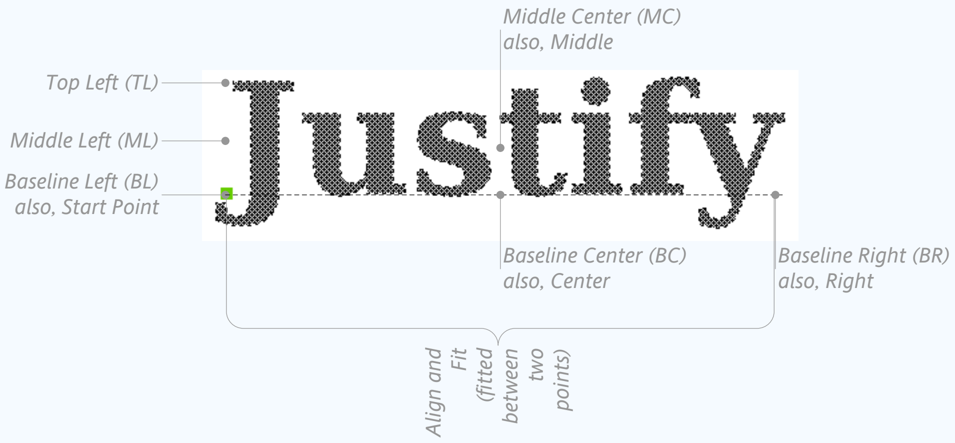
Command	DdEdit
Alias	ED
Shortcut	*double-click text*

Oops! “Pool” should read “Pond.”

To change the wording, use the **DdEdit** command. You could enter the command name at the : prompt, but there is a better way/

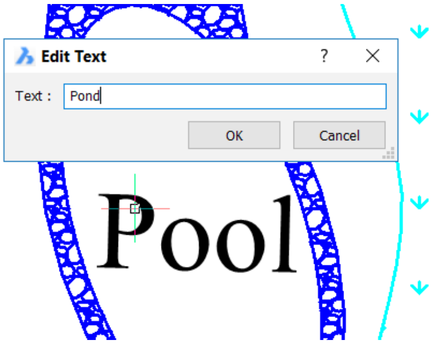
QUICK SUMMARY OF TEXT JUSTIFICATION MODES

BricsCAD can specify justification (alignment) for text in many different ways:



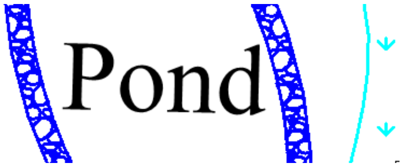
Justification	Meaning
Start point	Baseline left
Align	Fitted between two points
Fit	Fitted with constant text height
Center	Baseline center
Middle	Exact center of text
Right	Baseline right
TL	Top left
TC	Top center
TR	Top right
ML	Middle left
MC	Middle center (equivalent to Middle)
MR	Middle right
BL	Baseline left (equivalent to Start point)
BC	Baseline center (equivalent to Center)
BR	Baseline right (equivalent to Right)

1. Double-click the word **Pool**. This action activates the **DdEdit** command (short for "dynamic dialog editor"). Notice the Edit Text dialog box.



Text : Pond

2. Replace "Pool" with **Pond**.
3. Click **OK**. Notice that the word is corrected in the drawing.



PLACING ROTATED TEXT

Now we'll add more text, this time placing it sideways. BricsCAD can do that! First, though, perform a **Zoom All** to see the entire drawing.

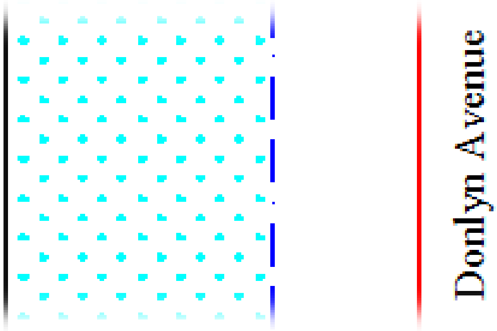
Restart the **Text** command, and then follow these steps:

1. Enter **Text**:

```
: text
Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>: (Pick a point on
street)
```

2. Specify a rotation angle of 90 degrees, and then enter the text:

```
Rotation angle of text <0>: 90
Text: Donlyn Avenue
Text: (Press Enter)
```



By specifying a rotation angle of 90 degrees, BricsCAD draws the text sideways. You can, of course, place text at any angle — from 0 degrees to 359 degrees.

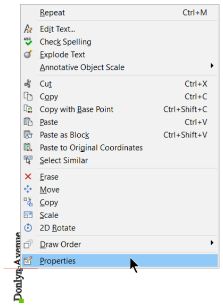
Changing the Angle of Rotated Text

If you placed the text at the wrong rotation angle, you change it after the fact.

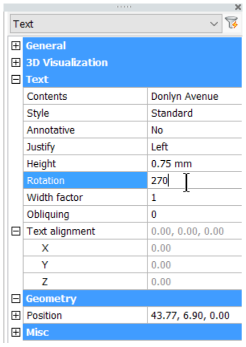
1. Use the **Zoom OBject** command to take a closer look at the text.
2. Now pick the text...

```
: (Pick text)
```

3. ..and then right-click the text. From the shortcut menu, choose **Properties**.



4. In the Properties palette:
   a. Look for the **Rotation** field under the Text section.
   b. Change 90 to **270**, and then press **Tab.**



Text

General
3D Visualization
Text

Contents	Donlyn Avenue
Style	Standard
Annotative	No
Justify	Left
Height	0.75 mm
Rotation	270
Width factor	1
Obliquing	0
Text alignment	0.00, 0.00, 0.00
X	0.00
Y	0.00
Z	0.00

Geometry

Position	43.77, 6.90, 0.00

Misc

5. Notice that BricsCAD rotates the text by 180 degrees. Press **Esc** to exit grips editing.

Placing Multiple Lines of Text

Now you use the Text command to place more callouts throughout the drawing. First, though, use **Zoom All** to see the entire drawing, and then follow these steps:

1. Use the **Text** command to add multiple callouts to the drawing.

```
: text
Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>: (Pick near the bottom
of the drawing)
```

2. Remember to change the rotation angle back to 0 degrees.

```
Rotation angle of text <90>: 0
```

3. Enter two lines of text, as follows:

```
Text: 4486 Donlyn Avenue (Press Enter)
Text: Anytown BC (Press Enter)
Text: (Press Enter to end the command)
```

Notice how BricsCAD places the second line of text precisely below the first.

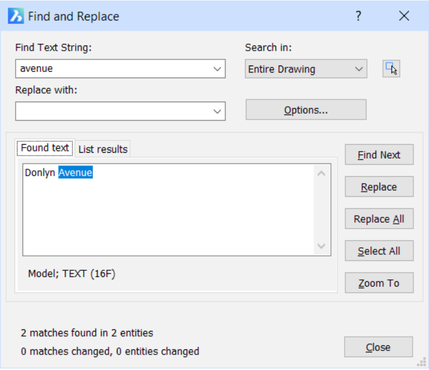
4. Repeat the command to add labels to trees, such as Birch, Aspen, Yellow Pine, or Western Red Cedar.

At this point, it is a good idea to save your work using the **Save** command.

FINDING AND REPLACING TEXT

Command	Find
Keyboard Shortcut	Ctrl+F
Menu Bar	Edit | Find
Ribbon	Draw | Text | Find

To find text in drawings and/or replace it with different text, use the **Find** command. This displays the following dialog box.


Find and Replace

Find Text String: avenue

Search in: Entire Drawing

Replace with:

Donlyn Avenue

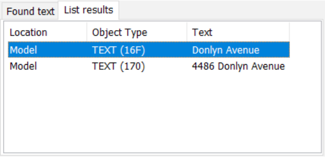
Model; TEXT (16F)

2 matches found in 2 entities

0 matches changed, 0 entities changed

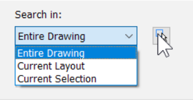
In the **Find Text String** field, enter the text for which you are looking, and then click **Find Next**. The dialog box offers two tabs with results:

- **Found Text** tab — displays the current instance of found text
- **List Results** tab — shows all instances



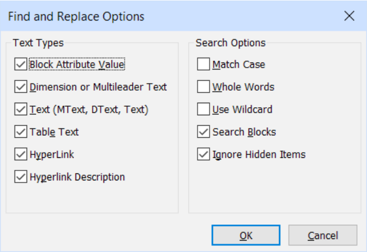
Location	Object Type	Text
Model	TEXT (16F)	Donlyn Avenue
Model	TEXT (170)	4486 Donlyn Avenue

When there is too much text in the drawing, you can narrow the search by searching only a specific area of the drawing. Click the **Search In** droplist and then choose from...


Search in: Entire Drawing

- **Current Selection** — searches for the text among currently selected entities
- **Current Layout** — searches for text only in the current layer, either Model space or a specific layout
- **Entire Drawing** — searches the entire drawing

Or, you can limit the search to a specific type of text. Click the **Options** button, and then choose the type of text from the list presented in the dialog box:



Find and Replace Options

Text Types

☑ Block Attribute Value
☑ Dimension or Multileader Text
☑ Text (MText, DText, Text)
☑ Table Text
☑ HyperLink
☑ Hyperlink Description

Search Options

□ Match Case
□ Whole Words
□ Use Wildcard
☑ Search Blocks
☑ Ignore Hidden Items

Text Types	Description
Block Attribute Value	Attribute text stored in blocks
Dimension Annotation Text	Text in dimensions, including leaders
Text (MText, DText, Text)	All regular kinds of text, including field text
Table Text	Text found in the cells of tables
Hyperlink	Text in URLs (Web links)
Hyperlink Description	Text in the descriptions of hyperlinks

And then there are even more options for narrowing the searches further:

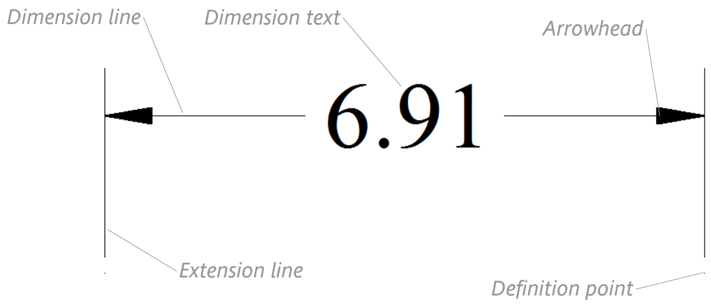
Search Options	Description
Match Case	Text must match the pattern of upper and lower case
Whole Words	Text has spaces or punctuation on either side, not part of larger words
Use Wildcard	Any text matching wildcards, such as * = any number of characters ? = any single character
Search Blocks	Searches for text inside of blocks
Ignore Hidden Items	Does not search for text on hidden and frozen layers

Dimensions in Drawings

With callouts firmly placed in the drawing, let's turn to dimensioning. This is done with commands that start with "Dim" (short for "dimensioning").

Now, a drawing properly created would not need dimensions. You can use the CAD program's Dist command to determine distances and angles, but this gets tedious. It really is much easier to see all important distances at a glance by dimensioning drawing. And so today, in this computer age, we still place dimensions in drawings.

Dimensions are made up of several elements, and the CAD program takes care of drawing all of them for us. The names of elements employed by all linear dimensions are shown below.



The bad news is that dimensions require dozens and dozens and *dozens* of settings to specify the look. Is the dimension text above, below, or in-line with the dimension line, centered or not centered; if above, how far above. And in what color? The list of options runs to over a hundred.

The settings are needed so that drawings to adhere to drafting standards set by standards organizations. The good news is that BricsCAD comes pre-configured with most settings set to reasonable values, in both metric and imperial units.

Settings are controlled by *dimensioning variables*, or "dimvars" for short. You can change dimvars in several ways:

- Adjusting values in the Dimension Styles section in the Drawing Explorer dialog box
- Directly entering their names at the ' : ' prompt
- After the dimension is placed, aspects of it can be changed with dimension editing commands

In the following tutorials, you use both methods.

TIP BricsCAD draws all components of dimensions automatically, according to the dimvar settings.

PREPARING DRAWINGS FOR DIMENSIONING

```
System Variable    DimScale
```

Like text, hatches, and linetypes, the scale of a dimension is relative to the printed size. Otherwise, arrowheads and text will be too small or too large to read. Before any drawing dimension, you should set the scale, like this:

1. To set the dimension scale, enter the **DimScale** dimvar like a command:

   ```
   : dimscale
   ```

2. The same scale factor that you calculated for text works here: **4** for metric units (or **142**, for imperial units).

   ```
   Enter new value for DIMSCALE <1>: 4
   ```

TIP The *height* of dimension text is controlled by **Style**, and is unaffected by the **DimStyle** scale factor.

3. With the **Layer** command, create a new layer named “Dims” with black color and continuous linetype.
4. Make the layer current.
5. If you wish, freeze unnecessary layers, such as Text, Lawn, Plants, and Pond. This keeps them from getting in the way of your dimensioning work. (For the illustrations below, I left some of these layers on so to make it easier to see the locations in the drawing where the dimensions are being applied.)
6. Because dimensioning often takes place at intersections, turn on **INTersection** entity snap mode, as follows:

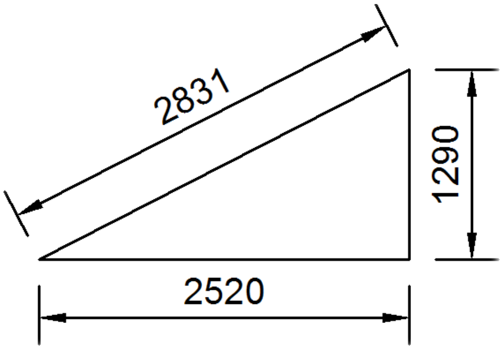
   ```
   : intersection
   ```

DIMENSIONING THE YARD

```
Command    DimLinear
Aliases    dli, dimlin, dimhorizontal, dimrotated, dimvertical
Menu Bar   Dimension | Linear Dimension
Ribbon     Draw | Dimensions | Linear
Toolbar    Draw | Leaders | Linear
```

```
Command    DimContinue
Aliases    dco, dimcont
Menu Bar   Dimension | Continuous
Ribbon     Draw | Dimensions | Continuous
Toolbar    Dimension | Continuous
```

By tradition, most dimensions are either horizontal or vertical. In rare cases, they are placed at an angle (“rotated”).



In this tutorial, you begin dimensioning with the most used dimensioning command, **DimLinear**. It draws horizontal, vertical, and rotated linear dimensions — depending on how you move the cursor.

BricsCAD determines how to construct the dimension from how you pick points in the drawing. Start DimLinear, pick a first point, then move the cursor, and you see the dimension jumping between horizontal and vertical. Depending on how you pick the second point, a horizontal or vertical dimension is drawn by BricsCAD:

- **Horizontal dimensioning** — pick two points that are roughly horizontal, BricsCAD draws the dimension perfectly horizontally
- **Vertical dimensioning** — pick two points roughly vertically, BricsCAD draws the dimension exactly vertical
- **Rotated dimensioning** — enter **R** to tell BricscAD that you want to place the dimension at an angle, and BricsCAD prompts you to specify the angle. (A more useful command for drawing dimensions at an angle, however, is DimAligned, which determines the angle automatically.)

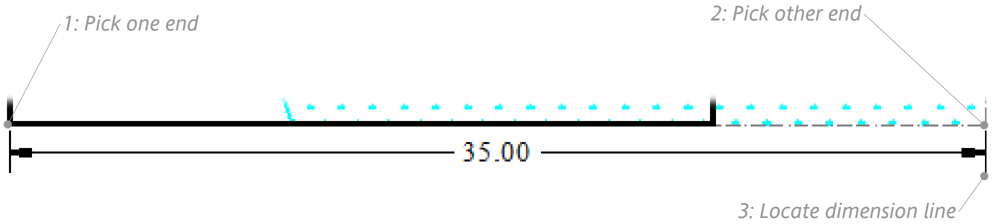
Start dimensioning, like this:

1. Begin the DimLinear commands.

```
: dimlinear
```

2. Dimension the lower lot line horizontally, as follows:

```
ENTER to select entity/<Origin of first extension line>: (Pick intersection near #1)
Origin of second extension line: (Pick other intersection at #2)
Angle/Text/Horizontal/Vertical/Rotated: (Locate dimension line by picking a spot below
the lot line, near #3 in the figure below)
```



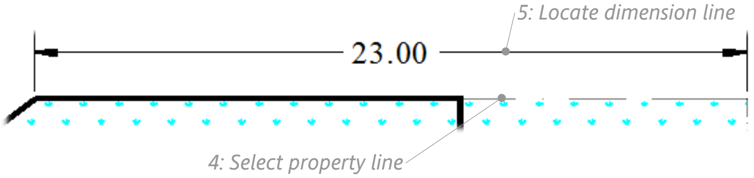
BricsCAD determines the length of the line from your two pick points, and then places all the elements of the dimension for you automatically, including determining the length between the two extension lines.

TIP The DimLinear command has several options
Angle — prompts you for a rotation angle for the text of the dimension
Text — lets you modify or replace the dimension text.
Horizontal — forces the dimension line to be horizontal
Vertical — forces the dimension line to be vertical
Rotated — prompts you for a rotation angle to the dimension line

Try another horizontal dimension of the top lot line. This time, use BricsCAD's *entity* dimensioning mode, in which you pick the entity and BricsCAD dimensions it. The method uses just two picks, instead of three.

3. Press the **Spacebar** to repeat the **DimLinear** command:

```
: (Press Spacebar to repeat command) _dimlinear
ENTER to select entity/<Origin of first extension line>: (Press Enter to select entities)
Select entity to dimension: (Pick property line at #4.)
Angle/Text/Orientation of dimension line: Horizontal/Vertical/Rotated: (Pick #5)
```

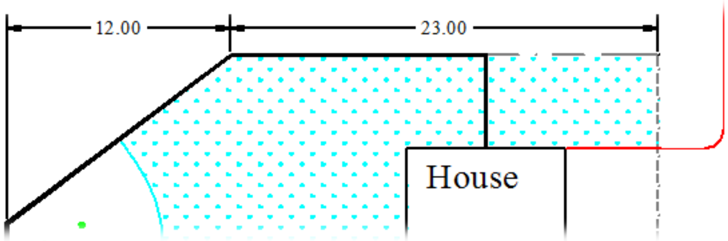


4. You can continue drawing horizontal dimensions with the **DimContinue** command. This is another way to dimension with fewer picks. When there is a lot of dimensioning to do, you want to minimize the number of picks! From the **Dimension** menu, select **Continue**:

```
: _dimcontinue
Continue:  ENTER to select starting dimension/<Origin of next extension line>: (Pick #6)
```

5. To end the command, it is important that you press **Esc**, because pressing Enter makes BricsCAD prompt you to choose the starting dimension.

```
Continue:  ENTER to select starting dimension/<Origin of next extension line>: (Press ESC
to end the command)
```



Because BricsCAD knows where the last extension line was, all it now needs to know the location of the *next* extension line to draw in the second dimension.

VERTICAL AND BASELINE DIMENSIONS

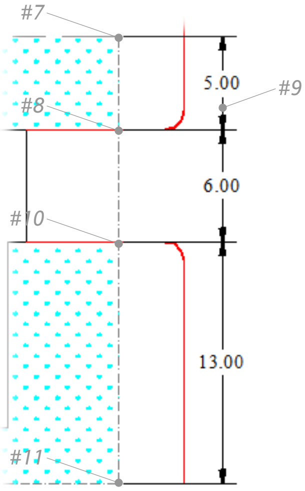
As I noted earlier, the DimLinear command also draws vertical dimensions. When you move the cursor in a vertical direction, BricsCAD knows to draw the dimension vertically.

1. To draw vertical dimensions, selec**t Dimension | Linear**, and then follow the numbered pick point from the illustration below.

```
: _dimlinear
ENTER to select entity/<Origin of first extension line>: (Pick #7)
Origin of second extension line: (Pick #8)
Angle/Text/Orientation of dimension line: Horizontal/Vertical/Rotated: (Pick #9)
```

2. Use **DimContinue** to continue the vertical dimensions along the right side of the lot at points 10 and 11.

```
: dimcont
Continue:  ENTER to select starting dimension/<Origin of next extension line>: (Pick #10)
Continue:  ENTER to select starting dimension/<Origin of next extension line>: (Pick #11)
Continue: ENTER to select starting dimension/<Origin of next extension line>:(Press Esc)
```



Baseline Dimensioning

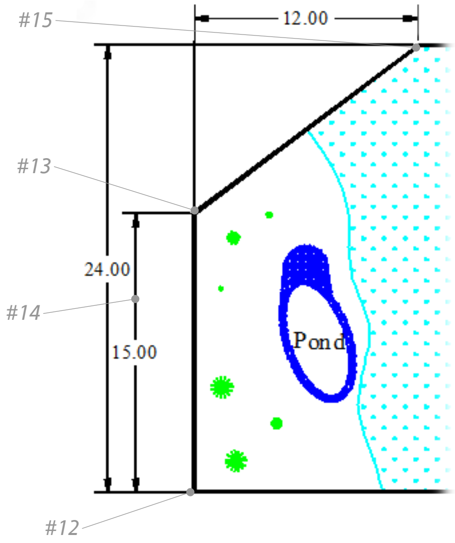
```
Command    DimBaseline
Aliases    dba, dimbase
Menu Bar   Dimensions | Baseline
Ribbon     Draw | Dimensions | Continue | Baseline
Toolbar    Dimensions | Baseline
```

A variation on the DimContinue command is the **DimBaseline** command. Rather than continuing a dimension from the *previous* extension line, DimBaseline continues dimensions from the *original* extension line (a.k.a. "the base line"). To see how it works, first place a vertical dimension:

3. With the **DimLinear** command, place the first extension line at #12. Place the second extension line at #13, and then the dimension line at #14.
4. Now try out the DimBaseline command.

```
: dimbaseline
Baseline:  ENTER to select starting dimension/<Origin of next extension line>: (Pick #15)
Baseline:  ENTER to select starting dimension/<Origin of next extension line>: (Press Esc)
```

Pressing **Esc** exits the command.



The DimBaseline and DimContinue commands also work with other linear and angular dimensions.

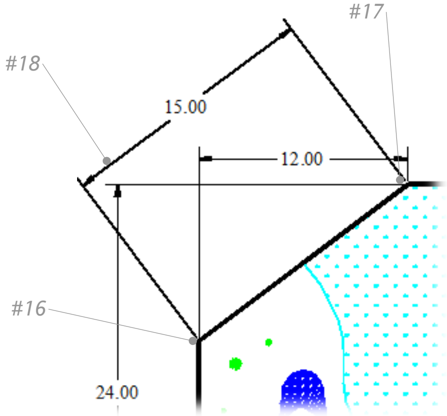
ALIGNED AND RADIAL DIMENSIONS

Command	DimAligned
Aliases	dal, dimali
Menu Bar	Dimension | Aligned Dimension
Ribbon	Draw | Dimensions | Aligned
Toolbar	Dimension | Aligned Dimension

So far, you have dimensioned the straight and angled portions of the lot line with horizontal and vertical dimension commands. To dimension an angled line, you use the **DimAligned** command.

1. Enter the **DimAligned** command:

```
: dimaligned
ENTER to select entity/<Origin of first extension line>: (Pick #16)
Origin of second extension line: (Pick #17)
Angle/Text/<Location of dimension line>: (Pick #18)
```



The dimension you draw may look different, depending on how the angled line was drawn.

Radial Dimensions

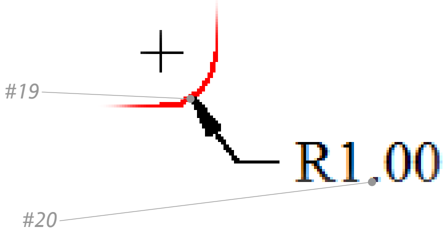
Command	DimRadius
Aliases	dra, dimrad
Menu Bar	Dimensions | Radius
Ribbon	Draw | Dimensions | Circle | Radius
Toolbar	Dimensions | Radius

So far, all dimensioning commands have presented pretty much the same series of prompts. Now here is one that's different. The **DimRadius** command dimensions arcs and circles. The related DimDiameter command places dimensions across diameters of circular entities.

2. (Zoom in for a closer look, if necessary.) Enter the **DimRadius** command, and then follow the prompts:

```
: dimradius
Select arc or circle: (Pick #19)
Dimension text = 1
Angle/Text/<Location of dimension line>: (Pick #20)
```

The DimRadius command gives you some flexibility as to where you want to place the dimension text. As you move the cursor, BricsCAD ghosts in the leader and text.



3. Save your work, and then print out a copy.

DIM, THE SUPER DIMENSION COMMAND

Command	Dim
Alias	dimension

I've had you using individual dimensioning commands, such as for linear and radial dimensioning, to understand how they work. Once you are expert at how they work, I suggest that you switch to Dim, the super dimensioning command.

It is "super" because

- It handles nearly all forms of dimensioning in a single command (just MLeader is missing)
- It knows the entity under the cursor and so draws the correct dimension for it
- It operates in entity mode, so you make only two picks: (1) the entity and (2) the dimension line
- It repeats itself automatically, so you don't need to pause to reload the appropriate command

When you enter the command, try not to be overwhelmed by the many options it presents, the longest of any command in BricsCAD:

```
: dim
Dimensioning command [HORizontal/VErtical/ALigned/ANgular/Leader/OBlique/ROtated/CEnter/
Diameter/RAdius/Baseline/COntinue/ORdinate/Position/DIStribute/UPdate dimensions/variable
STatus/OVerride/SEttings.../LAyer]:
```

Yip, that's 20 options! I counted them for you. The options are there in case you want to override its automatic entity-detection mode (the first 13 options), or else to edit an existing dimension. You've already used most of the dimension drawing commands, and so here is a summary of the editing options:

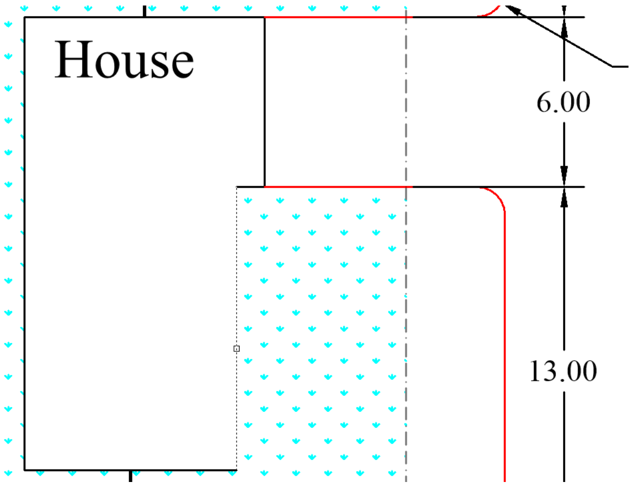
Dim Command Option	Description
Position	Repositions dimension text anywhere along the dimension lines, as well as by: **specify Angle** relative to dimension line **Left justify** along dimension line **Center justify** **Right justify** **REstore** text to original position
DIStribute	Spaces parallel or concentric dimensions by a specified distance
UPdate dimensions	Applies the current dimstyle to selected dimensions `
variable STatus	Lists the names of all dimvars and their current values
OVerride	Changes the value of a dimvar
SEttings...	Displays the Dimension Styles section of the Drawing Explorer
LAyer	Changes the name of DimLayer, the layer on which dimensions are placed

Let’s use Dim to dimension a line and an arc.

1. Start the **Dim** command:

```
: dim
Dimensioning command: (Move cursor over an entity)
```

2. Move the cursor over one of the lines making up the outline of the house. Notice that the line becomes dashed, as BricsCAD tells you it has located it.



In the command line, BricsCAD prompts you to click:

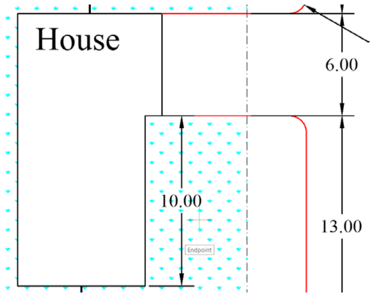
```
Select entity to dimension: (Click on the vertical house line)
```

3. Notice that BricsCAD draws the dimension. Now you need to position it:

```
Location of dimension line [Angle/Text]: (Move the cursor to position the dimension line)
```

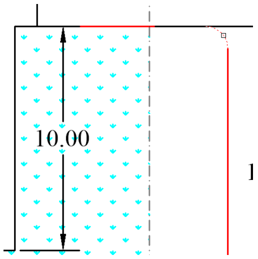
(The Angle and Text options let you set an angle for the text, and to edit the text.)

4. Click to locate the dimension line.



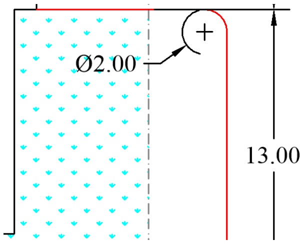
5. Notice that the Dim command is ready for another dimension. Now move the cursor over to one of the arcs of the driveway's curb return:

```
Dimensioning command: (Move cursor over the arc)
```



6. As before, click on the arc, and then position the text at the end of the leader line:

```
Select arc or circle to specify diameter or [Radial/Angular]: (Click on arc, and then posi-
tion the Leader)
Dimension text: = 2.00
```



(The **Radial** option switches between radius and diameter dimensions, while the **Angular** option switches to 3-point angular dimensioning.)

9. Press **Enter** to end the Dim command:

```
Dimensioning command: (Press Enter to end the command)
```

Annotatively-Scaled Text and Dimensions

In this and the previous chapter, you needed to work out the scale factor for *annotative* entities, specifically text, dimensions, hatch patterns, and linetypes. The scale of these entities takes into account the size of paper on which the drawing will be printed. If the paper size never changes, then all is fine.

Sometimes, however, the paper size does change, such as sometimes from A- (A4) to D-size (A1) paper. You need a handy way of changing the size of annotative entities, preferably automatically. BricsCAD provides it through annotative scaling.

When you turn on the annotative scaling option, BricsCAD determines the correct height to use for text, the correct scale factor for hatch patterns, and so on. Indeed, it can store several annotative scales per entity, hiding those scale(s) that are currently unneeded.

Think of annotative scaling as a "master scale factor" that affects only annotative entities, and then only in three areas of drawings:

- **Annotative entities** — whose size needs to change according to the output size
- **Viewports** — whose scale factor is related to the output size
- **Plots** — whose scale factor affects the output size

To understand this, I have put together a rule:

```
Annotative scale = Viewport scale = Plot scale
```

The scale at which you plot the drawing determines the scale factor of annotative entities. (If your drawing uses viewports, the same scale factor applies.)

A FEW EXTRA THINGS ABOUT ANNOTATIVE SCALING

Annotative scaling is tricky enough that some users avoid applying it to their drawings. Here are some things to watch out for.

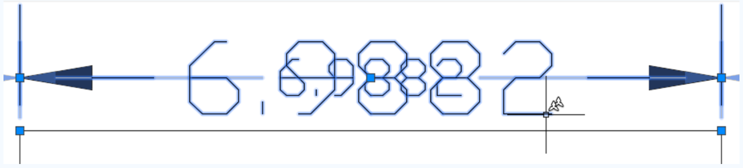
WHAT TO DO WHEN ANNOTATIVELY-SCALED ENTITIES GO MISSING

Annotative scaling can cause objects to disappear. The problem occurs when the viewport scale no longer matches the entities' annotative scale. This, however, makes sense: BricsCAD drawings can have multiple layouts, each with a different scale factor so that drawings can be printed on a variety of paper sizes. (This is why all drawings have just one Model tab but many Layout tabs: each Layout tab is normally assigned a different plot scale and/or a different view of the drawing.)

Annotatively-scaled entities only appear when the correct scale is set in model space.

VIEWING ALL ANNOTATIVE SCALES IN DRAWINGS

There is one exception to disappearing entities, and that's when multiple copies appear in the drawing. This occurs when more than one annotative scale is applies to entities. When the **AnnoAllVisible** system variable is turned on (= 1), then all variations appear, as illustrated below.



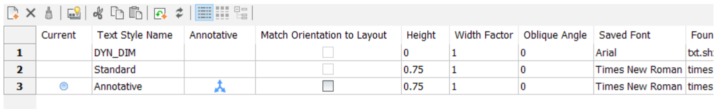
This can lead to some confusion as multiple-sized versions of text and other entities appear. The variable is meant for debugging: it's handy when you need to see all scales assigned to an annotative entity. To keep from seeing in double or triple vision, BricsCAD leaves AnnoAllVisible turned off by default.

HOW TO SPOT ANNOTATIVELY SCALED ENTITIES

An entity affected by annotative scaling shows the icon when you pass the cursor over it. When the entity has been assigned more than one annotative scale, then the double version of the icon is displayed. The icon represents the end view of an engineer's scale ruler, which allows engineers to work with multiple scales.

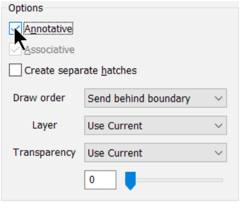
To use annotative scaling, you have to turn it on in styles by clicking the Annotative item in styles for text, multilines, dimensions, multiline leaders, and blocks.

Below, I clicked the **Annotative** property in the Drawing Explorer for text styles; I happened to name the text style “Annotative.”



	Current	Text Style Name	Annotative	Match Orientation to Layout	Height	Width Factor	Oblique Angle	Saved Font	Foun
1		DYN_DIM		□	0	1	0	Arial	txt.sh
2		Standard		□	0.75	1	0	Times New Roman	times
3	●	Annotative	⅄	□	0.75	1	0	Times New Roman	times

The same works for blocks, dimensions, multilines, and multiline leaders. For entities that don’t have styles, such as hatches and attributes, the dialog box with which you create them has the Annotative check box, as shown by the fragment of the Hatch and Gradient dialog box illustrated below.



Options

☑ Annotative

☑ Associative

□ Create separate hatches

Draw order: Send behind boundary

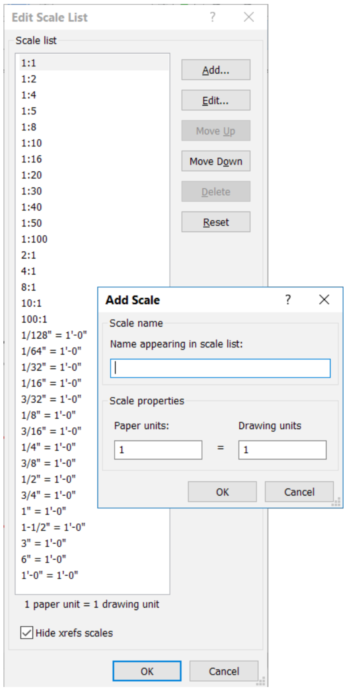
Layer: Use Current

Transparency: Use Current

0

Linetypes are a bit different, in that they use the **MsLtScale** variable to match the current annotative scale factor. Some other considerations:

- Annotative scales apply to entities, not to layers.
- You can’t just type one in any annotative scale factor; you are limited to the ones provided by BricsCAD. You can, however, edit them with the **ScaleListEdit** command to add and remove scale factors that your office uses or doesn’t use.



Edit Scale List

Scale list

1:1
1:2
1:4
1:5
1:8
1:10
1:16
1:20
1:30
1:40
1:50
1:100
2:1
4:1
8:1
10:1
100:1
1/128" = 1'-0"
1/64" = 1'-0"
1/32" = 1'-0"
1/16" = 1'-0"
3/32" = 1'-0"
1/8" = 1'-0"
3/16" = 1'-0"
1/4" = 1'-0"
3/8" = 1'-0"
1/2" = 1'-0"
3/4" = 1'-0"
1" = 1'-0"
1-1/2" = 1'-0"
3" = 1'-0"
6" = 1'-0"
1'-0" = 1'-0"

1 paper unit = 1 drawing unit

☑ Hide xrefs scales

Add Scale

Scale name

Name appearing in scale list:

Scale properties

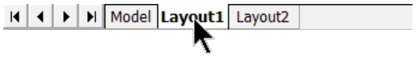
Paper units: 1 = Drawing units: 1

USING ANNOTATIVE SCALING

So far, I've been doing a lot of talking. Let's see how annotative scaling works for real. In the following tutorial, I have you switching back and forth between model and layout modes. *Layout mode* shows you what the drawing looks like on the paper you'll be printing it on; it is like an interactive print preview mode. (I've frozen the Dims layer so that the dimensions do not clutter the drawing.)

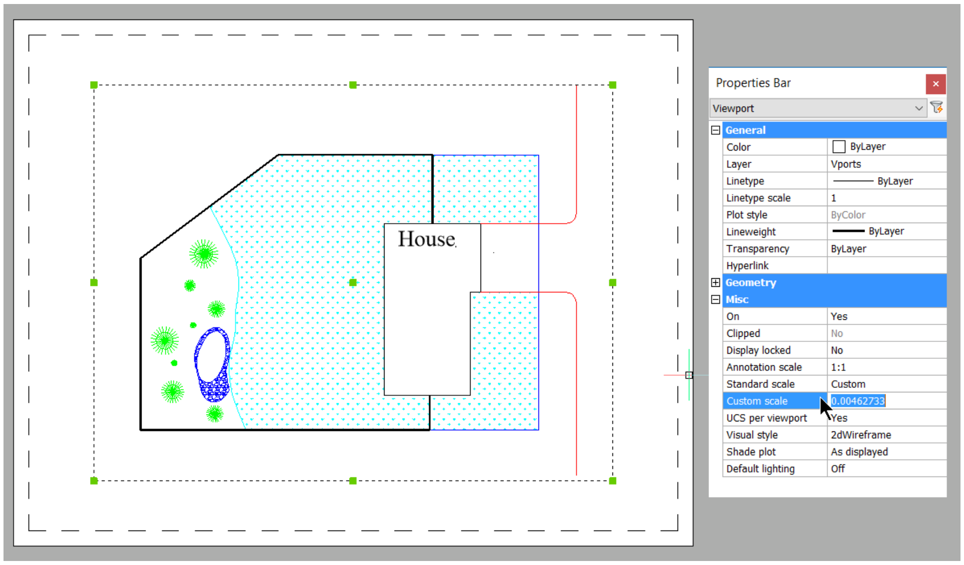
You find tabs for switching between the model and layout modes at the bottom of the drawing area.

1. Open the *Imp-Tutorial-06.dwg* file.
2. Switch to layout mode by clicking the **Layout1** tab.



Notice that a layout looks different from the default drawing area (a.k.a. model space):

- White rectangular area represents the paper size
- Dashed rectangle is the *margin*, the area at the edge of the paper on which the printer cannot print
- Black rectangle is the *viewport*, in which you see the model. In the figure below, the viewport has green grips and dashed lines, because I have selected it



Properties Bar

Viewport

General	
Color	□ ByLayer
Layer	Vports
Linetype	ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	
Geometry	
Misc	
On	Yes
Clipped	No
Display locked	No
Annotation scale	1:1
Standard scale	Custom
Custom scale	0.00462733
UCS per viewport	Yes
Visual style	2dWireframe
Shade plot	As displayed
Default lighting	Off

3. When the drawing is first opened in a viewport, it most likely is displayed at a non-standard scale. BricsCAD fits the model's extents inside the area of the viewport, and so this drawing has an initial scale of 0.00462733 (or 1:216.1073, expressed as a scale *factor*). To see the model's scale, select the viewport border, and then check the **Custom Scale** value in the Properties panel.

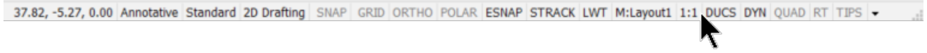
   Because 1:216.1073 is a non-standard scale factor, you have to change it to meet these two criteria:

   - The entire drawing must fit the viewport (usually somewhat smaller than the viewport)
   - The drawing must be at a standard scale

TIP To convert the *scale* into the *scale factor*, divide 1 by the scale. In this case
0.00462733 = 1 / 0.00462733 = 1:216.1073

4. You change the oddball scale to a standard one through a button on the status bar or the Properties panel. On the status bar, the annotative scale button appears near the right end next to DUCS, and probably looks like **1:1** right now.

   (If you do not see it, click the arrow at the right end of the status bar, and then turn on the **Annotative Scale button.)**

   The annotation scale button does double-duty of setting the scale for the viewport and for annotative objects.

5. Enter the model area by double-clicking inside the viewport border (the black rectangle), because annotative scaling works only in model space.

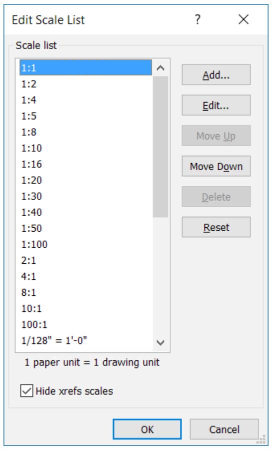
6. Use the **Zoom Extents** command to ensure that the drawing fits the viewport snugly. This gives you the largest scale permissible.

7. For the scale factor, pick the next largest whole number to 1:216.1073, because standard scale factors are always whole numbers, such as 1:250. Choosing the next largest number makes the drawing slightly smaller in the viewport. You don't want some of the drawing hidden by the edge of the viewport! (If the scale factor were something like 1:7.25, then pick 1:8.)

   There is, however, no scale factor listed at 1:250, so you need to add it with the ScaleListEdit command, as follows:

   a. Enter the **ScaleListEdit** command. Notice the Edit Scale List dialog box.

   ```
   : scalelistedit
   ```

   Edit Scale List

   Scale list

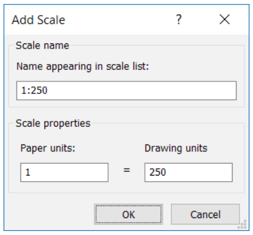
   1:1
   1:2
   1:4
   1:5
   1:8
   1:10
   1:16
   1:20
   1:30
   1:40
   1:50
   1:100
   2:1
   4:1
   8:1
   10:1
   100:1
   1/128" = 1'-0"

   1 paper unit = 1 drawing unit

   ☑ Hide xrefs scales

   b. In the Edit Scale List dialog box, click **Add**.

   c. For the 'Name appearing in scale list' you can type anything. For this tutorial, enter **1:250**.

   Add Scale

   Scale name

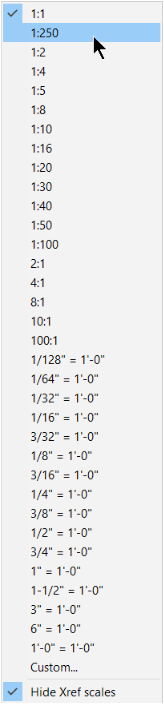
   Name appearing in scale list: 1:250

   Scale properties

   Paper units: 1 = Drawing units 250

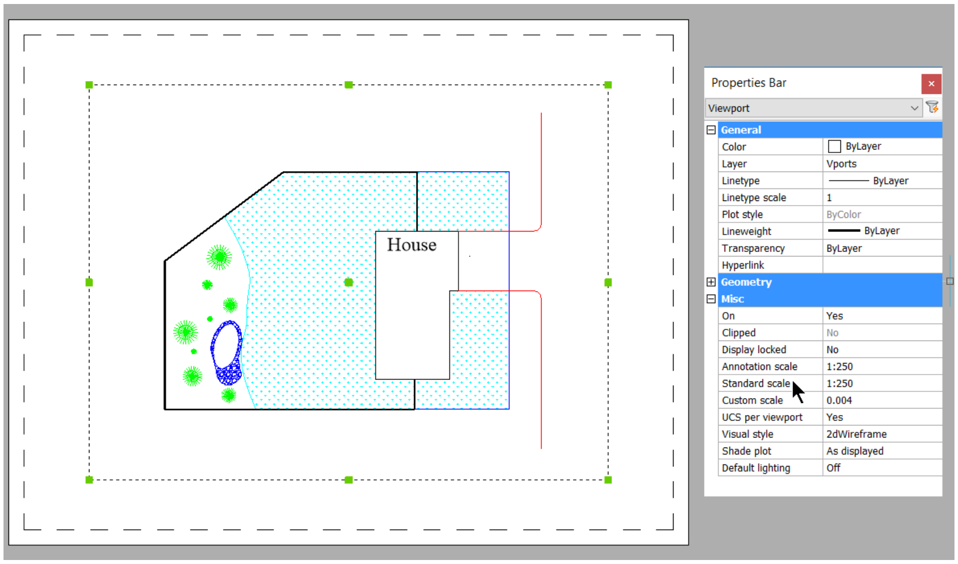
   d. For 'Drawing Units' enter **250**. Leave the Paper Units set to 1. Click **OK**.

8. To pick the annotative scale factor, right-click the **Annotation Scale** button (**1:1**), and then select **1:250**.



You know that you have picked the correct scale when the drawing becomes slightly smaller, but not too much smaller. (See figure below.) Notice also that the reported scale matches the annotation factor you selected. **From now on, the viewport scale is locked to the annotation scale.** This means that any change you make to the annotation scale changes the viewport scale automatically.

If the drawing does not change its size in the viewport automatically, you will need to make the change manually. In the Properties panel, change the value of **Viewport Scale** to match **Annotation Scale**, or 1:4 for this tutorial.



Viewport

⊟ General

Color	□ ByLayer
Layer	Vports
Linetype	ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	

⊞ Geometry

⊟ Misc

On	Yes
Clipped	No
Display locked	No
Annotation scale	1:250
Standard scale	1:250
Custom scale	0.004
UCS per viewport	Yes
Visual style	2dWireframe
Shade plot	As displayed
Default lighting	Off

9. With the annotative scale factor set up, test the annotation scaling feature by placing two kinds of text in Model tab, standard and annotative. First the standard text:

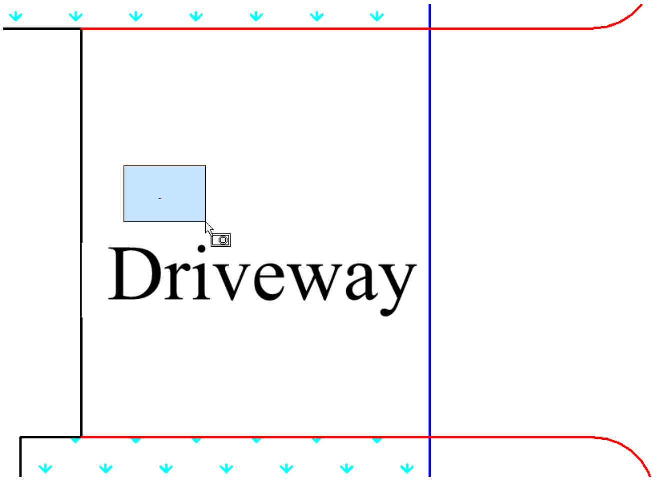
a. Click the **Model** tab.

TIP Why place the annotative text in model tab? (Why not in the layout tab?) My CAD-expert friend Bill Fane explains, "If you zoom and pan while entering annotative text in a layout tab, you mess up the viewport's scale; when you go back into model space, the annotations are missing, because their scale no longer matches the scale of the viewport. For this reason, it is best to enter annotative text in Model tab."

b. Enter the **Text** command to place regular text with the "Standard" style (which is non-annotative) at a height of 1/8" (0.125"):

```
: text
Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>: (Pick a point)
Height of text <0.2000>: 0.125
Rotation angle of text <0>: (Press Enter)
Text: Driveway
Text: (Press Enter)
```

Notice that the text looks very tiny. It is so tiny that I needed to highlight it with a blue selection rectangle; look for the short dash in the center. It is so tiny because it is placed at the size we normally use for text in drawings, 1/8". Recall from earlier in the chapter that we would normally scale it up by 250x and so draw it 31.25" high!



10. Now place the annotative text:

a. Restart the **Text** command, and then use the **Style** option to change the text style to an annotative one.

```
: text
Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>: s
Text style to use (or '?'): Annotative
Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>: (Pick a point)
```

b. Again place text with a height of 1/8".

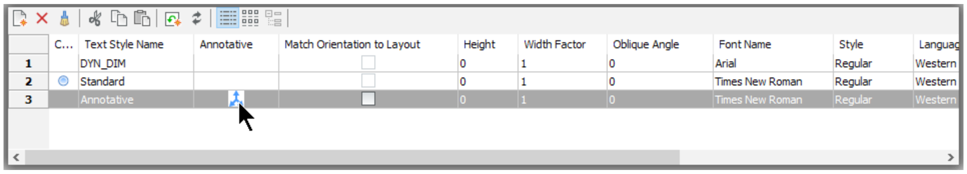
```
Height of text <0.2000>: 0.125
Rotation angle of text <0>: (Press Enter)
Text: Driveway
Text: (Press Enter)
```

Notice that the text looks a reasonable size. BricsCAD did the scale calculation for us and determined that the text should be 31.25" tall.

11. Any text you place now in Model tab will be scaled appropriately for the viewport and the scale factor. To see that the text appears at the correct size in the viewport, switch to **Layout1**. The text placed with the Standard style is effectively invisible, but the annotative text "Driveway" is correctly visible.

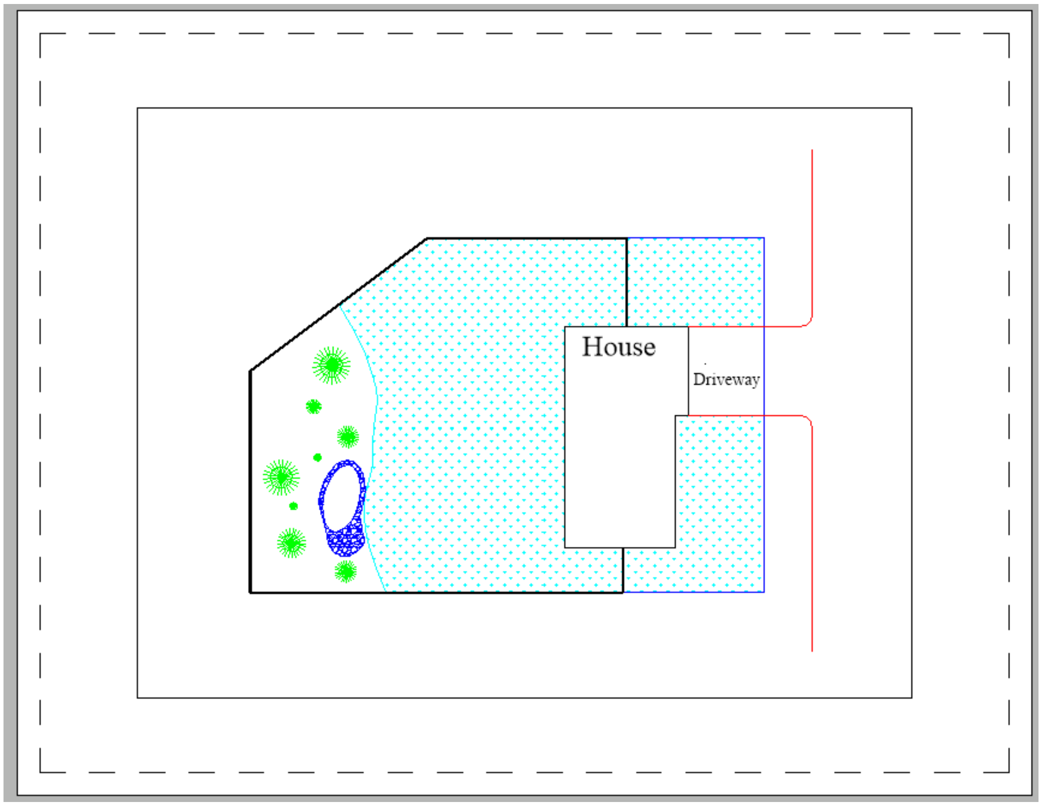
TIP If the drawing does not contain an annotative text style, then you can create one like this:

1. Enter the **Style** command.
2. Make a copy of an existing style by clicking the **New** button.
3. Click the **Annotative** option. It's that simple!



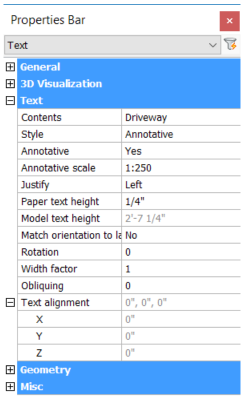
	C...	Text Style Name	Annotative	Match Orientation to Layout	Height	Width Factor	Oblique Angle	Font Name	Style	Languag
1		DYN_DIM		□	0	1	0	Arial	Regular	Western
2	○	Standard		□	0	1	0	Times New Roman	Regular	Western
3		Annotative	✓	□	0	1	0	Times New Roman	Regular	Western

4. Give the style a clever name, such as "Annotative," and then exit the Drawing Explorer.



12. Select the annotative text, and notice that the Properties panel reports two scales:

- Paper text height **1/4"**
- Model text height **2'7-1/4"** (same as 31.25")



Properties Bar

Text

General
3D Visualization
Text

Contents	Driveway
Style	Annotative
Annotative	Yes
Annotative scale	1:250
Justify	Left
Paper text height	1/4"
Model text height	2'-7 1/4"
Match orientation to la	No
Rotation	0
Width factor	1
Obliquing	0
Text alignment	0", 0", 0"
X	0"
Y	0"
Z	0"

Geometry
Misc

TIP Should you need to plot the drawing at a second scale, then I recommend you set up a second layout tab, set the new annotation scale, and then go back to model tab. Select the text and other annotations needing scaling, and then apply the new annotation scale factor.

Leaders and Multileaders

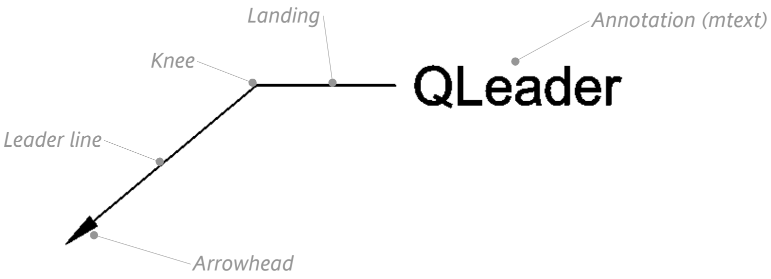
Sometimes you want to point out things in the drawing, and so this is where leaders come in. They point an arrow at the item, and have explanatory text at the other end of a connecting line.

BricsCAD offers commands for two types of leaders:

- **QLeader** command draw a single-line leader, with settings that allows lots of variations, such as curved lines and different shapes of arrowhead.
- **MLeader** command draws single- and multi-lined leaders, which are useful for pointing a single line of text at more than one item.

In both cases, the text at the end of the leader line is mtext. (There is also a Leader command, but it has fewer options than QLeader.)

Here are the parts of a typical leader:



In this tutorial, you place both kinds of leader in the drawing.

SINGLE-LINE LEADERS

```
Command    QLeader
Ribbon     Draw | Leaders | Leaders | QLeader
```

In this tutorial, you place a leader in the drawing to point out the fencing and specify its perimeter length, 77m (260 feet) .

1. Start the **QLeader** command using one of the methods listed above. (The ‘Q’ in QLeader is short for “quick,” a bit ironic in that this command has many more options than Leader.)

   ```
   : qleader
   ```

2. Pick a convenient point along the fencing. This is where BricsCAD will place the arrowhead, which is one end of the leader.

   ```
   Start of leader <Settings>: (Pick a point at the fence)
   ```

QUICK SUMMARY OF QLEADER

```
: qleader
Start of leader <Settings>:
Next point:
To point: <Undo>:
First line of annotation text:
Next line of annotation text:
```

Start of leader — starts drawing the leader line at the arrowhead end

Settings — displays the Settings dialog box

Next point — draws knee of the leader line, where it bends

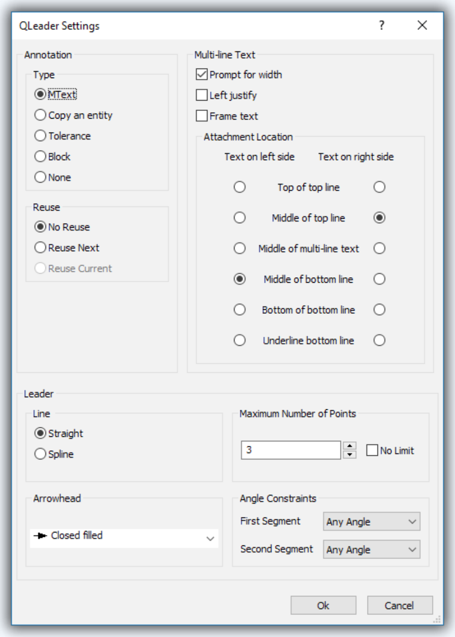
To point — continues drawing the leader line with a knee at each pick point, until you press **Enter**

Undo — reverses the drawing of the last leader line

First line of annotation text — enter text for the leader

Next line of annotation text — enters additional lines of text until you press **Enter**

SETTINGS DIALOG BOX



QLeader Settings

Annotation

Type
- ● MText
- ○ Copy an entity
- ○ Tolerance
- ○ Block
- ○ None

Reuse
- ● No Reuse
- ○ Reuse Next
- ○ Reuse Current

Multi-line Text
- ☑ Prompt for width
- □ Left justify
- □ Frame text

Attachment Location

Text on left side		Text on right side
○	Top of top line	○
○	Middle of top line	●
○	Middle of multi-line text	○
●	Middle of bottom line	○
○	Bottom of bottom line	○
○	Underline bottom line	○

Leader

Line
- ● Straight
- ○ Spline

Maximum Number of Points: 3 □ No Limit

Arrowhead: Closed filled

Angle Constraints
- First Segment: Any Angle
- Second Segment: Any Angle

Annotation — specifies the type of annotation to add: mtext (default), an entity from the drawing, a tolerance specification, a block, or no annotation at all.

Reuse — determines if the annotation used for the previous leader should be reused.

Leader — draws the leader as straight segments or as a spline

3. Pick another point. This is the "knee," where the leader line bends to become horizontal. This location is not crucial.

```
Next point: (Pick a point where you want the leader line to bend)
```

4. Press **Enter** to stop drawing the leader line. (If you keep clicking points, the leader line continues to be drawn with more knees.) This horizontal line is known as the "landing line."

```
To point: <Undo>: (Press Enter)
```

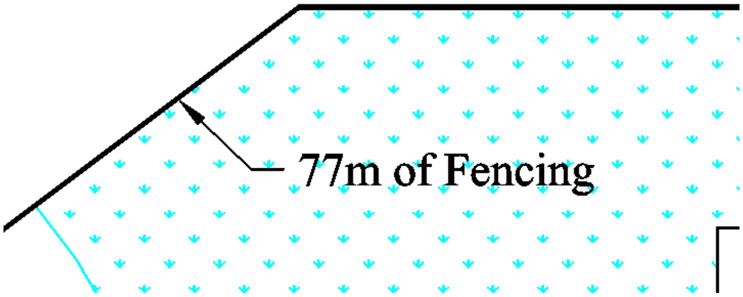
5. With the leader line drawing business done, now it is time to enter text.

```
Specify text width <0.00>: (Press Enter to accept)
First line of annotation text: 77m of fencing
```

6. Press **Enter** after entering '77m of fencing.' BricsCAD prompts you to enter another line of text:

```
Next line of annotation text: (Press Enter to end the command)
```

You can enter more text, or else just press **Enter** to end the command.



You can add more leaders to point out other parts of the landscape plan. To change the look of the leader, enter the **Format** option.

MULTI-LINE LEADERS

Command	MLeader
Menu Bar	Dimension | Multileader
Ribbon	Draw | Leaders | Leaders | Multileader
Toolbar	Dimensions | Multileader

Command	MLeaderEdit
Menu Bar	Dimension | Edit Multileader
Ribbon	Draw | Leaders | Leaders | Edit Multileader

In this tutorial, you place multiple leaders in the drawing to point out the stepping stones. First, you place one leader, and then you add a second one to the same annotation. This is known as a multi-leader. It is useful for pointing out multiple items in drawings.

1. Start the **MLeader** command:

```
: mleader
```

2. Pick a point at one of the stepping stones, roughly in the middle of them. BricsCAD will place the arrowhead here, just like the QLeader command.

```
Specify leader arrowhead location or [leader Landing first/Content first/Options] <Options>:
(Pick a point to start the mleader)
```

3. Pick a point for the knee:

```
Specify the leader landing location: (Pick a point)
```

4. Notice that BricsCAD goes immediately into annotation mode — without a prompt. The MText toolbar appears and you can start entering text. Enter text, such as...

   ```
   12" Stepping Stones
   ```

   ...and then click **OK** to end the command.

5. To add another leader line, start the **MLeaderEdit** command:

   ```
   : mleaderedit
   ```

6. Select the mleader you just drew:

   ```
   Select a multileader: (Select the mleader)
   ```

QUICK SUMMARY OF MLEADER

```
: mleader
Specify leader arrowhead location or [leader Landing first/Content first/Options]
<Options>:
```

Specify leader arrowhead location — starts drawing the leader line at the arrowhead end

leader Landing first — starts drawing the leader line in reverse, from the landing

Content first — start the leader by placing first the annotation

Options — continues drawing the leader line with a knee at each pick point, until you press **Enter**

OPTIONS OPTIONS

```
Enter an option [Leader type/leader lAnding/Content type/Maxpoints/First angle/Sec-
ond angle/eXit options] <eXit options>:
```

Leader type — specifies the leader type, straight, spline or none

Leader landing — toggles the leader landing

Content type — select the annotation types, multiline text, block or none

Maxpoints — specifies the maximum leader line nodes; default = 2

First and **second angle** — specifies the multiple* angle for the first leader segment; default = 0 degrees

Exit options — exits the options part of this command

* The angle specified here is used as a multiple for the actual angle. For instance, enter 15 to place the line at increments of 15 degrees, such as 15, 30 ,and 45.

ADDITIONAL MLEADER COMMANDS

MLeaderEdit adds and removes leader lines from multileaders.

MLeaderEditExt adds and removes leader lines and vertices from multileaders.

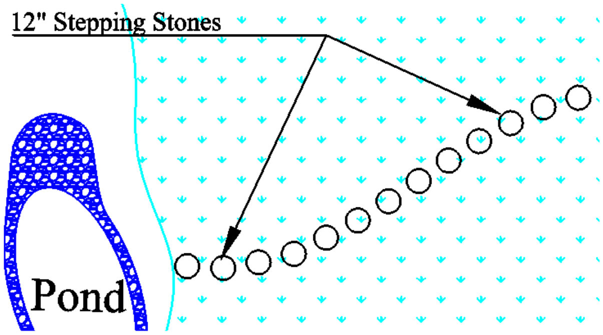
MLeaderStyle creates, edits, and deletes multileader styles (operates in Drawing Explorer).

aiMLeaderEditAdd adds multi-legged leaders.

aiMLeaderEditRemove removes leader lines.

7. As you drag the mouse, notice that the second leader line is drawn automatically, following the cursor like a puppy dog. Pick a point near another stepping stone to locate the arrow:

```
Specify leader arrowhead location or [Remove leaders]: (Pick a point at another
stepping stone)
```



8. Right-click to end the command. (You cannot press Enter to end the command, unlike most other repeating commands.)

```
Specify leader arrowhead location or [Remove leaders]: (Press the right mouse button)
```

Alternatively, you can use the undocumented **aiMLeaderEditAdd** command to add leader lines. This command differs from MLeaderEdit in that it adds multi-legged leaders.

The related **aiMLeaderEditRemove** command removes leader lines; be aware that the lines are removed *after* you press **Enter** to end the command — and not as you select them. This command also is undocumented.

This lesson showed how to add text and dimensions to drawings.

This tutorial is now complete. The following lessons describe other 2D CAD topics in greater detail, such as working with attribute data,and constructing 2D regions.

Notes

CHAPTER 8

Bills of Material

One of the most powerful aspects of CAD is its ability to generate information from drawings. You saw a hint of this in an earlier lesson when you used the Dist and List commands to find distances, lengths, areas, and totals of them.

In this chapter, you learn how to add information to drawings, and then extract it to create a bill of materials (BOM) in the drawing and in a spreadsheet.

IN THIS CHAPTER

- Defining attributes (custom data)
- Attaching attributes to blocks
- Exporting attributes and block data from drawings, then importing data into spreadsheets for analysis
- Importing spreadsheets into drawings as tables
- Automatic bills of materials from 3D component drawings

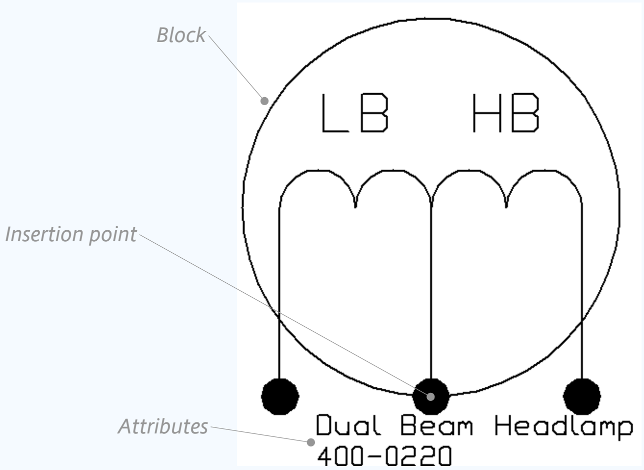
KEY TERMS IN THIS CHAPTER

Attribute — consists of text data attached to blocks

Attribute extraction — describes a process that exports attribute and block data to files

Balloon — leader that identifies parts in BOM tables

Block — describes BricsCAD's term for a symbol. (Other CAD software packages call blocks "components," "cells," "shapes," "symbols," or "parts.")



BOM — bill of materials

Data link — link established between an external data file and a table in a drawing

Donut — consists of a solid-filled circle

Insertion point — specifies the place at which blocks are inserted in drawings

Prompt — refers to the text displayed by BricsCAD during block insertions

Tag — identifies attributes by name

Template file — defines the format of data in export files

Value — specifies the default values of attribute data

NEW COMMANDS

Command	Aliases	Menu Bar	Ribbon Tab
AttDef	at	Tools | Attributes | Define Attributes	Insert | Block | Define Attribute
AttDisp	ad	Tools | Attributes | Attribute Display	Insert | Block | Attribute Display
bmBalloon	...	Assembly | Balloon	Assembly | Inquire | Balloon
bmBom	...	Assembly | Bill of Materials	Assembly | Inquire | Bill of Materials
bmExplode	...	Assembly | Assembly Explode | Explode	Assembly | Tools | Explode
DataExtraction	...	Tools | Data Extraction	...
DataLink	...	...	...
DataLinkUpdate	...	...	...
Donut	do, doughnut	Draw | Donut	Draw | Draw | Polygons | Donut
Table	...	Draw | Table	Draw | Tables | Table

About Attribute Data

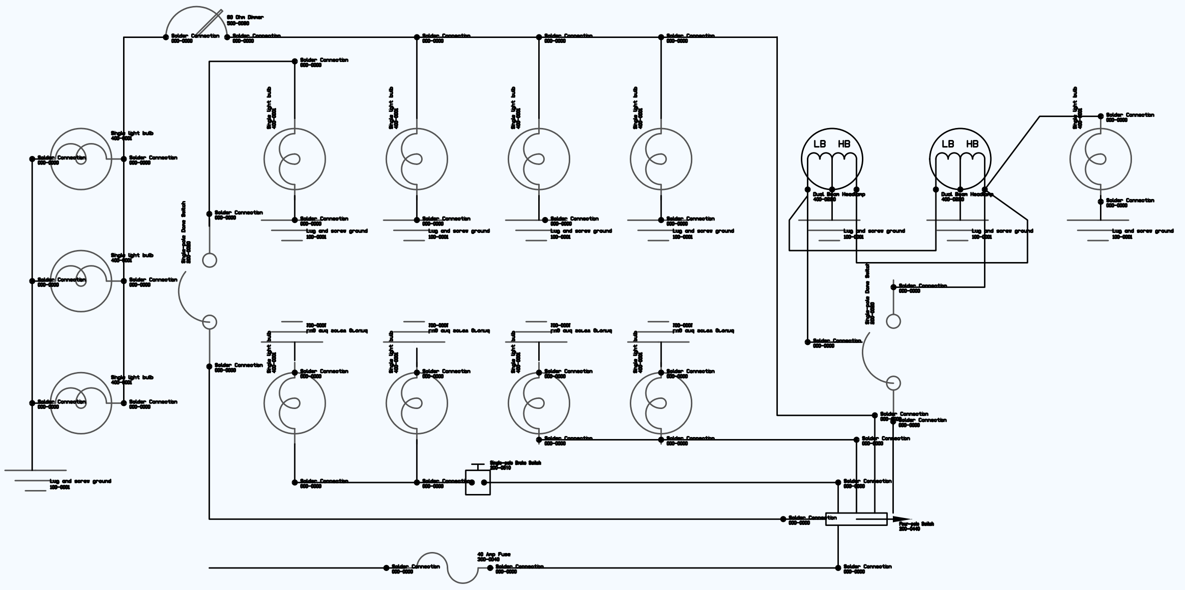
CAD can be used for many different kinds of drawings, floor plans, building construction, mechanical design, electrical layouts, and so on. Drawings usually are made from standard parts, such as desks, windows, bolts, and switches. When the parts contain *attribute data*, then drawings become truly useful.

In this chapter, you create a schematic drawing of an automobile's electrical system. The drawing contains numerous components, such as battery, fuse, several kinds of light bulbs, and quite a few ground and solder connections. You could count these components by hand, but I'd wager that you'd miss a few and end up with the wrong total. (Well, *you* might not miscount, but I know I would.)

It's faster and 100% accurate to let the software to do the counting. Part of the counting process is called "attribute extraction," because BricsCAD *extracts* attribute data from the drawing and then stores it in a data file. The data file can then be imported into a spreadsheet to count the components, determine costs, and so on. As well, the parts count can be placed in drawings as tables.

THIS CHAPTER'S DRAWING

During this lesson, you work with the following drawing file, *electric.dwg*.



This is very powerful stuff, and is known in many disciplines as BOM (short for "bill of materials") or in architecture as FM (short for "facilities management").

The attribute extraction process progresses through these steps:

Step 1: **Define Attributes** — define attributes with the AttDef command

Step 2: **Add Attributes** — use the Block command to merge attributes into symbols, the the Insert command to add the attributed blocks to the drawing

Step 3: **Export Attributes** — export the attributes with the DataExtraction command to a file or to a table in the drawing

In the first tutorial, you create a headlamp block, and then attach attribute data to it. In the following tutorials, you open a copy of the completed *electric.dwg* file, and then extract the attribute data from it. Later, you open the attribute data file in a spreadsheet program to count the parts, and finally you bring that data back into BricsCAD as a table.

Download the files for this chapter from
https://my.pcloud.com/publink/show?code=XZn98mkZ7T7f1pynxJYb7vDyVl6DwyqK1Ysk.

Creating Blocks with Attributes

The example drawing for this lesson is part of the electrical schematic of an antique automobile. See the drawing shown in the previous page. (Wondering which auto? It's adapted from the electrical system of my first car, the 1965 Volkswagen Beetle.) If you have repair manuals or other schematic drawings laying around, you can use them to produce your own schematic with BricsCAD. If not, then follow along with the schematic sketch I provide with this book.

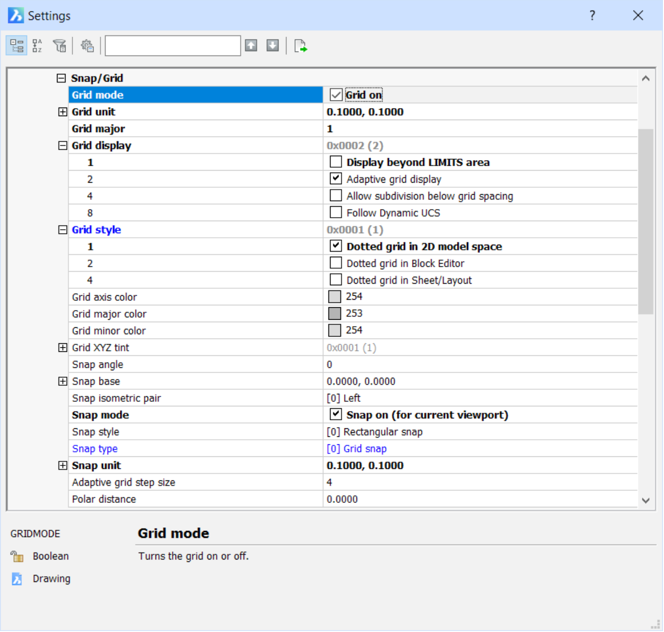
To begin, you create a single block and then attach attributes to it. Working with attributes is pretty tedious, which is why I won't get you to create *all* of the blocks. I've put them into a drawing file that you can download.

1. Start BricsCAD with a new drawing using the *Drawing* workspace (in metric units) or the *Default-cm.dwt* template file.
2. Because you will be drawing the block to an accuracy of 0.1 unit, it makes sense to change the snap spacing to this value, as well as to turn on the grid. Follow these steps:
   a. On the status bar, right-click **SNAP**, and then choose **Settings** from the shortcut menu.

   b. In the Settings dialog box, make these changes:

```
Grid Mode                  ✓ Grid on
Grid Unit                    0.1,0.1
Grid Major                   1
Grid Display                 Turn off Display beyond LIMITS area
Grid Style                 ✓ Dotted Grid in 2D model space

Snap Mode                  ✓ Snap on
Snap Unit                    0.1,0.1
```

The result should be like the bold-faced entries in the Settings dialog box illustrated below.



Settings

⊟ **Snap/Grid**	
Grid mode	☑ **Grid on**
⊞ **Grid unit**	**0.1000, 0.1000**
Grid major	**1**
⊟ **Grid display**	0x0002 (2)
1	☐ **Display beyond LIMITS area**
2	☑ Adaptive grid display
4	☐ Allow subdivision below grid spacing
8	☐ Follow Dynamic UCS
⊟ **Grid style**	0x0001 (1)
1	☑ **Dotted grid in 2D model space**
2	☐ Dotted grid in Block Editor
4	☐ Dotted grid in Sheet/Layout
Grid axis color	▢ 254
Grid major color	▨ 253
Grid minor color	▢ 254
⊞ Grid XYZ tint	0x0001 (1)
Snap angle	0
⊞ Snap base	0.0000, 0.0000
Snap isometric pair	[0] Left
Snap mode	☑ **Snap on (for current viewport)**
Snap style	[0] Rectangular snap
Snap type	[0] Grid snap
⊞ **Snap unit**	**0.1000, 0.1000**
Adaptive grid step size	4
Polar distance	0.0000

GRIDMODE

Boolean

Drawing

Grid mode

Turns the grid on or off.

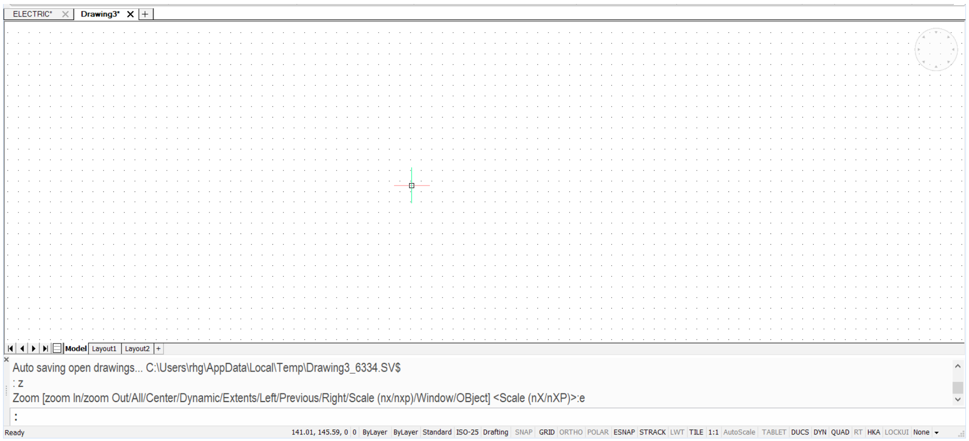
c. Click **X** when done to close the dialog box. Notice that the screen fills with a fine grid of lines. (Recall that the the extents of the grid display are determined by the limits.)

3. A couple more steps, and you will be ready to draw:

a. There is no need for the UCS icon, so turn it off with the **UcsIcon** command:

```
: ucsicon
[ucs icon ON/ucs icon OFf/display in All views/display at ORigin/display in Corner]
<ON>: off
```

b. Perform a **Zoom All** to center the drawing on the screen. All you see is the grid, for now.



```
Auto saving open drawings... C:\Users\rhg\AppData\Local\Temp\Drawing3_6334.SV$
: z
Zoom [zoom In/zoom Out/All/Center/Dynamic/Extents/Left/Previous/Right/Scale (nx/nxp)/Window/OBject] <Scale (nX/nXP)>:e
:
```

DRAWING BLOCKS

Command	Donut
Alias	doughnut
Menu Bar	Draw | Donut
Ribbon	Draw | Draw | Polygons | Donut
Toolbar	Draw | Polygons | Donut

The symbol that is used for solder connections is one that's really easy to draw. It is just a fat dot, and so is best drawn with the **Donut** command. This command creates solid looking circles out of wide polyline arcs. To draw one, you specify an outer and an inner diameter.

1. Start the **Donut** command.

```
: donut
```

2. The solder connection is 0.1 units in diameter. To make a solid-filled donut, you specify an inside diameter of zero. Enter the following sizes at the prompts:

```
Inside diameter of donut [2 Point/3 Point/Tangent Tangent Radius] <0.5>: 0

Outside diameter of donut <1>:  0.1
```

3. BricsCAD prompts you to place the donut, as follows:

```
Center of donut: (Pick anywhere in the screen)

Center of donut: (Press ENTER to exit the command)
```

The Donut command repeats itself until you exit it by pressing **Enter**.

4. That 0.1-unit donut sure is tiny. Enlarge the drawing area with the **Zoom Extents** command:

```
: zoom

Zoom [zoom In/zoom Out/All/Center/Dynamic/Extents/Left/Previous/Right/Scale (nx/nxp)/
Window/OBject] <Scale (nX/nXP)>:  e
```

(If the donut looks like it has straight sides, then use the **Regen** command to smooth the edges.)

```
: regen
```

DEFINING ATTRIBUTES

Command	AttDef
Menu Bar	Tools | Attributes | Define Attributes
Ribbon	Insert | Blocks | Define Attributes
Toolbar	Tools | Attributes | Define Attributes

With one solder connection drawn as a donut, you now create the attribute data for it. The process takes these steps:

Step 1: **AttDef** command defines attribute definitions

Step 2: **Block** command attaches attribute definitions to blocks

Step 3: **Insert** command defines attribute data during block insertion.

Optional step: **AttEdit** command edits attribute data after insertion (not in this tutorial)

Attributes are customized data that store information, such as part numbers and prices. Drawings are useful to showing you *how* to assemble the electrical system, but the bill of materials (derived from the attributes) tells you *how many* parts are needed for the assembly.

TIP Attributes attach only to blocks. It is possible to add data to other kinds of entities, but the process is more technical, because it involves *xdata* (short for "extended entity data"). Nevertheless, BricsCAD can export all data — such as attributes and properties — from all entities for further processing by data management software.

A single block can contain one or more attributes. Just as blocks are graphical descriptions, attributes are textual descriptions. Think of attributes as custom data, or user-defined properties. And only text or numbers can be used; graphical data, such as images and other entities, cannot be included in attributes.

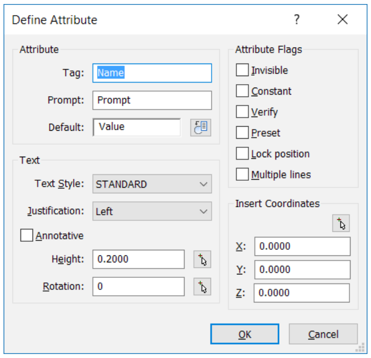
Attributes can be used to describe the block's part number, its manufacturer, the price, and any other text-based information you want to include. These attributes can have descriptive labels, such as "Product name," "Manufacturer," "Model number," "Stock number," "Serial number," and "Material."

You could include "Price" as an attribute field; but since prices tend to change, it is better to deal them in a spreadsheet program. Instead, I suggest that if you need prices in attributes that you use a price code instead, such as AA, which is substituted later with the latest price.

Attributes are created with the **AttDef** command (short for "attribute definition").

1. Start the **AttDef** command. Notice the Define Attribute dialog box.

```
: attdef
```



Define Attribute

Attribute
Tag: Name
Prompt: Prompt
Default: Value

Text
Text Style: STANDARD
Justification: Left
□ Annotative
Height: 0.2000
Rotation: 0

Attribute Flags
□ Invisible
□ Constant
□ Verify
□ Preset
□ Lock position
□ Multiple lines

Insert Coordinates
X: 0.0000
Y: 0.0000
Z: 0.0000

2. In the **Attribute** section, enter the following text:

Attribute Section	Value	Meaning
Tag	**Product**	Identifies the attribute to the program
Prompt	**Product**	Prompts the user about the data to enter
Default	**Solder Connection**	Default value of the data; can be overridden by the user

3. In the Text section of the dialog box, set the **Height** (of the text) to a tiny, unobtrusive value, such as **0.01**. You don't actually want to see attributes in drawings, and so you work with them only through dialog boxes.

(Attributes can be hidden entirely from view, but by being tiny you can tell where they are, but they don't get in the way.)

Text Section	Value	Meaning
Height	.01	Specifies the height of the attribute text

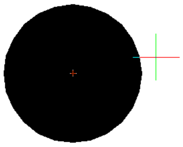
4. The **Insert Coordinates** section determines the point in the drawing at which the attribute text starts. A logical location would be on or near the block. To obtain the x,y coordinates, follow these steps:

   a. In the Insert Coordinates section, click the ↖ button. The dialog box disappears temporarily.

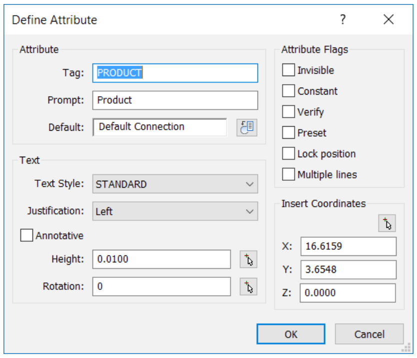
   b. In the command bar, notice that BricsCAD prompts you for the location of the insertion point.

   ```
   Specify insertion point: (Pick a point to the right of the donut)
   ```

   c. Pick a point to the right of the connection block:



As soon as you do, the dialog box returns. Its X, Y, and Z fields should now be filled out with values that look similar to the following:



Define Attribute ? ×

Attribute

Tag: PRODUCT

Prompt: Product

Default: Default Connection

Text

Text Style: STANDARD

Justification: Left

□ Annotative

Height: 0.0100

Rotation: 0

Attribute Flags

□ Invisible

□ Constant

□ Verify

□ Preset

□ Lock position

□ Multiple lines

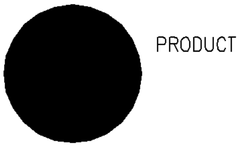
Insert Coordinates

X: 16.6159

Y: 3.6548

Z: 0.0000

5. Click **OK**. Notice that BricsCAD shows the tag next to the donut.



TIP Although blocks can contain multiple attributes, the **AttDef** command creates just one attribute at a time. You have to repeat the command to define additional attributes, up to a maximum of 245.

ADDING MORE ATTRIBUTES

For the Stock Number, you can add it as a second attribute directly below the first, like this:

1. Press the spacebar to repeat the **AttDef** command:

```
 : (Press spacebar)
 _attdef
```

And the Define Attribute dialog box reappears.

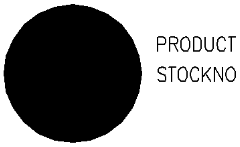
2. Fill in the following data in the Attribute section:

Attribute Field	Value
Tag	Stockno
Prompt	Stock No.
Value	000-0000

Ensure that the tag name “Stockno” is a single word, with no spaces.

3. Click the button, and then pick a point below the previous tag. (You find this easier to do if you turn off snap mode for now: click SNAP on the status bar.)

4. Back in the dialog box, click **OK**. Notice that BricsCAD adds the second attribute below the first.



The two attributes are identified by their *tags*, “PRODUCT” and “STOCKNO.” This text will change when the attributes are later combined with the donut to create a block, as you see in the next tutorial.

COMBINING ENTITIES AND ATTRIBUTES INTO BLOCKS

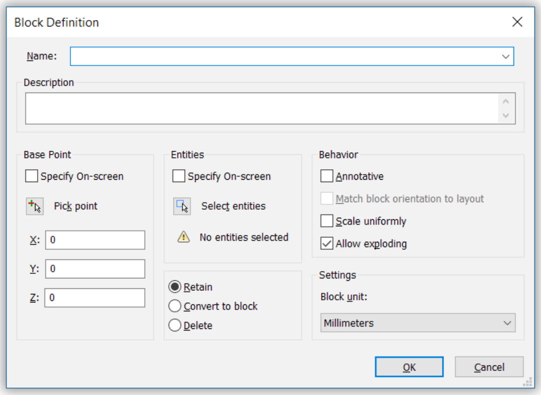
Command	Block
Alias	b
Menu Bar	Tools | Create Block
Ribbon	Insert | Blocks | Create Block
Toolbar	Tools | Create Block

You now have a donut with two attributes. The next step is to combine the donut and attributes into a single entity. This is done with the Block command.

1. Start the **Block** command.

```
: block
```

Notice the Block Definition dialog box.



Block Definition

Name:

Description

Base Point
□ Specify On-screen
Pick point
X: 0
Y: 0
Z: 0

Entities
□ Specify On-screen
Select entities
⚠ No entities selected
● Retain
○ Convert to block
○ Delete

Behavior
□ Annotative
□ Match block orientation to layout
□ Scale uniformly
☑ Allow exploding

Settings
Block unit: Millimeters

2. The first thing to do is enter a name for the block. In this case, enter “Connect” in the **Name** field:

Block Section	Value	Meaning
Name	**Connect**	Identifies the block to you and the program

You can ignore the **Description** field.

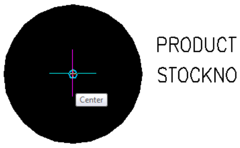
3. The most important aspect to blocks is their base point. The *base point* is the spot where the block will be inserted in drawings — either with coordinates or a cursor pick. The default is at the origin of the drawing at 0,0,0. However, other points might be more convenient, as described in the boxed text on the next page.

For this block, change the base point to the center of the donut:

a. In the **Base Point** section, click the **Pick Point** button. The dialog box disappears temporarily.

b. In the command bar, notice that BricsCAD prompts you for the location of the insertion point. Pick a point to the right of the connection block.

```
Insertion point for new block: (Pick the center of the donut)
```



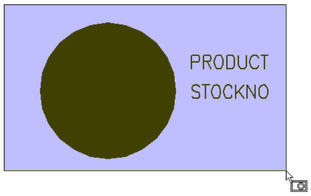
As soon as you pick the point, the dialog box returns.

4. BricsCAD needs to know which entities will be part of the block:

a. In the Entities section, click the **Select Entities** button. The dialog box disappears temporarily.

b. In the command bar, notice that BricsCAD prompts you to select the entities that you want to be part of the block. Select the donut and the text.

```
Select entities for block: (Select donut and two attribute entities)
```

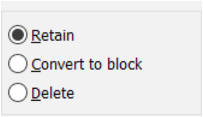


You can press **Ctrl+A** (or **Cmd+A** on Macs) to select all entities in the drawing, or else use windowing, like I did in the figure above.

c. Press **Enter** to end entity selection and then return to the dialog box.

```
Select entities for block: (Press ENTER)
```

5. Choose the **Retain** option. This determines what happens to the entities after you click **OK**.



● Retain
○ Convert to block
○ Delete

Option	Meaning
Retain	Retains the entities after defining them as a block
Convert to Block	Creates a block from the entities, and then inserts them in the drawing; this option combines two steps: erases the entities and inserts the block
Delete	Erases the entities after defining them as a block; this option saves using the Erase command to later remove the entities

6. Click **OK**. The drawing looks no different, but behind the scenes BricsCAD has converted the three entities into a *block definition*. Block definitions are stored in the *.dwg* file, and are used to insert them into the drawings, making them visible. See the next tutorial.)
7. Save the drawing with **Ctrl+S**, giving it the name "Symbols."

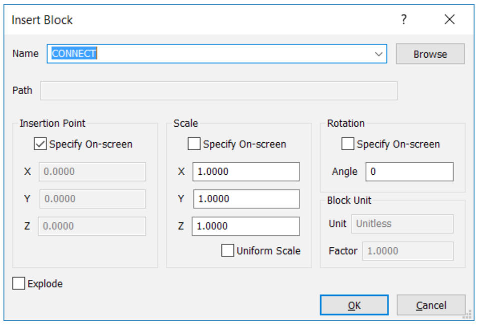
Inserting Blocks with Attributes

In an earlier lesson, you placed blocks with the Insert command. To see how attributes work, you will use is this command with the Connect block.

1. Start the **Insert** command:

```
: insert
```

2. Notice the Insert Block dialog box. Check that **Name** field contains "Connect." If not, select it from the droplist.



Insert Block

Name: CONNECT

Path:

Insertion Point
☑ Specify On-screen
X 0.0000
Y 0.0000
Z 0.0000

Scale
□ Specify On-screen
X 1.0000
Y 1.0000
Z 1.0000
□ Uniform Scale

Rotation
□ Specify On-screen
Angle 0

Block Unit
Unit Unitless
Factor 1.0000

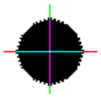
□ Explode

3. Change the following options in the dialog box:

Attribute Field	Value	Meaning
Insertion Point	✓ Specify On-screen	Point at which block is inserted in drawing
Scale X	*(off; scale = 1)*	Size of the block
Rotation	*(off; angle = 0)*	Angle at which the block is placed

4. Click **OK**.

Notice that the cursor is located at the block's center. This is the base point you defined earlier with the Block command; now it is known as the "insertion point."



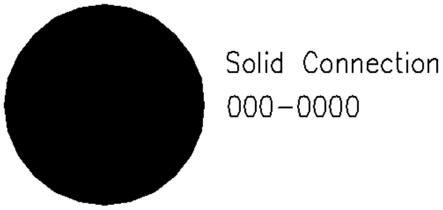
5. In the command bar, BricsCAD asks for the location of the block:

```
Insertion point for block: (Pick a point)
```

Notice that BricsCAD does not prompt you for the scale or rotation angle, because you specified these parameters in the dialog box (scale = 1, rotation = 0).

6. BricsCAD does, however, prompt you to enter the attribute values. You can press **Enter** to accept the default values:

```
Product <Solder Connection>: (Press Enter)
Stock No. <000-0000>: (Press Enter)
```

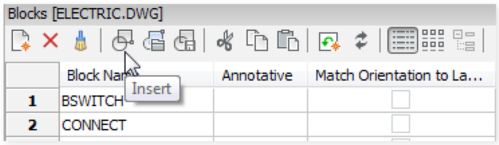


Notice that the attribute values appear next to the connection block in exactly the same location where you placed the tags.

ALTERNATIVES TO THE INSERT COMMAND

In addition to the Insert command, BricsCAD has several other methods for placing blocks in drawings:

- **Drawing Explorer** (ExpBlocks command) **—** accesses blocks stored in other drawings and inserts them into drawings with the Insert button



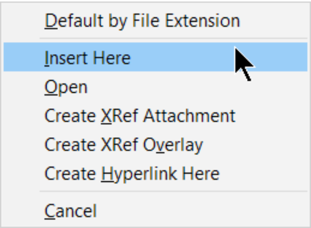
	Block Name	Annotative	Match Orientation to La...
1	BSWITCH		□
2	CONNECT		□

- **-Insert** command — displays insertion options in the command bar, and is meant for use in scripts and macros
- **Drag'n drop** technique — drags *.dwg* files from the operating system's file manager right into the drawing

Inserting Blocks with Drag'n Drop

When you use drag'n to drag *.dwg* files from the file manager and drag them into the drawing, BricsCAD reacts differently depending on which mouse button you hold down during the action:

- Hold down the **left** mouse button: the file opened as a drawing, like using the Open command
- Hold down the **right** mouse button: BricsCAD gives you options through this shortcut menu:



The functions of these options are as follows:

Default by File Extension — places the drawing files according to their file types, with the appropriate commands:

File Extension	Command Activated	Result
Drawing *.dwg*	**Open** command	Inserted as a block
Raster images	**ImageAttach** command	Attached as a raster image
Unknown file types	**InsertObj** command	Inserted as an OLE entity (Windows only)

Insert Here — places the file as a block through the -Insert command

Open — opens the file as a drawing in a new window through the Open command

Create XRef Attachment — places the file as an externally-referenced drawing through the -XRef Attach command

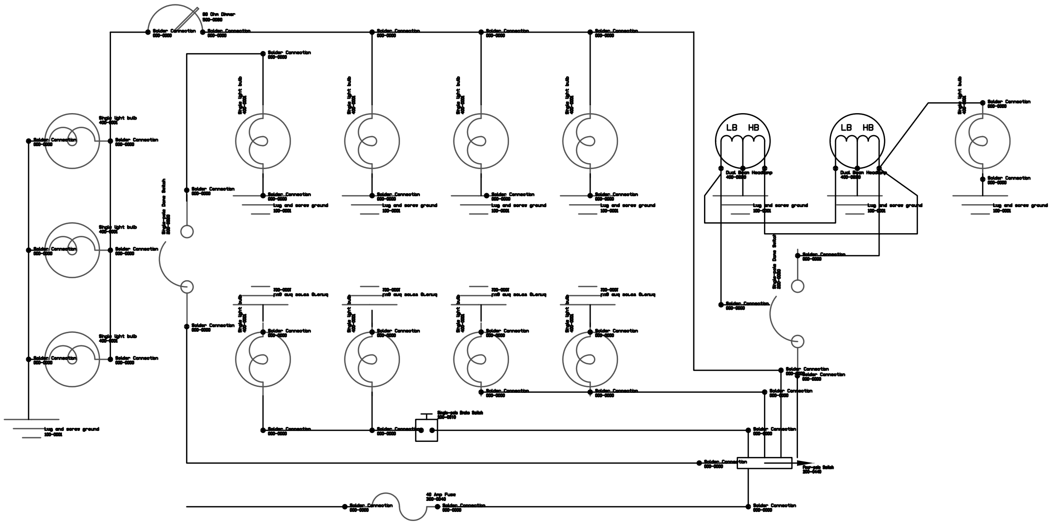
Create XRef Overlay — places the file as an overlain xref through the -XRef Overlay command

Create Hyperlink Here — prompts you to select one or more entities, and then attaches a URL to them that points to the file through the -HyperLink command

Cancel — cancels the operation

Exporting Data from Drawings

Adding attributes to blocks is tedious work, and so to save you time and trouble, I prepared a drawing to be used by the next set of tutorials. It contains all needed blocks and attributes. Get your copy of the *electric.dwg* file from my cloud folder: https://my.pcloud.com/publink/show?code=XZn98mkZ7T7f1pynxJYb7vDyVl6DwyqK1Ysk.



DATA EXTRACTION

```
Command    AttDisp
Alias      ad
Menu bar   Tools | Attributes | Attribute Display
Ribbon     Insert | Blocks | Attribute Display
Toolbar    Tools | Attributes | Attribute Display

Command    DataExtraction
Menu Bar   Tools | Data Extraction
```

To export data from drawings, you use the **DataExtraction** command. It saves the data in a CSV file, which is short for "comma delimited values." The CSV format separates each piece of data with a character, such as a comma. Software like spreadsheets and databases recognize the comma as the separator, and place the pieces of data into their own cells or fields. This make CSV a kind of universal exchange format for data.

Once the data from the CSV file is in a spreadsheet or database program, it can be processed further, such as adding up columns and creating charts. The processed data can then be imported back into BricsCAD. When the data comes from the Excel brand of spreadsheet, it is a linked table. This means that when you make a change in the Excel spreadsheet, the table in the drawing is updated. (Sorry, but this function is not available with LibreCalc and other competitors to Excel.)

(This command supersedes the aging AttExt command, because it does not need template files to be prepared first.)

To extract the attribute data from the drawing, follow these steps:

1. Launch BricsCAD, and then open the *electric.dwg* file.
2. To see *all* of the attributes, including invisible ones, use the **AttDisp** command. This step is not necessary for extracting attributes, because BricsCAD extracts both visible and invisible attributes. But turning them on makes it easier for you to monitor the task.

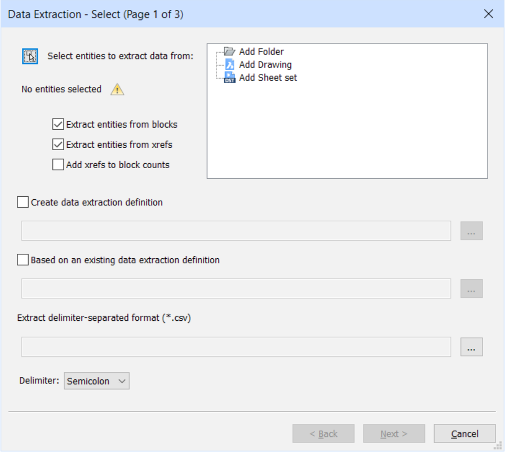
```
: attdisp
Attribute display mode: ON/OFF/Normal <Normal>: ON
```

TIP The **AttDisp** command changes the way BricsCAD displays attribute text:
Normal — displays attributes, except for those set to Invisible mode.
On — displays all attributes, including invisible ones.
Off — hides all attributes.

3. Start the attribute extraction entering the **DataExtraction** command.

```
: dataextraction
```

Notice the Data Extraction dialog box.



Data Extraction - Select (Page 1 of 3)

Select entities to extract data from:

- Add Folder
- Add Drawing
- Add Sheet set

No entities selected

☑ Extract entities from blocks
☑ Extract entities from xrefs
☐ Add xrefs to block counts

☐ Create data extraction definition

☐ Based on an existing data extraction definition

Extract delimiter-separated format (*.csv)

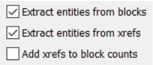
Delimiter: Semicolon

4. Click the **Select** button to select all of the entities in the drawing, like this:

```
Select entities: all
Entities in set: 124
Select entities: (Press Enter to return to dialog box)
```

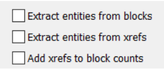
The number of entities found may differ in your drawing from mine, depending on how you drew it.

5. Back in the dialog box, notice the first set of options:



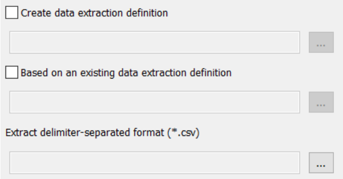
☑ Extract entities from blocks
☑ Extract entities from xrefs
☐ Add xrefs to block counts

You are only interested in extracting attribute *data* from blocks, so turn off all options. The section now looks like this:



☐ Extract entities from blocks
☐ Extract entities from xrefs
☐ Add xrefs to block counts

6. The second set of options specify files that are needed. Let's go through them one by one:



☐ Create data extraction definition

☐ Based on an existing data extraction definition

Extract delimiter-separated format (*.csv)

Create Data Extraction Definition — I recommend that you turn on this option.

When on, BricsCAD saves the options you chose to a DXD file (short for "data extraction definition"). The next time you run this data extraction command, you don't have to set things up all over again. You just use the "Based on an existing data extraction definition" option. You'll see the benefit of this when you get to the next dialog box!

Based on an Existing Data Extraction Definition — Reuse the DXD file from a previous session; this option does not apply the first time you run this command on a drawing.

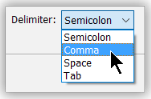
Extract Delimiter-separated Format — Specify the file name and folder location for the extracted data. Data is saved in CSV format, which separates data values with commas: one record per row, with fields separated by commas. You have to fill out this option, as BricsCAD prevents you from moving further without it by keeping the Next button deactivated.

For ease of reference, specify these names for each file:

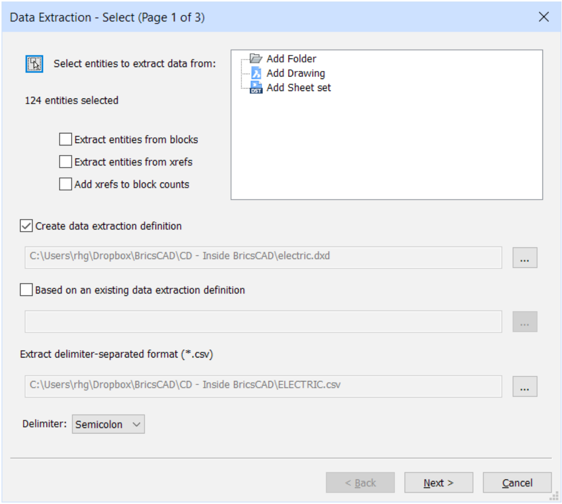
`Create Data Extraction Definition`	**`electric.dxd`**
`Based on an Existing Data Extraction Definition`	*`(ignore this time around)`*
`Extract Delimiter-separated Format`	**`electric.csv`**

For each of the options that you are using, click the ... **Browse** button, and then choose a convenient folder location. Specify the file names as 'electric' with appropriate extensions (*.dxd* and *.csv*).

7. Click the **Delimited** droplist, and change it to 'Comma." A *delimiter* is punctuation that separates data values in the CSV file. It can be a semi-colon, a comma, a space, or a tab. I recommend using the comma.



When you are done, the dialog box should look something like this.


Data Extraction - Select (Page 1 of 3)

Select entities to extract data from:

124 entities selected

☐ Extract entities from blocks

☐ Extract entities from xrefs

☐ Add xrefs to block counts

☑ Create data extraction definition

C:\Users\rhg\Dropbox\BricsCAD\CD - Inside BricsCAD\electric.dxd

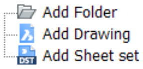
☐ Based on an existing data extraction definition

Extract delimiter-separated format (*.csv)

C:\Users\rhg\Dropbox\BricsCAD\CD - Inside BricsCAD\ELECTRIC.csv

Delimiter: Semicolon

(The area in the upper right is useful for making a massive extraction of data by adding entire folders of drawings, individual drawings, and sheet set collections of drawings.)

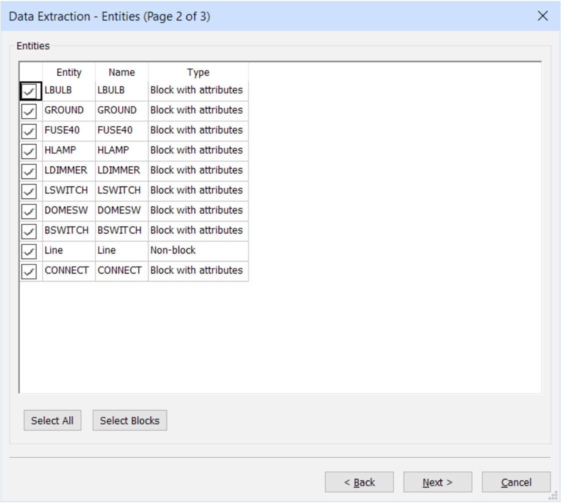


8. Click **Next**.

   Notice that the Page 2 dialog box lists the name of blocks and of line entities. You want data about all of them extracted, so leave everything as it is. Even the line entities are useful, because from them you get the length wiring.

   Here is the meaning of the options:

   - **Check boxes** — to exclude an item, click the check box
   - **Select all** — selects all items in the list (turns on all check boxes)
   - **Select blocks** — selects just blocks, and avoids non-block entities (like the lines)

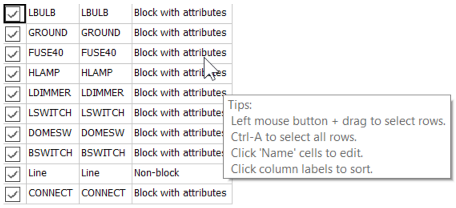


Data Extraction - Entities (Page 2 of 3)

Entities

	Entity	Name	Type
☑	LBULB	LBULB	Block with attributes
☑	GROUND	GROUND	Block with attributes
☑	FUSE40	FUSE40	Block with attributes
☑	HLAMP	HLAMP	Block with attributes
☑	LDIMMER	LDIMMER	Block with attributes
☑	LSWITCH	LSWITCH	Block with attributes
☑	DOMESW	DOMESW	Block with attributes
☑	BSWITCH	BSWITCH	Block with attributes
☑	Line	Line	Non-block
☑	CONNECT	CONNECT	Block with attributes

TIP When you pause the cursor over the data fields, a tooltip advises you on additional operations that are possible in this dialog box:

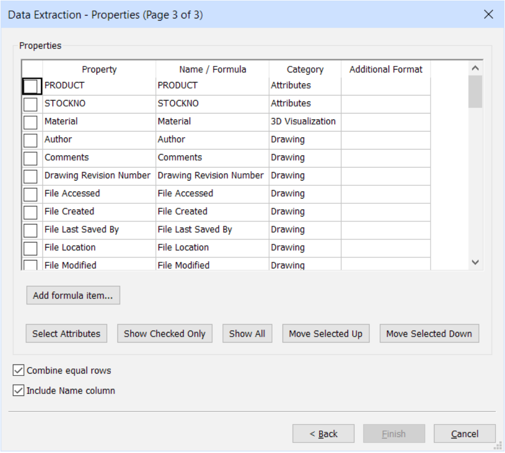


☑	LBULB	LBULB	Block with attributes
☑	GROUND	GROUND	Block with attributes
☑	FUSE40	FUSE40	Block with attributes
☑	HLAMP	HLAMP	Block with attributes
☑	LDIMMER	LDIMMER	Block with attributes
☑	LSWITCH	LSWITCH	Block with attribute
☑	DOMESW	DOMESW	Block with attribute
☑	BSWITCH	BSWITCH	Block with attribute
☑	Line	Line	Non-block
☑	CONNECT	CONNECT	Block with attributes

Tips:
Left mouse button + drag to select rows.
Ctrl-A to select all rows.
Click 'Name' cells to edit.
Click column labels to sort.

9. Click **Next** to carry on.

When the Page 3 dialog box appears, it looks scarey, as it lists every property of every entity, as well as attributes. The options in this dialog box have the following meaning:



Data Extraction - Properties (Page 3 of 3)

Properties

	Property	Name / Formula	Category	Additional Format
☐	PRODUCT	PRODUCT	Attributes	
☐	STOCKNO	STOCKNO	Attributes	
☐	Material	Material	3D Visualization	
☐	Author	Author	Drawing	
☐	Comments	Comments	Drawing	
☐	Drawing Revision Number	Drawing Revision Number	Drawing	
☐	File Accessed	File Accessed	Drawing	
☐	File Created	File Created	Drawing	
☐	File Last Saved By	File Last Saved By	Drawing	
☐	File Location	File Location	Drawing	
☐	File Modified	File Modified	Drawing	

☑ Combine equal rows
☑ Include Name column

- **Checkbox** — selects an item to be exported to the CSV file
- **Select Attributes** — selects only attributes in the list, and excludes other items
- **Show Checked Only** — shows only the items that are selected, and hides the rest from the list
- **Show All** — shows all items, checked and unchecked
- **Move Selected Up / Down** — moves the selected item(s) up and down the list; this affects the order in which items appear in the CSV file

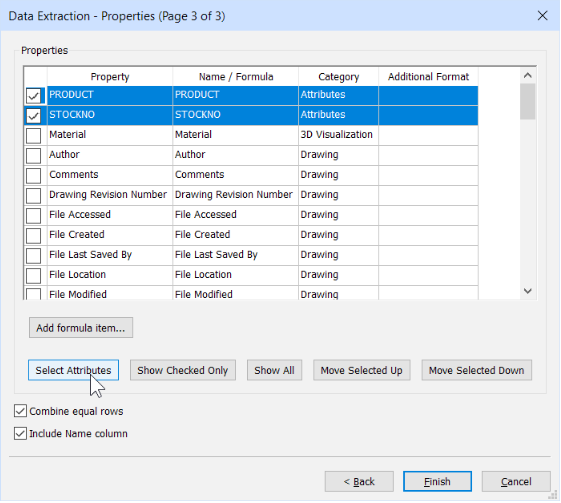
TIP To turn off (or on) all properties at once, select the first property in the list, hold down the **Shift** key, select the last property, and then click the checkbox.
To sort the columns alphabetically, click the header of each one. When you click the invisible header of the checkboxes column, it sorts by which boxes are and are not checked.

Your big job now is to turn off all properties, except for the following ones:

Length
Product
StockNo

To do this, follow these steps:

a. Click the **Select Attributes** button. This highlights the attributes in the list.



	Property	Name / Formula	Category	Additional Format
☑	PRODUCT	PRODUCT	Attributes	
☑	STOCKNO	STOCKNO	Attributes	
□	Material	Material	3D Visualization	
□	Author	Author	Drawing	
□	Comments	Comments	Drawing	
□	Drawing Revision Number	Drawing Revision Number	Drawing	
□	File Accessed	File Accessed	Drawing	
□	File Created	File Created	Drawing	
□	File Last Saved By	File Last Saved By	Drawing	
□	File Location	File Location	Drawing	
□	File Modified	File Modified	Drawing	

☑ Combine equal rows
☑ Include Name column

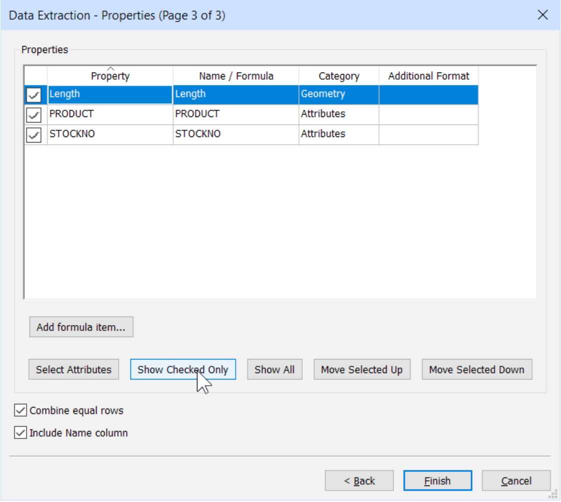
b. Click a checkbox in front of an attribute name. This action selects both of them.

c. Click the **Property** header. This sorts the items alphabetically by property name.

d. Go through the list until you find **Length**, and then click its checkbox.

e. Click **Show Checked Only** to ensure that you have the correct ones.

Here is the result of your actions:

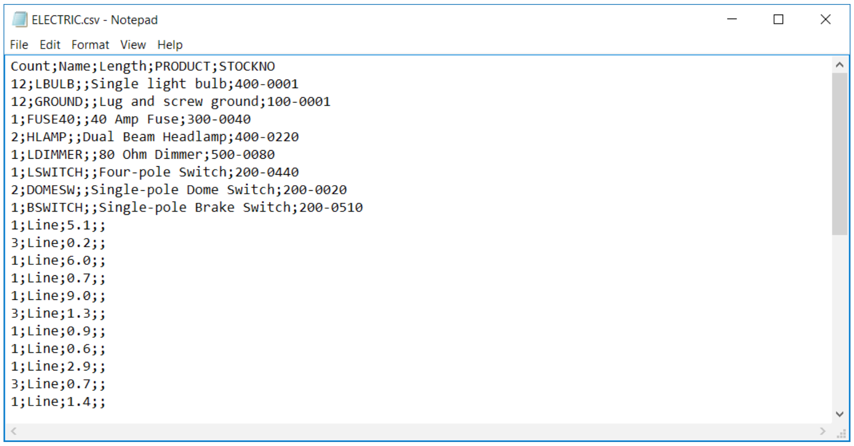


	Property	Name / Formula	Category	Additional Format
☑	Length	Length	Geometry	
☑	PRODUCT	PRODUCT	Attributes	
☑	STOCKNO	STOCKNO	Attributes	

☑ Combine equal rows
☑ Include Name column

10. Click **Finish**. After a second or two, the results are deposited in the *electric.csv* file.

11. To see the result, open the *electric.csv* file with a text editor. The content should look something similar to that illustrated below.



```
Count;Name;Length;PRODUCT;STOCKNO
12;LBULB;;Single light bulb;400-0001
12;GROUND;;Lug and screw ground;100-0001
1;FUSE40;;40 Amp Fuse;300-0040
2;HLAMP;;Dual Beam Headlamp;400-0220
1;LDIMMER;;80 Ohm Dimmer;500-0080
1;LSWITCH;;Four-pole Switch;200-0440
2;DOMESW;;Single-pole Dome Switch;200-0020
1;BSWITCH;;Single-pole Brake Switch;200-0510
1;Line;5.1;;
3;Line;0.2;;
1;Line;6.0;;
1;Line;0.7;;
1;Line;9.0;;
3;Line;1.3;;
1;Line;0.9;;
1;Line;0.6;;
1;Line;2.9;;
3;Line;0.7;;
1;Line;1.4;;
```

For each block and line in the Electric drawing, BricsCAD lists the following pieces of data:

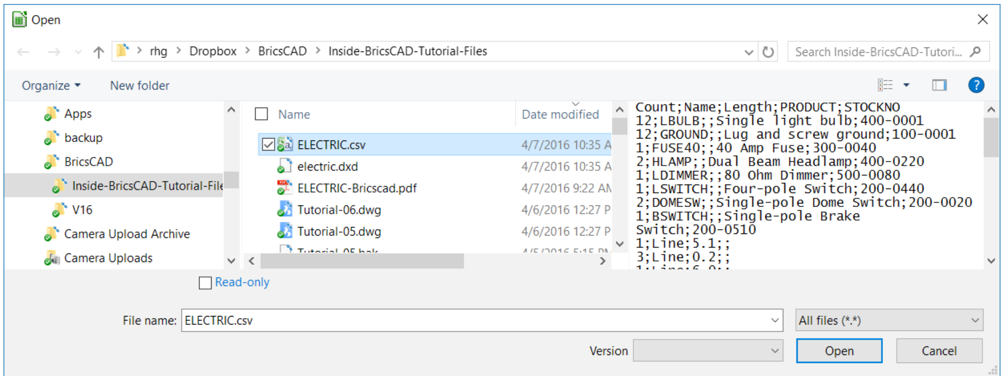
Field	Meaning	Example Value
Count	Number of occurrences in the drawing	12 (light bulbs)
Name	Name of the block or entity	LBULB
Length	Length of line (does not apply to attributes)	5.1
Product	Block's attribute value (does not apply to lines)	Single light bulb
StockNo	Block's attribute value (does not apply to lines)	400-0001

You've created a rudimentary bill of material, which can be printed out on your printer or imported to a spreadsheet — as described next. Later you learn how to bring the spreadsheet into the drawing as a bill of materials.

IMPORTING DATA FILES INTO SPREADSHEETS

To format the raw data and perform calculations, you can import the exported data into a spreadsheet. Shown in this tutorial is LibreOffice Calc, part of a free software package available from https://www.libreoffice.org/discover/calc/. Other spreadsheet programs, such as Excel, may have analogous steps.

1. Launch the spreadsheet program.
2. To open the data file correctly, please follow these steps:


Open
rhg › Dropbox › BricsCAD › Inside-BricsCAD-Tutorial-Files
Search Inside-BricsCAD-Tutori...
Organize ▾
New folder
Apps
backup
BricsCAD
Inside-BricsCAD-Tutorial-File
V16
Camera Upload Archive
Camera Uploads
Name
Date modified
ELECTRIC.csv 4/7/2016 10:35 A
electric.dxd 4/7/2016 10:35 A
ELECTRIC-Bricscad.pdf 4/7/2016 9:22 AM
Tutorial-06.dwg 4/6/2016 12:27 P
Tutorial-05.dwg 4/6/2016 12:27 P
□ Read-only
Version
Open
Cancel

```
Count;Name;Length;PRODUCT;STOCKNO
12;LBULB;;Single light bulb;400-0001
12;GROUND;;Lug and screw ground;100-0001
1;FUSE40;;40 Amp Fuse;300-0040
2;HLAMP;;Dual Beam Headlamp;400-0220
1;LDIMMER;;80 Ohm Dimmer;500-0080
1;LSWITCH;;Four-pole Switch;200-0440
2;DOMESW;;Single-pole Dome Switch;200-0020
1;BSWITCH;;Single-pole Brake
Switch;200-0510
1;Line;5.1;;
3;Line;0.2;;
```

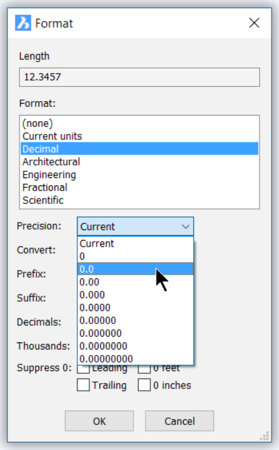
File name: ELECTRIC.csv

All files (*.*)

QUICK SUMMARY OF ADDITIONAL FORMATS

The Data Extraction Step 3 dialog box has a column labeled **Additional Format** column. Its purpose is to specify different formats for values. You don't want too much formatting, because that might confuse the spreadsheet program. Nevertheless, here is how to use it:

1. Next to "Length," click the blank area under **Additional Format**. Notice the dialog box.



Format

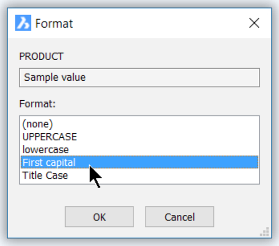
Length: 12.3457

Format: Decimal

Precision: Current

The **Length** field shows you what the format looks like, either the default or else after you make changes. In the figure shown, I had changed the "Format" to **Decimal** and the "Precision" to **0.0**. I was tempted to specify **mm** for "Suffix," but then I realized that would make the spreadsheet unhappy — it would think the number was text.

2. The Product and StockNo attributes are text, and I changed the "Format" to **First capital**.



Format

PRODUCT: Sample value

Format: First capital

3. Click **OK** to exit the Format dialog box.

The formatting codes used by the DataExtraction command are the same ones used for field text and the Field command.

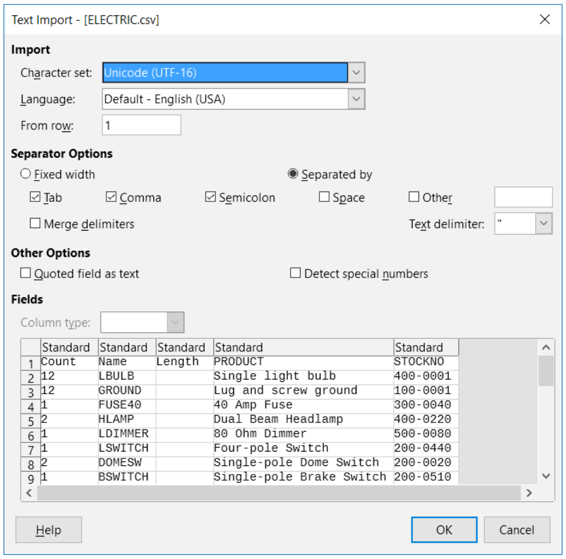
---

a. From the spreadsheet's **File** menu, select **Open** to display the Open dialog box.

b. Navigate to the folder holding the *electric.csv* file.

c. Select the *electric.csv* file, and then click **Open**.

3. Notice that LibreOffice displays the Text Import dialog box, which allows you to specify the format of files being imported.



Text Import - [ELECTRIC.csv]

Import

Character set: Unicode (UTF-16)

Language: Default - English (USA)

From row: 1

Separator Options

○ Fixed width ● Separated by

☑ Tab ☑ Comma ☑ Semicolon □ Space □ Other

□ Merge delimiters Text delimiter: "

Other Options

□ Quoted field as text □ Detect special numbers

Fields

Column type:

	Standard	Standard	Standard	Standard	Standard
1	Count	Name	Length	PRODUCT	STOCKNO
2	12	LBULB		Single light bulb	400-0001
3	12	GROUND		Lug and screw ground	100-0001
4	1	FUSE40		40 Amp Fuse	300-0040
5	2	HLAMP		Dual Beam Headlamp	400-0220
6	1	LDIMMER		80 Ohm Dimmer	500-0080
7	1	LSWITCH		Four-pole Switch	200-0440
8	2	DOMESW		Single-pole Dome Switch	200-0020
9	1	BSWITCH		Single-pole Brake Switch	200-0510

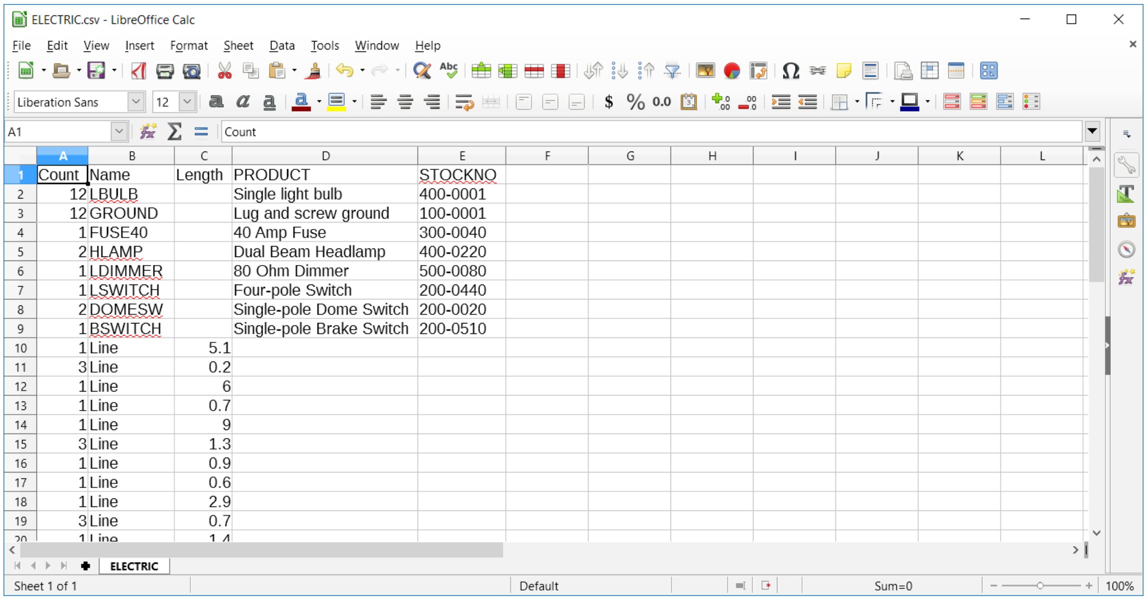
All the default options are sufficient, but make sure the following ones are set:

```
Import
From row:                    1

Separator Options
Separated by:                Tab Comma Semicolon
```

4. Click **OK**.

Notice that LibreCalc loads the *electric.csv* file, and then displays each field in its own column.

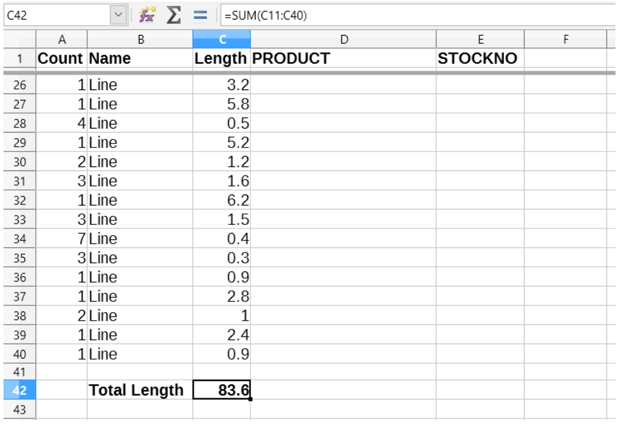
If it is necessary to adjust the column widths, select the four columns, then choose from the menu bar **Format | Column | Optimal Width**.



ELECTRIC.csv - LibreOffice Calc
File Edit View Insert Format Sheet Data Tools Window Help
Liberation Sans
12
A1
Count

	A	B	C	D	E
1	Count	Name	Length	PRODUCT	STOCKNO
2	12	LBULB		Single light bulb	400-0001
3	12	GROUND		Lug and screw ground	100-0001
4	1	FUSE40		40 Amp Fuse	300-0040
5	2	HLAMP		Dual Beam Headlamp	400-0220
6	1	LDIMMER		80 Ohm Dimmer	500-0080
7	1	LSWITCH		Four-pole Switch	200-0440
8	2	DOMESW		Single-pole Dome Switch	200-0020
9	1	BSWITCH		Single-pole Brake Switch	200-0510
10	1	Line	5.1		
11	3	Line	0.2		
12	1	Line	6		
13	1	Line	0.7		
14	1	Line	9		
15	3	Line	1.3		
16	1	Line	0.9		
17	1	Line	0.6		
18	1	Line	2.9		
19	3	Line	0.7		
20	1	Line	1.4		

5. If you wish, add price and extension fields, a totals row, and format the text for lovely output. For example, I used the SUM() function to find the total length of wires (lines).



C42 =SUM(C11:C40)

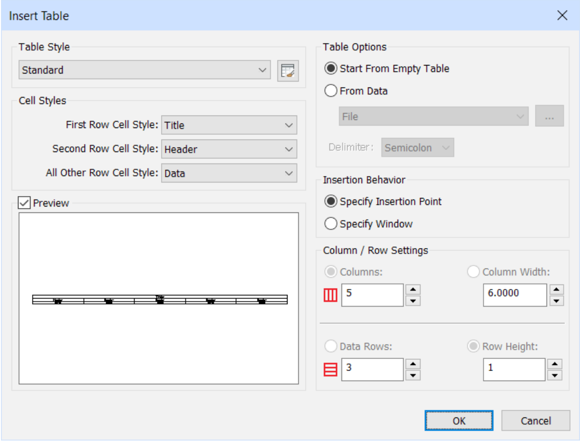
	A	B	C	D	E	F
1	Count	Name	Length	PRODUCT	STOCKNO	
26	1	Line	3.2			
27	1	Line	5.8			
28	4	Line	0.5			
29	1	Line	5.2			
30	2	Line	1.2			
31	3	Line	1.6			
32	1	Line	6.2			
33	3	Line	1.5			
34	7	Line	0.4			
35	3	Line	0.3			
36	1	Line	0.9			
37	1	Line	2.8			
38	2	Line	1			
39	1	Line	2.4			
40	1	Line	0.9			
41						
42		Total Length	83.6			
43						

6. Remember to save the spreadsheet file.

Placing Data in Drawings as Tables

Command	Table
Menu Bar	Draw | Table
Ribbon	Draw | Tables | Table
Ribbon	Draw | Table

Data that you export as CSV files can be placed in drawings as tables. You can use this to place bills of materials right in the drawing. BricsCAD's Table command imports data in several formats.



Insert Table

Table Style: Standard

Cell Styles
First Row Cell Style: Title
Second Row Cell Style: Header
All Other Row Cell Style: Data

☑ Preview

Table Options
● Start From Empty Table
○ From Data
File
Delimiter: Semicolon

Insertion Behavior
● Specify Insertion Point
○ Specify Window

Column / Row Settings
○ Columns: 5
○ Column Width: 6.0000
○ Data Rows: 3
● Row Height: 1

OK Cancel

- **CSV** — output from BricsCAD's DataExtraction command, as well as other programs
- **XLS** — spreadsheet files saved by Excel 2003 and earlier
- **XLSX** — encapsulated spreadsheet files saved by Excel 2007 and newer
- **XML** — non-product specific spreadsheet format using XML codes (extended markup language), and can be exported by LibreOffice Calc and Excel. (Don't confuse XML with XLS, the latter being Excel's native format.)

You place these data files in drawings as tables with the Table command. In the following tutorial, you import the *electric.csv* file you created earlier.

1. Start the **Table** command. Notice the Insert Table dialog box.
2. In the Table Options section, choose **From Data**.

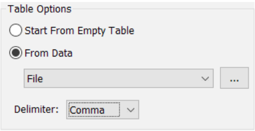


Table Options

○ Start From Empty Table

● From Data

File

Delimiter: Comma

3. Open the *electric.csv* file you created earlier:

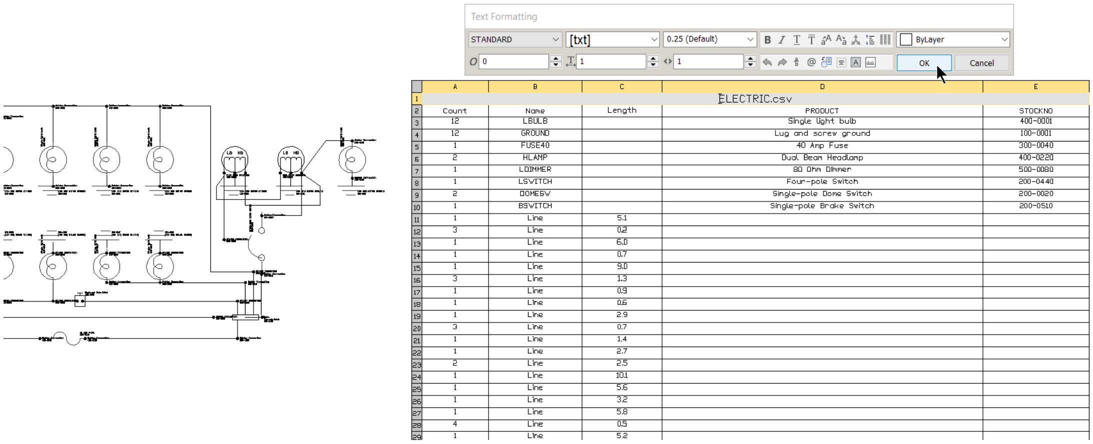
   a. Click the ... **Browse** button.

   b. In the Open dialog box, navigate to the folder in which the *electric.csv* file is stored.

   c. Choose it, and then click **Open**.

4. If necessary, change the **Separator** field to "Comma" or which ever character is used to separate fields in the file. If you specify the wrong separator character, BricsCAD will later complain "Invalid format."
5. Click **OK**. Notice that BricsCAD prompts you to pick an insertion point:

```
   Specify insertion point or [Style/Width/Height]: (Pick a point)
```



Text Formatting

STANDARD | [txt] | 0.25 (Default) | ByLayer

0 | 1 | 1

	A	B	C	D	E
1	ELECTRIC.csv				
2	Count	Name	Length	PRODUCT	STOCKNO
3	12	LBULB		Single light bulb	400-0001
4	12	GROUND		Lug and screw ground	100-0001
5	1	FUSE40		40 Amp Fuse	300-0040
6	2	HLAMP		Dual Beam Headlamp	400-0220
7	1	LDIMMER		80 Ohm Dimmer	500-0080
8	1	LSWITCH		Four-pole Switch	200-0440
9	2	DOMESW		Single-pole Dome Switch	200-0020
10	1	BSWITCH		Single-pole Brake Switch	200-0510
11	1	Line	5.1		
12	3	Line	0.2		
13	1	Line	6.0		
14	1	Line	0.7		
15	1	Line	9.0		
16	3	Line	1.3		
17	1	Line	0.9		
18	1	Line	0.6		
19	1	Line	2.9		
20	3	Line	0.7		
21	1	Line	1.4		
22	1	Line	2.7		
23	2	Line	2.5		
24	1	Line	10.1		
25	1	Line	5.6		
26	1	Line	3.2		
27	1	Line	5.8		
28	4	Line	0.5		
29	1	Line	5.2		

6. The table is placed in the drawing, and BricsCAD immediately goes into formatting mode, should you wish to further change the look of the table.

The data you inserted in the drawing is static. It will change only if you edit the table.

LINKING DATA IN DRAWINGS WITH TABLES

Command DataLink

Sometimes you want the spreadsheet data in drawings to remain static — it shouldn't change, and so you would use the Table command's From Data option.

Other times, you want the data to update every time the spreadsheet changes. For instance, you might want to make prices dynamic: change the price in the spreadsheet and the price shown in the drawing's table updates.

Or the parts in the drawing change, and so you want the spreadsheet updated.

To integrate this kind of automation, you need to create a link between the spreadsheet and the table. The link is established with the DataLink command, and then when the data needs to be updated, you use the DataLinkUpdate command.

The command links to the following formats of files:

- **CSV** — output from the BricsCAD DataExtraction command and other programs
- **XLS** — spreadsheet files saved by Excel 2003 or earlier
- **XLSX** — encapsulated spreadsheet files saved by Excel 2007 or newer

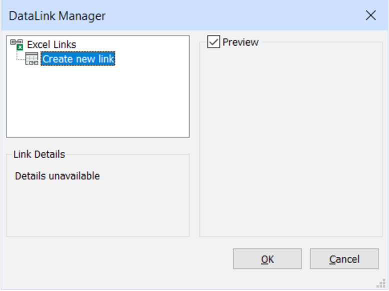
TIP There is one limitation to be aware of: **DataLink** works fully only with the full Excel program, and so you cannot use it with Excel Viewer, LibreCalc or any other spreadsheet program, unfortunately.

To create a linked table, follow these steps:

0. If necessary, open the *electric.csv* file in Excel, and then save it as an XLS or XLSX file.
1. Open BricsCAD with the *electric.dwg* file.
2. Start the **DataLink** command:

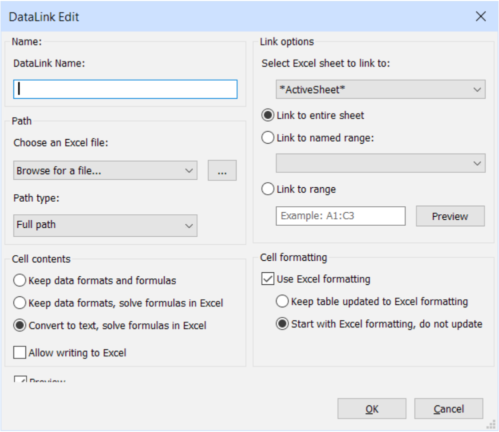
```
: datalink
```

3. Notice the Data Link Manager dialog box. It creates new data links with spreadsheets and manages existing ones.



Choose **Create New Link**.

4. Notice the Data Link dialog box:



DataLink Edit

Name:

DataLink Name: |

Path

Choose an Excel file: Browse for a file...

Path type: Full path

Cell contents

○ Keep data formats and formulas

○ Keep data formats, solve formulas in Excel

● Convert to text, solve formulas in Excel

☐ Allow writing to Excel

Link options

Select Excel sheet to link to: *ActiveSheet*

● Link to entire sheet

○ Link to named range:

○ Link to range

Example: A1:C3

Cell formatting

☑ Use Excel formatting

○ Keep table updated to Excel formatting

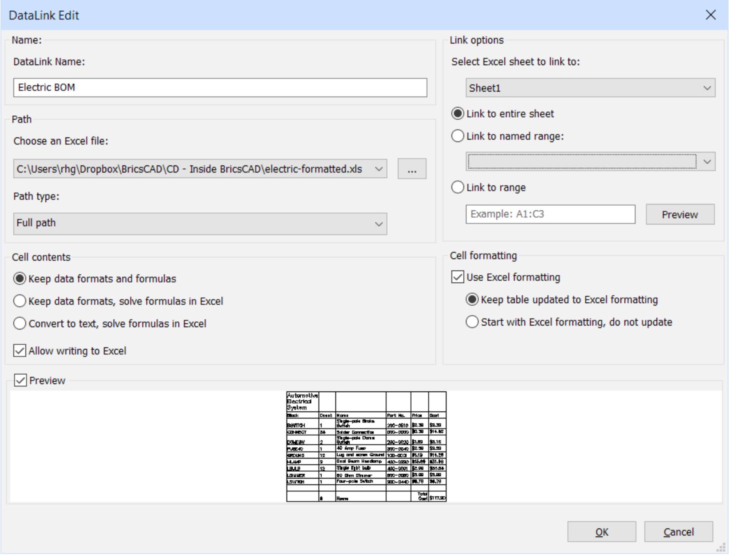
● Start with Excel formatting, do not update

Enter the following information into the dialog box.

Field	Data	Meaning
DataLink Name	**Electric BOM**	This name identifies the data link; it can be any useful name
Choose an Excel File	**electric.xls**	Source of data can be a CSV, XLS, or XLSX file
Allow writing to Excel	**Yes**	Changes made in the BricsCAD table can be written back to Excel
Select Excel sheet	**ActiveSheet**	This is the sheet that is currently displayed by Excel
Link to entire sheet	**Yes**	BricsCAD imports only the cell range that is in use
Use Excel formatting	**Yes**	BricsCAD uses the Excel formatting, instead of a table style

TIP When you choose a CSV file, many of the Excel-related options become unavailable (grayed out).

5. When you are finished, the dialog box should look like this:



DataLink Edit

Name:

DataLink Name: Electric BOM

Path

Choose an Excel file: C:\Users\rhg\Dropbox\BricsCAD\CD - Inside BricsCAD\electric-formatted.xls

Path type: Full path

Cell contents

● Keep data formats and formulas

○ Keep data formats, solve formulas in Excel

○ Convert to text, solve formulas in Excel

☑ Allow writing to Excel

☑ Preview

Link options

Select Excel sheet to link to: Sheet1

● Link to entire sheet

○ Link to named range:

○ Link to range

Example: A1:C3

Cell formatting

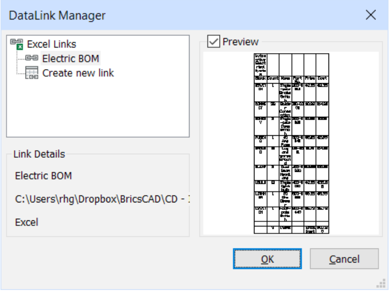
☑ Use Excel formatting

● Keep table updated to Excel formatting

○ Start with Excel formatting, do not update

Click **OK**.

6. Notice that you are back in the original DataLink Manager dialog box, and that “Electric BOM” is added to the list of Excel Links.



DataLink Manager

- Excel Links
  - Electric BOM
  - Create new link

☑ Preview

Link Details

Electric BOM

C:\Users\rhg\Dropbox\BricsCAD\CD - :

Excel

Click **OK**.

7. To place the linked BOM, you use the Table command again, with a small twist:

   a. Start the **Table** command:

```
: table
```

   b. In the Table Options section, choose **DataLink**:

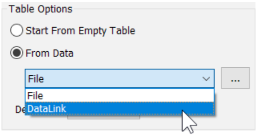


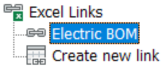
Table Options

○ Start From Empty Table

● From Data

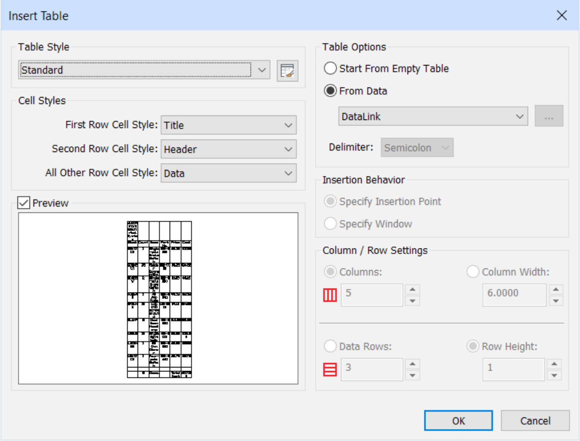
File

   c. The Data Manager dialog box pops up immediately. Choose “Electric BOM” and then click **OK**.



- Excel Links
  - Electric BOM
  - Create new link

   d. BricsCAD brings back in the Table dialog box. It should look like this:



Insert Table

Table Style: Standard

Cell Styles

First Row Cell Style: Title

Second Row Cell Style: Header

All Other Row Cell Style: Data

☑ Preview

Table Options

○ Start From Empty Table

● From Data

DataLink

Delimiter: Semicolon

Insertion Behavior

● Specify Insertion Point

○ Specify Window

Column / Row Settings

● Columns: 5

○ Column Width: 6.0000

○ Data Rows: 3

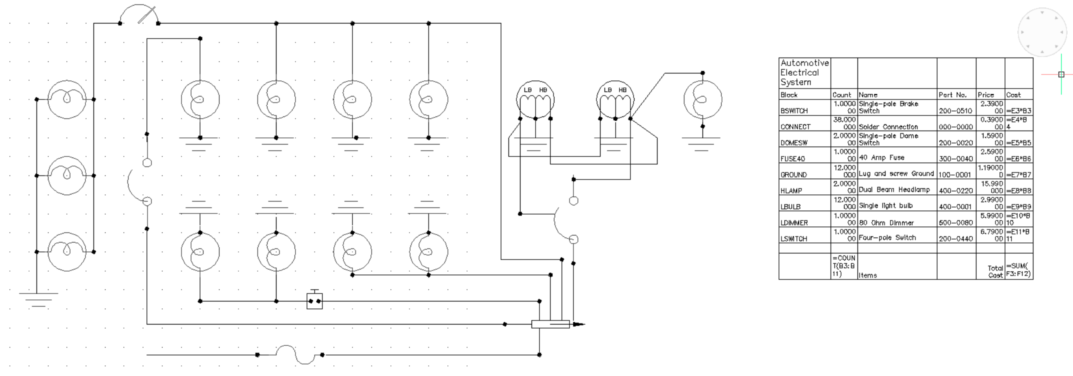
● Row Height: 1

Click **OK**.

8. In the command bar, BricsCAD prompts you to place the table:

```
Specify insertion point or [Style/Width/Height]: (Pick a point)
```

Pick a point to place the table, which might look something like this:



Automotive Electrical System					
Block	Count	Name	Part No.	Price	Cost
BSWITCH	1.000000	Single-pole Brake Switch	200-0510	2.390000	=E3*B3
CONNECT	38.000000	Solder Connection	000-0000	0.390000	=E4*B4
DOMESW	2.000000	Single-pole Dome Switch	200-0020	1.590000	=E5*B5
FUSE40	1.000000	40 Amp Fuse	300-0040	2.590000	=E6*B6
GROUND	12.000000	Lug and screw Ground	100-0001	1.190000	=E7*B7
HLAMP	2.000000	Dual Beam Headlamp	400-0220	15.990000	=E8*B8
LBULB	12.000000	Single light bulb	400-0001	2.990000	=E9*B9
LDIMMER	1.000000	80 Ohm Dimmer	500-0080	5.990000	=E10*B10
LSWITCH	1.000000	Four-pole Switch	200-0440	6.790000	=E11*B11
	=COUNT(B3:B11)	Items		Total Cost	=SUM(F3:F12)

Warning. The table is *locked*. This means you cannot edit its content, such as text and number. It is locked, because the content is controlled by Excel.

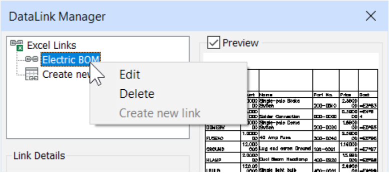
```
Cell contents are locked
```

To change the content, do so in Excel, and then use the DataLinkUpdate command, as described next.

You still can format the table, such as change the look of text and lines.

Editing Datalinks

To edit a datalink, right-click its name and then choose **Edit**.



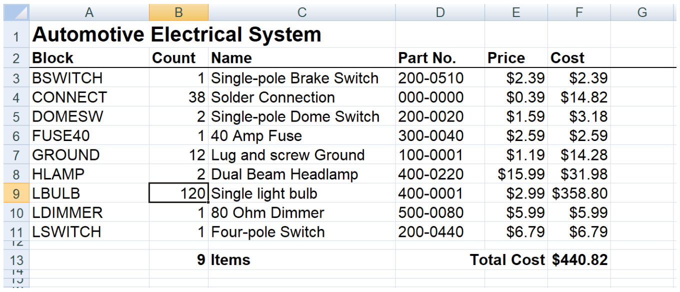
The Data Link dialog box appears; make changes and then click **OK**.

Updating the Table

```
Command    DataLinkUpdate
```

With the table in BricsCAD linked to the spreadsheet in Excel, you can make changes in the spreadsheet, and then force the table to update itself. To reflect the changes, the update is made with the DataLinkUpdate command, like this:

1. To start this tutorial, open Excel with the *electric.xls* spreadsheet, and then make an obvious change. For example, change the number of LBULB (light bulbs) from 12 to 120. This forces the cost to update, as well.)



	A	B	C	D	E	F	G
1	Automotive Electrical System						
2	Block	Count	Name	Part No.	Price	Cost	
3	BSWITCH	1	Single-pole Brake Switch	200-0510	$2.39	$2.39	
4	CONNECT	38	Solder Connection	000-0000	$0.39	$14.82	
5	DOMESW	2	Single-pole Dome Switch	200-0020	$1.59	$3.18	
6	FUSE40	1	40 Amp Fuse	300-0040	$2.59	$2.59	
7	GROUND	12	Lug and screw Ground	100-0001	$1.19	$14.28	
8	HLAMP	2	Dual Beam Headlamp	400-0220	$15.99	$31.98	
9	LBULB	120	Single light bulb	400-0001	$2.99	$358.80	
10	LDIMMER	1	80 Ohm Dimmer	500-0080	$5.99	$5.99	
11	LSWITCH	1	Four-pole Switch	200-0440	$6.79	$6.79	
12							
13		9	Items		Total Cost	$440.82	
14							
15							

2. Save the spreadsheet, and then exit Excel.
3. In BricsCAD, start the DataLinkUpdate command:

```
: datalinkupdate
```

4. Enter **u** to update the links, and then press Enter.

```
Select an option [Update data link/Write data link]: u
```

5. You are asked to select the table:

```
Select objects or [Update all]: (Press Enter to select the table)
```

6. Wait for a few moments as BricsCAD processes the request. When the command prompt returns, the table shows the updated numbers for LBULB and Cost.

Automatic BOMs from 3D Components

(**PLATINUM AND ULTIMATE EDITIONS ONLY**) BricsCAD Platinum can generate BOMs with a single click, when drawings are made with components. A *component* is a special kind of block, about which BricsCAD knows. In this tutorial, you take a 3D sample drawing, "explode" it, generate the BOM, and then tag parts with balloons.

THe commands in this section work only with BricsCAD Platinum or Ultimate, and with drawings that have inserts of mechanical components. In the following tutorial, you work through the following stages:

Stage 1 — Explode an assembly into parts

Stage 2 — Generate a BOM automatically

Stage 3 — Label parts with balloons

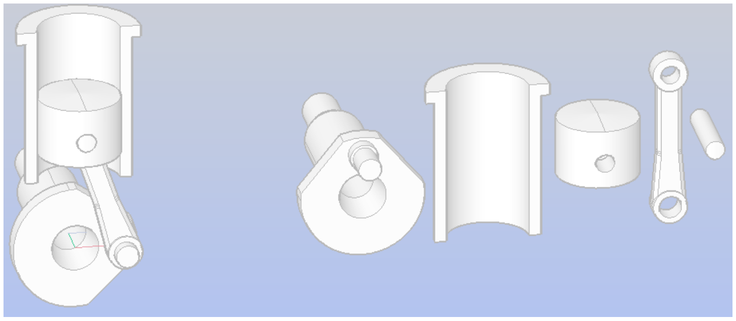
EXPLODING ASSEMBLIES

Command	bmexplode
Transparent	'bmexplode
Menu bar	Assembly | Assembly Explode | Explode
Ribbon	Assembly | Tools | Explode
Toolbar	Assembly | Assembly Explode | Explode

Complex 3D models are usually made from many parts. Each part is designed separately, as described later in this book, and then assembled to make the entire design — whether a remote control or an entire automobile. A part can be as simple as a pin or a as complex as a crankshaft.

To document the design, it is common to explode the assembly. Exploding separates each part from the other, so that each one can be seen clearly; no parts are hiding any other parts. This kind of exploding has nothing to do with the Explode command used in 2D drafting.

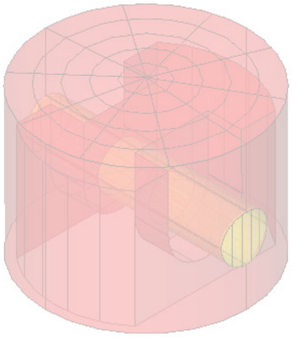
In BricsCAD Platinum and Ultimate, assemblies are exploded with the **bmExplode** command. The "bm" prefix to this command name (and others in this section) indicates "BricsCAD Modeling," and that these commands are specific to 3D modeling.



Left: *An assembly of parts;* ***right:*** *parts exploded*

In part 1 of this tutorial, you open a 3D assembly model supplied as a sample file with BricsCAD Platinum, and then separate the parts.

1. Start BricsCAD Platinum, and then open the 3D model *piston-pin.dwg*. This drawing contains two components in an assembly.



QUICK SUMMARY OF bmEXPLODE

```
: bmexplode
Select explosion algorithm [Table by Level/Table by Types/Manual/Settings] <Manual>:
Select a position of the exploded representation:
```

Table by Level (TL) — places components according to their level in the assembly tree in the same row

Table by Types (TT) — places components of the same type together in the same row"

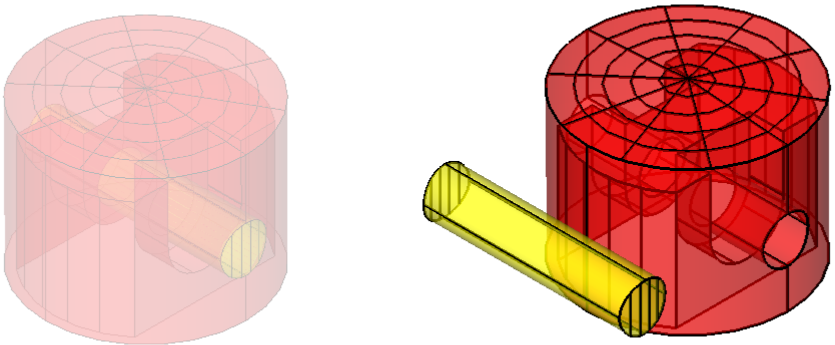
Manual — places components to be moved apart manually

Settings — determines how assembly is exploded:

- **Top** — explodes only top level components to suppress minor components
- **Bottom** — explodes components
- **Name** — assigns a name to the settings so that they can be used for future explosions

2. Enter the **bmExplode** command.

```
: bmexplode
Select explosion algorithm [Table by Level/Table by Types/Manual/Settings] <Manual>: TL
```



Left: *An assembly of two parts;* ***right:*** *parts exploded*

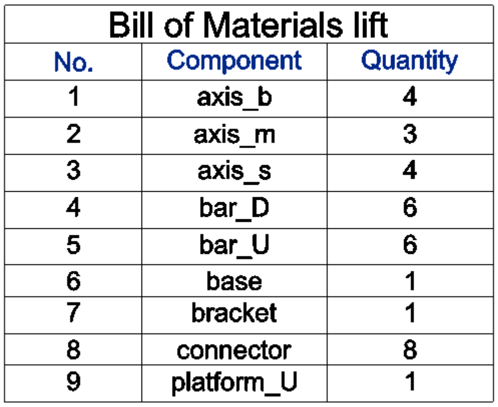
3. Notice the BricsCAD turns the exploded parts into a single block. This makes it easier to place many parts at once. Pick a point in the drawing to place the exploded components.

```
Select a position of the exploded representation: (Pick a point in the drawing away from
the source components)
```

GENERATING BOMs

Command	bmbom
Transparent	'bmbom
Menu bar	Assembly | Bill of Materials
Ribbon	Assembly | Inquire | Bill of Materials
Toolbar	Assembly | Bill of Materials

BricsCAD automatically generates a bill of material from any drawing made with components with the bmBom command. Bills of material are listed in table form, like this. The "Quanity" column in the table tells the manufacturer how many parts to manufacture of each item (named in the "Component" column. The "No." column identifies the part in the drawing, which has a balloon next to it (see the tutorial following).



Bill of Materials lift

No.	Component	Quantity
1	axis_b	4
2	axis_m	3
3	axis_s	4
4	bar_D	6
5	bar_U	6
6	base	1
7	bracket	1
8	connector	8
9	platform_U	1

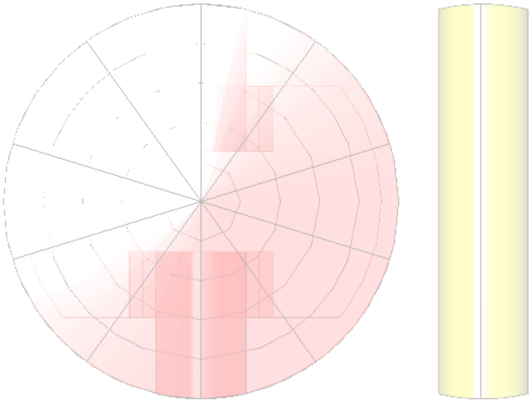
The bmBom command does nearly all the work for you. All you have to do is specify the BOM table's location in the drawing. The command creates a table that is a regular BricsCAD table that you can edit anyway you see fit. It is subject to the properties of the TableStyle command.

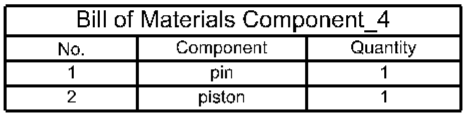
For the bmBom command to work, it needs at least one component in a drawing. The components do not need to be assembled, nor does the assembly need to be exploded.

Here is how you generate a bill of materials from the parts:

1. Continue with the model from the previous tutorial.
2. Enter the **bmBom** command:

```
: bmbom
Insertion point [Name/Top level/Bottom level/Configure]: (Pick a point in the drawing to
place the table)
```





Bill of Materials Component_4

No.	Component	Quantity
1	pin	1
2	piston	1

That's it, you're done! BricsCAD does all the work creating the table for you. If you want to change the look, then use the **TableStyle** command.

QUICK SUMMARY OF bmBOM

```
: bmbom
Insertion point [Name/Top level/Bottom level/Configure]:
```

Insertion point — places the table in the drawing by its upper-left corner

Name — adds a name to the title, such as "Lift Parts."

Top level — makes the BOM table from the top level components only; this is useful for suppressing many minor components.

Bottom level — makes the BOM table from all components.

Configure — specifies which columns are included in the table: Description, Density, Volume, Mass, Material, Thickness (only for sheet metal parts), and Parameters.

When you want to change the content of the table, then use the **bmBom** command’s **Configure** option, which lets you choose among these columns:

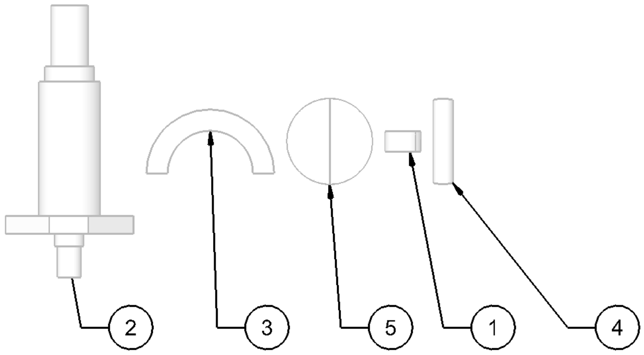
- Description
- Density
- Volume
- Mass
- Material
- Thickness
- Parameters

The command gets this information from the parts themselves, because they are 3D solid models.

ATTACHING BALLOONS

```
Command     bmballoon
Transparent         'bmballoon
Menu bar    Assembly | Balloon
Ribbon      Assembly | Inquire | Balloon
Toolbar     Assembly | Balloon
```

BricsCAD semi-automatically attaches balloons to components using the **bmBalloon** command. We use balloons to identify components, making a visual link between the geometry and the text in the BOM table.



During the bmBalloon command, you pick the component and then specify the location of the balloon. The command identifies the component, and then assigns the ID number, which it gets from the BOM table. *Balloons can be attached only after the BOM is generated*. This command can be used only in model space.

BricsCAD constructs balloons from multileaders, and so they are subject to the properties of the MLeaderStyle command. You can edit the balloons like multileaders, such as pointing multiple leaders to multiple, identical components.

Traditionally, balloons are lined up to make it easier to read and find them, and so BricsCAD accommodates this: hold down the **Shift** key while placing them. This lines up the balloons, vertically or horizontally.

Here is how to attach reference balloons to the parts:

1. Start the bmBalloon command:

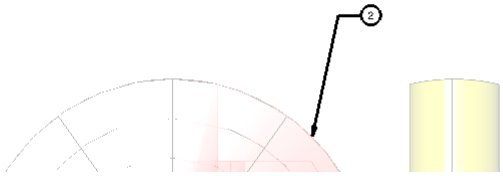
   ```
   : bmballoon
   ```

2. Pick one of the components. Notice that the arrowhead end of the balloon attaches itself to the component.

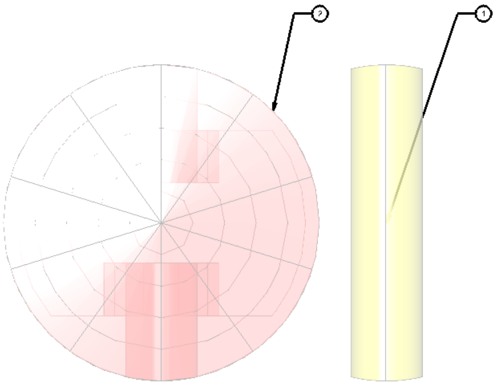
   ```
   Select a component insert [select other Table/choose balloon Frame]: (Pick a component)
   ```

3. Place the rest of the balloons at a convenient location in the drawing, away from the other components:

   ```
   Pick a point to place balloon: (Pick a point)
   ```



4. Repeat the **bmBalloon** command to place a balloon for the other component. At the 'Pick point to place balloon' prompt, hold down the **Shift** key so that the second balloon lines up with the first one.



Bill of Materials Component_4

No.	Component	Quantity
1	pin	1
2	piston	1

QUICK SUMMARY OF bmBALLOON

```
: bmballoon
Select a component insert [select other Table/choose balloon Frame]:
Pick point to place balloon:
```

Insert — places the arrowhead end of the balloon, preferably on the component

select other Table — chooses the BOM table to use, should more than one exist in the drawing

choose balloon Frame — choose the balloon style: circle, rectangle, triangle, or hexagon.

Pick point — places the balloon end; hold down the Shift key to align balloons

You learned how to export attribute data to spreadsheets, and then bring formatted spreadsheet data back into BricsCAD. You also learned about creating, populating, and editing tables. At the end of the chapter, you learned how to explode a 3D assembly, have BricsCAD generate a BOM automatically from components, and then place identifying balloons. You learn more about 3D in later chapters.

Next, you learn about a more sophisticated 2D entity known as the region.

CHAPTER 9

Working with 2D Regions & Booleans

Here you learn how to increase your CAD efficiency by working with regions and with Boolean operations. These two are BricsCAD features that are often used to construct complex 2D and 3D shapes, and then to analyze them for information useful to designers, such as centers of gravity. BricsCAD does the calculations far quicker than if you were to perform them by hand.

In this chapter you learn to use the Boundary and Region commands, and commands related to boolean operations. (The Region command is available in the Pro, Platinum, and Ultimate editions of BricsCAD.)

IN THIS CHAPTER

- Converting entities into regions
- Applying Boolean operations to regions
- Finding the mass properties of regions

KEY TERMS IN THIS CHAPTER

Boolean — refers to logical operations, such as AND, OR, and NOT

Mass property — reports the properties of a mass, such as its area, centroid, and radius of gyration

Point filter — returns a single coordinate

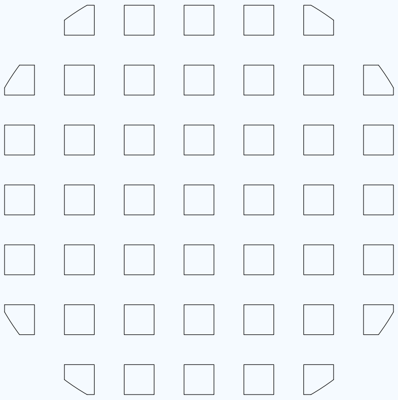
Region — consists of a closed 2D area

NEW COMMANDS

Command	Shortcut	Menu Bar	Ribbon
DelObj	...	...	...
Intersect	in	Modify | Solid Editing| Intersect	Solid | Edit | Intersect
MassProp	...	Tools | Inquiry | Region/Mass Properties	Tools | Inquiry | Mass Properties
Region	reg	Draw | Region	...
Subtract	su	Modify | Solid Editing | Subtract	Solid | Edit | Subtract
Union	uni	Modify | Solid Editing| Union	Solid | Edit | Union

THIS CHAPTER'S DRAWING

By the end of this lesson, your drawing will look like this one:

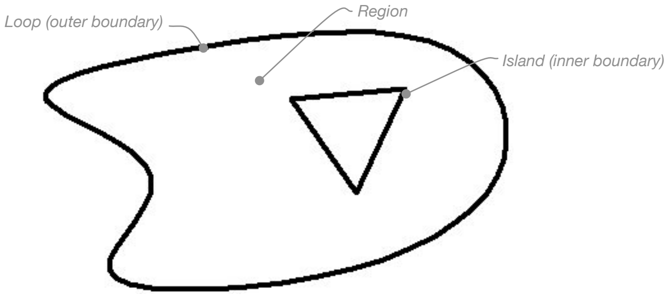


RegionTutorial drawing

About Regions

(PRO, PLATINUM, ULTIMATE EDITIONS ONLY) BricsCAD works with *regions*. These are closed 2D areas of almost any shape. Regions are interesting, because two or more can be combined using *Boolean* operations. Boolean operations join, intersect, and subtract one region from the other. This lets you create a single, complex entity that has holes of any shape in it, and then lets you analyze its physical properties.

Regions have their own jargon. The outer boundary of a region goes by the name of "loop." The holes inside of regions are "islands." The loops and islands can be of any shape, from a simple triangle to flowing curves.



Technically, regions are ACIS entities, the same technology BricsCAD uses for 3D solid models.

HOW TO CREATE REGIONS

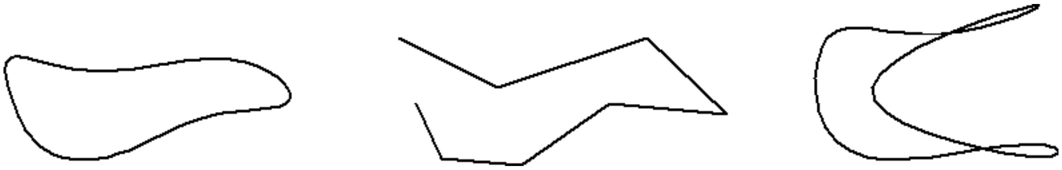
In a curious twist of fate, BricsCAD cannot create regions from scratch. There is no version of the Region command that draws them, as there is with the PLine command for drawing polylines. Rather, the Region command converts *existing* closed entities into regions. To do this, BricsCAD provides you with two commands, **Boundary** and **Region.** They are subtly different:

- **Boundary** command — displays a dialog box, then prompts you to pick a point inside a closed area. BricsCAD finds the boundary, and then makes a region or a polyline along its edges. (This command is, in fact, a subset of the Hatch command, and so you get the island options and gap tolerance as a bonus.)
- **Region** command — operates at the command line and prompts you to select entities that make up a closed area, and then generates a region entity.

Either way, both commands create a region entity from a closed area, which can be made of one or more entities. So, it takes two steps to create a region:

Step 1: Draw the shape using drawing commands such as **PLine**, **Arc**, and **Circle**. To create a region successfully, the shape must be closed when used with the Region command; the Boundary command allows gaps of up to 5000 drawing units in size. Self-intersecting curves are made into multiple regions. See figure below for examples.

(Left to right) You can end up with one region, no region, or three regions:



Left: *Closed non-self-intersecting shape become a single region;*
center: *Unclosed shapes cannot become regions;*
right: *Self-intersecting curve becomes multiple regions*

Step 2: Convert the shape into a region with the **Boundary** or **Region** commands. The results of the commands differs for the shapes shown in the figure below:

Command	Closed Entity	Open Entity	Self-intersecting Entity
Boundary	1 region	0 or 1 regions *	1 region per pick
Region	1 region	0 regions	3 regions with one pick

* The Boundary command can make a region from an open area if the gap is smaller than the tolerance.

The two commands aren't able to combine non-overlapping entities into one. For instance, select three circles, each an inch apart, and they become three regions. (This behavior may change in a future release of the software.)

Differences Between Boundary and Region Commands

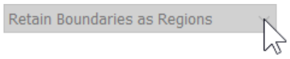
The Boundary and Region commands work differently from each other, and so one might be more useful to you in certain situations than the other. Because of the differences in how the two commands select detect boundaries, the shape of the resulting region can differ.

Boundary. First of all, the Boundary command's dialog box does not let you switch between creating polylines and boundaries. Initially, the dialog box is set up to create polylines; the droplist is grayed out and does not work, for some reason.

To change it to regions, you need to change the **HpBound** system variable to off, either at the command line or in the Settings dialog box:

```
 : hpbound
New current value for HPBOUND [1 for ON/0 for Off] <1 for ON>: off
```

HpBound	Meaning
0	Boundary and Hatch commands make boundaries out of regions
1	Boundaries are made of polylines



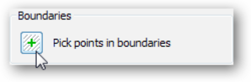
Retain Boundaries as Regions

QUICK SUMMARY OF BOUNDARY OPTIONS

The Boundary dialog box contains options that control how boundaries are created.

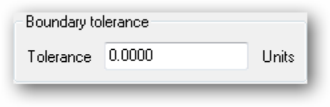
BOUNDARIES

The Pick Points button dismisses the dialog box temporarily, and the prompts you to pick a point in the drawing or select entities. You can pick inside more than one boundary.



BOUNDARY TOLERANCE

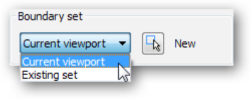
The Boundary Tolerance option specifies the acceptable size of gaps. For example, when you set the tolerance to 0.1", then gaps as large as 0.1" will be ignored (or bridged) by the boundary-seeking algorithm.



Tolerance 0.0000 Units

BOUNDARY SET

Boundary Set section determines where BricsCAD should look for entities from which to create boundaries.



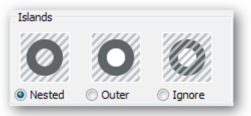
The default is the entire current viewport, but you can change this with the **New** button to smaller areas, such as the existing selection set or the current viewport. For instance, after you click the **New** button and choose a number of entities, the boundary seeking algorithm limits its work to the entities you selected.

BOUNDARY RETENTION

The Boundary Retention option does not work in this dialog box. Use the **HpBound** variable to set the boundary as a polyline or region.

ISLANDS

When entities contains other entities (a.k.a. "islands"), then you can tell the boundary-seeking algorithm to include or ignore them.



● Nested ○ Outer ○ Ignore

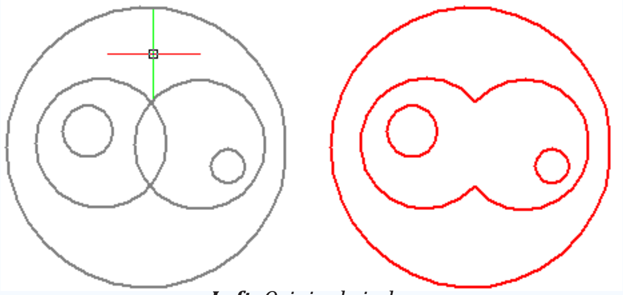
The circles shown below illustrate the difference between the Nested, Outer, and Ignore options. Notice the location of the cursor, and that the pick point is the same in each case.

Continued...

...continued.

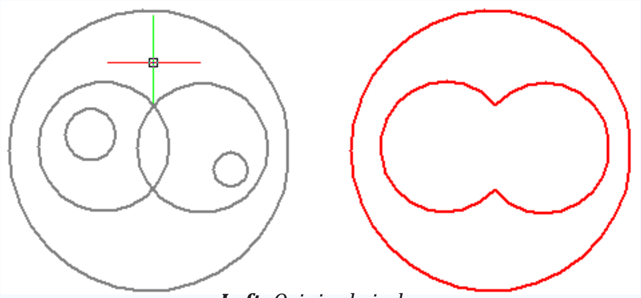
Entities created by Boundary can look invisible, because they are created on the current layer. To see them, create a new layer with a different color.

Nested — all entities (the circles) form boundaries, as shown in red at the right.



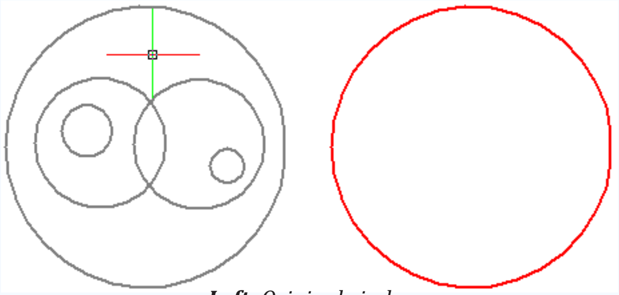
***Left:** Original circles*
***Right:** All circles turned into boundaries*

Outer — the outermost circles form boundaries (shown in red below); the innermost circles are ignored.



***Left:** Original circles*
***Right:** Circles nearest to pick point turned into boundaries*

Ignore — only the outermost circle is used to form the boundary; the others are ignored.



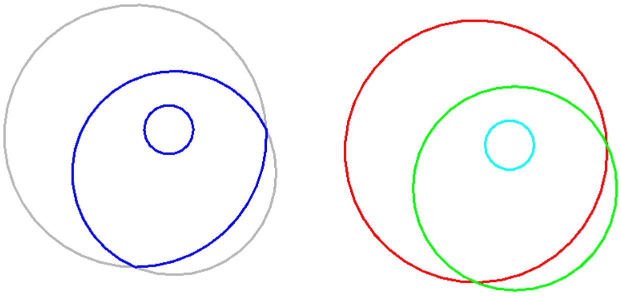
***Left:** Original circles*
***Right:** Only outermost circle turned into a boundary; other circles ignored*

When you click the **Pick Points in Boundaries** button, you are prompted at the command line:

```
Pick a point to define a boundary or hatch area or [Select entities/Undo]: (Enter an option)
```

Option	Meaning
Pick a point	Uses boundary detection to find regions around the pick point
Select entities	Picks individual entities, and then forms regions out of them
Undo	Unselects the last pick area or entity

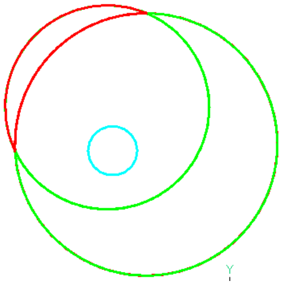
I've drawn three circles to show you how the resulting regions differ according to how you select them.



Left: *Blue region entities created by picking a point*
Right: *Red, green, and cyan region entities created by selecting each circle entity*

Region. The Region command creates only regions. At the command prompt, it asks you to pick entities:

```
Select entities or boundaries: (Select one or more entities)
```



In the figure above, the command created the two red and green crescent moon-like regions, plus the circular cyan-colored region.

Summary. I've put together this table to highlight the differences between the two commands:

Activity	Boundary	Region
Entities created (Hpbound)	Polylines Regions	Regions only
Islands detected	Yes, multi-level	Yes, only one level
Allow gaps (HpGapTol)	Yes, up to 5000 units	No
Selection method	Pick enclosed areas Select entities	Select entities only
Retain source entities	Optional	Optional

In summary, the Boundary command is more flexible with more options, but the Region command is simpler with no options.

What To Do With the Leftovers

Both commands leave the source entities in place. To change this behavior, change the **DelObj** (short for "delete objects") variable. To have Region erase the source entities automatically, set it to -2 or 2:

DelObj	Meaning
-2	Prompts if source entities should be erased with Region
-1	Prompts if source entities should be erased during commands like Extrude, Revolve, and Sweep
0	Leaves source entities in place
1 (default)	Erases source entities only during commands like Extrude, Revolve, and Sweep
2	Erases source entities with Region

TIPS Regions are always closed; there are no open regions.

The **Explode** command change region entities back to their original components.

After entities are converted to regions, they look no different, but they do have different properties, of which you learn about later.

In the following tutorials, you use the Boundary command and then the Region command to see how they work. Later, you apply boolean operations.

TUTORIAL: CREATING BOUNDARIES

```
Available in all versions of BricsCAD

Command    Boundary
Alias      bo
Menu Bar   Draw | Boundaries | Boundary Polyline
Ribbon     Draw | Draw | Polygons | Boundary
Toolbar    Draw | Boundaries
```

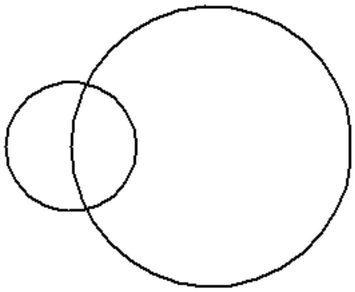
The **Boundary** command draws a boundary around the inside of selected areas and/or entities. It draws them as polylines or as regions — your choice, although in this tutorial you want regions.

(This command is a subset of the Hatch command. To place hatch patterns correctly, the Hatch command first draws invisible boundaries around the areas to be hatched; the hatching is applied inside the boundary, and then the boundary is optionally erased. The Boundary command draws the same outline, but keeps it.)

The Boundary command displays a dialog box that looks like a simplified version of the Hatch and Gradient dialog box, but the hatch components are missing.

To create boundaries as regions, follow these steps:

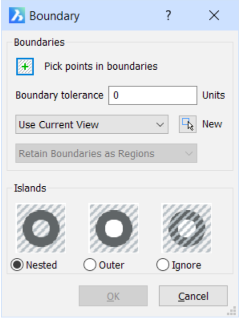
1. Start BricsCAD with a new drawing. The workspace or template does not matter.
2. Use the **Circle** command to draw two overlapping circles, as illustrated below. The size does not matter.



3. Turn off the HpBound system variable to ensure the resulting entities will be regions:

```
: hpbound
New current value for HPBOUND [1 for ON/0 for OFf] <1 for ON>: off
```

4. Start the **Boundary** command. Notice the Boundary dialog box; the boxed text on a nearby page explains its functions.

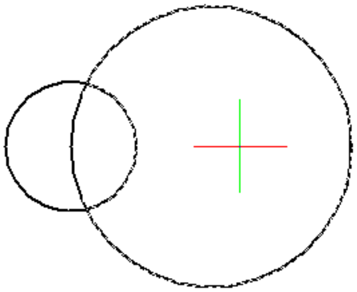


5. Click the **Pick Points in Boundaries** ⊞ button. It lets you pick a point inside the circles or the circles themselves. Based on your pick point, BricsCAD determines the boundary(ies) automatically.

6. Notice that the dialog box disappears (temporarily), and that you are prompted at the command bar.

```
Pick a point to define a boundary or hatch area or [Select entities/Undo]: (Pick inside
one of the circles)
```

Pick a point *inside* one of the circles, as illustrated below. It doesn't matter which one, just don't select a circle itself.

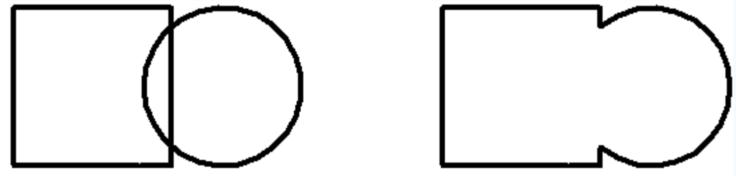


7. Press **Enter** to return to the dialog box:
8. And then click **OK** to exit the dialog box.

QUICK SUMMARY OF BOOLEAN OPERATIONS

UNION COMMAND

Union joins two or more regions into one. This allows you to create complex entities from simple ones.

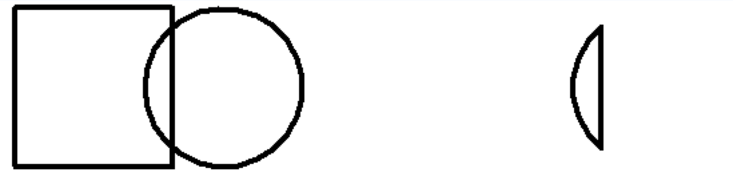


***Left**: Original objects (two regions); **right**: square unioned with circle as a single object*

In boolean terms, the union operation returns everything in region #1 OR in #2.

INTERSECT COMMAND

Intersect removes all but the overlapping portions of two or more regions. This allows you to find the areas in common between entities.



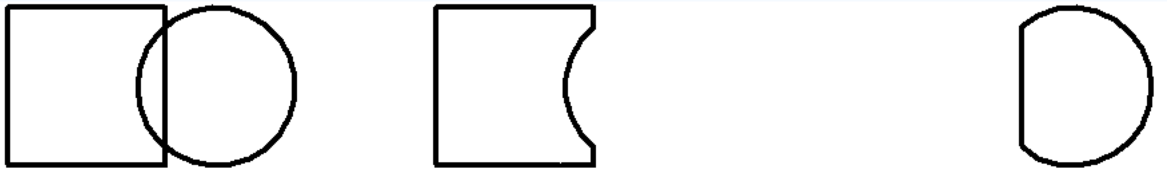
***Left**: Original objects (two regions); **right**: circle intersected with square*

In boolean terms, the intersection operation returns everything that's in region #1 AND in region #2.

SUBTRACT COMMAND

Subtract subtracts one set of regions from another. This allows you to remove parts of entities.

There are two outcomes possible from subtraction. The outcome depends on the order in which you select the regions, as shown by the figure below. In the center result, the circle was removed from the square; in the right result, the opposite occurred: the square was removed from the circle. (Selection order does not matter for the union and intersect operations.)

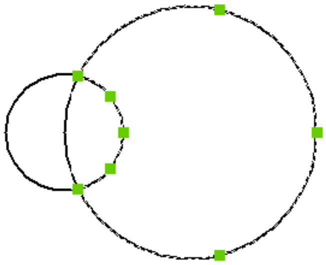


***Left**: Original objects (two regions); **center**: circle subtracted from square; **right**: square subtracted from circle*

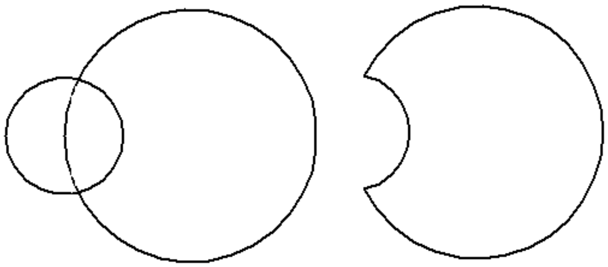
In boolean terms, the subtraction operation returns everything that's in region #1 but NOT in region #2.

9. The circles look no different, yet BricsCAD has drawn an entity inside one of them. To see it, drag it away from the circles, like this:

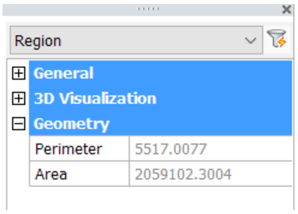
   a. Click the circle that surrounds your pick point made during the Boundary command. Notice that when BricsCAD highlights it, the highlight looks like a crescent moon. The moon is the region entity.



   b. Drag the region away from the two circles. Notice that it is an independent entity; the two original circles remain in place.



10. Use the Properties panel to determine that the entity is in fact a region. Notice that the Properties panel also reports the area and perimeter of the region, a useful side effect!



Region

Perimeter	5517.0077
Area	2059102.3004

As you can see from the result of this tutorial, regions are a handy way to create unusual shapes as a single entity. In the next tutorial, you use the Region command.

TIP After the boundary (region or polyline) is created, the Boundary command does not delete source entities, even when the **DelObj** variable (short for "delete entity") is set to 2 to force deletion.
The command line version is **-Boundary**, and is meant for use by scripts and programming routines.
Yes, you can use Boundary on regions, and Region on polylines made by the Boundary command, and Region on regions. In all cases, an identical copy is made of the region or polyline.

Boolean Operations

Available only in the Pro, Platinum, Ultimate versions of BricsCAD

When drawings contain two or more regions or 3D solids, you can perform *boolean* operations on them. These operations allow you to combine separate regions and 3D solids and make them one — or to separate one into many. “Boolean” is a named after Mr Boole, whose mathematics describe the logical operations that you may have learned in school, such as AND, OR, and NOT.

Together with Boundary and Regions commands, boolean commands are employed to construct complex shapes with BricsCAD. Indeed, boolean operations work only with region entities and 3D solids. They do not work with regular 2D and 3D entities like lines and 3D polymeshes; you need to first convert regular entities into regions or 3D solids.

BricsCAD provides three boolean commands to combine and separate regions and solids. (See the details in the boxed text on a nearby page.) Here are the names of the commands, and what they do to regions:

- **Union** command — combines two or more regions into one region (AND operation)
- **Intersect** command — finds the area common to two or more overlapping regions (OR operation)
- **Subtract** command — removes the area of one or more regions from other overlapping ones (NOT operation)

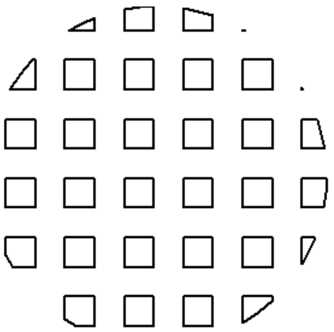
Technically, these commands employ ACIS modeling to perform their work, which is why they are not available in BricsCAD Classic, unfortunately.

The trickiest of the boolean operations is subtraction, which is why you get to employ it in the following tutorial.

TUTORIAL: CREATING A WAFFLE SHAPE

```
Commands   ArrayClassic and -Array
Aliases    ar, -ar
Menu Bar   Modify | Array | Array Dialog Box

Command    Region
Alias      reg
Menu Bar   Draw | Region
Ribbon     Surface | Create | Region
Toolbar    Draw | Region

Command    Subtract
Alias      su
Menu Bar   Model | 3D Solids Editing| Subtract
Ribbon     Solid | Edit | Boolean | Subtract
Toolbar    3D Solid Editing | Subtract
```

In this tutorial, you create a waffle shape as illustrated below. After drawing some intersecting rectangles, you will apply the Subtract command to form the waffle shape. To determine the net area, you find the mass properties of the shape; this would be very difficult to do without regions.

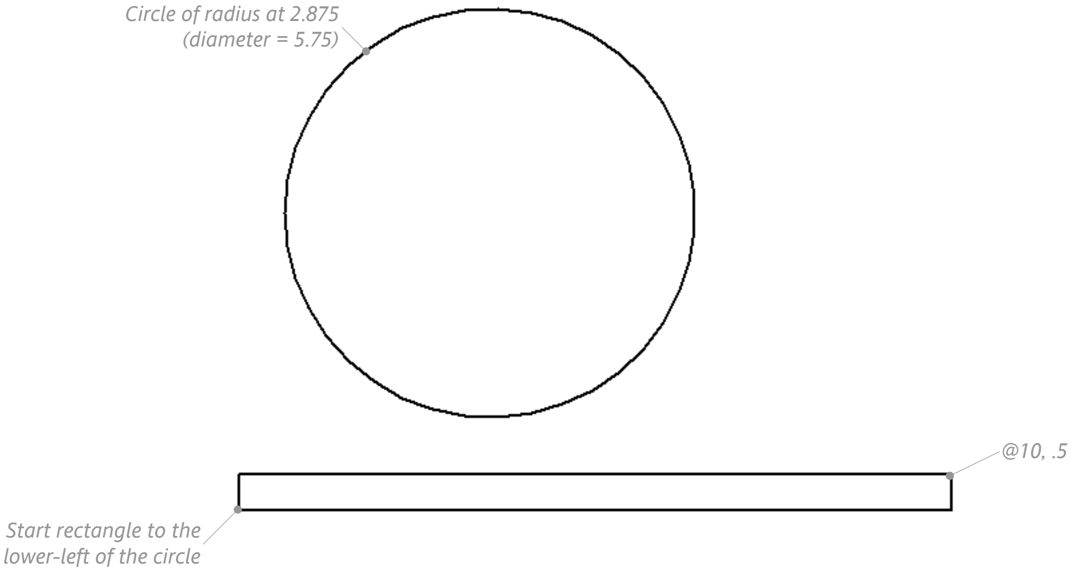


1. Start BricsCAD with a new drawing.

2. Using the **Circle** command, draw a circle with a radius of **2.875** units at the origin (0,0), as follows:

```
 : circle
Select center of circle or [2Point/3Point/TTR/Arc/Multiple]: 0,0
Set Radius or [Diameter]: 2.875
```

(If necessary, use the **Zoom Extents** command to see the entire circle, and then employ **Zoom 0.5x** to get some space around it.)



3. Now draw a rectangle with the **Rectang** command:

```
 : rectang
```

The rectangle's first corner is not crucial, except that it should be to the lower-left of the circle. See the figure above.

```
Select first corner of rectangle or [Chamfer/Fillet/Rotated/Square/Elevation/Thickness/
Width of line/Area/Dimensions]: qua
Snap to quadrant of: (Pick a point to the lower-left of the circle)
```

Enter relative coordinates to position the other corner. The width (x) does not matter, but the height (y) should be 0.5 units. I'm going to use a width of 10 units. I'll specify the other corner using relative coordinates, as follows:

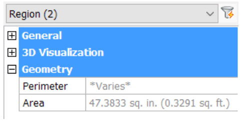
```
Other corner of rectangle:  @10,.5
```

4. The next step is to convert the circle and rectangle to region entities using the **Region** command:

```
: region
Select entities or boundaries: all
Select entities or boundaries: (Press ENTER to end entity selection)
2 regions(s) created
```

After the entities are converted to regions, they look no different. The circle and rectangle are, however, now *region* entities that are circular and rectangular in shape. The Boolean operations that you carry out later in this tutorial would not work on actual circles and rectangles, and so they had to be converted to regions.

If necessary, use the Properties panel to convince yourself the entities are now regions!



Region (2)

⊞ General

⊞ 3D Visualization

⊟ Geometry

Perimeter	*Varies*
Area	47.3833 sq. in. (0.3291 sq. ft.)

5. Create copies of the rectangle region to cover the circle.

Now, you could use the Copy command for this task, but it is much faster to deploy the **-Array** command. In this tutorial, you array the rectangle twice. The first time you array the rectangle vertically; the second time, you rotate one by 90 degrees (with the Mirror command) and then array it horizontally.

To start the -Array command, enter the **-ar** alias:

```
: -ar
```

6. Choose the rectangle:

```
Select entities to array: (Pick the rectangle)
Select entities to array: (Press Enter to end entity selection)
```

This command can make arrays that are polar (circular) or rectangular, but you want rectangular:

```
Type of array [Polar/Rectangular] <Rectangular>: (Press Enter to accept Rectangular)
```

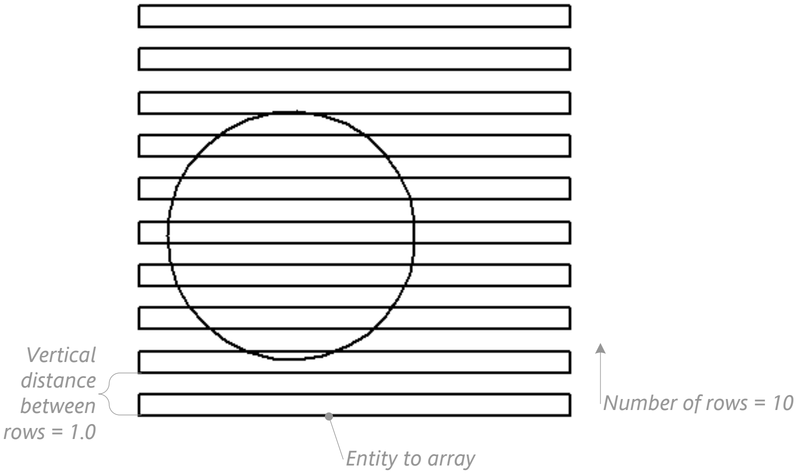
The number of rows in the first array is somewhat arbitrary, because you want to cover the entire circle with rectangles. If there are too many, you can just erase the extras; if too few, then it's a pain to restart the Array command, so draw too many in the first place!

```
Number of rows in the array <1>: 10
Number of columns <1>: (Press Enter to accept default, 1)
```

The distance between rows is 1 unit:

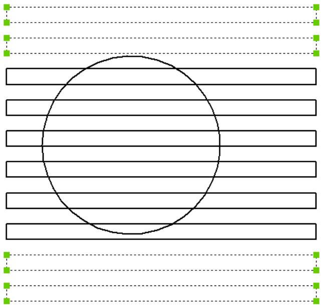
```
Vertical distance between rows, or spacing rectangle: 1
```

Notice that BricsCAD instantly creates ten copies of the rectangle.



(The reason I use command-line oriented -Array command instead of the dialog box-toting Array is because it can be faster to enter values at the command line than hunting through a dialog box.)

7. Use the **Erase** command to remove superfluous rows, in other words, those that don't cover the circle.



8. Now you want a second set of rectangles at 90 degrees. Create the first one by mirroring an existing rectangle with the **Mirror** command:

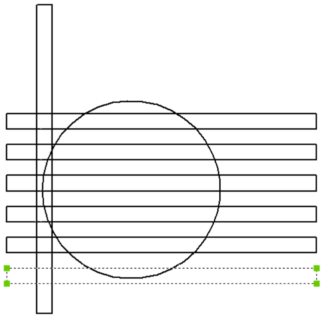
```
: mirror
```

Select the lowest rectangle, like this:

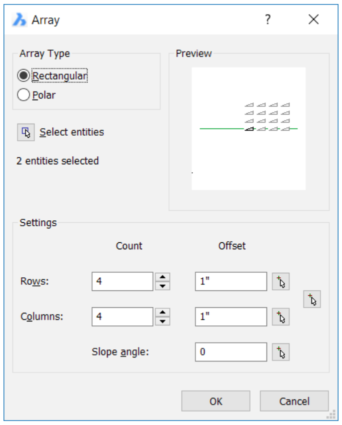
```
Select entities to mirror: (Pick the rectangle, indicated by the figure below)
Select entities to mirror: (Press ENTER to end entity selection)
```

BricsCAD needs an imaginary line called the "mirror line" about which to mirror the rectangle. The placement of the mirror line is not crucial, as long as it is at 45 degrees to the rectangle. The easy way to do this is to employ the following relative polar coordinates: @1<45 as the mirror line's end point:

```
Start of mirror line: 0,0
End of mirror line: @1<45
Delete the original entities? [Yes-delete entities/No-keep entities] <No-Keep entities>:
(Press Enter to accept default, N)
```



9. With the vertical rectangle in place, start the **ArrayClassic** command. This is the dialog box version of the command.



Array

Array Type

● Rectangular

○ Polar

Select entities

2 entities selected

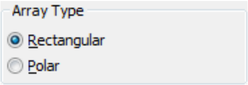
Preview

Settings

	Count	Offset
Rows:	4	1"
Columns:	4	1"
	Slope angle:	0

OK Cancel

a. For Array Type, ensure **Rectangular** is chosen:



Array Type

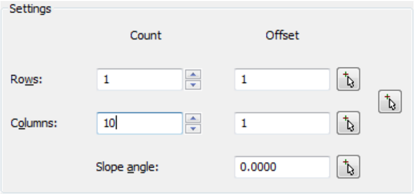
● Rectangular

○ Polar

b. Click **Select Entities** and then enter ‘L’ to select the last-chosen entity:

```
Select entities to array: l
Entities in set: 1
Select entities to array: (Press Enter to return to the dialog box)
```

c. Specify the number of copies to make:

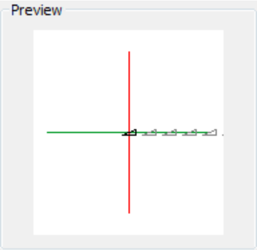


Settings

	Count	Offset
Rows:	1	1
Columns:	10	1
Slope angle:		0.0000

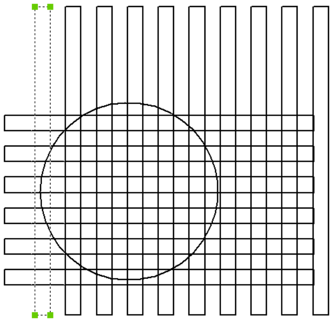
Rows Count:	**1**
Columns Count	**10**
Column Offset:	**1**

Notice that the preview window shows several columns, one row high.

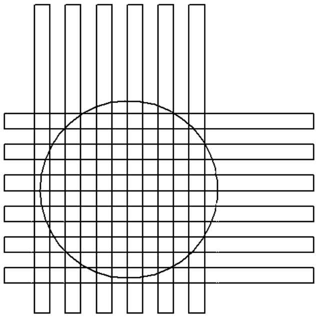


Preview

d. Click **OK**, and BricsCAD instantly creates ten copies of the vertical rectangle.

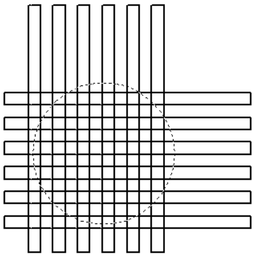


10. Erase the rectangles that don’t lie on the circle.



11. Now you finally get to use a Boolean. With the **Subtract** command, remove the rectangular regions from the circular region, as follows:

```
: subtract
Select ACIS entity to subtract from: (Pick circle)
Select ACIS entity to subtract from: (Press Enter to end “From” entity selection)
```



To pick all of the rectangles more quickly, follow these steps:

a. First, use **All** selection mode to select everything in the drawing.

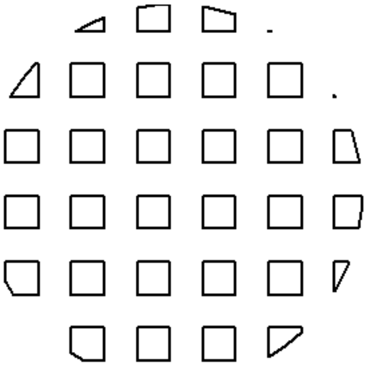
```
Select ACIS entities to subtract: all
```

b. Second, use the **R** (remove) option to remove just the circle from the selection set.

```
Select ACIS entities to subtract: r
Subtract entities from selection set: (Pick circle)
Subtract entities from selection set: (Press ENTER to end the command)
```

That’s just two selections, instead of 12!

Notice the result: the **Subtract** command removes the overlapping regions, producing the waffle effect — yet those 32 parts are in reality a *single* entity. This again shows you one benefit of working with regions.



Measuring Regions

Command	MassProp
Menu Bar	Tools | Inquiry | Mass Properties
Ribbon	Tools | Inquiry | Mass Properties
Toolbar	Inquiry | Mass Properties

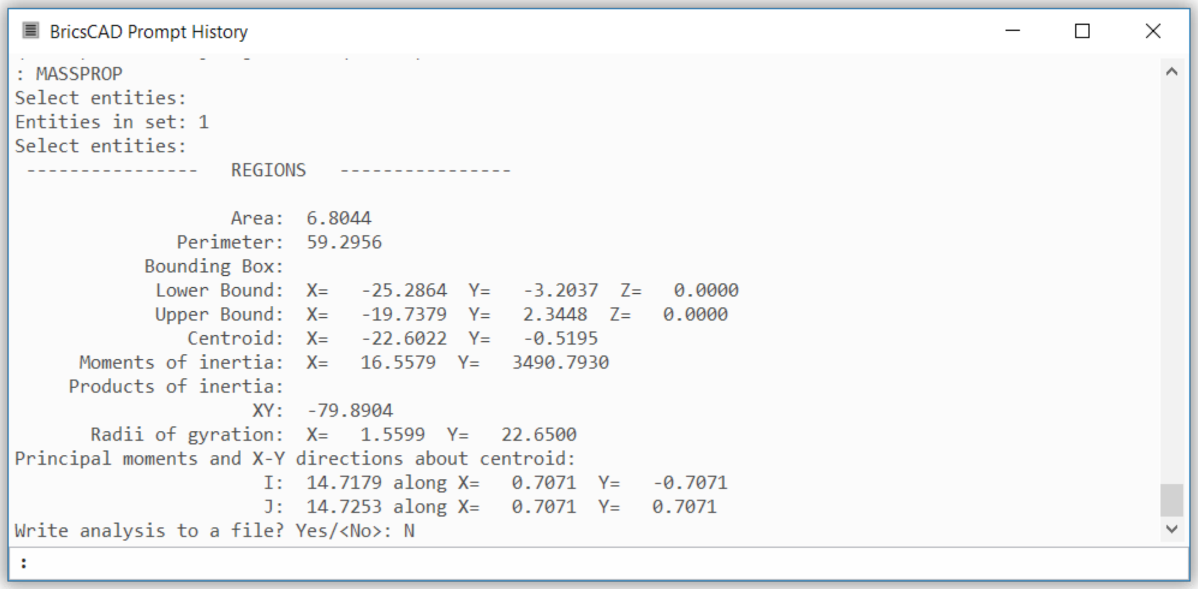
You've seen in the first two tutorials how region entities are unique in BricsCAD, by consisting of unusual shapes and even a series of seemingly disconnected shapes.

Another benefit we get from regions is that we can easily determine their physical properties — especially easy for ones made of many separate parts, like the waffle shape. If the waffle were instead made of 32 regular 2D entities, then we would have to add up the areas of all individual parts, and then find the total. In contrast, the waffle region is a single entity, and so easy to measure.

1. To find the total area of the waffle pattern, use the **MassProp** command, as follows:

```
: massprop
Select entities: (Pick the waffle shape)
Select entities: (Press Enter)
```

2. BricsCAD displays the results of the analysis in the text window. If necessary, press **F2**:

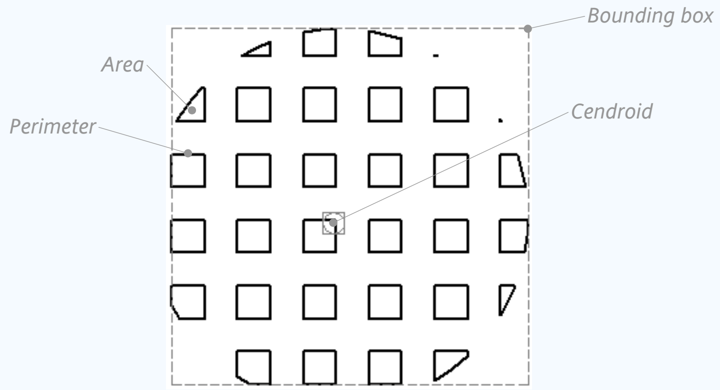


```
: MASSPROP
Select entities:
Entities in set: 1
Select entities:
 ----------------   REGIONS   ----------------

                    Area:  6.8044
               Perimeter:  59.2956
            Bounding Box:
             Lower Bound:  X=   -25.2864  Y=   -3.2037  Z=   0.0000
             Upper Bound:  X=   -19.7379  Y=   2.3448  Z=   0.0000
                Centroid:  X=   -22.6022  Y=   -0.5195
      Moments of inertia:  X=   16.5579  Y=   3490.7930
     Products of inertia:
                      XY:  -79.8904
       Radii of gyration:  X=   1.5599  Y=   22.6500
Principal moments and X-Y directions about centroid:
                       I:  14.7179 along X=   0.7071  Y=   -0.7071
                       J:  14.7253 along X=   0.7071  Y=   0.7071
Write analysis to a file? Yes/<No>: N
:
```

So now you know: The area of the waffle pattern is 6.8044 square units, precisely. The value you see for your region may differ due to the way you placed the rectangles.

QUICK SUMMARY OF MASS PROPERTIES

The MassProp command uses terms that have the following meaning:



- **Area** — cumulative area of all 24 waffles.
- **Perimeter** — cumulate perimeters of all 24 waffles.
- **Bounding Box: Lower Bound and Upper Bound** — rectangular limits of the region; the coordinates describe the lower-left and upper-right corners of the corners of an imaginary rectangle that tightly encloses the region.
- **Centroid** — center of mass of the region's 24 waffles, excluding the open areas.
- **Moments of inertia** — measure of the region's resistance to angular acceleration.
- **Product of Inertia: XY** — measures the region's resistance to change in rotation.
- **Radius of Gyration** — distance from the axis that the entire region can be concentrated to obtain the same mass moment of inertia.
- **Principle moments and X-Y directions about centroid** — equivalent to torque.

This lesson introduced you to regions and Boolean operations. These concepts extend to working with 3D models in BricsCAD Professional and other 3D software packages.

Notes

CHAPTER 10

3D Modeling & Direct Editing

Here you learn how to create 3D models using traditional and modern editing techniques. Traditional techniques use *commands* to create and manipulate 3D models; modern techniques manipulate 2D and 3D entities *directly*.

In this chapter, you learn how to create 3D models by directly manipulating elements, without necessarily using a command. (Direct modeling and editing are available only in the Pro, Platinum, and Ultimate editions of BricsCAD.)

IN THIS CHAPTER

- Learning about 3D solid models
- Extruding profiles (2D entities) into bodies
- Rotating 3D viewpoints
- Subtracting 3D entities from one another
- Modeling with direct editing
- Aligning UCSes
- Adding fillets to 3D edges

KEY TERMS IN THIS CHAPTER

Direct modeling and editing — creates and edits 3D parts directly, without entering commands

Profiles — describes 2D entities that define 3D parts.

Quad — multi-tiled cursor with common commands

Shell — refers to a hollowed out 3D solid model

Sub entities — describes entities that make up 3D solid models, such as faces and edges

Union — joins two or more solid models into a single body

USEFUL ABBREVIATIONS

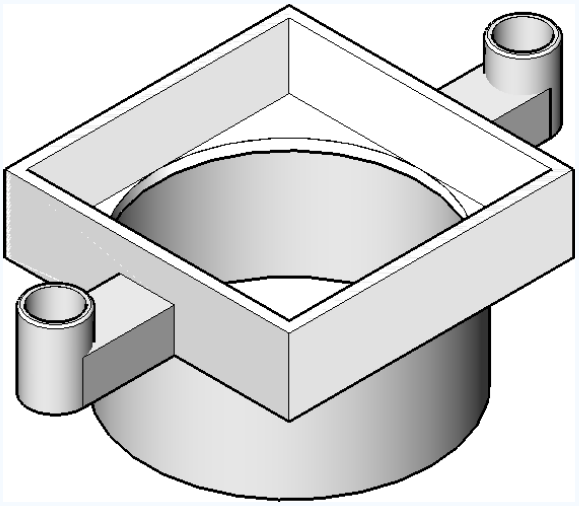
QUAD	Quad cursor
SUB	Sub-entity selection

NEW COMMANDS

Command	Shortcut	Menu Bar	Ribbon Tab
DelObj	...	...	...
Extrude	Ext	Model | 3D Solids | Extrude	Solid | Create
QuadDisplay	F12	...	...
SolidEdit	...	Model | 3D Solid Editing | Shell	Solid | Edit
UCS	...	...	...

THIS CHAPTER'S MODEL

By the end of this lesson, your 3D model will look like this:



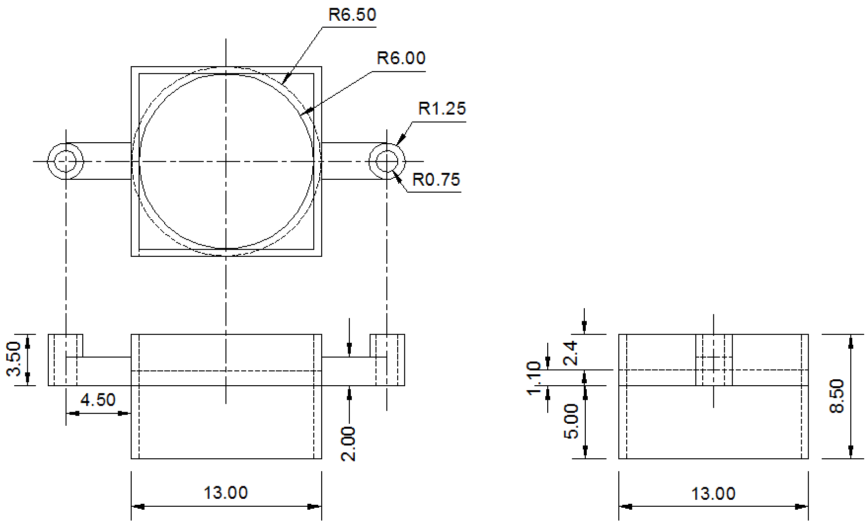
Focus-bracket drawing 3d-part.dwg

About 3D Solid Models

(PRO, PLATINUM, ULTIMATE EDITIONS ONLY) BricsCAD Pro, Platinum, and Ultimate allow you to create 3D models from *solids*. Solids are 3D entities that are solid on the inside, which distinguishes them from other 3D entities that are not solid, such as surfaces and polyface meshes. Being solid through and through is important, because they accurately mimic real world entities. For instance, using third-party tools like FEA (finite element analysis) and CFD (computational fluid dynamic), designs can be tested on the computer to ensure they work properly before they are manufactured.

BricsCAD offers several methods for creating 3D models from solids, and here you learn about two of them, traditional commands and direct modeling.

In this chapter's tutorials, you model the focus ring mount for a webcam. The 2D plans are illustrated below and finished 3D model is shown on the facing page. The dimensioned 2D drawings illustrate a further benefit to 3D: it is easier for non-experts to visualize products in 3D than in 2D.



The 2D drawing is *Tutorial-10-2D.dwg* and the completed 3D model is *3d-part.dwg*, and both are available through this cloud folder:
https://my.pcloud.com/publink/show?code=XZn98mkZ7T7f1pynxJYb7vDyVl6DwyqK1Ysk

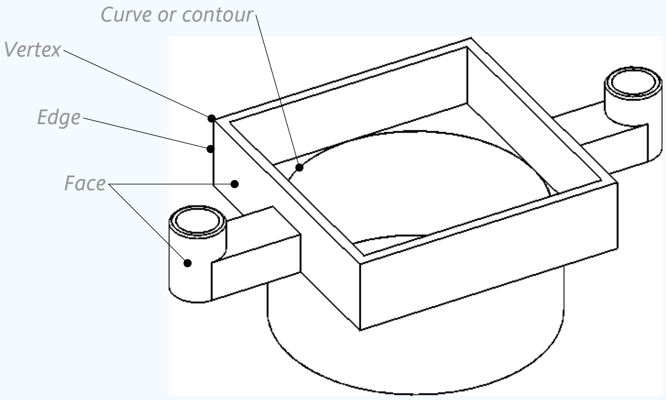
PLANNING AHEAD

There are specific differences between 2D drafting and 3D modeling. One is that 3D has a limited number of ways to create parts that make up models, whereas there are many ways in 2D. For instance, to create a box shape in 3D, you use the **Box** command or extrude a rectangle; that's pretty much it; in 2D, by contrast, you can draw a rectangle with lines, polylines, traces, with **Rectangle**, **Polygon** commands, and so on.

ELEMENTS OF 3D MODELS

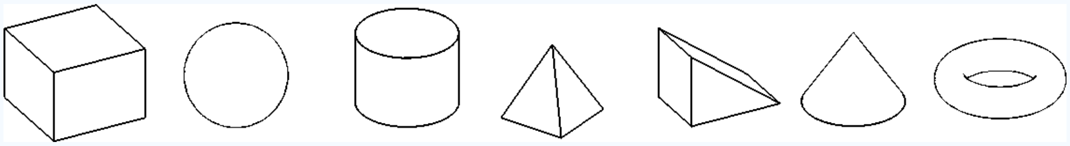
Three-dimensional modeling employs a jargon of its own. Here is a visual reference to some common terms.

PARTS OF A 3D MODEL



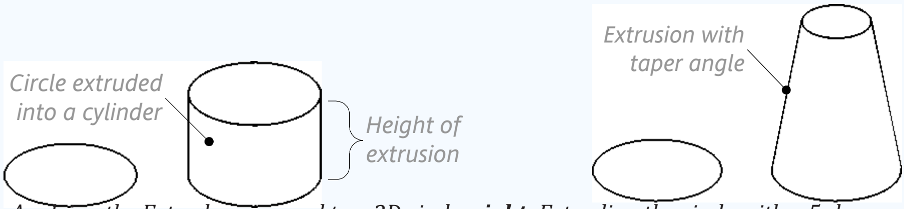
CREATE 3D MODELS FROM 3D PRIMITIVES

Primitives are basic 3D parts are made with **Box, Sphere, Cylinder, Pyramid, Wedge,** and **Torus** commands.

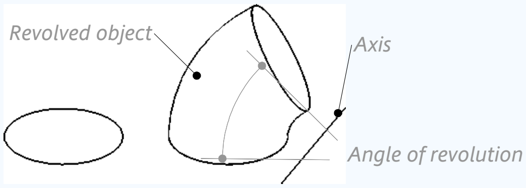


CREATE 3D MODELS FROM 2D ENTITIES

Extrude and **Revolve** commands turn 2D entities into 3D ones:

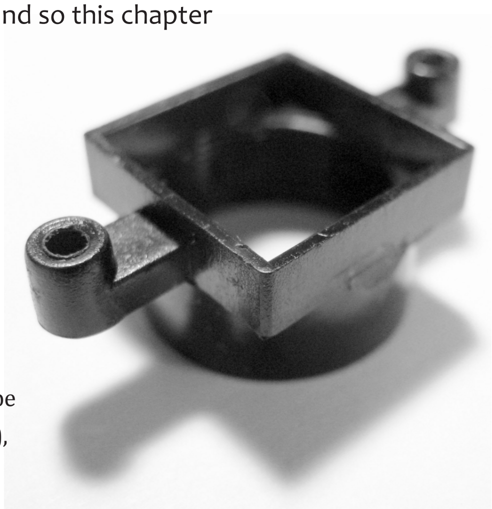


Left: *Applying the Extrude command to a 2D circle;* ***right:*** *Extruding the circle with a 5-degree taper*



Extruding a circle to 45 degrees with the Revolve command, about an axis

Another difference is that you need to plan ahead to construct 3D models; this differs from 2D, where you can simply start drawing. There is, however, a Catch-22 here: you become effective in planning ahead only after you gain experience with 3D modeling, and so this chapter shows you some of the tricks in putting together 3D models.



Planning Ahead by Deconstructing 3D

One way to plan ahead is to deconstruct the model. Here are some tips:

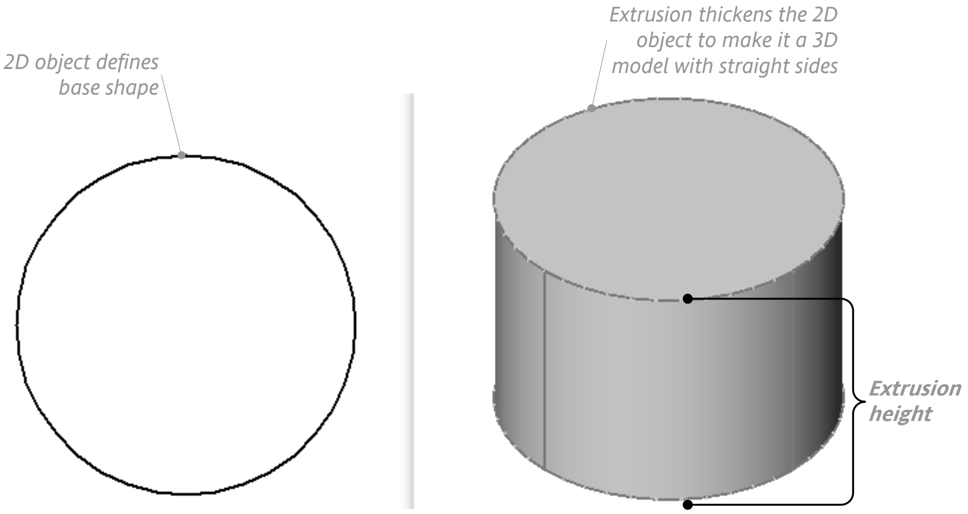
- Look to see what kinds of sub-parts exist. "Sub-parts" are portions of the model that look like boxes and cylinders.
- Holes are formed from cylinders.
- Figure out which common operations can be applied. For instance, parts can be repeated (made once then copied), mirrored (made once then copy-mirrored), or made uniquely (made once).
- Rounded edges are added with the **Fillet** command.
- 2D drawings determine the dimensions of the part.

Examine the photograph (right) and identify sub-parts (a.k.a "primitives") that could be modeled first. Notice that there are some are square ones and some round ones. How would these be modeled? (Hint: with the **Box** and **Cylinder** commands.)

TRADITIONAL EXTRUSION METHOD

Because the sides are parallel, much of this part can be constructed from *extrusions*. An "extrusion" takes a 2D outline and then makes it taller to make it 3D.

To create an extrusion, draw the outline of the part in 2D (on the x,y-plane), and then use the **Extrude** command to thicken it in the z-direction. See the figure below for how this works. Extrusions have either straight or slanted sides. (If you want a part to have curved sides, then *revolve* the 2D entity with the Revolve command.)



2D object defines base shape
Extrusion thickens the 2D object to make it a 3D model with straight sides
Extrusion height

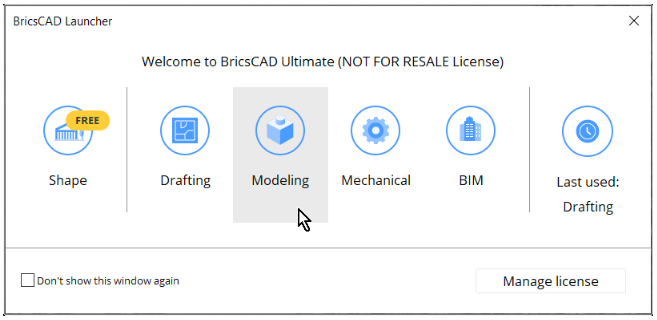
Left: *2D circle defining the diameter of the cylinder;* ***right:*** *Circle extruded to become a 3D cylinder*

In this set of tutorials, you use variations on a few commands to model the webcam focus ring holder. In the first tutorial, you use the traditional **Extrude** command, which in CAD dates back to the late 1980s; in a later tutorial, you get to use a newer approach to do the same thing known as "direct modeling."

Preparing the Drawing

Here are the steps you need to take to prepare the drawing for 3D modeling.

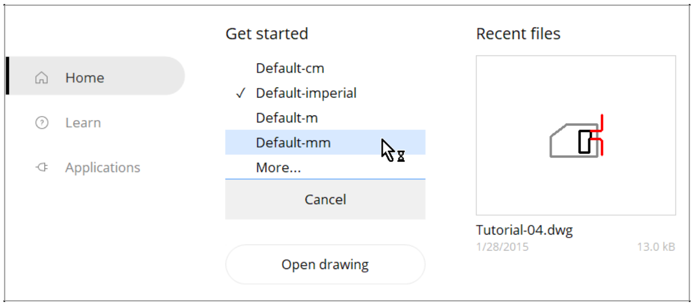
1. Start BricsCAD.
2. Notice the Launcher dialog box. Click **Modeling**.



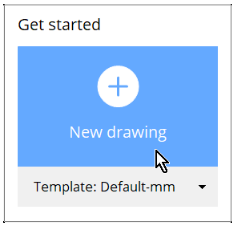
BricsCAD Launcher

Welcome to BricsCAD Ultimate (NOT FOR RESALE License)

3. In the Getting Started tab, change the template file to "Default-mm." This ensures the drawing will be made with metric units.



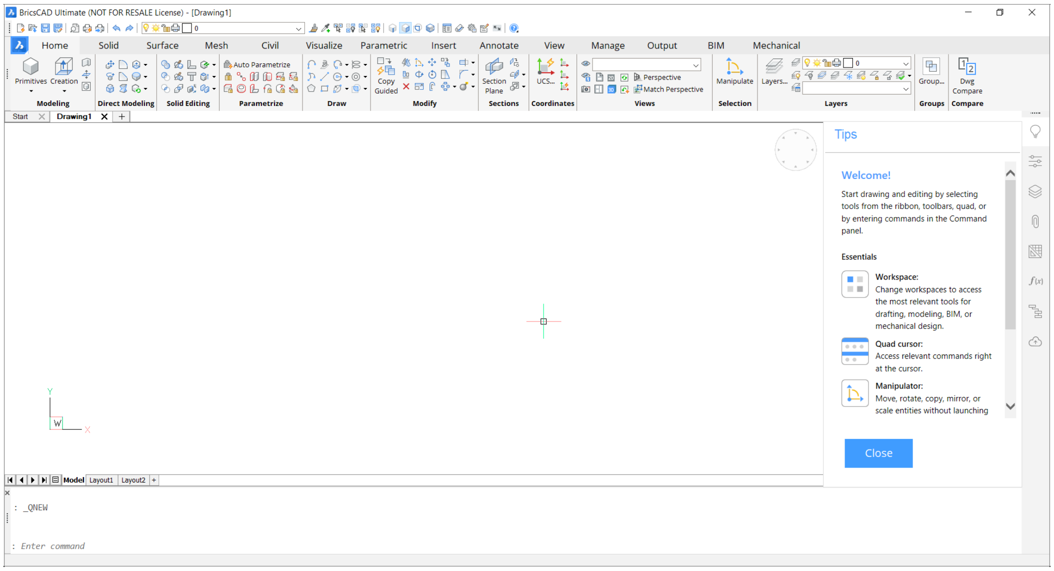
4. Click **New Drawing**.



Notice that BricsCAD opens in the Modeling workspace.

5. Prepare this drawing by changing the following settings in the status bar:

Setting	Toggle	Comments
SNAP	Off	You use entity snaps and dynamic dimensions, in place of snap spacing
GRID	Off	The grid is not useful for this project
ESNAP	On	Turn on **INTersection** snap; turn off all others



Welcome!

Start drawing and editing by selecting tools from the ribbon, toolbars, quad, or by entering commands in the Command panel.

Essentials

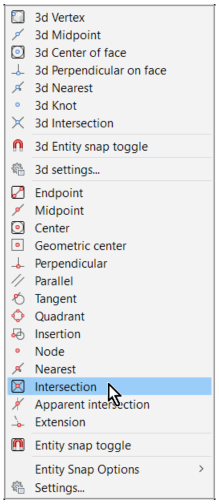
Workspace: Change workspaces to access the most relevant tools for drafting, modeling, BIM, or mechanical design.

Quad cursor: Access relevant commands right at the cursor.

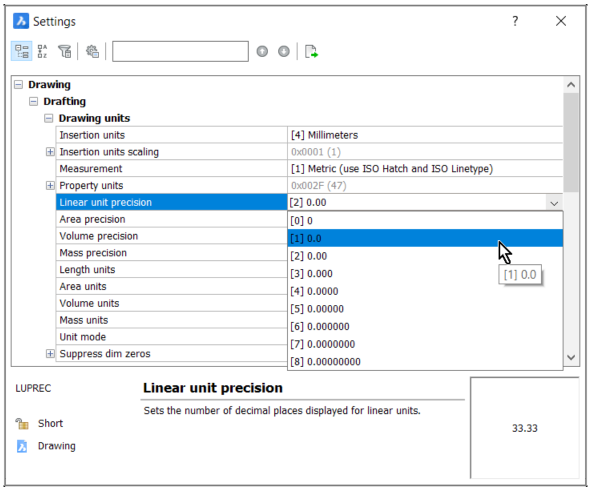
Manipulator: Move, rotate, copy, mirror, or scale entities without launching

```
: _QNEW

: Enter command
```

To turn on INTersection entity snap, right-click **ESNAP** on the status bar, as shown below. Ensure all other esnaps are turned off.



6. Use the **Units** command to change the display precision of decimal places (linear units) to 0.0mm. You will be working with dimensions to the nearest 0.5mm.



Drawing
Drafting
Drawing units

Insertion units	[4] Millimeters
Insertion units scaling	0x0001 (1)
Measurement	[1] Metric (use ISO Hatch and ISO Linetype)
Property units	0x002F (47)
Linear unit precision	[2] 0.00
Area precision	[0] 0
Volume precision	[1] 0.0
Mass precision	[2] 0.00
Length units	[3] 0.000
Area units	[4] 0.0000
Volume units	[5] 0.00000
Mass units	[6] 0.000000
Unit mode	[7] 0.0000000
Suppress dim zeros	[8] 0.00000000

[1] 0.0

LUPREC

Short

Drawing

Linear unit precision

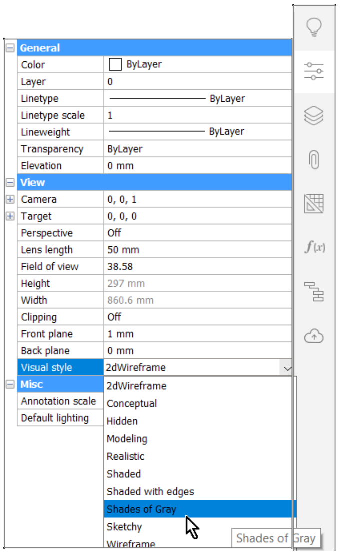
Sets the number of decimal places displayed for linear units.

33.33

7. Close the Settings dialog box by clicking the **X**.
8. Change the visual style to "Shades of Gray," which I like the best for 3D modeling. You do this through the Properties panel:
   a. If the panel is not open, then enter the **Properties** command.

   b. In the **View** section, click the **Visual Style** droplist.

   c. Choose "Shades of Gray." (By the way, there are 256 shades of gray, not just 50.)



General	
Color	□ ByLayer
Layer	0
Linetype	———— ByLayer
Linetype scale	1
Lineweight	———— ByLayer
Transparency	ByLayer
Elevation	0 mm
View	
Camera	0, 0, 1
Target	0, 0, 0
Perspective	Off
Lens length	50 mm
Field of view	38.58
Height	297 mm
Width	860.6 mm
Clipping	Off
Front plane	1 mm
Back plane	0 mm
Visual style	2dWireframe
Misc	
Annotation scale	
Default lighting	

9. Change the value of variable **DelObj** to **0**. This ensures 2D entities are retained after 3D objects are made from them.

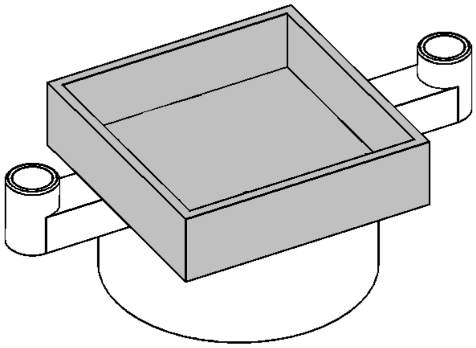
   Recall from a previous lesson that this variable determines what happens to 2D entities after you convert them to 3D models. (When set to 1 or 2, the program erases them; I find it useful to keep them around.)
10. Finally, save the drawing by any name that you like, such as "3d-part.dwg".

With the drawing prepped, you are ready to start modeling!

Modeling a Box

```
Command    Extrude
Alias      ext
Menu       Solids | Solids | Extrude
Ribbon     Solid | Create | Extrude
Toolbar    3D Solids | Extrude

Command    SolidEdit
Menu       Solids | Solid Editing | Shell
Toolbar    3D Solid Editing | Solid Edit
Ribbon     Solid | Edit | Shell
```

The top of the part is a square open box. It is shown in gray by the figure below.



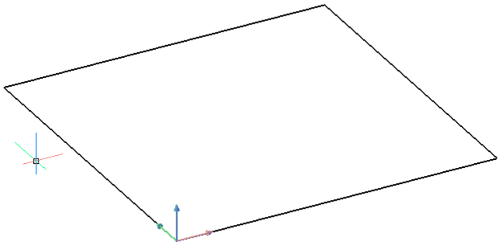
To make it, you'll work through three commands, in this order:

i. **Rectangle** command — defines the overall size and the base of the square

ii. **Extrude** command — changes the 2D square into a solid 3D box

iii. **SolidEdit** command — hollows the 3D box by *shelling* it

1. First, use the **Rectangle** command's **Dimension** option to draw a 2D square sized 13x13mm:

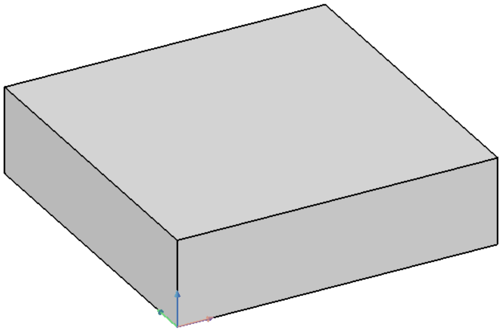


```
: rectangle
Choose first corner of rectangle or [Chamfer/Elevation/Fillet/Rotated/Square/Thickness/
Width/Area/Dimensions]: d
Length to use for rectangles <13.0>: 13
Width to use for rectangles <13.0>: 13
Select first corner of rectangle or [Chamfer/Elevation/Fillet/Rotated/Square/Thickness/
Width/Area/Dimensions]: 0,0
Other corner of rectangle: (Pick a point; it's not crucial where you pick it)
```

2. If necessary, use the **Zoom Extents** to see it better.

3. Now use the **Extrude** command to convert the square into a short 3D box. Specify a height of 3.5mm, as follows:

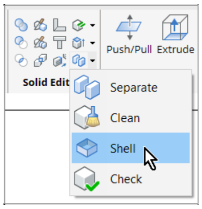
```
: extrude
Select entities/subentities to extrude or set [MOde]: (Pick the square)
Select entities/subentities to extrude or set [MOde]: (Press ENTER to end entity selection)
Specify height of extrusion or [Direction/Path/Taper angle]: 3.5
```



4. "Shell" is an option of the SolidEdit command that turns solid entities into ones with "walls" — it uniformly removes the insides. Follow these steps to make a hollow box:

   a. Because the SolidEdit command has many options, it is faster to get to the Shell option via the ribbon. (Don't enter the "shell"command, because it doesn't do what you need!)

   From the ribbon's **Solids** tab, look in the **Solid Editing** panel, and then click **Shell**. (It's a bit hard to find it, but Shell is among the ones that drop down from the last icon, Separate.)



QUICK SUMMARY OF EXTRUDE COMMAND

Extrude commands thickens 2D entities into 3D solids or surfaces (included in BricsCAD Pro and Platinum).

```
: extrude
Select entities/subentities to extrude or set [MOde]:
Specify height of extrusion or [Direction/Path/Taper angle]:
```

Entities/subentities — selects 2D closed entities or faces of 3D solids

MOde — specifies the result of the extrusion, solid or surface

Height — specifies the height of the extruded entities; positive values extrude in the positive z direction, negative values go "downwards"

Direction — controls the direction of the extrusion, up or down along the z axis

Path — allows curved extrusions by specifying an entity that determines the path

Taper angle — gives sloping sides to the extrusion; positive angle slope inwards, negative angles outwards

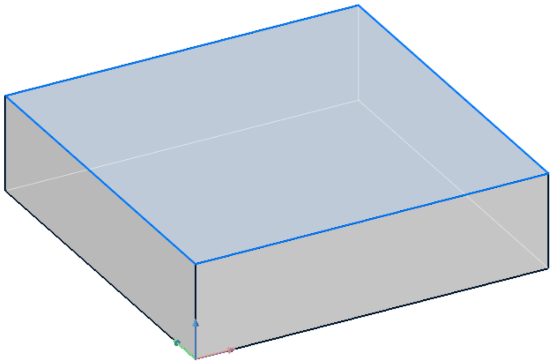
```
: _solidedit
Enter a solids editing option Face/Edge/Body/Undo/eXit: _body
Enter a body editing option Imprint/seParate/Shell/cLean/Check/Undo/eXit: _shell
```

b. As prompted, select the extruded box:

```
Select 3d solid: (Pick anywhere on the box)
```

Notice that the box turns blue, as BricsCAD confirms your selection.

c. Next, pick the top face, which I shown in blue in the figure below.



```
Select face to remove or [Add/Undo/ALL]: (Pick the face on top of the box)
Select face to remove or [Add/Undo/ALL]: (Press Enter to exit entity selection)
```

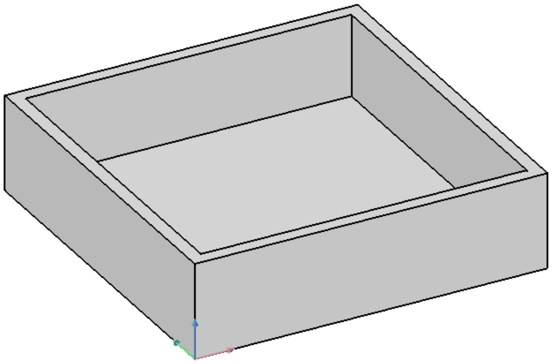
d. Specify the thickness of the walls, **0.5mm**, for the remaining sides:

```
Enter the shell offset distance: .5
```

e. And finally press **Enter** twice to exit the command:

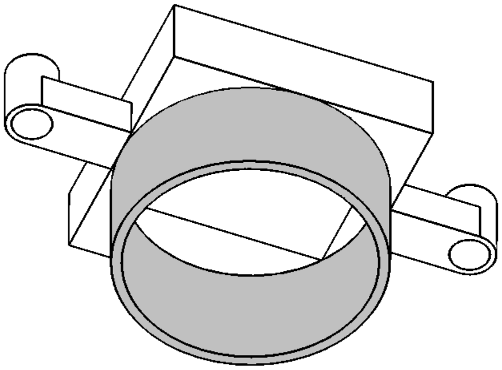
```
Enter a body editing option Imprint/seParate/Shell/cLean/Check/Undo/eXit: (Press Enter)
Enter a solids editing option Face/Edge/Body/Undo/eXit: (Press Enter)
```

Notice that the result is a hollow box; it no longer has a top.



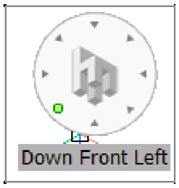
3D View Rotation

The next step is to add the round part underneath the box, shown in gray below.



But to do this, you need to spin the model so that you can see the bottom of it. There are several methods you could employ, but the easiest is the most direct way: real-time rotation. Here's how:

1. Hold down the **Ctrl** key.
2. On the Lookfrom widget, click the triangle at the 7pm position.



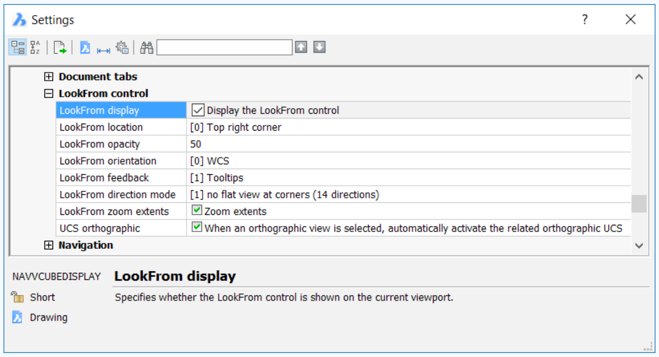
QUICK SUMMARY OF LOOKFROM COMMAND

```
: lookfrom
LookFrom [ON/OFF/Settings] <ON>: on
```

ON — turns on the Look From widget

OFF — turns off the Look From widget

Settings — displays the Settings dialog box at the Look From Control section



⊞ **Document tabs**

⊟ **LookFrom control**

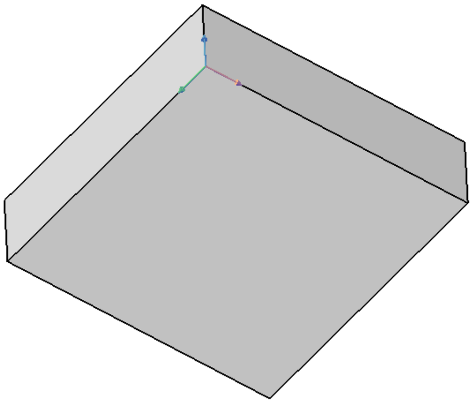
LookFrom display	☑ Display the LookFrom control
LookFrom location	[0] Top right corner
LookFrom opacity	50
LookFrom orientation	[0] WCS
LookFrom feedback	[1] Tooltips
LookFrom direction mode	[1] no flat view at corners (14 directions)
LookFrom zoom extents	☑ Zoom extents
UCS orthographic	☑ When an orthographic view is selected, automatically activate the related orthographic UCS

⊞ **Navigation**

NAVVCUBEDISPLAY **LookFrom display**

Specifies whether the LookFrom control is shown on the current viewport.

Notice that BricsCAD swirls the box around so that we now see the bottom of it.



3. If necessary, use the **Zoom Extents** command to see the entire model.

EXTRUSION BY DIRECT MODELING

Now that you have a taste of modeling using commands, we'll switch to direct modeling — modeling without commands.

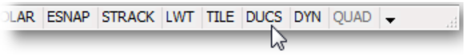
The bottom of the part has a hollow cylinder. In this part of the tutorial, you draw two circles and then extrude them without using commands to do the work. But first, this word of explanation of how to draw in 3D space...

Applying Dynamic UCS

A box has six sides in 3D. CAD, however, can only draw on one side at a time: the side that is in the current x,y-plane. No matter which 3D CAD package you use, they all fall back to drawing on the x,y-plane. And so a crucial aspect to direct modeling is working on a plane that is correctly-positioned.

Fortunately, CAD systems make it easy today to place the x,y-plane so that it matches the area on which you want to work, pretty much automatically. In BricsCAD, this feature is called *dynamic* UCS. Here is how it works for drawing a 2D circle on a 3D box:

1. On the status bar, ensure **DUCS** is turned on. The letters should look black, not gray.

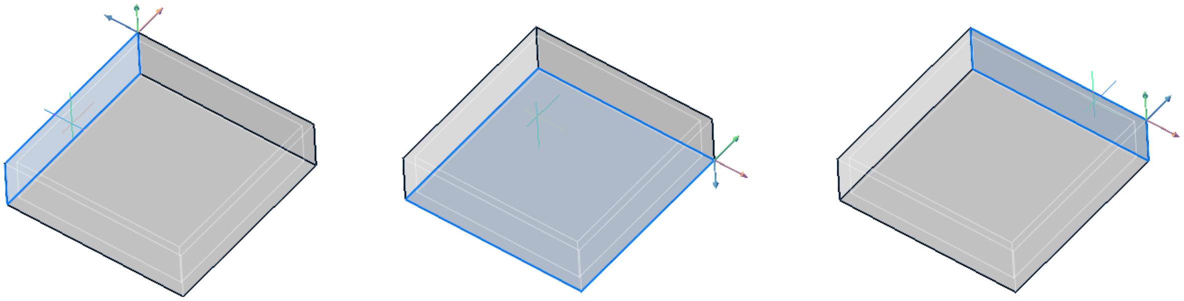


2. Start to draw a circle on the bottom of the box. Use the **Circle** command and its **2P** option, as follows:

```
: circle
Select center of circle or [2Point/3Point/TanTanRad/Arc/Multiple]: 2p
```

3. Now you get to see the effect of dynamic UCS: move the cursor around the visible faces of the box. Notice that two things happen: the face turns blue, and the UCS icon jumps to a corner of the face. Move the cursor and another face turns blue.

This is dynamic UCS at work: BricsCAD automatically relocates the x,y drawing plane to the face that you pick.



Left to right: *As cursor passes over a face, it turns blue and the UCS icon (tri-color icon) is relocated*

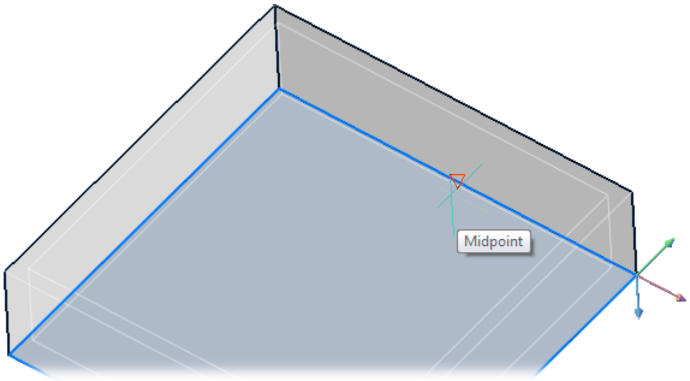
In summary: As the cursor passes over a face during DUCS, the face turns blue to tell you that it is the current temporary x,y drawing plane. The UCS icon is positioned at the current temporary origin (0,0,0). DUCS works during drawing and editing commands, because most of them operate only on an x,y plane.

TIP If you want to see a more dramatic effect of dynamic UCS at work, turn on the grid display. (Click **GRID** on the status bar.) As you move the cursor from face to face, the grid jumps from face to face.

4. To locate the circle, pick two points on the edges of the box, right on the middle of two facing edges; see the following figures.

   a. Position the cursor over the bottom face of the box. Notice that it turns blue.

   b. Move the cursor close to the edge (illustrated below) near where the word "Midpoint" appears, but do not go beyond the edge! (Were you to go beyond the edge, then the DUCS would jump to the adjacent face, telling BricsCAD that you want to draw the circle on a different face.)



   c. Enter a temporary MIDpoint entity snap mode:

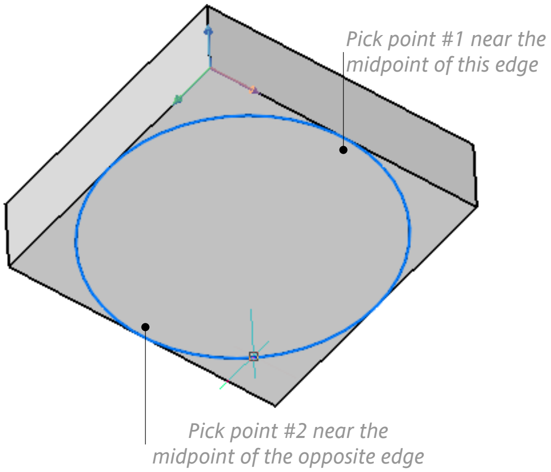
```
First point on diameter: mid
```

   d. Pick a point near the edge:

```
Snap to midpoint of: (Pick near point 1)
```

e. Repeat the MID ensap mode, and pick a point near the opposite edge:

```
Second point on diameter: mid
Snap to midpoint of: (Pick near point 2)
```



Pick point #1 near the midpoint of this edge

Pick point #2 near the midpoint of the opposite edge

QUICK SUMMARY OF DUCS ENTITIES

Dynamic UCS works with many kinds of entities during drawing, editing, and dimensioning:

- All 3D solid faces
- Most 2D entities

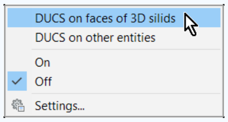
DUCS does not work with curved 3D entities, such as spheres, as they have no flat faces, and so the UCS cannot be determined.

(NEW IN V20) DUCS works with the following 2D entities: points, lines, 2D and 3D polylines, rays and xlines, arcs, circles, ellipses, splines, text and mtext, 2D solids, 3D faces, traces, blocks, viewports (in paper space), mlines, leaders and mleaders, hatches, helixes, cameras and lights, sections, PDF and image underlays, and shapes.

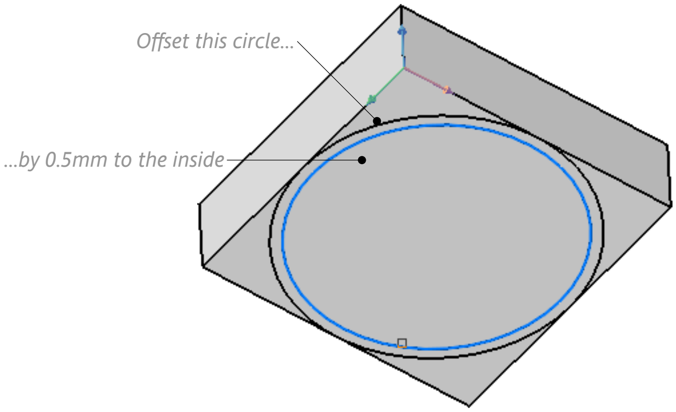
BricsCAD lets you determine whether DUCS supports 3D, 2D, or both through the **UcsDetect** variable:

UcsDetect	Meaning
3	Detect 2D entities and 3D faces
2	Detect only 2D entities
1	Detect only 3D faces
0	DUCS is off

Alternatively, right-click DUCS on the status bar, and then choose the entity types DUCS should work with:



5. A second circle is needed to define the thickness of the 0.5mm wall. But instead of drawing it, you make the copy using the **Offset** command.



```
: offset
Enter offset distance or [Through point/Erase/Layer] <Through point>: .5
Select entity/subentity or [Exit] <Exit>: (Select the black circle)
Select side for parallel copy or [Both sides/Multiple]: (Pick a point inside the circle)
Select entity/subentity or [Exit] <Exit>: (Press Enter to end the command)
```

Notice that the second circle appears.

Extruding Directly with the Quad Cursor

With the pair of 2D circles in place, you extrude them to create a 5mm-tall hollow cylinder. One way to make cylinders is to extrude circles.

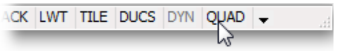
This time, you use the Quad cursor to start the extrusion. The Quad is unique to BricsCAD, giving you fast access to commands right at the cursor, as described earlier in this book. Making a hollow cylinder takes these steps:

i. Extrude the inside circle into a cylinder that is more than 5mm in height.
ii. Extrude the outside circle to exactly 5mm.
iii. To make the hole, subtract the taller cylinder from the shorter one.

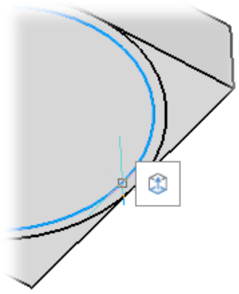
You extrude the inside circle by a distance longer than 5mm, because its height does not matter; making it taller makes it easier to erase it to create a hole.

Follow these steps:

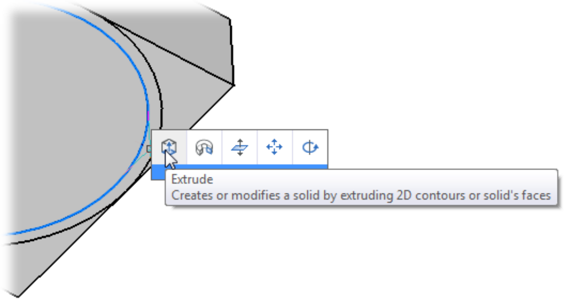
1. On the status bar, ensure that **QUAD** is turned on.



2. Move the cursor over the smaller circle. Notice that it turns blue, and that a second cursor appears. This is the Quad cursor, sporting a single icon (for now).



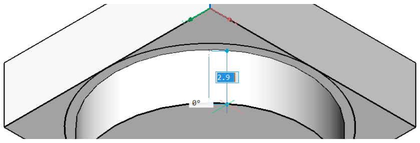
3. The icon on the cursor indicates the Extrude command. To confirm, pass the cursor over the icon. Notice that the Quad cursor expands to show more commands; after a moment, a tooltip appears, explaining the name and purpose of the button.



Extrude
Creates or modifies a solid by extruding 2D contours or solid's faces

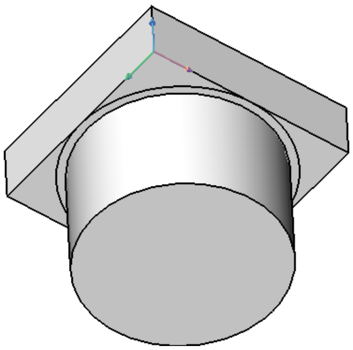
4. Click the **Extrude** button, and then move the cursor. As you do, notice that the circle extrudes into a cylinder (i.e., thickens in 3D). The direction you move the cursor determines the direction the circle extrudes, upwards or down.

5. Notice the *dynamic dimension* next to the cylinder. It consists of a pair of arrowheads and text with blue background. It allows you to enter precise distances; in this case, however, you don't care about the *precise* height of this cylinder, because its only purpose is to be cut out of the larger one to form a hole. But it is useful to ensure the cylinder ends up longer than 5mm.



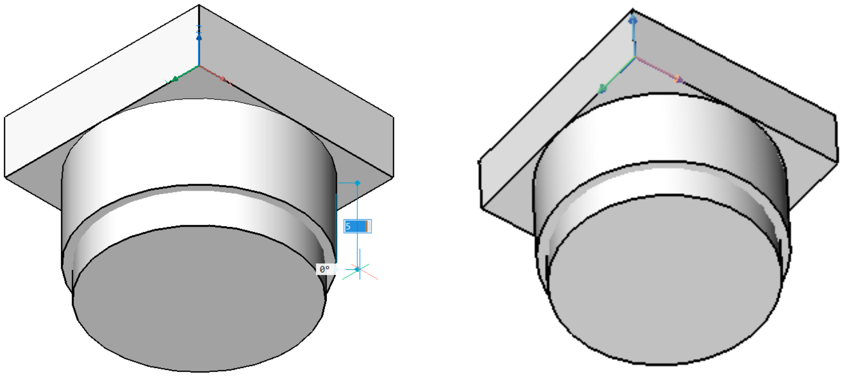
Watching the dynamic dimension, drag the extrusion so that it is longer than 5mm — like 7 or 8mm.

6. Then click. The cylinder is formed — without entering commands.



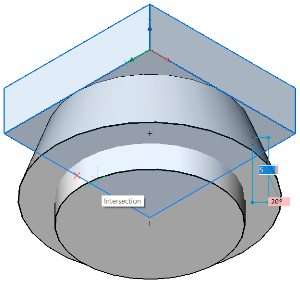
7. Repeat the direct modeling procedure for the larger circle. This time, however, enter **5** in the direct dimension.

TIP If you have difficulty selecting the circle, hover the cursor over a part of the circle, and the press **Tab** until the Quad reports "Circle."



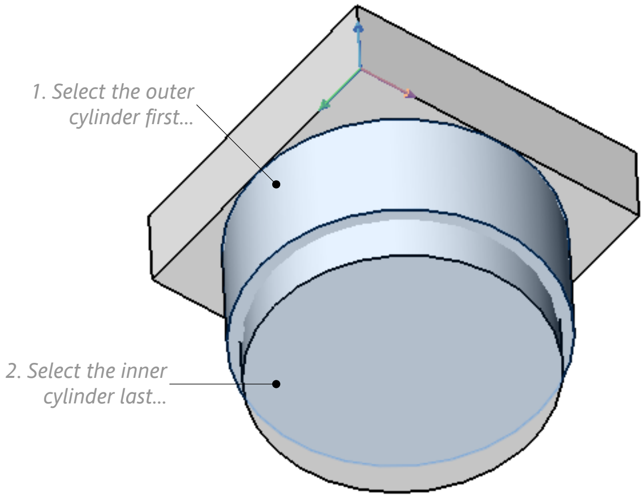
Left: *Extruding the outer circle into a 5mm cylinder;* ***right:*** *Two cylinders in place*

TIP There is also a dynamic angle, shown as 0 in the figures above. Its purpose is to extrude entities at an angle; press **Tab** to edit the angle value.



Subtracting Solids To Make Holes

You learned about Boolean operations in a previous lesson, where it was applied to 2D regions. The same operations can be applied to 3D solids: union, subtract, and intersect. In this tutorial, you create a hole by subtracting one cylinder from another.


1. Select the outer cylinder first...
2. Select the inner cylinder last...

TIP The **Subtract** command is sensitive to the order in which entities are selected. Make sure you pick the correct ones in the correct order: first the one that stays, and then the one that will be removed (subtracted).

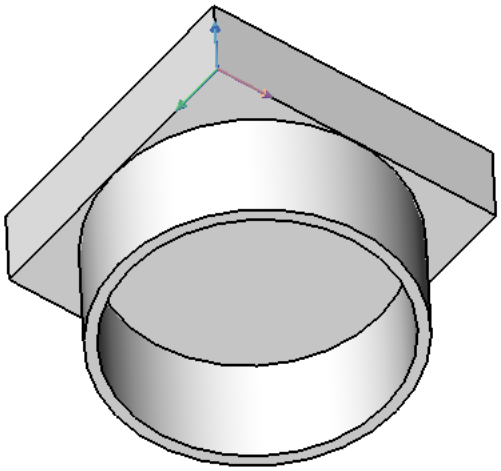
1. With the two cylinders looking somewhat like a wedding cake, it is time to subtract the smaller one from the larger one. Start the **Subtract** command, and then choose the cylinders in correct order:

```
 : subtract
Select ACIS entity to subtract from: (Select the fatter, shorter cylinder)
Select ACIS entity to subtract from: (Press Enter to end entity selection)
```

2. Now pick the cylinder to be removed (erased):

```
Select ACIS entities to subtract: (Select the smaller, longer cylinder)
Select ACIS entities to subtract: (Press Enter to end the command)
```

After you press **Enter**, the Subtract command ends by hollowing out the cylinder.

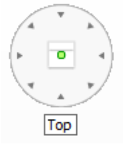


PushPull Modeling

A hole needs to be punched through the thin wall that's between the cylinder and the box. While developing this tutorial, I tried a number of approaches to make the opening, and found the only good way was to again draw a circle and then use the PushPull function to turn it into a hole.

1. The easiest way to draw a circle (and other 2D entities) is in plan view. The quick way to switch between static viewpoints — such as the plan and isometric viewpoints — is to use the Look From widget. To use it, follow these steps:

   a. Click in the center of the widget. Notice that the viewpoint changes.



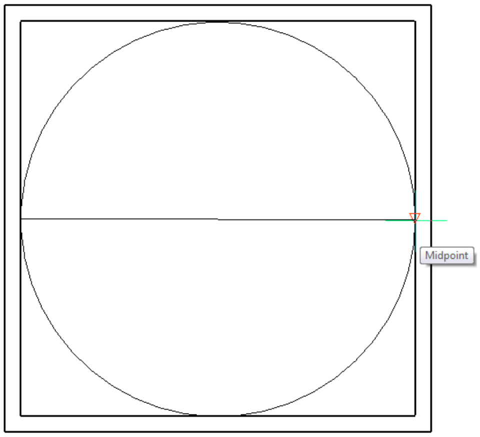
   b. You may need to do a **Zoom Extents** to see the entire model.

TIP If the Look From widget is turned off, you can turn it on with the **LookFrom** command:

```
: lookfrom
LookFrom [ON/OFF/Settings] <ON>: on
```

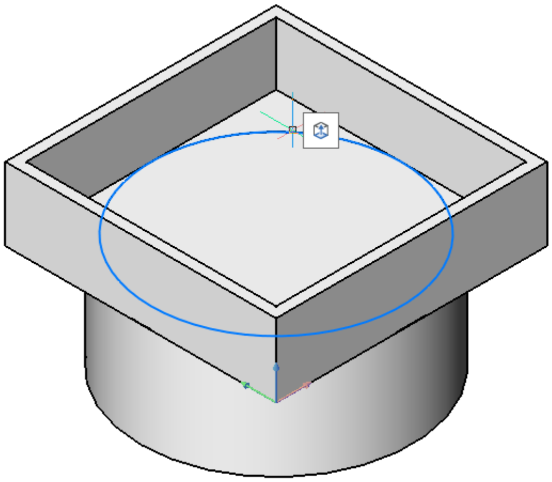
2. To draw the circle, use the same technique as before: employ the **Circle** command with the **2P** option and **MIDpoint** esnaps:

```
: c
Select center of circle or [2 Point(2P)/3 Point(3P)/Tangent-Tangent-Radius(TTR)/
Turn arc into circle(A)/Multiple circles]: 2p
First point on diameter: mid
Snap to midpoint of: (Pick one edge)
Second point on diameter: mid
Snap to midpoint of: (Pick the opposite edge)
```

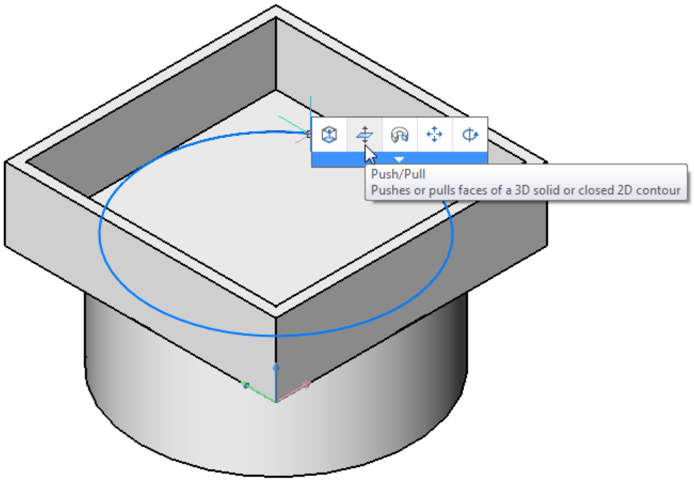


3. Using the Quad cursor, punch out the circle using direct modeling. You may find it easier to control the extrusion by clicking the **Top Front Left** position on the Look From widget.

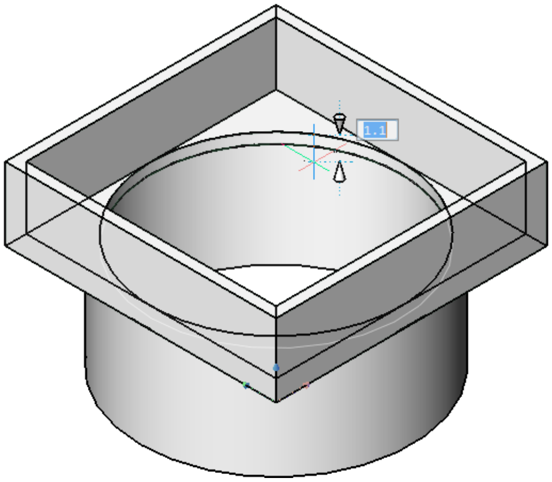
a. Pass the cursor over the newly added circle. Notice that it turns blue.



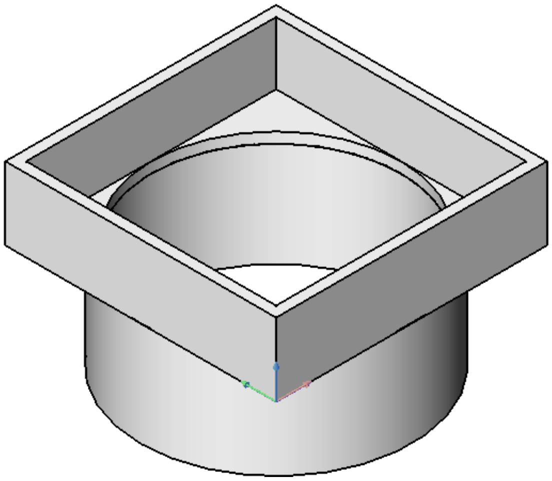
b. Move the crosshair cursor into the Quad cursor. Notice that it expands to show more commands. Choose the **PushPull** button.



c. Drag the cursor down so that a hole appears. The distance you drag does not matter; all you need is for the hole to appear.



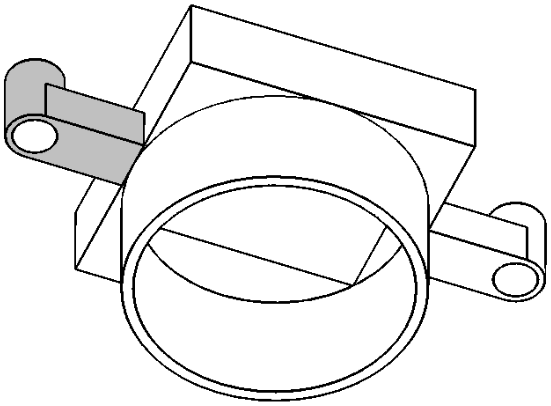
d. Click to finish the hole. This was quicker and easier than using the Extrude command!



Aligning the UCS

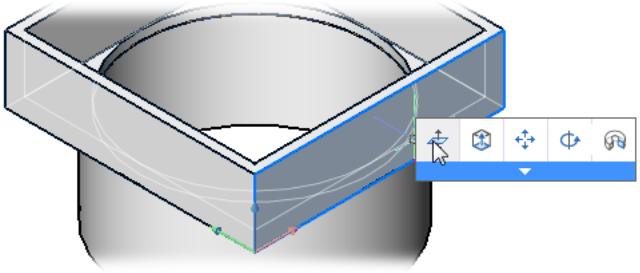
```
Command    UCS, Face option
```

The next step is to draw the rectangle needed for the arms that hold the two screw holes, one of which is shown in gray in the figure below. This time you use PushPull to create an entity, instead of a hole.

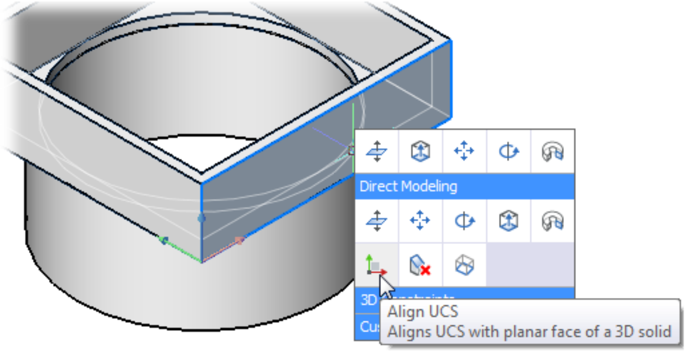


In this case, the 2D entity is a rectangle. Before drawing the rectangle, however, you align the UCS on the face by another method: align UCS. This permanently reorients the UCS plane to a face — permanent until you change it again. (This is different from the temporary, dynamic UCS alignment you used in the earlier part of this tutorial.)

1. Ensure **QUAD** is turned on in the status bar, and then move the crosshair cursor over a rectangular face. Notice that it is highlighted in blue, and that the Quad cursor appears.
2. Move the crosshair into the Quad. The crosshair turns into an arrow cursor.

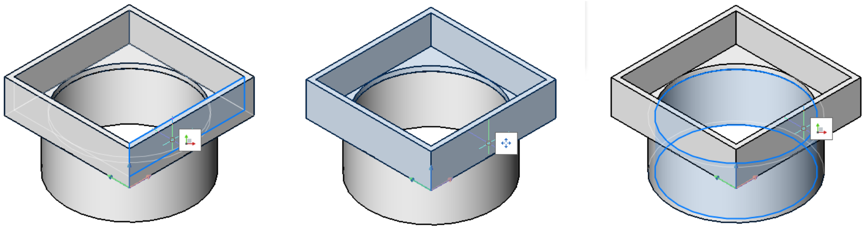


3. Move the arrow cursor into the blue Direct Modeling title bar. Notice that the Quad cursor expands to show more icons.



4. Choose the **Align UCS** button, and then press **Enter**. The UCS plane is now fixed on the face.

TIP If BricsCAD does not highlight the feature in which you are interested, then press the **Tab** key. Each time you press Tab, a different feature laying under the cursor is highlighted.
Press Tab enough times, and the selection cycles around to the first one highlighted. See figure below.



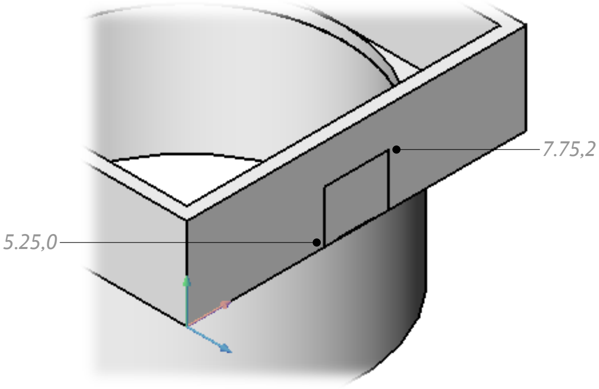
Pulling, Instead of Pushing

Command	dmPushPull
Menu	Model | Direct Modeling | Push/Pull
Ribbon	Model | Edit | Push/Pull
Toolbar	Direct Modeling | Push/Pull

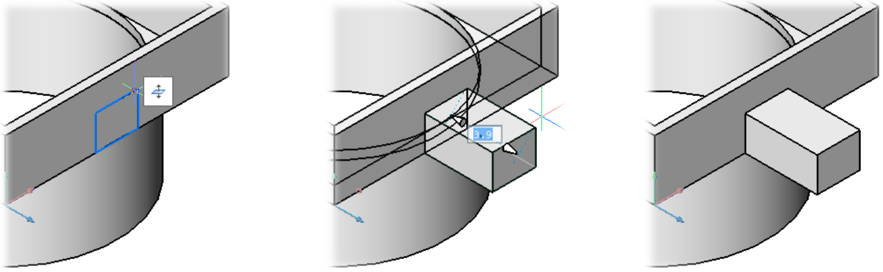
Here you will draw a rectangle on a face, then pull it out with the direct PushPull operation. Later, when it comes to the screw holes I want you to make a mistake so that I can show you that BricsCAD also performs direct *editing* — in this case, changing the diameter of a 3D hole without using a command.

1. The arm has a rectangular cross-section of **2.5mm** wide (length) by **2.0mm** high (width). Draw its profile with the **Rectang** command. I'll give you the dimensions:

```
: rectang
Choose first corner of rectangle or [Chamfer/Elevation/Fillet/Rotated/Square/Thickness/
Width/Area/Dimensions] 5.25,0
Other corner of rectangle: 7.75,2
```

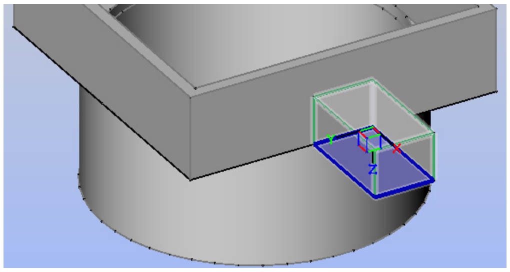


2. Use the **dmExtrude** command to pull the profile to a length of **4.5mm**.



Left to right: *Choose the rectangle (a.k.a "profile");*
start the dmExtrude command and specify extrusion distance of 4.5mm;
done!

3. The screw hole hangs on the wing. It is made of a pair of cylinders, which (again) are made from a pair circles. First, though, align the UCS to the bottom face of the wing, as illustrated below:



Remember to press **Tab**, if you find you have difficulty selecting the bottom face.

4. Drawing 2D entities in 3D can be tricky when the viewpoint is the plan view. By default, Bricsys will snap to the nearest geometric feature; in 3D, this can be the one you don't want too easily. To solve the problem, turn on the **OsnapZ** system variable:

```
: osnapz
New current value for OSNAPZ (Off or On) <Off>: on
```

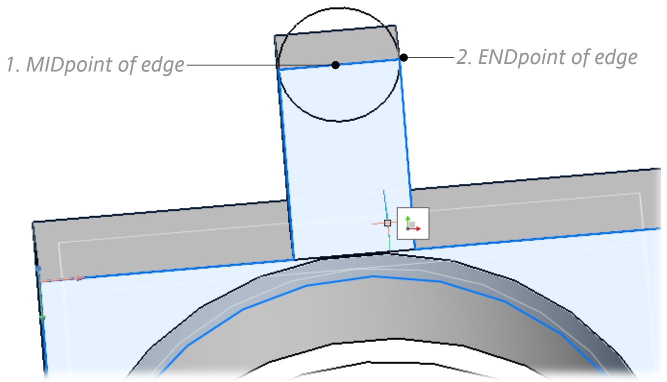
When on, all entity snaps set z = 0, so that the snapping takes place on the x,y-plane — and not at some other higher or lower location.

OsnapZ	Meaning
Off	Snaps to the nearest z coordinate
On	Snaps to the elevation setting (usually 0; z = elevation)

5. Using the LookFrom widget, change the viewpoint to the top (or plan) view.

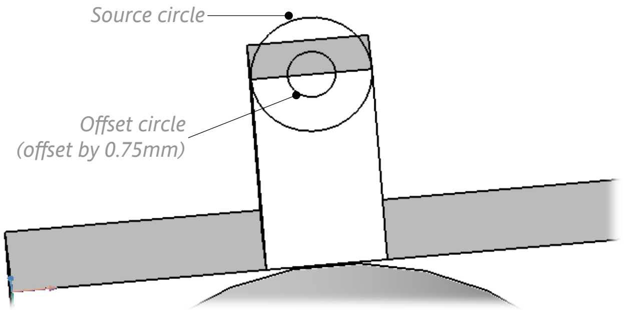
6. Use the **Circle** command to draw the round profile:

```
: circle
Select center of circle or [2 Point(2P)/3 Point(3P)/Tangent-Tangent-Radius(TTR)/Turn arc
into circle(A)/Multiple circles]: mid
Snap to midpoint of: (Pick the midpoint of the edge.)
Set Diameter or [Radius] <1.3>: end
Snap to endpoint of: (Pick the end of the edge)
```

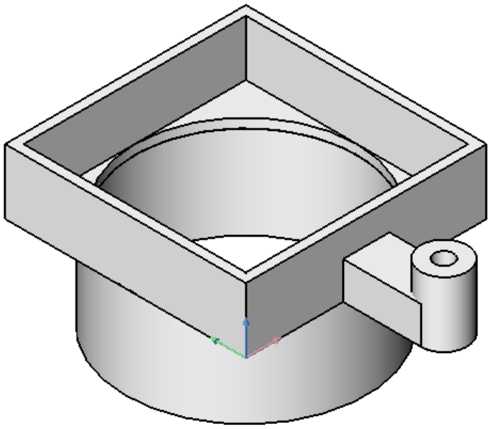


When you rotate the view back to isometric, you see the circle placed on the bottom face, thanks to OsnapZ.

7. Use the **Offset** command to place a second circle, offset by **0.75mm**. (This value is deliberately incorrect, so that you can correct it later using direct editing.)

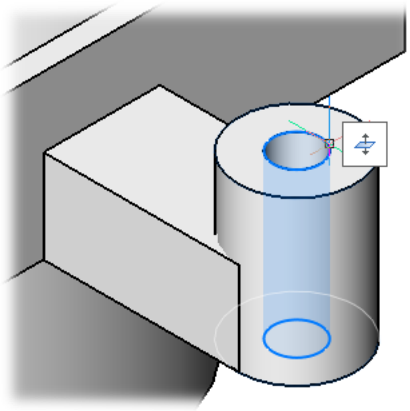


8. As in the earlier tutorials, use direct modeling to extrude the two circles into cylinders:
   a. Use Extrude to turn the outer circle in a cylinder with a height of precisely 3.5mm.
   b. Use Extrude again to turn the inner one to any height taller than 3.5mm; notice that it creates the hole automatically — no need to use the Subtract command!

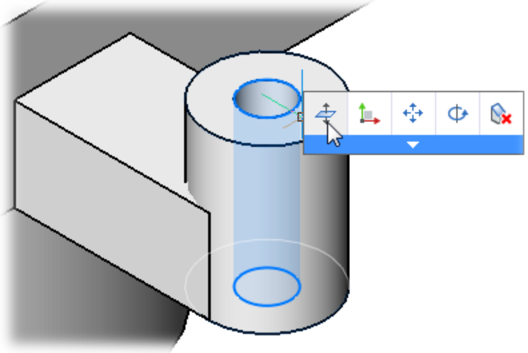


9. The diameter of the hole is too large. It should be 1mm, but is currently 0.5mm. BricsCAD can edit solid models using direct editing. Here is how to correct the diameter of the hole:

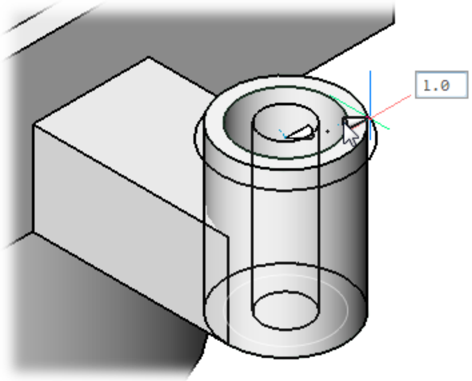
   a. Move the cursor over the edge of the hole. If the hole does not highlight in blue, then press the **Tab** key until it does.



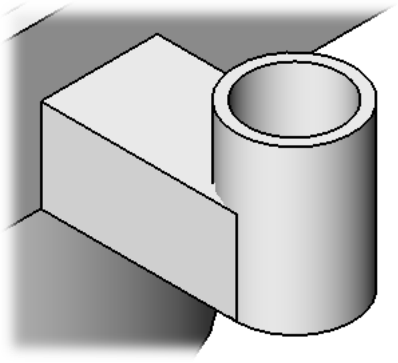
   b. In the Quad cursor, choose the **dmExtrude** button.



   c. The hole must be changed to a diameter of 1.0mm. Drag the hole dynamically, or enter 1.0 in the dynamic dimension.



The hole is the correct size. Press **Ctrl+S** to save your work.



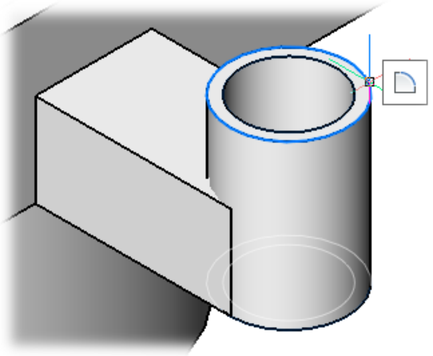
TIP To change the UCS back to “normal,” enter the **UCS** command, and then choose the **World** option:

```
: ucs
Specify origin of UCS or [Face/NAmed/Entity/Previous/View/X/Y/Z/ZAxis/Move/World]<World>: w
```

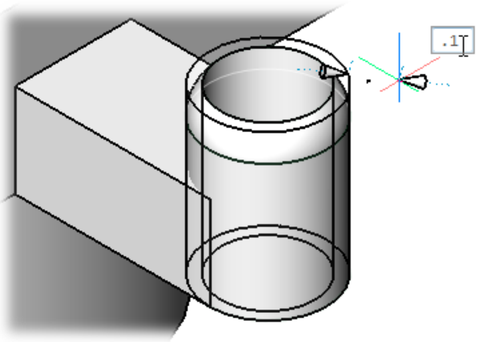
Filleting 3D Solids

The outer edge of the screw hole has a small fillet around the edge. With direct editing, you can apply fillets interactively. Here’s how:

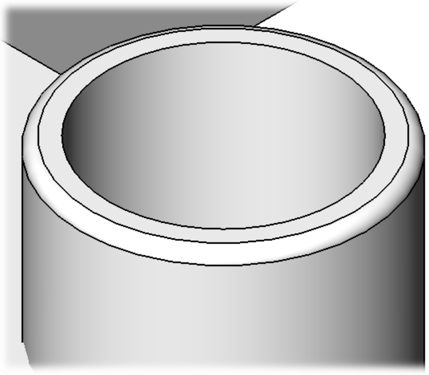
1. Move the cursor over the edge of the cylinder. Remember: if you cannot select the outer edge, then press **Tab** until BricsCAD highlights it. See the figure below.



2. Choose the **Fillet** button on the Quad cursor, and then enter **0.1** as the fillet radius.

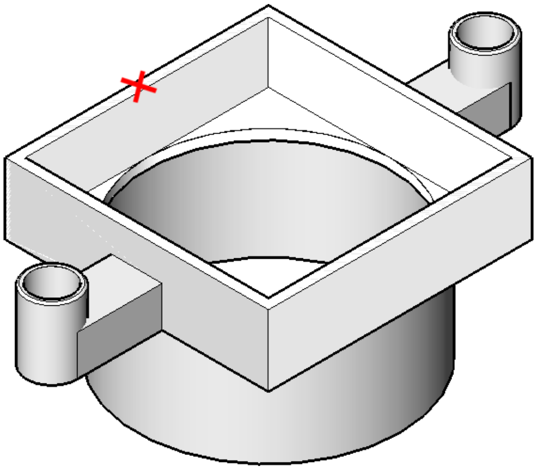


...and then press **Enter**.



3. Mirror the two new parts to place them on the other side, like this:

```
: mirror
Select entities to mirror: (Select the wing part and screw hole)
Select entities to mirror: (Press Enter to continue)
Start of mirror line: mid
Snap to midpoint of: (Pick middle of rectangular part, as illustrated below)
End of mirror line: mid
Snap to midpoint of: (Pick a point on the other side)
Delete the original entities? [Yes-delete entities/No-keep entities] <No-Keep entities>: n
```



Joining Parts with Union

The 3D model consists of six solids. You can leave them as individuals, or join them into a single body with the Union command.

1. Use the **Union** command to merge all solids into one:

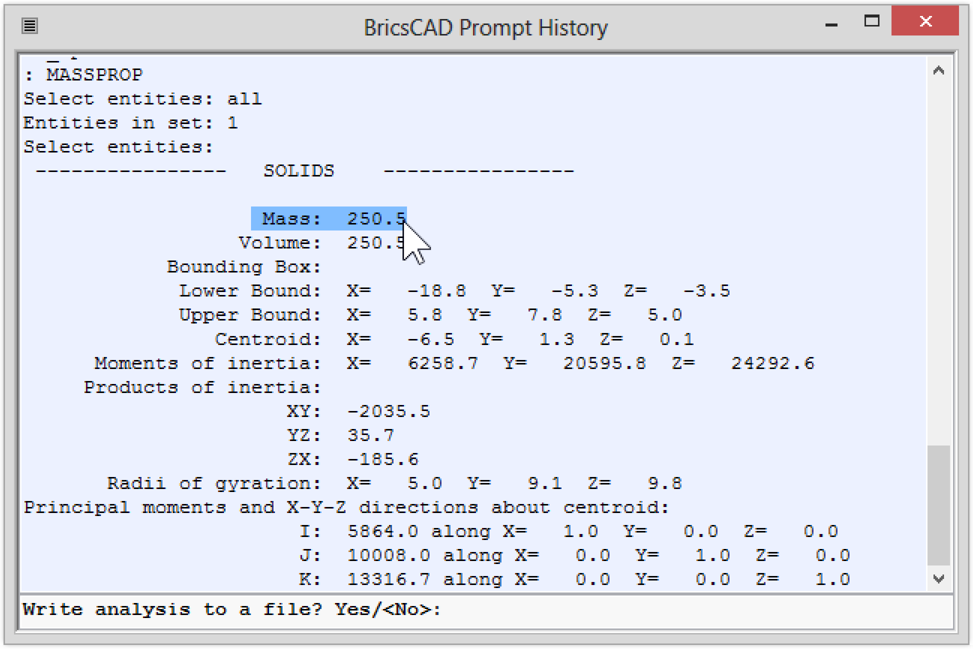
```
: union
Select ACIS entities to union: all
Select ACIS entities to union: (Press Enter to end the command)
```

TIP You can continue to directly edit the model, even after all its parts are joined by the **Union** command.

2. Now that the model is one unified part, you can apply commands like **MassProp** and **Properties** to find out the properties of this focusing bracket.

```
: massprop
Select entities: all
Select entities: (Press Enter to continue)
Write analysis to a file? Yes/<No>: y
```

The report you see in the text window can be saved to a *.mpr* (mass properties report) file.



```
: MASSPROP
Select entities: all
Entities in set: 1
Select entities:
 ----------------   SOLIDS    ----------------

                    Mass:  250.5
                  Volume:  250.5
            Bounding Box:
             Lower Bound:  X=   -18.8  Y=   -5.3  Z=   -3.5
             Upper Bound:  X=   5.8  Y=   7.8  Z=   5.0
                Centroid:  X=   -6.5  Y=   1.3  Z=   0.1
      Moments of inertia:  X=   6258.7  Y=   20595.8  Z=   24292.6
     Products of inertia:
                     XY:  -2035.5
                     YZ:  35.7
                     ZX:  -185.6
       Radii of gyration:  X=   5.0  Y=   9.1  Z=   9.8
Principal moments and X-Y-Z directions about centroid:
                      I:  5864.0 along X=   1.0  Y=   0.0  Z=   0.0
                      J:  10008.0 along X=   0.0  Y=   1.0  Z=   0.0
                      K:  13316.7 along X=   0.0  Y=   0.0  Z=   1.0
Write analysis to a file? Yes/<No>:
```

3. Save the drawing.

Generating 2D Drawings

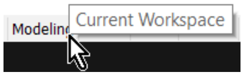
Command	`ViewBase`
Menu	`Model | Direct Modeling | Drawing Views | Generate Drawing Views`
Ribbon	*`(Mechanical workspace)`* `Annotate | Views | Base Views`
Toolbar	`Drawings Views | Base View`

With the 3D model completed, it is time to generate *engineering drawings* from it. These are 2D plans that are used to check dimensions and give instructions to manufacturers. BricsCAD generates these drawings semi-automatically: all you need to do is pick the spots where you want the views placed. BricsCAD figures out the orientations of the views, and then generates the drawings.

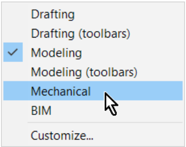
Let's see how it works.

1. The Drawings Views panel needed for this tutorial is not found in the Modeling workspace, so you need to switch to the Mechanical workspace:

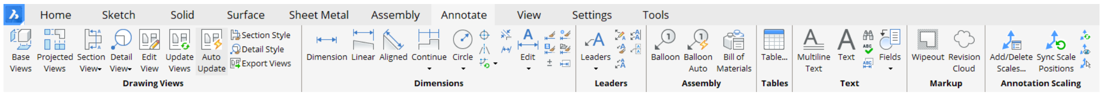
   a. On the status bar, right-click **Modeling**.



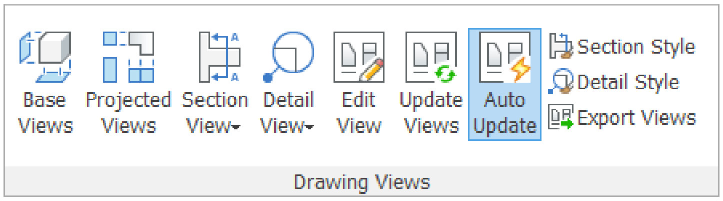
   b. From the shortcut menu, choose **Mechanical**.



   c. Wait a moment as BricsCAD changes the user interface. Then, on the ribbon, click the **Annotate** tab.



   Focus on the **Drawing Views** panel, as it contains the commands you need for this tutorial.



2. Click the **Base Views** button. (It runs the ViewBase command). This command semi-automatically generates 2D views of the 3D model — front, side, top, isometric, etc — in a new layout.

```
: _viewbase
Preset: "None",  View scale: "Adapt to paper size"
```

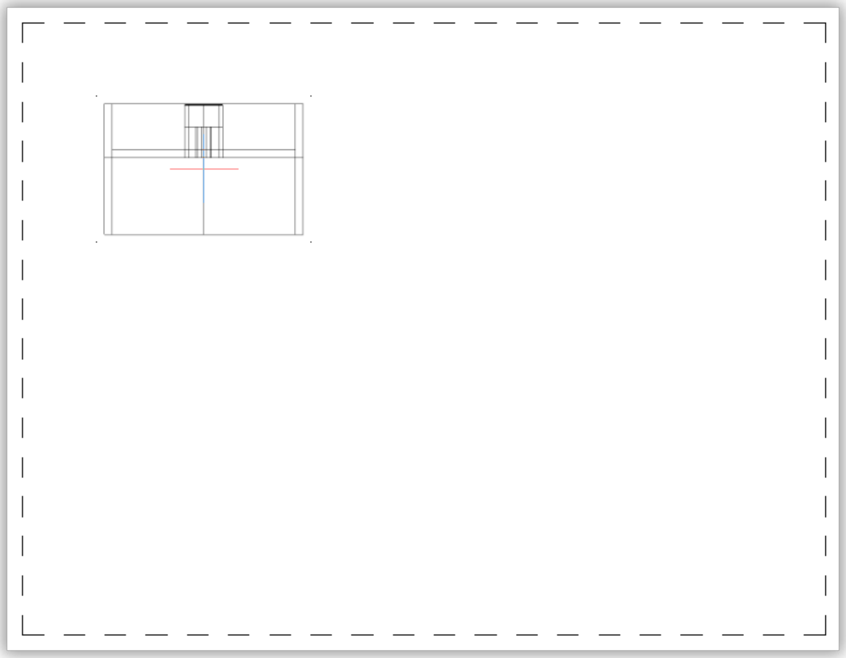
3. Press **Enter** to make drawings from the entire model:

```
Select objects or [Entire model/preseTs] <Entire model>: (Press Enter)
```

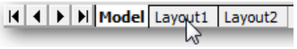
4. At the next prompt, press **Enter** to use Layout1 for the drawings:

```
Enter new or existing layout name to make current <Layout1>: (Press Enter)
```

5. Notice that BricsCAD switches to Layout1 automatically, and then previews the 2D drawing being generated from the 3D model.



(If it does not switch automatically, then click the **Layout1** tab at the bottom of the drawing area.)

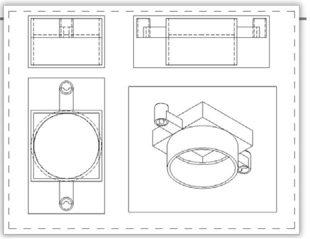


As you move the cursor, the 2D preview moves and changes its projection to match the location. In this step of the tutorial, position the cursor in the upper left quadrant, and then click. See figure above.

```
Select position for base view [Scale/Hidden lines/Tangent edges/Orientation/Projection
type/Isometric style/sElect] <Cancel>: (Move the cursor to upper left, and then click)
```

The quadrant in which you click determines the viewpoint generated by BricsCAD automatically:

Quadrant	2D View Generated
Upper left	Top view
Upper right	Side view
Lower left	Front view
Lower right	Isometric view



5. The command repeats its prompt. Repeat the process: move the cursor to a quadrant of the layout, and then click to position the view.

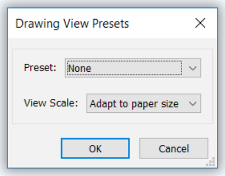
```
Select position for projected view [Done] <Done> (Move the cursor to lower left
quadrant, and then click to place the front view)

Select position for projected view [Done] <Done> (Move the cursor to upper right
quadrant, and then click to place the side view)
```

QUICK SUMMARY OF VIEWBASE OPTIONS

```
: viewbase
Select objects or [Entire model/preseTs/Special view] <Entire model>:
Enter new or existing layout name to make current <LayoutName>:
Select position for base view [Scale/Hidden lines/Tangent edges/Orientation/Projec-
tion type/view Quality/Isometric style/sElect]<Cancel>:
Select position for projected view [Done] <Done>:
```

PRESETS OPTION

Displays the Drawing View Presets dialog box:



Drawing View Presets

Preset: None

View Scale: Adapt to paper size

Preset droplist offers these options:

- **None**
- **Architectural Views** — generates Front, Right, Back, Left (elevations) and Top views; three section views (two horizontal section plans); and one vertical section.
- **Mechanical Views** — generates standard Front, Top and Left (first-angle projection) or Right (third-angle projection) views. The projection type: first or third angle. The projection type held by the ProjectionType variable.

View Scale droplist offers automatic scaling or one of the scales presided over by the ScaleListEdit command.

SPECIAL VIEW OPTION

Offers to make the view exploded or not exploded (default).

SCALE OPTION

Sets the scale of all views to fit the following arrangements:

- **fit 4 views** — adjusts scale to fit four orthographic views: Front, Top, Left, and Right. The Front view (base view) is defined by the Orientation option.
- **9 views** — adjusts scale to fit five orthographic views and four isometric views.
- **5 views** — adjusts scale to fit five orthographic views: Front (*), Top, Left, Right, and Back.
- **10 views** — adjusts scale to fit six orthographic views and four isometric views.
- **Standard scales** — displays the scales lorded over by the ScaleListEdit command.
- **Custom** — prompts for any scale factor.

HIDDEN LINES OPTION

Toggles the representation of hidden lines.

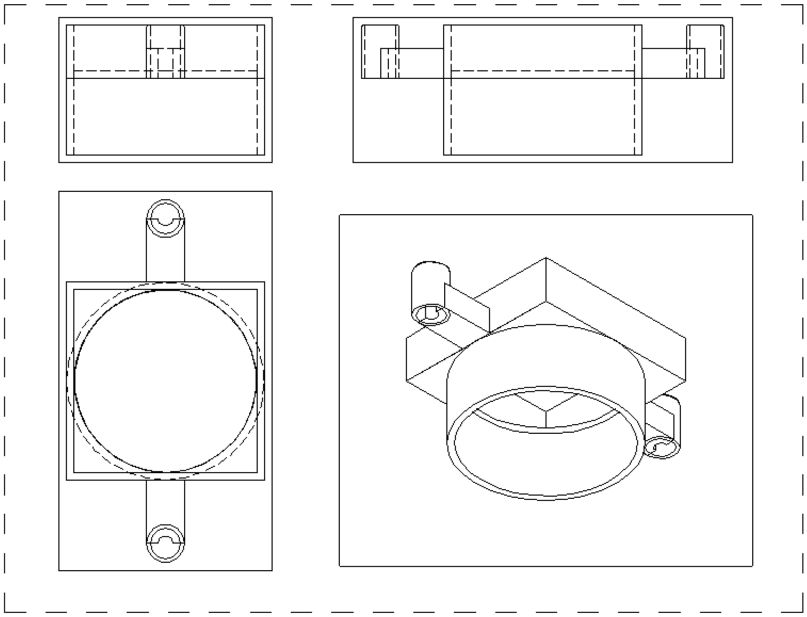
TANGENT EDGES OPTION

Determines whether tangent edges between tangent faces are displayed or not; tangent edges are always displayed in isometric views:

Continued...

```
Select position for projected view [Done] <Done> (Move cursor to lower right quadrant,
and then click to place the isometric view)
```

The result should look like the figure below. BricsCAD automatically adjust the scale factor so that the resulting viewports nicely fit the page.



Continued...

HIDDEN LINES OPTION

Determines whether hidden lines are shown.

ORIENTATION OPTION

After you specifies the main view, BricsCAD rotates the 3D model so that the main view is projected on the vertical projection plane. Choose from Front, Back, left, right, top, or bottom views.

PROJECTION TYPE OPTION

Determines the projection type: first angle (or European projection) or third angle (American projection).

VIEW QUALITY

Toggles the quality of the projected view between low (faster to display) and high (default)

ISOMETRIC STYLE OPTION

View style for isometric views: rendered using Conceptual visual style, wireframe, or any other visual style.

SELECT OPTION

Adds and removes model space elements to and from the projected view.

QUICK SUMMARY OF VIEWSECTION OPTIONS

```
: viewsection
Select drawing view:
Specify start point of section line or select type [Full/Half/Offset/Aligned] <Full>:
Specify next point of section line [Done] <Done>:
Select position for section view:
Select option [Scale/Hidden lines/Tangent lines/anChor/Geometry/Annotation/Depth/
Projection/Rotate view] <Cancel>:
```

TYPE OPTION

- **Full** — draws a section of the entire viewport
- **Half** — draws a section of half the viewport
- **Offset** — draws a section specified by points that you pick
- **Aligned** — draws a section per perpendicular to the section line

SCALE OPTION

Sets the scale of the section view; choose from the standard scale factors,specify a custom scale, or inherent the scale of the parent view:

GEOMETRY OPTION

Defines the visual style for the section view: conceptual 3D view or 2dWireframe visual style; alternately, select a visual style from the Properties panel. (Hidden Line option is moved to the ViewBase command.)

TANGENT LINES OPTION

Toggles display of tangent lines, on or off.

ANCHOR OPTION

Determines what happens to the section view when it is updated after the 3D model changes:

- **Yes** — anchors the center of the viewport so that the viewport grows and shrinks around that point.
- **No** — fixes the position of the geometry; the **AutoVpFitting** variable specifies whether the viewport size is adjusted to fit the extents of the 3D model automatically (default = on).

ISOMETRIC STYLE OPTION

Determines style for isometric views: rendered using Conceptual visual style, wireframe, or another one.

ANNOTATION OPTION

Prompts you to override that automatic annotations of section view:

- **Identifier** — specifies the view identifier, such as A.
- **Label** — toggles the display of the view label.

DEPTH OPTION

Specifies clipping of the section view, full or limited. You can define the depth with your mouse.

PROJECTION OPTION

Toggles between normal and orthographic projection of the view.

ROTATE VIEW OPTION

Rotates section viewports to be horizontal, vertical, or at a custom angle.

6. When you are finished placing view, press **Enter** to end the command. Don't press Esc, because all your carefully placed views are removed!

```
Select position for projected view [Done] <Done>: (Press Enter)
```

ADDING SECTION VIEWS AND DETAILED VIEWS

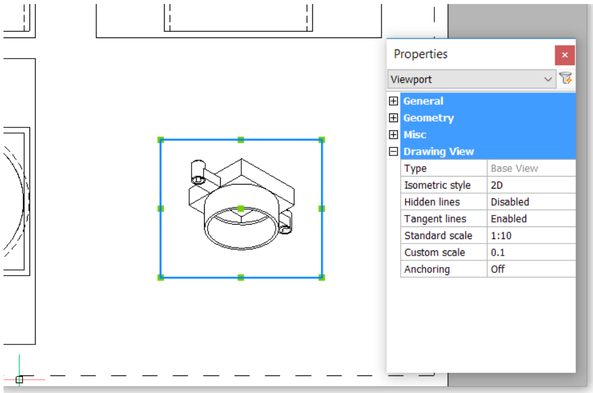
Command	ViewSection
Menu	Model | Drawing Views | Section Types
Ribbon	*(Mechanical workspace)* Annotate | Drawing Views | Section View
Toolbar	Drawing Views | Section Types
Command	ViewDetail
Menu	Model | Drawing Views | Detail View Types
Ribbon	*(Mechanical workspace)* Annotate | Drawing Views | Detail View
Toolbar	Drawing Views | Detail View

With the 2D drawings in place, it is common to add section views that show the insides of models, which use hatch patterns to indicate the materials with which they should be manufactured. When necessary, greatly enlarged detail views are also added to show complex areas.

Resizing Views

Before you can add a cross-section view, you have to adjust the existing views, because the page is full — no room for another view! One solution is to make the existing views smaller. Here is how to do this:

1. Select a viewport border.
2. In the Properties panel, click **Custom Scale**.


Properties

Viewport

- General
- Geometry
- Misc
- Drawing View

Type	Base View
Isometric style	2D
Hidden lines	Disabled
Tangent lines	Enabled
Standard scale	1:10
Custom scale	0.1
Anchoring	Off

3. Enter a new value, such as **.1**.
4. Notice that the viewport is smaller. Move the viewport by dragging it (with its center grip) or through the Move command.

Another options is to erase unwanted views. To erase a view, select the viewport border, and then press the **Del** key on the keyboard.

QUICK SUMMARY OF VIEWDETAIL OPTIONS

```
: viewdetail
Select drawing view:
Specify detail center or select boundary type [Circular/Rectangular] <Circular>:
Select radius of detail view:
Select position for detail view [Scale] <Cancel>:
Select option [Scale/Hidden lines/Tangent lines/anChor/Annotation/Boundary/model
Edge] <Cancel>:
```

CIRCULAR OPTION

Creates a circular boundary, prompting you for the radius.

RECTANGULAR OPTION

Creates a rectangular boundary. You can still change the boundary type with the **Boundary** option in the last prompt of this command.

SCALE OPTION

Sets the scale of the detail view; choose from the standard scale factors,specify a custom scale, or inherent the scale of the parent view:

HIDDEN LINES OPTION

Toggles hidden line display of the detail view on and off, or inherited from the parent view.

TANGENT LINES OPTION

Toggles display of tangent lines, on or off.

ANCHOR OPTION

Determines what happens to the detail view when it is updated after the 3D model changes:

- **Yes** — anchors the center of the viewport so that the viewport grows and shrinks around that point.
- No — fixes the position of the geometry; the AutoVpFitting variable specifies whether the viewport size is adjusted to fit the extents of the 3D model automatically (default = on).

ANNOTATION OPTION

Prompts you to override automatic annotations of detail views:

- **Identifier** — specifies the view identifier, such as A.
- **Label** — toggles the display of the view label.

BOUNDARY OPTION

Determines if the boundary of the detail view is a circle or a rectangule.

MODEL EDGE OPTION

Specifies how the connection line is drawn between the two views (detail and in the parent view), smooth with the border or with the connection line.

Making Section Views

Okay, with space freed up, you can go ahead to make the section view:

1. From the ribbon's **Annotate > Drawing Views** panel, click the **Section View** button (or else enter the **ViewSection** command).

```
: _viewsection
```

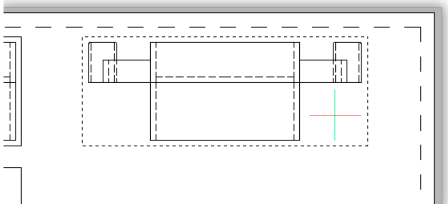
2. BricsCAD prompts you to choose a drawing view. The section view will be generated from the view you pick, so choose carefully!

   a. For this tutorial, move the cursor to the side view, which is in the upper left quadrant (see figure below).

   b. Click inside the viewport.

TIP When selecting the view, don't select the viewport's rectangle, as BricsCAD will just repeat the prompt. Click *inside* the viewport, as shown by the crosshair cursor below.

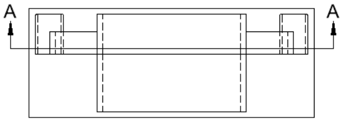
```
Select drawing view: (Pick inside a viewport)
```



Notice that BricsCAD highlights the selected viewport with dashed lines.

3. With the drawing view selected, it's time to show BricsCAD the drawing that you want to section. At the next prompt, pick two points outside the viewport, shown by the arrowheads in the figure below:

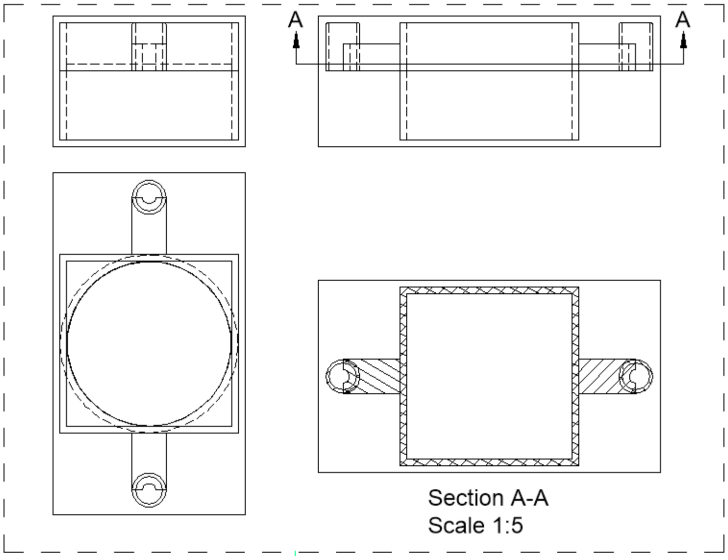
```
Specify start point of section line or select type [Full/Half/Offset/Aligned] <Full>:
(Pick a point outside the viewport)

Specify next point of section line [Done] <Done>: (Pick a second point on the other
side of the viewport)
```



TIP I recommend holding down the **Shift** key to enforce ortho mode. The result is a section line that's precisely horizontal or vertical.

4. The final step is to position the section view:

```
Select position for section view: (Pick a point in the drawing)
```

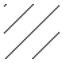
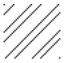
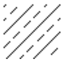
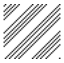
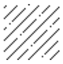
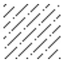
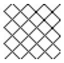
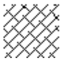


BricsCAD automatically labels the section using the standard notation of "A-A." Engineers use the A-A to see what is referenced elsewhere in the drawing. The label also indicates the scale factor of the section, 1:5 — also automatically determined by BricscAD.

Hatching is applied automatically to the cross section. Hatching is used by drafters to describe two types of information:

- The **presence** of hatching shows that areas of the are solid; areas without hatching are empty air.
- The **style** of hatching indicates the type of material; "ANSI31" hatching is the default pattern, and represents iron.

Here is the meaning of the ANSI hatch patterns included with BricsCAD:

Pattern Name	Example	Pattern Meaning
ANSI31		Iron, general purpose hatch
ANSI32		Steel
ANSI33		Bronze, brass, copper, composites
ANSI34		Rubber, plastic, electrical insulation
ANSI35		Defined by the legend
ANSI36		Defined by the legend
ANSI37		White metal, zinc, lead, babbit, and alloys of them
ANSI38		Magnesium, aluminum, and alloys of them

Creating Detail Views

A detail view is an enlargement of one part of a drawing. It makes it easier to see what is going on with complex parts. Details are placed at a larger scale factor by the **ViewDetail** command.

Here you make a detail of one of the small round parts:

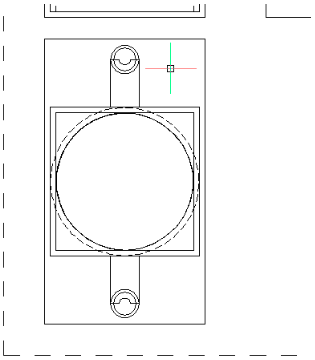
1. Enter the enter the **ViewDetail** command:

   ```
   : viewdetail
   ```

2. BricsCAD prompts you to choose a drawing view:

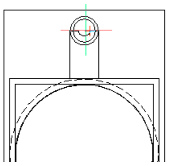
   Select drawing view: *(Pick inside a viewport)*

   This is the drawing view from which the detail will be taken. For this tutorial, click inside the lower left quadrant's viewport, as shown below.



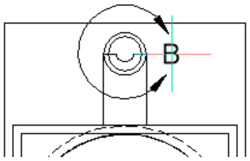
3. Pick the center of the detail, as BricsCAD by default uses a circle as the view's boundary:

   Specify detail center or select boundary type [Circular/Rectangular] <Circular>:*(Pick the center of what will be detailed)*

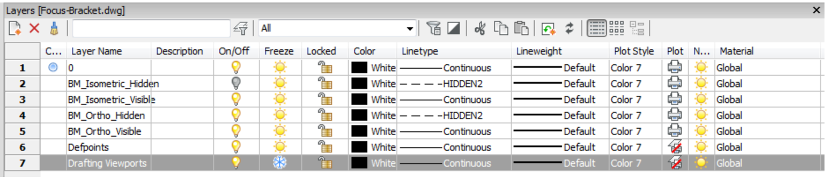


4. Now size the circle to indicate the extent of the detail. You can see the circle in the figure below, it already has the "B" reference attached.

   Select radius of detail view: *(Drag the circle to size the detail area)*



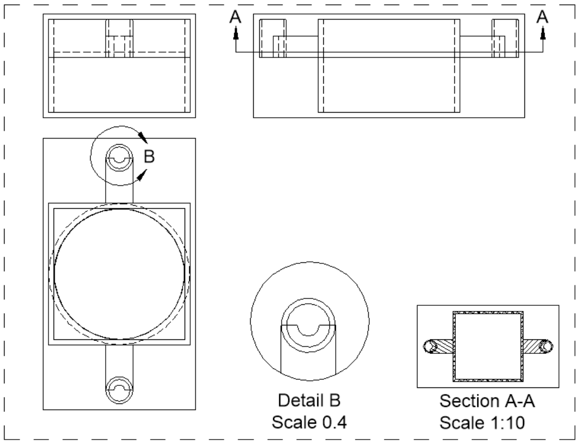
TIPS You can make sections of details, and details of sections, and details of details.
To hide the rectangles that make up the viewports, freeze the "Drafting Viewports" layer. You can use the Layer command (shown below) or the Layers droplist in the Properties panel.



	C...	Layer Name	Description	On/Off	Freeze	Locked	Color	Linetype	Lineweight	Plot Style	Plot	N...	Material
1	●	0		💡	☀	🔓	■ White	——— Continuous	——— Default	Color 7	🖶	☀	Global
2		BM_Isometric_Hidden		💡	☀	🔓	■ White	– – – HIDDEN2	——— Default	Color 7	🖶	☀	Global
3		BM_Isometric_Visible		💡	☀	🔓	■ White	——— Continuous	——— Default	Color 7	🖶	☀	Global
4		BM_Ortho_Hidden		💡	☀	🔓	■ White	– – – HIDDEN2	——— Default	Color 7	🖶	☀	Global
5		BM_Ortho_Visible		💡	☀	🔓	■ White	——— Continuous	——— Default	Color 7	🖶	☀	Global
6		Defpoints		💡	☀	🔓	■ White	——— Continuous	——— Default	Color 7	⊘	☀	Global
7		Drafting Viewports		💡	❄	🔓	■ White	——— Continuous	——— Default	Color 7	⊘	☀	Global

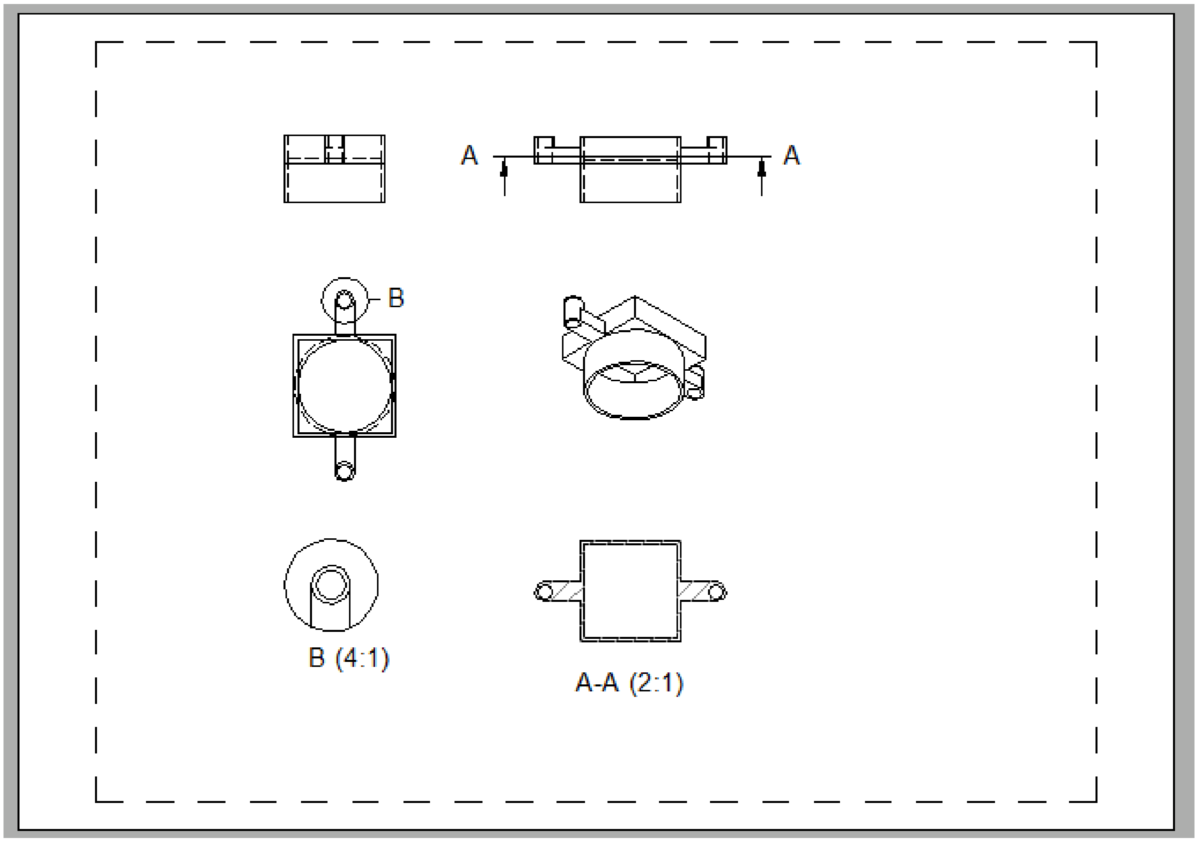
5. Position the detail view somewhere in the drawing:

```
Select position for detail view [Scale] <Cancel>: (Pick a point)
```



BricsCAD automatically labels the detail using the standard method of "B," which is used to referenced the source of the detail drawing. It also indicates the scale factor of the section, 4:1. You can use the Scale option to make the detail larger or smaller.

Here is how the final 2D plan could look. The viewports are turned off and some viewports are moved (with the Move command) and resized (with the Custom Scale property). The next step would be to dimension the parts.



You learned how to construct 3D models using traditional and modern commands for creating and editing bodies. You also saw how to generate 2D drawings from the model, semi-automatically.

Next, you learn how to control the size and positioning of entities through constraints and parameters.

Notes

CHAPTER 11

Dimensional & Geometric Constraints

Here you learn to create drawings that are *constrained.* In these drawings, dimensions determine the sizes of entities — not the other away around, as you learned in an earlier chapter. In addition, constraints can be used to lock geometry into place to locate entities relative to one other. Together with constraints, parameters determine the positions of entities through formulae.

In this chapter, you learn how to apply dimensional and geometric constraints to 2D drawings.

IN THIS CHAPTER

- Applying dimensional constraints
- Using 2D geometric constraints
- Controlling constraints through parameters

KEY TERMS IN THIS CHAPTER

Constraint bar — describes the small tooltips that report the constraints applied to entities

Dimensional constraint — specifies the size of entities in a dimension-like manner

Geometric constraint — determines the geometric relationship between entities, much like a semi-permanent entity snap

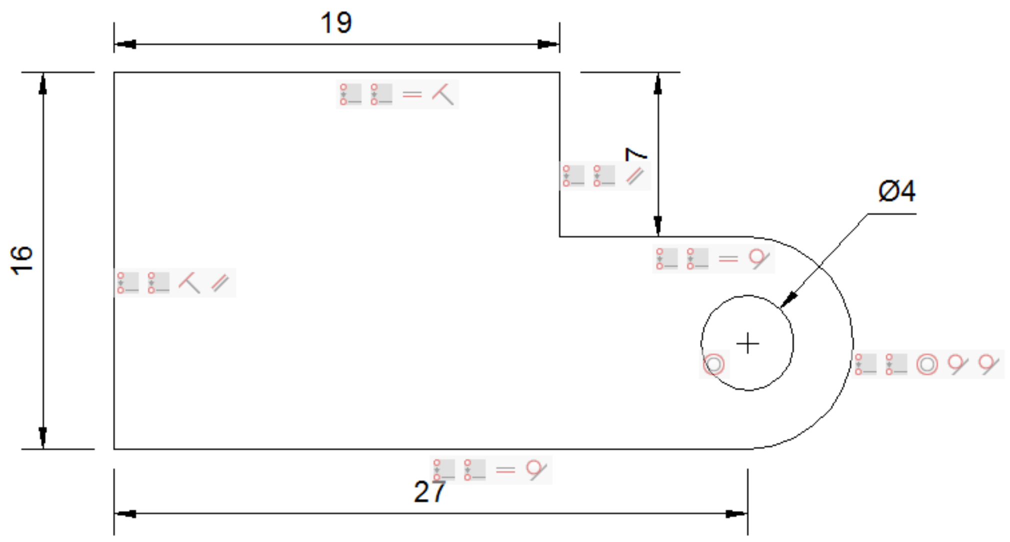
Parameter — specifies a formula that determines relationships between constraints

NEW COMMANDS

Command	Menu Bar	Ribbon
MaterialBrowserOpen	Parametric | Mechanical Browser	...
ConstraintBar	Parametric | 2D Constraints Bar	2D Constraints Bar
DimConstraint	Parametric | 2D Constraints	2D Constraints panel
GeomConstraint	Parametric | 2D Constraints	2D Constraints panel

THIS CHAPTER'S MODEL

By the end of this lesson, your parametric drawing will look like this one:



Drawing file begun as bracket-ragged.dwg and ended as bracket-constrainted.dwg

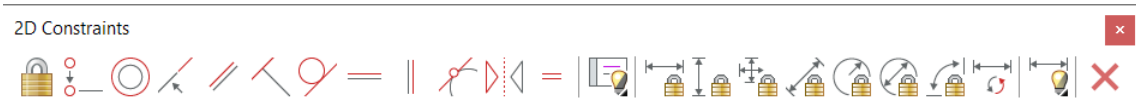
Working with Constraints

BricsCAD allows you to control the size of entities in drawings through constraints. There are two types of constraints: one specifies the *size* of entities, the other locates their *positions*.

- **Dimensional constraints** regulate the sizes of entities, and the distances between them
- **Geometric constraints** determine the position of entities relative to others

BricsCAD Classic and Pro provide 2D constraints. BricsCAD Platinum and Ultimate add 3D constraints, which are applied to faces and edges of 3D bodies; they are not meant for 2D entities. (Three-dimensional constraints are not covered by this book.)

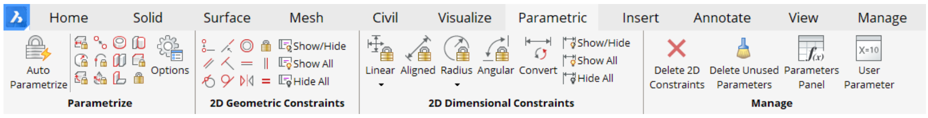
There are a number of commands for applying and removing constraints, but I find it easiest to just use the 2D Constraints toolbar. To see this toolbar, right-click any toolbar or the ribbon, and then choose **BRICSCAD | Toolbars | 2D Constraints**:



Buttons geometric constraints (left half) and for dimensional constraints (right half)

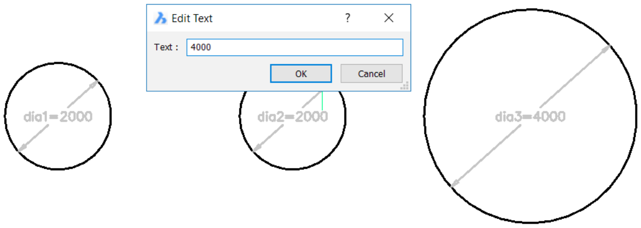
TIP If you find the buttons on the toolbar too small, then right-click the toolbar, and then choose **Toolbar Size | Large Icons**. This trick does not work for the ribbon.

If you prefer the ribbon, then switch to the "Modeling" workspace, and then choose the ribbon's **Parametric** tab:



ABOUT DIMENSIONAL CONSTRAINTS

The great thing about dimensional constraints is when you use them to control the size of entities. For instance, apply a diameter dimensional constraint to a circle. Increase the value of the constraint, and BricsCAD forces the circle to become larger. To edit the value, you just double-click the dimension text.



Text : 4000

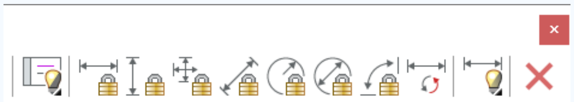
Left: *Circle with diameter of 2000 units;* ***center:*** *Editing the value of the dimensional constraint;* ***right:*** *New 4000-value of parameter forces circle to grow larger*

QUICK SUMMARY OF DIMENSIONAL CONSTRAINTS

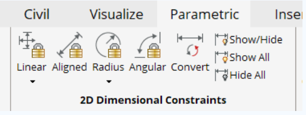
There are two sets of commands for applying constraints. One is the all-purpose **DimConstraint** command, which is useful for seeing a list of all modes, The other set consists of the individual commands, one for each dimensional constraint:

DIMCONSTRAINT COMMAND

```
: dimconstraint
Select associative dimension to convert or [LLinear/Horizontal/Vertical/Aligned/ANgu-
lar/Radial/Diameter] <LLinear>: (Enter an option.)
```



__Above:__ 2D dimensional constraints on right half of the toolbar; __below:__ 2D dimensional constraints on ribbon



DIMENSIONAL CONSTRAINT COMMANDS

The other set of commands consists of ones specific to each constraint mode, as listed in the table below. Notice that the command names all begin with ‘dc’, short for dimensional constraint.

	Type of		
Icon	Constraint	Command	Constraining Action
	LInear	dcLinear	Horizontal or vertical distance
	Horizontal	dcHorizontal	Horizontal (X) distance between two points
	Vertical	dcVertical	Vertical (Y) distance between two points
	Aligned	dcAligned	Distance between two points at any angle
	ANgular	dcAngular	Angle between two lines or linear polyline segments; or angle of an arc or polyline arc; or angle between three points on an entity
	Radial	dcRadius	Radius of a circle, arc, or polyline arc
	Diameter	dcDiameter	Diameter of a circle, arc, or polyline arc
	...	DimConvert	Converts associative dimensions to dim constraints

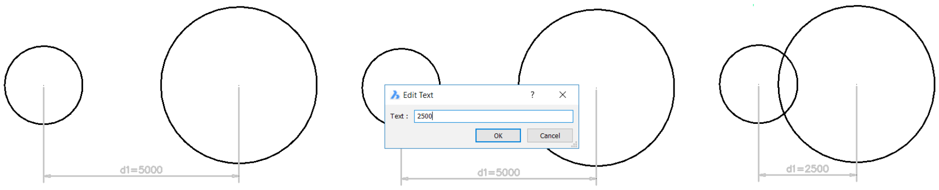
Dimensional constraints are applied in the current UCS. Icons for dimensional constraints are found at the right end of the 2D Constraints toolbar or ribbon tab:

CONVERTING DIMENSIONS TO CONSTRAINTS

Dimensional constraints are very much like associative dimensions and, in fact, existing associative dimensions can be converted to constraints with the DimConstraint command’s **Select Associative Dimension to Convert** option — or the DimConvert command.

The command converts like to like. For instance, linear dimensions become linear constraints, radial dimensions become radial constraints, and so on.

To control how far apart entities are, you place linear dimensional constraints between them. Decrease the value of the constraint, and BricsCAD moves the entities closer together; increase the value, and they are moved further apart. To edit the value, just double-click the dimension text.

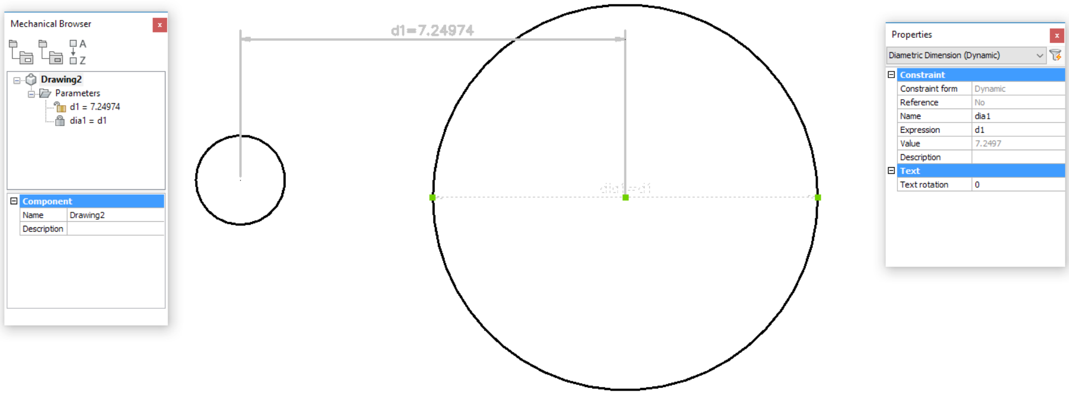


Text : 2500

Left: *Two circles separated by 5000 units;* ***center:*** *Distance being edited;* ***right:*** *Circles now separated by 2500 units*

Better yet, the values of constraints can be determined by other constraints, and even by formulas. For instance, you can make the diameter of a circle equal to the linear distance between it and another entity. When you change one dimension, the linked ones follow suit.

For example, the figure below shows that I made the value of *dia1* (diameter of one circle) equal to *d1* (distance between circles) using a simple formula, *dia1=d1*.



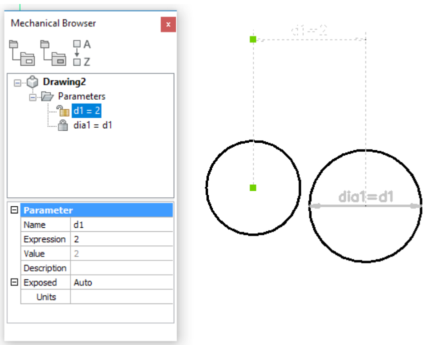
Drawing2
Parameters
d1 = 7.24974
dia1 = d1

Component	
Name	Drawing2
Description	

Diametric Dimension (Dynamic)

Constraint	
Constraint form	Dynamic
Reference	No
Name	dia1
Expression	d1
Value	7.2497
Description	
Text	
Text rotation	0

To change the values of dimensional constraints, open the Mechanical Browser with the **MechanicalBrowserOpen** command. (You can also edit values with the Properties panel.) As I change the value of d1 (the distance between the two circles), the diameter of the big circle changes.



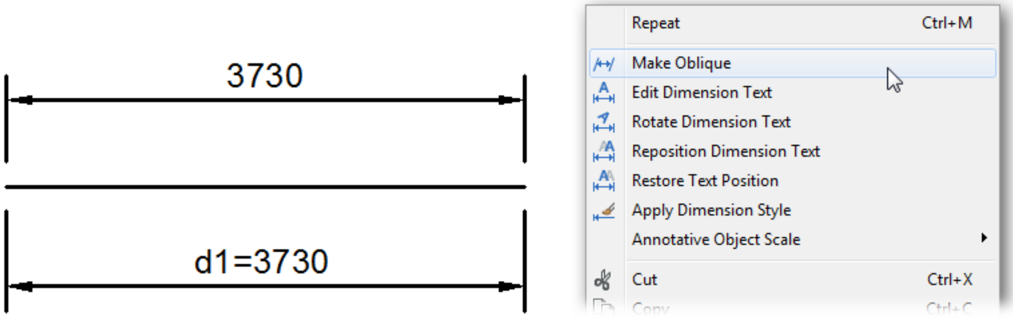
Drawing2
Parameters
d1 = 2
dia1 = d1

Parameter	
Name	d1
Expression	2
Value	2
Description	
Exposed	Auto
Units	

To remove a dimensional constraint, simply select it and then press **Del** (or else use the **Erase** command). A concise reference to all dimensional constraints is found in the boxed text on the facing page.

Using Dimensional Constraints

Using dimensional constraints in drawings is very much like placing regular dimensions. Except for the text, they look exactly like one another; you even can apply dimension styles to them. The only visual difference between the two is the 'd1=' text that prefixes the constraint value; this is how you identify dimensions that are constraints.

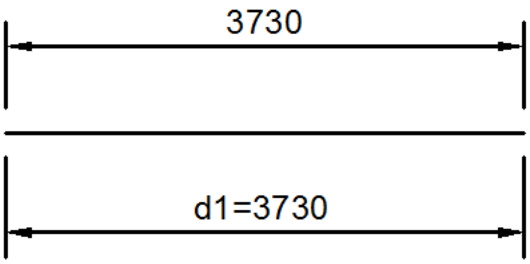


Left: *Associative dimension (top) and dimensional constraint (bottom) applied to the same line;*
right: *Editing options for dimensional constraints are the same as for associative dimensions*

The difference between the two types of dimensions is this: whereas associative dimensions are controlled *by* the entity, dimensional constraints *do* the controlling. They specify the sizes of entities, overruling what ever you may have drawn. Here is a tutorial to illustrate the differences.

1. Start BricsCAD with a new drawing.
2. Draw a line with the **Line** command. The length is not critical.
3. Dimension the line with the **DimLinear** command.
4. Use the **dcLinear** command to apply a dimensional constraint to the same line. ('dc' is short for dimensional constraint.)

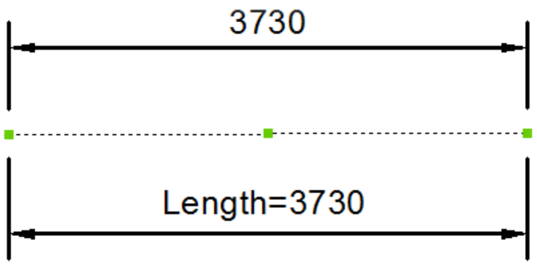
```
: dclinear
Specify first constraint point or [Entity] <Entity>: (Press Enter)
Select an entity: (Choose the line)
Specify dimension line location: (Move the cursor, and then click)
Dimension text <3730>: (Press Enter)
```



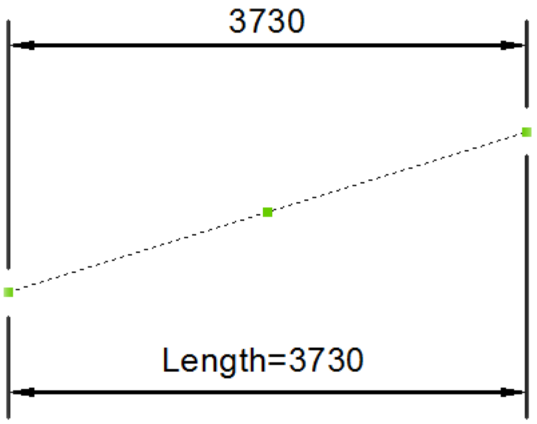
In the figure above, the two linear dimensions look identical — other than the 'd1=' name. The 'd' is short for "distance," and the '1' indicates this is the first distance constraint placed in the drawing. You can change this name through the Properties panel.

Continuing with the tutorial, you now edit the line and the dimensions to see how one affects the other.

5. Edit the line by stretching one of its end grips; see figure below. Notice that you cannot. This is because the length of the line is "locked" by the dimensional constraint.

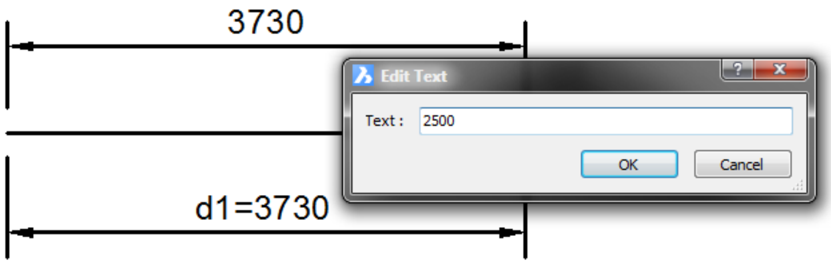


6. You can, however, use the center grip to move the line and its dimension as a unit. As well, you can stretch the line vertically and it will change its angle, because the position and the height are not locked — only the horizontal length is locked by dcLinear.



7. Now change the value of the constraint to see what happens to the line:

   a. Enter the **DdEdit** command, and then select the constraint value.



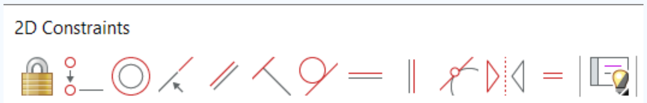
Text : 2500

   b. Enter a new value, such as **2500**, and then twice press **Enter** to exit the dialog box.

QUICK SUMMARY OF 2D GEOMETRIC CONSTRAINTS

As with dimensional constraints, there are two sets of commands for applying geometric ones. One is the all-purpose **GeomConstraint** command, which is useful for seeing a list of all modes:

```
: geomconstraint
Enter constraint type [Horizontal/Vertical/Perpendicular/PArallel/Tangent/SMooth/
Coincident/CONcentric/COLlinear/Symmetric/Equal/Fix] <CONcentric>:
```



The other set of commands consists of ones specific to each constraint mode, as listed in the table below. The names of geometric constraint commands all begin with ‘gc’.

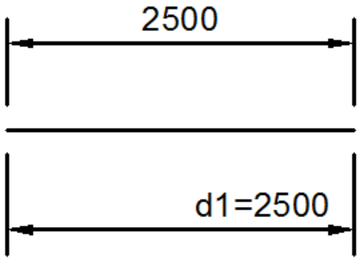
Icon	Type of Constraint	Command	Constraining Action
=	Horizontal	gcHorizontal	Keeps entities horizontal (parallel to the x-axis)
‖	Vertical	gcVertical	Keeps entities vertical (parallel to the y-axis)
∠	Perpendicular	gcPerpendicular	Keeps entities perpendicular to one other
//	PArallel	gcParallel	Keeps entities parallel to one another
	Tangent	gcTangent	Keeps circular and straight entities tangent
	SMooth	gcSmooth	Keep splines smooth with splines, lines, arcs, polylines
⁝	Coincident	gcCoincident	Keeps end points attached, such as of two lines
◎	CONcentric	gcConcentric	Keeps circles, arcs, ellipses, and elliptical arcs centered
	COLlinear	gcCollinear	Makes linear entities to lie in the same line
▷◁	Symmetric	gcSymmetric	Keeps entities or points symmetric about mirror lines
=	Equal	gcEqual	Gives curved entities the same radius; open entities the same length
🔒	Fix	gcFix	Keeps entities fixed in-place in the drawing

ACCEPTABLE GEOMETRY

Constraints apply to lines, segments of polylines, circles, arcs, ellipses, elliptical arcs, and splines.

Entity	Constraint Points
Arcs	Endpoints, center points, and midpoints
Arrays	Parameters
Block insertions	Insertion point
Circles	Center points
Ellipses	Center points
Elliptical arcs	Endpoints, center points, and midpoints
Lines	Endpoints and midpoints
Polyline line and arc segments	Endpoints, midpoints, center points of arc segments
Splines	Endpoints
Tables	Insertion point
Text, mtext, and attributes	Insertion point
Xref attachments	Insertion point

Notice that the line changes its length to match the new value of the constraint, as does the distance measured by the associative dimension.



With it comes to dimensional constraints, their *values control the entities' sizes*, and so the entities cannot be edited directly.

ABOUT GEOMETRIC CONSTRAINTS

Whereas dimensional constraint control the size of entities and distances between them, geometric constraints control their positions in the drawings. They act like semi-permanent entity snaps. Whereas esnaps are in effect only at the time that you create or edit an entity, geometric constraints remain in effect forever — until you remove them.

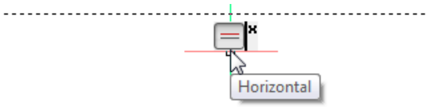
For instance, *horizontal* constraints force entities to be horizontal. Draw a line at any angle, and then apply the horizontal constraint: it snaps flat; see figure below.

Left: *Line drawn at an arbitrary angle;* ***right:*** *Line snapped flat by horizontal constraint*

When you try to rotate the entity, you can't. It can be moved, shortened (or lengthened), but not rotated, because it is constrained. When you copy a constrained entity, the copy takes on the same constraint(s).

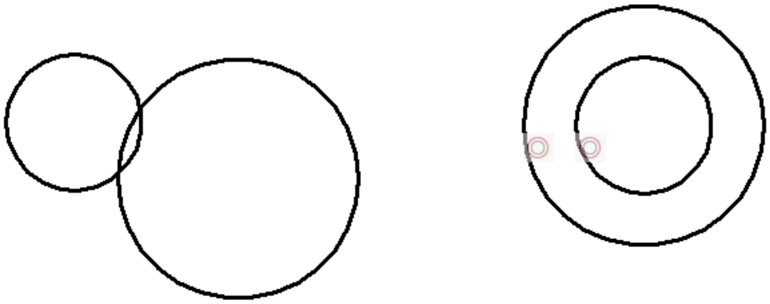
Identifying Geometric Constraints

Geometric constraints are identified by the small icon that floats near the associated entity. Pass the cursor over the icon and three things appear:



- Tooltip identifies the name of the constraint, "Horizontal"
- **X** appears next to the tooltip; clicking the X makes the icon disappear (the constraint remains in effect)
- The associated entity is highlighted

Here is another for-instance: **concentric** constraints force circles to be concentric. Move one, and the other moves with it.



Left: *Two circles placed arbitrarily in the drawing;* ***right:*** *Larger circle made concentric to the first one*

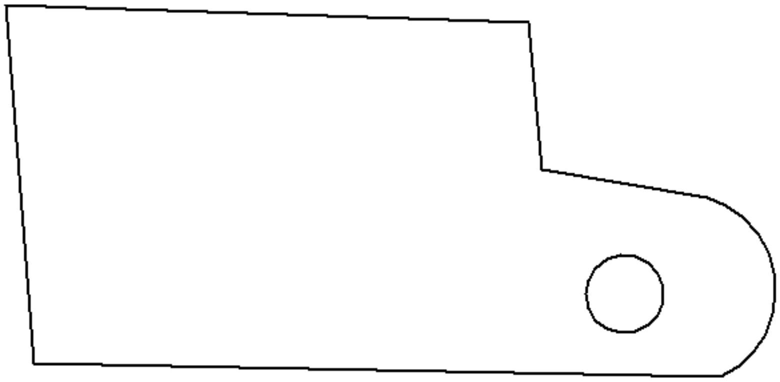
The gcConcentric command is order-dependent: the first entity you select is the master, meaning that the second entity you select follows it. (Unlike dimensional constraints, geometric constrains do not take formulae.)

To remove a geometric constraint, you have to use the **DelConstraint** command; you cannot simply "erase" geometric constraints.

All the kinds of dimensional constraints found in BricsCAD are listed in the boxed text on the facing page.

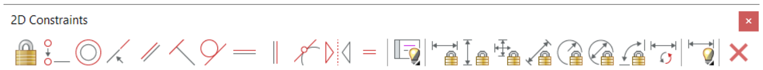
Using Geometric Constraints

The easiest way to understand geometric constraints is to work through a tutorial. Below, I drew a rough sketch of a bracket. I drew it badly deliberately to show off the beneficial effect of constraints. You can access the *bracket-ragged.dwg* file from my public cloud folder: https://my.pcloud.com/publink/show?code=XZn98mkZ7T7f1pynxJYb7vDyVl6DwyqK1Ysk



1. Start BricsCAD with a new drawing.
2. Draw a shape similar to the one illustrated above with the **Line**, **Arc**, and **Circle** commands. The exact size and exact placement of entities is not critical.

3. The 2D Constraints toolbar assists you in assigning geometric constraints to entities. Open it, as follows:
   a. Right-click any toolbar.
   b. From the shortcut menu, choose **BRICSCAD**, choose **Toolbars**, and then **2D Constraints**.



4. The first step, usually, is to connect all lines with the **gcCoincident** command. This particular constraint mode forces the endpoints of lines and arcs to stick together. The good news is that you don't need to pick entities individually:
   a. From the 2D Constraints toolbar, choose the Coincident button.

```
: _gccoincident
```

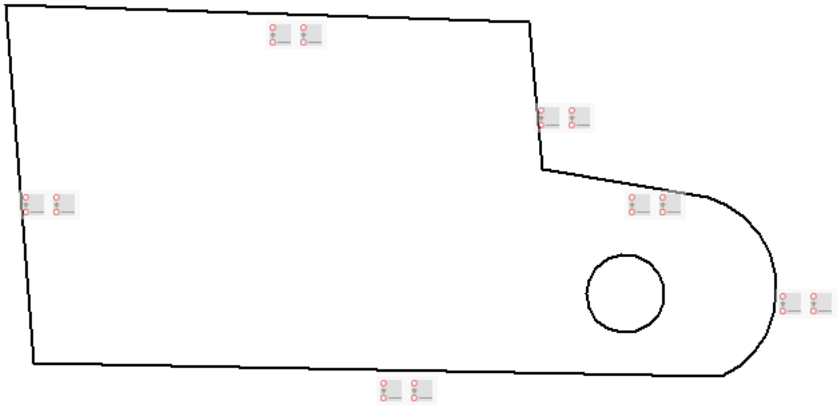
   b. Use the command's **Autoconstrain** option to connect all lines in one fell swoop:

```
Select first point or [Entity/Autoconstrain] <Entity>: a
Select entities: all
```

   c. Press **Enter** to end the command:

```
Select entities: (Press Enter)
```

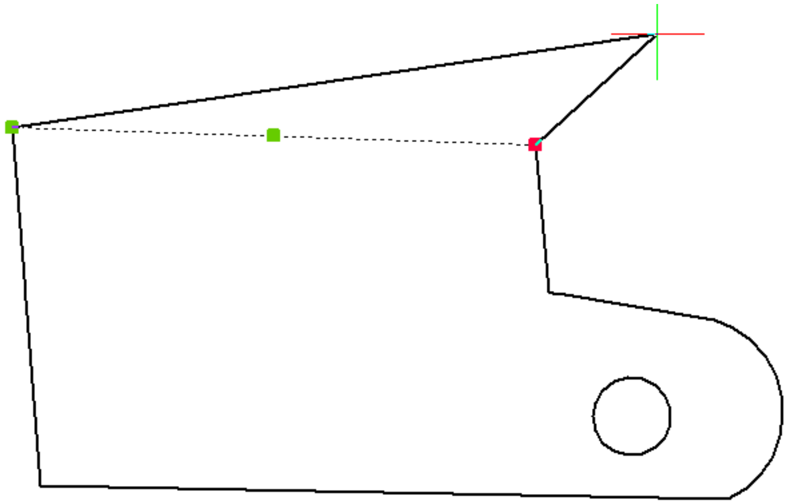
5. Notice that tooltip-like tags appear all over the drawing. They are known as "constraint bars," and report the types of geometric constraints applied to entities. In this case, the tags are all the same, because of the coincident constraint.



6. Me, I find these bars annoying, because they clutter the drawing, and so I close them. You can, too:
   - To turn off bars individually, click the **x** that appears in the upper right corner of each bar
   - To turn them off all at the same time, use the **ConstraintBar** command's **Hide** option, as follows:

```
: constraintbar
Select entities: all
Select entities: (Press Enter to continue)
Select option to [Show/Hide/Reset] constraints: h
```

7. To see the effect of the Coincident constraint, drag a line. Notice the other lines that are connected to it. This shows that Coincident is like a sticky bit of glue, making lines act like polylines.



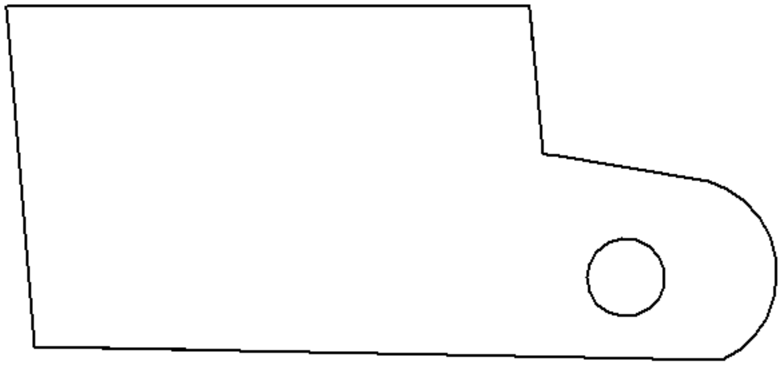
8. Enter the **U** command to return the drawing as it was before.
9. To straighten out lines that are supposed to be horizontal, use the **gcHorizontal** command on the nearly horizontal ones, as follows:
   a. In the 2D Constraints toolbar, click the ═ **Horizontal** button, and then follow the prompts in the command bar:

   ```
   : _gchorizontal
   ```

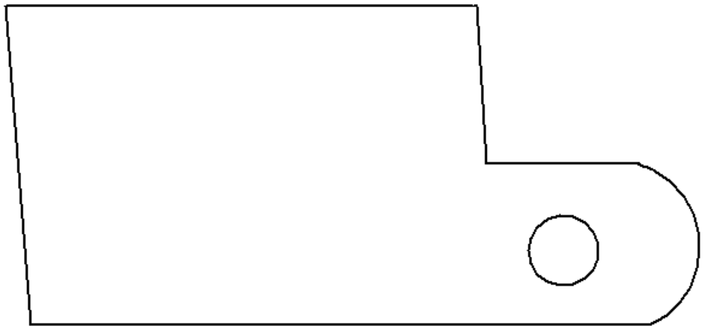
   b. Select a nearly-horizontal line:

   ```
   Select an entity or [2Points] <2Points>: (Choose a line)
   ```

   Notice that it immediately goes precisely horizontal. The line changes its length as needed, so that it remains attached to its neighbors.



c. Repeat the command to make other lines horizontal.



Now you need to straighten out the vertical lines. Here you have some options. You could apply any of these constraints:

- Two **vertical** constraints to make all the near-vertical lines truly vertical
- Two **perpendicular** constraints to make vertical lines perpendicular to the horizontal ones
- One **perpendicular** and one **parallel** constraint to make one vertical line perpendicular to one horizontal one, and then to make the remaining vertical one parallel to the first

Which approach you choose depends on your intent for the design: how do you want the lines to relate to each other?

10. I consider approach #2 is best, because I want vertical lines to all be perpendicular to the horizontals. Here is how to do this:

    a. Click the Perpendicular button to start the **gcPerpendicular** command.

    ```
    : _gcperpendicular
    ```

    b. For this type of constraint, the selection order *matters*. Essentially, the second entity selected is made perpendicular to the first one. (The first is the master, the second is the slave.) Select a horizontal line:

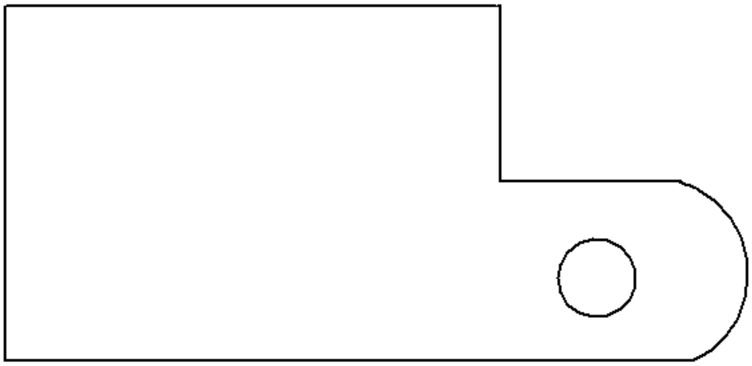
    ```
    Select first entity: (Choose a horizontal line)
    ```

    c. And then select an attached nearly-vertical one:

    ```
    Select second entity: (Choose the vertical one)
    ```

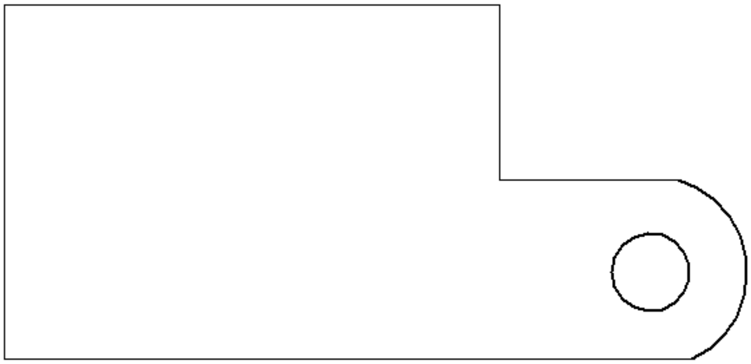
    Notice that the vertical line straightens out, because it is forced to be perpendicular to the horizontal one.

11. Repeat for the other nearly-vertical lines. The result should look similar to the figure below.



12. The circle needs a geometric constraint to fix its location. Use ◎ **gcConcentric** to match the center point of the circle to that of the arc, as follows:

    ```
    : _gcconcentric
    Select first entity: (Pick the circle)
    Select second entity: (Pick the arc)
    ```

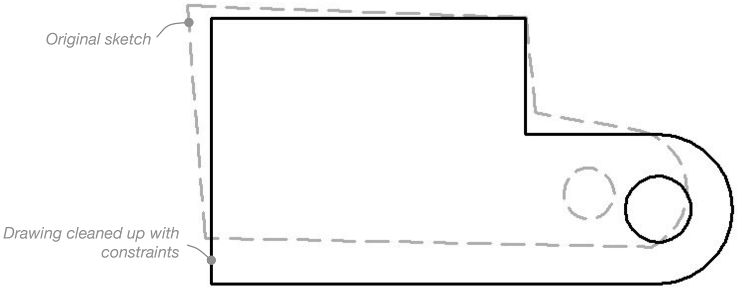
    Normally, selection order matters with Concentric constraint: the second entity shifts its position to be concentric with the first one selected. In this particular case, however, selection order did not matter, because the arc is fixed in place (through the earlier Coincident constraint), and so BricsCAD forced the circle to move.

13. Notice that the connection between the arc and the two tangent lines is not smooth. To force the arc's two ends smoothly meet the lines, use the **gcTangent** constraint, as follows:

    ```
    : _gctangent
    Select first entity: (Pick the arc)
    Select second entity: (Pick a line)
    ```

    Repeat for the second arc-line connection.

In the figure below, I overlapped the constrained drawing (shown in black) with the original rough sketch (shown in gray). It is easy to see how constraints fixed it up!



14. If you wish to see all the constraint bars again, turn them on like this:

    ```
    : constraintbar
    Select option to [Show/Hide/Reset] constraints: S
    Select entities or ENTER to select all: (Press Enter)
    ```

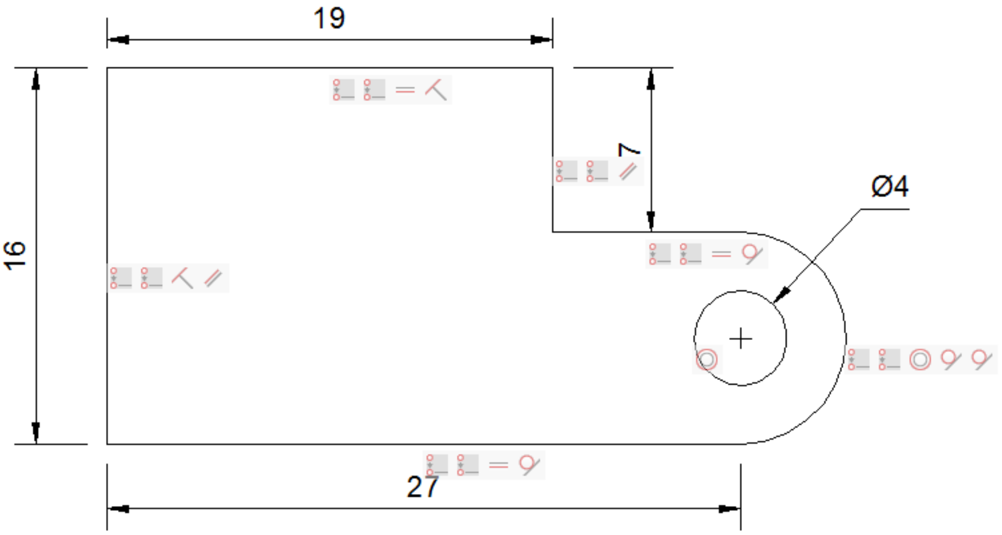
15. With all the geometry fixed in place relative to one another, you can use dimensional constraints to size the bracket. The drawing illustrated below provides the values of the dimensions.

    As you enter the values, notice that parts stretch their sizes to accommodate.

    - Use **dcHorizontal** for horizontal dimensional constraints

```
: _dchorizontal
Specify first constraint point or [Entity] <Entity>: (Press Enter)
Select an entity: (Pick a line)
Specify dimension line location: (Pick a point)
Dimension text <17.993713>: 19
```

- Use **dcVertical** for vertical dimensional constraints
- Use **dcDiameter** for diameter dimensional constraints



This concludes the tutorials on using constraints with BricsCAD. You learned how constraints can be used to clean up drawings, as well as to quickly change the sizes of them.

Notes

APPENDIX A

Concise Summary of Command Names

IN THIS APPENDIX, YOU CAN REFERENCE THE NAMES OF OVER 900 COMMANDS IN BRICSCAD. They are listed alphabetically by name, as well as in groupings of common commands, as follows:

- **ai-** commands
- **bim-** (building information modeling) commands
- **bm-** (BricsCAD mechanical) commands
- **Civil** commands (NEW TO V20)
- **Cloud-** commands (ex-Chapoo commands)
- **Dim-** (dimension) commands
- **dc-** (dimensional constraint) commands
- **dm-** (direct modeling) commands
- **gc-** (geometric constraint) commands
- **Layer** Commands
- **sm-** (sheet metal) Commands
- **VBA** (Visual Basic for Applications) commands
- **ViewBase** commands

Commands with a hyphen prefix, such as -Color, are ones that run at the command prompt, and have a complimentary command, such as Color, that displays a dialog box.

Command names new in V20 are shown in **blue**.

A Commands

About displays information about the program.

AcisIn imports 3D solids in SAT format (SAT is short for "save as text").

AcisOut exports 3D solids and surface entities in SAT format.

AddInMan displays the VBA COM Add-In Manager dialog box.

AddSelected creates a new entity of the same type as an existing entity.

Align aligns entities with other entities in 2D and 3D space.

AlignSpace adjusts viewport angle, zoom factor, and pan position based on alignment points specified in model space and paper space; operates in paper space only.

AniPath makes movies from views generated by a camera moving through 3D scenes.

AnnReset resets all scale representations to the entity's original positions

AnnUpdate updates annotative scale factors to match updates made with Style and DimStyle commands.

Aperture sets selection area for snapping to entities.

Apparent toggles Apparent intersection entity snap; snaps to the intersections of entities, even when they only appear to intersect in 3D space.

AppLoad loads DRX, LISP, and SDS applications to run inside BricsCAD; Mac and Linux load only LISP and SDS.

Arc draws arcs.

Area determines the area and perimeter of closed 2D objects; the area and length of open polylines and splines as if they were closed; the lengths only of lines, sketches, arcs, and elliptical arcs; and the areas of faces of 3D objects.

Array and **-Array** creates dynamic polar, path, and rectangular arrays of entities.

ArrayClassic runs the dialog box-based version of the Array command.

ArrayClose and **-ArrayClose** end the array editing session.

ArrayEdit edits entities and source entities of arrays.

ArrayEditExt edits entities in arrays.

ArrayPath distributes entity copies evenly along a path into multiple rows and levels.

ArrayPolar distributes entity copies evenly in a circular pattern about a center point or axis of rotation, using multiple rows and levels.

ArrayRect distributes entity copies into any number of rows, columns, and levels.

AttachmentsPanelOpen opens the Attachments panel for managing Xref, Raster Image, PDF, and Pointcloud attachments.

AttachmentsPanelClose closes the Attachments panel.

AttDef and **-AttDef** defines attributes for blocks.

AttDisp toggles the display of attributes through all, none, or those normally visible.

AttEdit edits the values and properties of attributes.

AttExt and **-AttExt** exports data from attributes to text files.

AttRedef redefines blocks and updates associated attributes.

AttSync synchronizes attribute definitions in all references to a specified block definition.

Audit repairs open drawings in case of data corruption.

AutoComplete sets the options for autocomplete mode on the command line.

Ai Commands

Ai_Box draws 3D boxes as mesh surfaces.

Ai_CircTan draws a circle tangent to three entities.

Ai_Cone draws 3D cones as mesh surfaces.

Ai_Cylinder draws 3D cylinders as mesh surfaces.

Ai_DeSelect unselects all selected entities.

Ai_Dish draws 3D dishes as mesh surfaces.

Ai_Dome draws 3D domes (half-spheres) as mesh surfaces.

Ai_DrawOrder changes the display order of overlapping entities.

Ai_Fms switches to the first layout tab and enters model space of the first viewport.

AiMleaderEditAdd adds leader lines to multi-leaders.

AiMleaderEditRemove removes leader lines from multi-leaders.

Ai_Molc makes the layer current of the selected entity (short for "make object layer current").

Ai_MSpace switches to model tab.

Ai_PSpace switches to the first layout tab.

Ai_Pyramid draws 3D pyramids as mesh surfaces.

Ai_SelAll selects all non-frozen entities in the current space, like Ctrl+A.

Ai_Sphere draws 3D spheres as mesh surfaces.

Ai_TileMode1 sets TileMode variable to 1 and then switches to model tab.

Ai_Torus draws 3D tori as mesh surfaces.

Ai_Wedge draws 3D wedges as mesh surfaces.

B Commands

Background displays the Background dialog box for creating background colors and images in shaded modes and renderings.

Base changes the drawing's insertion point when when it is inserted into other drawings.

BAttMan manages the attributes of block definitions (short for Block Attribute Manager).

BClose closes the Block Editor

BEdit and **-BEdit** open the Block Editor environment

BHatch and **-BHatch** fills closed areas with repeating patterns, solid colors, or gradients.

Blade opens the LISP editing environment

BlipMode enables and disables display of marker blips.

Block and **-Block** groups entities into blocks (symbols).

Blockify converts entities to blocks to save space and increase speed

BmpOut exports the current viewport as a BMP (bitmap) file.

Boundary and **-Boundary** draws a polyline that forms a boundary around the inside closed areas.

Box draws three-dimensional solid boxes.

Break removes portions of entities.

Browser opens the default Web browser.

BIM Commands

(Available as an add-on to the Platinum edition only; bim = building information modeling)

bimAddEccentricity controls relative positions of the axes in linear solids.

bimApplyProfile applies profiles to linear entities and linear solids.

bimAttachComposition attaches BIM compositions to solids.

bimAttachSpatialLocation locates the drawing in mapping references.

bimClassify classifies an entity as a building element with a name and an internal 'guid' (globally unique identifier).

bimCopy copies entities *normal* (at 90 degrees) to the selected face.

bimCurtainWall creates curtain walls made of planar quadrilateral panels from free-form surfaces.

bimDecompose decomposes composition-based solids into separate plies.

bimDisplayComposition toggles the display of compositions on and off.

bimDrag drags faces of solids; when dragging major faces, it preserves connections with minor faces; when dragging minor faces, it optionally connects minor faces to major faces of other solids.

bimFlip flips the starting face from which the layers of a composition are set out.

bimFlowConnect connects linear solids.

bimGrid creates rectangular and radial grids with automatically-applied labels.

bimIfy automatically classifies and spatial locates the entire bim model.

bimInsert and **-bimInsert** insert windows and doors in solids.

bimLinearSolid creates chains of linear solids.

bimList list names and properties of BIM entities in the current drawing.

bimMultiSelect selects one or more linear solids with coplanar and/or parallel axes based on the initial solid or face selected.

bimPatch reserves an of a BIM model for editing with the RefEdit command.

bimProfiles displays the Profiles dialog box for creating and editing profiles.

bimProjectInfo displays the BIM Project Info dialog box for specifying project library databases.

bimPropagate (replaces bimSuggest) maps details from selected solids to all similar solids, as well as on grids.

bimPropagateEdges propagates along the edges of planar solids, such as railings.

bimPropagateLinear propagates connections to linear elements,such as connections to walls and slabs.

bimPropagatePattern propagates a single element (such as a switch) to multiple locations and grids.

bimPropagatePlanar propagates connections to planar elements, such as walls, slabs, and roofs.

bimProperties displays the BIM Properties dialog box for specifying and editing properties of bim projects.

bimQuickDraw draws rooms and stories from rectangles and L-shapes.

bimRecalculateAxis recalculates the axes of structural elements back to their centroids.

bimRoom defines room areas with markers.

bimRoomBoundingElements determines which elements (walls, floors, etc) determine bounds of rooms.

bimSchedule generates linked schedule tables after analyze building elements in BIM models.

bimSection creates BIM section entities.

bimSectionOpen opens the drawing file related to a BIM section entity; or the 3D BIM model related to a BIM section drawing.

bimSectionUpdate updates and exports BIM sections.

bimSetReferenceFace controls the layout of plies through reference and opposing faces.

bimSplit splits segmented solids into separated solids automatically; splits solids using cutting faces.

bimStair creates stairs between two floors (as a rectangular parametric array).

bimStretch stretches BIM entities.

bimStructuralConnect connects linear solids.

bimTag tags BIM sections.

bimUpdateRoom updates data about the selected room.

bimUpdateThickness re-applies the overall thickness of a composition to the solid.

bimWindowCreate replaces closed entities with parametric window entities; displays the choose window style dialog box.

bimWindowPrint prints a specified area of the BIM model.

bimWindowUpdate updates openings made by windows or doors in solids in case the opening did not updated correctly automatically.

ClipDisplay toggles the clipped display property of a section plane or a BIM section entity.

BricsCAD Mechanical Commands

(Available in Platinum edition only; bm = BricsCAD mechanical)

bmBalloon associates balloon with assembly components in Model space and in generated views in layouts.

bmBom inserts bill of material (BOM) tables in the current drawing.

-bmCreateComponent creates a component from a selection set; add it to the library.

bmDependencies lists all files, containing component definitions inserted in the assembly, in the command window.

bmDissolve dissolves a mechanical component inserted in the current drawing.

bmExplode creates a block of an exploded representation of an assembly.

bmExplodeMove allows users to created exploded representations of assemblies.

bmExternalize converts local components to external components.

bmForm creates a new mechanical component and inserts it into the current drawing; if necessary, run bmMech to initialize the mechanical structure in the current drawing.

bmHardware and -**bmHardware** insert standard hardware parts as a mechanical component in the current drawing.

bmHide hides the visibility of mechanical components; hidden inserts are taken into account by commands such as bmBom and bmMassProp.

bmInsert and -**bmInsert** insert an existing mechanical component as a virtual component into the current drawing.

bmLispGet retrieve variables for blocks and parameters of components.

bmLocalize converts external components to local components.

bmMassProp computes mass properties for the current model using densities assigned to the components (defined by the density property of the components and subcomponents).

bmMech converts the current drawing into a mechanical component.

bmNew creates a mechanical component as a new drawing file.

bmOpen opens the source drawing of external mechanical components.

bmOpenCopy opens a copy of a component insert as a new drawing.

-bmParameters lists and edits parameters of inserted components.

bmRecover recovers broken mechanical structures.

bmReplace replaces a component insert.

bmShow shows previously hidden mechanical components.

bmUnlink breaks links between components.

bmUnmech converts the current mechanical component into a plain drawing.

bmUpdate reloads all referenced components from external files and updates BOM tables.

bmVStyle applies visual styles to mechanical component inserts.

bmXConvert converts X-Hardware solids in the current drawing to mechanical components.

C Commands

Cal displays the operating system's Calculator program.

Callout places callouts; can be used only from the SheetSet panel.

Camera changes the viewpoint to perspective.

Center toggles Center entity snap; snaps to the center of circles, arcs, and other circular entities.

CenterDisassociate disassociates center lines and marks from circles and arcs

Centerline places associative center lines on circles and arcs

Centermark places associative center marks on circles and arcs

CenterReassociate reassociates centerlines/marks with circles and arcs

CenterResetresets centerline and mark entities, if moved

Chamfer bevels entities.

Change changes the position and properties of entities: endpoint, color, elevation, layer, linetype, linetype scale, lineweight, and thickness.

ChProp changes just the properties of entities.

ChSpace moves entities from paper space to model space and vice versa.

Circle draws circles.

CleanScreenOn hides most user interface elements to maximize the drawing area.

CleanScreenOff restores the user interface to its default configuration.

CleanUnsedVariables clears unused variables from memory.

Close exits the current drawing, but not the program.

Color and **-Color** specifies the color for entities.

CommandLine and **CommandLineHide** open and close the command bar.

Commands reports the names of all commands supported by the program.

CommunicatorInfo reports the status of the Communicator add-on

ComponentsPanelOpen opens the Components panel for accessing symbols

ComponentsPanelClose closes the Components panel

Cone draws three-dimensional solid cones.

ContentBrowserClose and **ContentBrowserOpen** close and open the Content Browser panel.

ConvertCtb converts older CBT (color-based plot tables) files to newer STB (style-based plot tables) files.

ConvertOldLights converts old light definitions to the current format.

ConvertOldMaterials converts old material definitions to the current format.

ConvertPoly converts lightweight polylines to classic polylines (2D polylines) and vice versa.

ConvertPStyles converts drawings to from CTB (color-based plotting) to STB (plot styles).

ConvToMesh converts 3D solids and surfaces to mesh objects

ConvToSolid converts watertight meshes, circles, and closed polylines to 3D solids

ConvToSurface converts 3D solids, open polylines and other entities to 3D surfaces

Copy duplicates entities.

CopyBase copies entities with a specified reference point to the Clipboard.

CopyClip copies entities to the Clipboard.

CopyEData Copies extended entity data from one entity to others.

CopyGuided copies entities along guidelines.

CopyHist copies the command history to the Clipboard.

CopyToLayer copies selected entities to another layer.

CPageSetup edits the page setup of the current layout or model space.

CuiLoad and **CuiUnload** load and unload CUI and CUIX (user interface customization), MNU (menu), MNS (LISP code), and ICM (IntelliCAD menu) files.

Customize customizes user interface elements, such as menus, toolbars, and shortcuts.

CutClip copies entities to the Clipboard and deletes the entities.

Cylinder draws three-dimensional solid cylinders.

Civil Commands

(Civil engineering commands are part of Platinum)

Alignment creates horizontal and vertical alignments typically used to design roads.

AlignmentEdit edits horizontal and vertical alignments.

AlignmentView views alignment along TIN surfaces.

AlignmentVInitial creates vertical alignments.

Grading interactively modifies TIN surfaces to create grading effects, such as for roads and foundations.

LandXmlEexport exports the drawing in LandXML format

LandXmlImport imports LandXML files into the current drawing

Tin (triangulated irregular network) imports data from TIN files to create land surfaces, and converts Civil 3D surfaces to the BricsCAD format

TinEdit adds and removes points, break lines, and boundaries in TIN surfaces.

TinExtract creates a mesh or 3D solid between TIN surfaces or between a TIN surface and elevation.

TinMerge combines two or more TIN surfaces into one.

TinModify deforms and smooths selected TIN surfaces.

TinVolume creates a TIN volume surface between a base and comparison TIN surfaces or an elevation.

Cloud Commands

All Chapoo- commands were renamed Cloud- in V18

CloudAccount reports the status of the 24/7 account at the command bar.

CloudDownload downloads drawings from the 24/7 project to a local folder.

CloudLogoff logs off from the 24/7 project.

CloudLogon logs on to 24/7.

CloudOpen opens a drawing after downloading it from 24/7.

CloudProject opens the 24/7 project in the default browser.

CloudUpload uploads the current drawing to 24/7.

CloudWeb connects to the 24/7 website at https://www.bricsys.com/en-intl/247/.

D Commands

DataExtraction exports entity properties, block attributes and drawing information to CSV (comma separated values) file.

DataLink imports Excel spreadsheets and CSV files as linked table entities

DataLinkUpdate updates the data linked between a table and an external file

DbList lists information about all entities in the drawing (short for "database listing").

DdAttE edits the values of attributes through a dialog box (short for "dynamic dialog attribute editor").

DdEdit edits single-line text, multi-line text, attribute definitions, and attribute text (short for "dynamic dialog editor").

DdEModes sets default values for creating entities (short for "dynamic dialog entity modes").

DdFilter creates a selection set of the entities selected.

DdGrips specifies the properties of grips through the Settings dialog box.

DdPType specifies the look and size of point entities, through the Settings dialog box (short for "dynamic dialog point type").

DdSelect specifies the properties for selecting entities, through the Settings dialog box.

DdSetVar displays the Settings dialog box to change the values of variables.

DdSTrack Sets the properties for snap tracking, through the Settings dialog box (short for "snap tracking").

DdVPoint sets 3D viewpoints or plan view.

DefaultScaleList displays the Scale List Edit dialog box to edit the default scale factors

DesignTable creates new design tables for the Mechanical Browser.

-DesignTableEdit configures, replaces, exports, and deletes design tables at the command line.

Delay delays execution of the next command; for use with scripts only.

DelEData deletes extended entity data from the selected entity (short for "delete entity data").

DgnImport imports Microstation design files and converts them to entities.

DgnImportOptions opens the Settings dialog box at the DgnImport section.

Dish draws dishes (bottom half-spheres) from polygon meshes.

Dist reports the distance and angle between two points.

Distantlight places distant lights.

Divide places points or blocks along entities.

Dome draws domes (top half-sphere) from polygon meshes.

Donut draws circular polylines with width.

Drag moves faces.

DragMode controls the appearance of objects while being dragged.

DrawOrder changes the display order of overlapping entities.

DrawOrderByLayer controls the draw order of overlapping objects through layer names.

DSettings displays the Settings dialog box for drafting settings (short for "drafting settings").

DstConvert converts sheetset DST files to XML format.

DView changes the 3D viewpoint interactively, and turns on perspective mode (short for "dynamic view").

DwgCodePage changes the code page for text in drawings.

DwgCompare compares differences between two drawings, and visually merges drawings.

DwgProps opens the Drawing Properties dialog box, showing the general information and user defined properties stored with a drawing.

DxfIn and **DxfOut** imports DXF files (short for "drawing exchange format") and exports drawings in ASCII or binary DXF format.

Dimensioning Commands

(Dim = dimension)

Ai_Dim_TextAbove moves text above the dimension line.

Ai_Dim_TextCenter centers text on the dimension line.

Ai_Dim_TextHome moves text to its home position, as defined by the dimension style.

AiDimFlipArrow mirrors arrowheads on dimension lines.

AiDimPrec changes the precision of dimension text.

AiDimStyle creates dimension styles from a selected dimension.

Dim places and edits dimensions at the 'Dimensioning command:' prompt.

Dim1 executes a single dimension command at the 'Dimensioning command:' prompt.

DimAligned draws dimensions parallel to (aligned with) selected entities; works with lines, polylines, arcs, and circles.

DimAngular dimensions angles.

DimArc places arc length dimensions.

DimBaseline places multiple linear or angular dimensions starting at the same base point; command can only be used when at least one other dimension is already in the drawing.

DimCenter places center marks at the center points of circles and arcs.

DimContinue continues linear and angular dimensions from the endpoint of the previous dimension.

DimDiameter dimensions the diameter of circles and arcs, and places a center mark.

DimDisassociate removes associativity from selected dimension entities.

DimEdit changes wording and angle of dimension text; changes the angle of extension lines.

DimLeader draws leaders.

DimLinear places linear dimensions horizontally, vertically, or rotated.

DimOrdinate measures x and y ordinate distances from a common origin, specified by the current UCS origin.

DimOverride overrides the values of the current dimension style.

DimRadius dimensions the radii of arcs and circles.

DimReassociate reassociates or associates dimensions to entities or points on entities.

DimRegen updates associative dimensions (short for "dimension regeneration").

DimStyle and **-DimStyle** creates and modifies dimension styles through the Drawing Explorer.

DimStyleSet reports the current dimension style in the command bar.

DimTEdit changes the position of dimension text.

Dimensional Constraint Commands

(dc = dimensional constraint)

CleanUnusedVariables purges variables not used by constraint expressions and not linked to dimensions.

dcAligned constrains the distance between two defining points on entities.

dcAngular constrains the angle between three constraint points on entities; or between two lines; or between two polyline segments; or constrains the angles of arcs or polyline arcs.

dcConvert converts an associative dimension to a dimensional constraint.

dcDiameter constrains the diameters of circles, arc, or polyline arcs.

dcDisplay shows and hides dimensional constraints.

dcHorizontal constrains the horizontal distance between two defining points on entities.

dcLinear constrains horizontal or vertical distance between two defining points on entities.

dcRadial constrains the radius of circles, arcs, or polyline arcs.

dcVertical constrains the vertical distance between two defining points on entities.

DelConstraint removes all dimensional (and geometrical) constraints from an entity.

DimConstraint applies a dimensional constraint to an entity or between constraint points on entities; converts associative dimensions to dynamic dimensions.

Direct Modeling Commands

(Available for Pro or Platinum editions only; dm = direct modeling)

dmAngle3D applies angle constraints between the faces of a solid or of different solids.

dmAudit checks and fixes 3D models.

dmAuditAll also checks and fixes 3D ACIS models in externally-referenced drawings

dmChamfer creates an equal distance chamfer between adjacent faces.

dmCoincident3D applies coincident constraints between two edges, two faces, or an edge and a face of two different solids.

dmConcentric3D applies concentric constraints between two cylindrical, spherical, or conical surfaces.

dmConstraint3D applies geometric relationships and dimensional constraints between sub-entities (such as faces, surfaces, and edges) of 3D entities.

dmCopyFaces copies features like holes and ribs to the same or other 3D solids

dmDeformCurve deforms one or more connected faces of a 3D solid/surface by replacing their edges with given curves.

dmDeformMove deforms one or more connected faces of a 3D solid/surface by moving and rotating their edges.

dmDeformPoint deforms as smoothly as possible (using G1 or G2 continuity) a region, one or more connected faces of a 3D solid or a surface by moving a point lying on one of them in arbitrary 3D direction.

dmDelete deletes faces and solids.

dmDistance3D applies a distance constraint between two sub-entities of a solid or of different solids.

dmExtrude creates 3D solids by extruding closed 2D entities, regions or closed boundaries.

dmFillet creates a smooth fillet between adjacent faces sharing a sharp edge.

dmFix3D applies a fixed constraint to a solid or to an edge or a face of a solid.

dmGroup creates new groups, edits them, and dissolves groups.

dmMove moves the selected solids, or faces or edges of a solid using a vector.

dmParallel3D applies a parallel constraint between two faces of a solid or of different solids.

dmPerpendicular3D applies a perpendicular constraint between two faces of a solid or of different solids.

dmPushPull adds or removes volume from a solid by moving a face.

dmRadius3D applies a radius constraint to cylindrical surfaces or circular edges.

dmRepair fixes inconsistencies in 3D geometry supported by ACIS kernel (3D solids, surfaces).

dmRevolve creates 3D solids by revolution of closed 2D entities or regions about an axis.

dmRigidSet3D defines a set of entities or sub-entities as a rigid body.

dmRotate rotates faces of a solid around an axis.

dmSelect selects edges and faces of 3D solids or surfaces based on their geometric properties.

dmSelectEdges selects faces and edges of 3D solids.

dmSimplify simplifies the geometry and topology of 3D solid entities by removing unnecessary edges and vertices, merges seam edges, and replaces the geometry of faces and edges by analytic surfaces and curves, if possible within the user-specified tolerance. Run this command on imported 3D solid geometry.

dmSimplifyAll also unnecessary elements in externally referenced drawings

dmStitch converts a set of region and surface entities that bound a watertight area to a 3D solid.

dmTangent3D applies a tangent constraint between a face and a curved surface of different solids.

dmThicken creates 3D solids by thickening (i.e. adding thickness to) surfaces, their faces, and faces of 3D solids.

dmTwist twists 3D solids, surfaces, and regions by an angle.

dmUpdate forces 3D constraints to update.

E Commands

EAttEdit edits the value and most properties of attributes (short for "enhanced attribute editor").

EdgeSurf creates a 3D Coons mesh surface patch between four lines, forming a closed shape (short for "edge surface").

EditEData creates and edits extended entity data (short for "edit entity data").

Elev changes the default elevation and thickness.

Ellipse draws ellipses and elliptical arcs.

EndCompare ends the drawing compare session

Endpoint toggles endpoint entity snap; snaps to the ends of open entities, such as line, arcs, and open polylines.

Erase erases selected entities from drawings; alternatively, press the Del key.

eTransmit creates a package of a drawing file and all its dependencies, such as external references, images, font files, plot configuration files, plot style tables and font map files.

ExpBlocks opens the Blocks section of the Drawing Explorer dialog box (short for "explorer blocks").

ExpFolders opens the Drawing Explorer on the Folders tab.

ExpImages opens the Drawing Explorer at the Images section.

ExpLayers opens the Drawing Explorer at the Layers section.

Explode breaks complex objects into their component entities.

Explorer opens the Drawing Explorer dialog box, which controls Layers, Layer States, Linetypes, Multiline Styles, Multileader Styles, Text Styles, Dimension Styles, Table Styles, Coordinate Systems, Views, Visual Styles, Lights, Materials, Render Presets, Blocks, External References, Images, PDF Underlays, Dependencies, Page Setups, and Section Planes.

Export saves entities in other file formats.

ExportLayout exports visible objects from the current layout to model space of new drawings.

ExportPDF exports the current layout to a PDF file.

ExpPdfs opens the Drawing Explorer at the PDF section.

ExpUcs creates, modifies, deletes named UCSes through the Drawing Explorer (short for "explore user-defined coordinate systems").

ExpXrefs opens the Drawing Explorer at the XRefs section.

Extend extends entities to bounding edges defined by other entities.

Extension toggles extension entity snap, which snaps to the point where a line extended would intersect another entity.

Extrude extrudes closed entities as 3D solids and open ones as 3D surfaces.

F Commands

FbxExport and **-FbxExport** export 3D models in FBX format for rendering programs

Field inserts text that is updated automatically when system variables change.

FileOpen opens drawing (DWG), template (DWT), and interchange (DXF) files from the command line.

Files opens the operating system's file manager, such as Windows Explorer or Finder.

Fill fills areas with a solid color or color gradient

Fillet rounds entities.

Find finds and replaces text in notes, annotations, and dimension text.

Flatshot creates a hidden line representation of all 3D solids in model space as a block or a new drawing.

Flatten flattens 2D objects with thickness and allows to convert splines to polylines.

G Commands

GCE snaps the the geometric center of entities.

GenerateBoundary creates closed polylines from faces of 3D solids, as well as from boundaries detected when the Enable

Boundary Detection of SelectionModes is activated.

GeographicLocation sets the geographic location of the drawing.

GoToStart displays the Start tab.

Gradient and **-Gradient** fill closed areas with gradient fills of one or two colors.

GradientBkgOff and **GradientBkgOn** turn off and on the gradient displayed in the working area.

GraphScr switches from the text windows to the graphics windows (short for "graphics screen").

Grid turns the grid display on or off and sets other grid options.

Group and **-Group** creates and modifies named groups of entities.

Geometric Constraint Commands

(For 3D constraints, see Direct Modeling Commands section; gc = geometric constraints)

ConstraintBar shows, hides, and resets the display of geometric constraint icons.

DelConstraint removes all geometrical (and dimensional) constraints from an entity.

gcCenter snaps to the centroid of closed entities.

gcCoincident constrains points on entities coincidently; or constrains a point on an entity to another entity.

gcCollinear constrains lines collinearly.

gcConcentric constrains the center points of arcs, circles, ellipses, and/or elliptical arcs to be coincident.

gcEqual constrains lines to have the same length, or arcs and circles to have the same radius.

gcFix constrains points on entities to fixed positions.

gcHorizontal constrains lines or linear polyline segments, or pairs of points on entities to be parallel to the x axis in the current coordinate system.

gcParallel constrains two lines or linear polyLine segments to be parallel to each other.

gcPerpendicular constrains two lines or linear polyline segments to be perpendicular to each other.

gcSmooth constrains a spline to be fluidly continuous to another spline, or arc, or line, or polyline.

gcSymmetric constrains two entities, or two points on entities, to be symmetric about a line of symmetry.

gcTangent constrains one entity tangent to another.

gcVertical constrains lines or linear polyline segments, or pairs of points on entities to be parallel to the y axis in the current coordinate system.

GeomConstraint acts as a universal command that applies all available geometric constraint points.

H Commands

Hatch and **-Hatch** fills a selected boundary with a pattern.

HatchEdit and **-HatchEdit** edits hatch patterns and gradient fills.

HatchGenerateBoundary generates a boundary around a hatch or gradient fill.

HatchGripEdit adds and removes grips from hatches and gradients.

HatchToBack sets the draw order of all hatch entities in the drawing to display behind all other entities.

Helix draws 2D spirals or 3D helixes.

Help displays online help.

HelpSearch prompts for searching through the help files at the command prompt.

Hide removes hidden lines from 3D entities until the UnisolateObjects command is used.

HideObjects temporarily hides selected entities.

Hyperlink and **-Hyperlink** adds hyperlinks to entities or modifies existing hyperlinks.

HyperlinkOptions controls the display of the hyperlink cursor, shortcut menu, and tooltips.

I Commands

Id reports the x,y,z coordinates of a picked point.

Image inserts raster images in drawings through the Drawing Explorer.

ImageAdjust adjusts the properties of images through the Properties palette.

ImageAttach and -**ImageAttach** attache raster images to the drawing like xrefs.

ImageClip clips images.

ImageFrame toggles the frame around images.

ImageQuality determines the display quality of images attached to the drawing.

Import displays a dialog box for importing files into the drawing: DWG, DXF, DWT, and DAE (Collada) files. Platinum edition also imports IFC and SKP (SketchUp) files. Additional formats are imported with the optional Communicator module.

Imprint imprints 2D entities onto planar faces of 3D solids and surfaces; allows to create additional edges on planar faces.

Insert and **-Insert** inserts blocks or another drawing into the current drawing.

InsertAligned inserts blocks repeatedly, and inserts mirrored blocks.

InsertGuided and **-InsertGuided** inserts blocks along guide curves.

Insertion toggles Insertion entity snap; snaps to the insertion point of text and blocks.

InsertObj displays data from other programs in drawings, such as text documents, spreadsheets, and images.

Interfere checks interferences between solid models.

Intersect creates regions or 3D solids from the intersection of regions or 3D solids.

Intersection toggles Intersection entity snap; snaps to the intersections of entities.

IsolateObjects hides all other entities from view.

Isoplane controls the isometric plane (left, right, or top) when isometric snap is used.

J Command

Join joins lines, lwpolylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at common endpoints.

K Command

KeepMe visually merges drawings during the DrawingCompare command

L Commands

Layer: see Layer Commands below.

Layout creates, copies, renames, and deletes layouts.

LayoutManager displays the Layout Manager dialog box for creating, naming, and reordering sets of layouts

LConnect creates connections between faces of two solids

Leader draws leader lines that connect annotations to drawing entities.

Lengthen changes the length of open objects, such as lines and arcs.

LicenseManager provides access to all Bricsys software licenses, as shown below.

LicEnterKey enters the license key number (short for "licence enter key").

LicProperties reports the BricsCAD license information; modifies and deactivates single user and volume license keys.

LicPropertiesCommunicator reports license information for the optional extra-cost Communicator add-on.

Light places lights in drawings.

LightList displays the lighting palette.

Limits sets the extents of the drawing and the grid.

Line draws straight line segments.

LineType and **-LineType** creates, loads, and sets linestyles.

List lists the properties of selected entities at the command line.

LiveSection toggles the Live Section property of a section plane.

Load loads compiled SHX shape files into the drawing.

Loft creates 3D solids passing through two or more cross sections.

LogFileOff and **LogFileOn** turn off and on log file recording.

LWeight sets lineweight options.

Layer Commands

LayCur moves the selected entities to the current layer.

Layer and **-Layer** controls layers and layer properties.

LayerP undoes previously applied changes to layer settings when LayerPMode is on (short for "layer previous").

LayerPMode controls the tracking of changes made to layer settings.

LayersPanelClose and **LayersPanelOpen** closes and open the Layers panel.

LayerState saves and restores the properties of layers.

LayFrz and **LayThw** freeze and thaw the layers associated with entities selected in the drawing.

LayIso and **LayUnIso** isolate and restore layers associated with entities selected in the drawing; locks or turns off all other layers (short for "layer isolate").

LayLck and **LayUlk** lock and unlock the layers of selected entities.

LayMCur changes the working layer to that of a selected entity (short for "layer make current").

LayOff and **LayOn** turn off and on layers associated with entities selected in the drawing; off layers cannot be seen.

M Commands

Mail attaches the current drawing to a new message with your computer's default email client.

Manipulate launches the widget for rotating, copying, moving, mirroring, and scaling entities.

MapConnect sets up a connection with a Web Map Service, after the GeographicLocation command defines the geographic location in the drawing.

MassProp reports the area, perimeter, and other mathematical properties of 3D solids and 2D regions (short for "mass properties").

MatBrowserClose and **MatBrowserOpen** close and open the materials browser.

MatchPerspective changes the viewpoint in perspective mode to match a background image.

MatchProp assigns the properties of one entity to one or more other entities (short for "match properties").

MaterialAssign assigns materials from the Material Browser onto 3D objects

MaterialMap maps material definitions onto the surfaces of objects, with presets for boxes, planes, spheres, and cylinders.

Materials creates materials and edits their properties through the Drawing Explorer.

MatLib displays the Rendering Materials panel.

Measure places points or blocks along entities.

MechanicalBrowserClose closes the Mechanical Browser panel.

MechanicalBrowserOpen displays the Mechanical Browser panel.

Menu loads menu files to modify the user interface.

MenuLoad and **MenuUnload** load and unload CUIX and CUI (user interface customization), MNU (menu), MNS (LISP code), and ICM (IntelliCAD menu) files.

Midpoint toggles Midpoint entity snap; snaps to the middle of lines, arcs, and other open entities.

MInsert inserts a block as a rectangular array; combines the -Insert and Array commands (short for "multiple insertion").

Mirror draws mirror image copies of entities.

Mirror3D draws mirror images of entities about a plane in 3D space.

MLeader creates multileader entities using the current multileader style.

MLeaderAlign aligns multiple leaders

MLeaderCollect collects multiple leader blocks

MLeaderEdit adds leader lines to and removes leader lines from a multileader entity.

MLeaderEditExt adds and removes leader lines, adds and removes vertices from a multileader entity.

MLeaderStyle creates and manages multileader styles through the Drawing Explorer.

MLine draws multilines.

MLStyle creates and edits multiline styles.

ModelerProperties and **-ModelerProperties** controls the various settings of the ACIS modeler through the Settings dialog box.

Move displaces entities a specified distance in a specified direction.

MoveEData moves extended entity data from one entity to another.

MSlide makes SLD (slide) files from the current view.

MSpace switches to model space inside a viewport of layout tab.

MText and **-MText** opens the multi-line text editor interface for placing paragraph text.

Multiple command prefix forces commands to repeat themselves automatically.

MView creates viewports in layout tab.

MvSetup prepares sets of paper space viewports; superseded by the ViewBase command.

MTP snaps to the midpoint between two points.

N Commands

Navigate walks and flys through 3D models.

Nearest toggles Nearest entity snap mode; snaps to the nearest geometry on entities.

NetLoad loads .NET applications.

New starts new drawing files.

NewSheetSet creates a new sheet set.

NewWiz starts new drawings with the New Drawing Wizard.

Node toggles Node entity snap mode; snaps to point entities.

None turns off all entity snap modes.

Number adds incremented number tags for BIM entities

O Commands

ObjectScale and **-ObjectScale** adds or removes supported scales for annotative entities.

Offset offsets linear entities in parallel orientation.

OleLinks adjusts links of OLE entities embedded in or linked to drawings (short for "object linking and embedding").

OleOpen opens OLE objects for modification.

OnWeb opens the Bricsys home page in your computer's default Web browser.

Oops un-erases the last erased entity, including those erased by the Block command.

Open opens an existing drawing file.

OpenSheetSet and **-OpenSheetSet** open an existing sheet set.

Options configures program operating parameters.

Orthogonal constrains the pointer so it moves parallel to the axes of the current coordinate system.

OSnap and **-OSnap** sets entity snaps through the Settings dialog box or the command line (short for "object snap").

Overkill and **-Overkill** deletes duplicate entities and overlapping lines, arcs or polylines and unifies partly overlapping or contiguous ones.

P Commands

PageSetup creates and edits page setups for plotting drawings in the Drawing Explorer.

Pan and **-Pan** moves the drawing display in the active view tile.

Panelize command draws freeform surfaces as subdivision meshes, optionally planarizing the panels.

Parallel turns on parallel entity snap.

-Parameters create and edit constraint expressions and values.

ParametersPanelOpen opens the Parameters panel.

ParametersPanelClose closes the Parameters panel.

Parameterize adds constraints and parameters to models automatically.

ParametricBlock creates a parametric block from entities in the drawing; useful for BEdit.

PasteBlock inserts data from the Clipboard as block.

PasteClip inserts data from the Clipboard.

PasteOrig pastes entities from the clipboard at the coordinates from the source drawing.

PasteSpec pastes entities from the clipboard, after the user specifies the format.

PdfAdjust adjust the fade, contrast and monochrome settings of PDF underlays.

Pdfattach and **-PdfAttach** attaches PDF files as underlays into the drawing.

PdfClip clips PDF underlays.

PdfImport And **-PdfImport** imports PDF files and converts them to drawing entities.

PdfLayers controls the display of layers in PDF underlays.

PdfOptions controls the exporting of drawings in PDF format through the Settings dialog box.

PEdit edits polylines, 3D polylines, and 3D meshes (short for "polyline edit").

PEditExt edits vertices and segments of a polyline.

Perpendicular toggles perpendicular entity snap mode.

PFace draws 3D multi-sided meshes; meant for use by programs (short for "polyface mesh").

Plan sets plan view to construction plane.

PLine draws polyline lines, arcs, and splines with optional width (short for "polyline").

Plot and **-Plot** both execute the plot command at the command line.

PlotStamp specifies a header and footer for plotted output.

PlotStyle sets the current plot style; works only when plot styles are enabled in drawings.

PlotterManager creates customized parameter PC3 files for printers and other output devices; executes the PlotConfig.exe utility program.

Point draws point entities.

PointCloud displays the Point Cloud section of the Drawing Explorer

PointCloudAttach and **-PointCloudAttach** attach Bricsys-format point cloud files to the current drawing

PointCloudColorMap changes the colors of point based on their elevation.

PointCloudCrop and **PointCloudUncrop** crop the extents of the current point cloud, and undo the cropping.

PointCloudPointSize specifies the size of points in a point cloud.

PointCloudPointSize_Minus decreases the size of points in a point cloud.

PointCloudPointSize_Plus increases the size of points in a point cloud.

PointCloudPreprocess and **-PointCloudPreprocess** convert ASCII PTS, PTX, LAS, and other cloud files into the compressed binary Bricsys file format.

PointLight places point lights in drawings.

Polygon draws equi-sided polygons from polylines of 3 to 1,024 sides.

PolySolid creates 3D wall-like solids.

Preview shows a preview before printing the drawing.

Print plots the drawing to a plotter, printer, or file.

ProfileManager sets current, create, copy, delete, import and export user profiles.

ProjectGeometry projects geometry like curves, and edges onto regions, surfaces, and 3D solids.

Properties displays the Properties palette to change drawing entity properties.

PropertiesClose closes the Properties palette.

PSetupIn and **-PSetupIn** imports page setup definitions from another drawing.

PSpace switches from model to paper space (short for "paper space").

Publish and **-Publish** prints sheet lists of model space or paper space layouts; saves a sheet list to a file.

Purge and **-Purge** remove unused named entities from drawings, such as unused layers and linetypes.

Pyramid draws three-dimensional solid pyramids.

Q Commands

QLeader draws leaders; specifies properties through a dialog box.

QNew opens new drawings in BricsCAD (short for "quick new").

QPrint prints the drawing with the default plot configuration, without displaying the Print dialog box (short for "quick print").

QSave saves the drawing without displaying the Save dialog box (short for "quick save").

QSelect composes a selection set using filters.

QText toggles the display of text as rectangles (short for "quick text").

Quadrant toggles snaps to quadrant points of circles, arcs, and polyarcs.

Quick toggles snaps to the first entity geometry found; used together with at least one other entity snap mode.

Quit ends BricsCAD; optionally saves unsaved drawings.

R Commands

Ray draws semi-infinite construction lines

ReAssocApp associates extended entity data with applications (short for "reassociate application").

Recover repairs damaged drawings.

RecScript records keystrokes to an SCR file for playback with the Script command (short for "record script").

Rectang draws a rectangular polyline.

Redefine restores built-in commands that have been undefined using the Undefine command.

Redo reverses the effects of a previous U command.

Redraw refreshes the display of the active view tile.

RedrawAll refreshes the display of all currently-open view tiles.

RedSdkInfo reports on rendering related hardware and driver specifications (short for "Red software development kit information").

RefClose closes the in-situ block and xref editor.

RefEdit and **-RefEdit** edits blocks and externally-referenced drawings (short for "reference editor").

RefSet adds and removes entities from the block or external reference being edited.

Regen regenerates the current viewport.

RegenAll regenerates all viewports.

RegenAuto determines when BricsCAD regenerates the drawing automatically.

Region converts an entity enclosing an area into a region.

ReInit reloads the PGP alias file (short for "re-initialize").

Rename and **-Rename** changes the names of objects.

Render and **-Render** generates photorealistic renderings od 3D models using materials and lights.

RenderPresets creates and edits rendering presets, and to set the current render preset.

ReportPanelClose and **ReportPanelOpen** close and open the Report panel.

ResetAssocViews removes associative data from blocks

ResetBlock resets dynamic blocks to their default values.

Resume resumes an interrupted script.

RevCloud draws revision clouds commonly used for red-lining drawings.

Revolve draws 3D solids or surfaces by revolving 2D objects about an axis.

RevSurf creates 3D mesh surfaces by revolving open entities around a axis (usually a line).

Ribbon displays the ribbon user interface.

RibbonClose closes the ribbon.

Rotate rotates entities about a base point.

Rotate3D moves entities about a 3D axis.

RScript reruns the currently loaded SCR script file (short for "repeat script").

RtLook moves the viewpoint through a 3D scene (short for "real time looking").

RtPan pans the view in real time.

RtRot, **RtRotCtr**, or **RtRotF** rotate the viewpoint in real time.

RtRotX, **RtRotY**, or **RtRotZ** rotates the 3D viewpoint about the x, y, or z axis in real time.

RtUpDown tilts the viewpoint up, down, left, or right in real time.

RtWalk walk lefts, right, forward or backward through 3D scenes in real time.

RtZoom zooms into the drawing in real time.

RuleSurf draws ruled surfaces between two curves.

S Commands

Save saves the drawing under the current file name or a specified name.

SaveAll saves all open drawings.

SaveAs saves an unnamed drawing with a file name or renames the current drawing.

SaveAsR12 saves drawings in DWG R12 format.

SaveFileFolder opens the File Explorer to the folder in which the current drawing is saved.

Scale enlarges or reduces specified entities equally in the X, Y, and Z directions.

ScaleListEdit and **-ScaleListEdit** edits the list of scale factors used by annotative scaling, sheet scales, and plot scales.

Script loads and runs SCR script files.

Scrollbar toggles the display of the horizontal and vertical scroll bars.

Section creates a cross section based on the intersection of a plane and 3D solids**.**

SectionPlane creates a section entity that creates sections of 3D solids.

SectionPlaneSettings defines the properties of section plane entities in the Drawing Explorer**.**

SectionPlaneToBlock saves the selected section plane as a 2D cross section / elevation block or a 3D cutaway section block.

Security determines whether VBA macros can run automatically; not available in the 64-bit version.

SecurityOptions sets a password to protect the drawing.

Select places selected entities in the 'Previous' selection set.

SelectAlignedFaces selects all faces in a model which are coplanar with a selected face.

SelectAlignedSolids selects all solids in a model of which a face is coplanar with a selected face.

SelectConnectedFaces selects all faces in a model which are connected to a selected face.

SelectConnectedSolids selects all solids in a model which are connected to a selected face.

SelectSimilar selects entities of the same type and properties.

SelGrips prompts to selects entities and then displays grips.

Settings displays the Settings dialog box for changing the values of variables.

SettingsSearch opens the Settings dialog box at the specified category, variable name, or user preference.

SetUCS sets the UCS to a viewpoint specified through a dialog box.

SetVar displays and changes the values of system variables (short for "set variables").

Sh and **Shell** open the Windows command prompt window; runs other applications (short for "shell").

Shade shades the drawing mode.

ShadeMode sets the current visual style at the command line, such as Realistic, Conceptual, Edges, and X-ray.

-ShadeMode sets the old type of shade modes: 2D, 3D, Hidden, Flat, Flat with Edges, Gouraud, and Gouraud with edges.

Shape places shapes from SHX files in drawings.

SheetSet and **SheetsetHide** manage sheet sets, and closes the Sheet Set pane.

Singleton toggles whether multiple copies of BricsCAD can run at the same time.

Site imports terrain models from points and Civil 3D surfaces, or creates them from entities.

SiteEdit edits terrain sites.

Sketch draws freehand lines.

Slice slices 3D solids with a plane or surface.

Snap restricts pointer movements and pointing in the drawing to specified intervals.

Solid draws solid-filled 2D faces.

SolidEdit edits 3D solids and 2D regions.

SolProf creates hidden line representations of 3D solids in a layout viewport.

Spell checks the spelling of text in the drawing.

Sphere draws three-dimensional solid spheres.

Spline draws quadratic or cubic non-uniform rational Bezier spline (NURBS) curves.

SpotLight inserts spot lights into drawings.

Screenshot takes a screen grab of the current space, excluding all UI elements.

Start runs operating system applications.

StatBar toggles the display of the status bar.

Status reports status of the drawing's settings in the Text window.

StlOut export 3D models in STL format for 3D printing (short for "stereolithography").

StopScript stops recording of scripts begun with the RunScript command.

Stretch moves or stretches entities.

StandardPartsPanelClose and **StandardPartsPanelOpen** close and open the Standard Parts panel.

StructurePanel and **StructurePanelClose** open and close the Structure panel displaying tree structure of the drawing content.

+StructurePanel opens a CST structure tree configuration file.

Style and **-Style** creates and edits text styles through the Drawing Explorer.

StylesManager creates and attaches plot style files.

Subtract creates a composite region or a 3D solid by subtraction.

SunProperties edits sun properties through the Drawing Explorer.

SupportFolder opens the *C:\Users\<login>\AppData\Roaming\Bricsys\BricsCAD\V20x64\en_US\Support* folder.

SvgOptions controls the output as SVG files.

Sweep creates solid primitives or surfaces by sweeping two dimensional entities along a path.

SysWindows arranges windows.

Sheet Metal Commands

(Available for Mechanical edition; requires an additional license; sm = sheet metal)

LicPropertiesSheetmetal reports the license state of the sheet metal module.

smAssemblyExport converts 3D solid sheet metal parts to DXF files with unfolding information.

smBendCreate converts hard edges (sharp edges between flange faces) into bends.

smBendSwitch converts bends to lofted bends.

smConvert automatically recognizes flanges and bends in a 3D solid.

smDelete removes a bend or a junction by restoring the hard edge between two flanges; removes a flange with all the bends adjacent to it.

smDissolve removes sheet metal data from the selected features.

smExport2D exports unfolded representations of sheet metal bodies as 2D profiles in DXF or DWG files.

smExportOSM exports sheet metal solids to OSM files (short for "Open Sheet Metal") used by CADMAN-B CAM systems.

smExtrude extrudes polylines to sheet metal parts.

smFlangeBase creates base (initial) flanges of sheet metal parts from closed 2D entities.

smFlangeBend bends existing flanges along a line, taking into account the k-factor.

smFlangeConnect closes gaps between two arbitrarily oriented flanges.

smFlangeContour creates flange from a closed contour.

smFlangeEdge creates one or more flanges to a sheet metal part by pulling one or more edges of an existing flange.

smFlangeRotate rotates a selected flange of a sheet metal part with automatic selection of the rotation axis depending on the design intent.

smFlip switches flange sides to reverse reference faces.

smForm adds forms to sheet metal.

smHemCreate creates a variety of hems on sheet metal models.

smImprint uses imprinted edges to split thickness faces of sheet metal parts.

smJunctionCreate converts hard edges (sharp edges between flange faces) and bends into junctions.

smJunctionSwitch changes symmetrical junction features to overlapping faces.

smLispGet returns values related to sheet metal variables.

smLispSet changes values related to sheet metal variables.

smLoft creates sheet metal part with lofted bends and flanges from two non-coplanar curves.

smParametrize generates consistent sets of 3D constraints for sheet metal parts

smReliefCreate creates proper corner (three or more adjacent flanges) and bend reliefs (at the start and end of a flange edge).

smRepair restores the 3D solid model of a sheet metal part by thickening one of its sides: all thickness faces become perpendicular to flange faces.

smReplace replacing form features with ones from libraries.

smRibCreate adds associative rib (form) features on sheet metal parts based on 2D profiles.

smSelect selects hard edges and form features of sheet metal parts.

smSplit splits flanges and lofted bend; replaces the old smFlangeSplit command.

smTabCreate creates a tab between two flanges.

smUnfold generates unfolded 2D or 3D representations of sheet metal parts.

T Commands

Table and **-Table** draws tables in drawings.

Tabledit edits text in table cells.

TableExport exports the contents of a table entity to CSV (command separated values) files.

TableMod modifies the properties of table cells.

TableStyle creates and manages table styles through the Drawing Explorer.

Tablet configures and calibrates tablets, and toggles tablet mode (WINDOWS MODE).

TabSurf draws tabulated surfaces from a path curve and a direction vector.

Tangent toggles tangent entity snap; snaps to the tangency of circles, arcs, ellipses and elliptical arcs.

TConnect connects solids by their faces

TemplateFolder opens the *C:\Users\<login>\AppData\Local\Bricsys\BricsCAD\V20x64\en_US\Templates* folder.

Text and **-Text** places lines of text in the drawing.

TextScr displays the text window showing command history (short for "text screen").

TextToFront sets the draw order of all texts and dimensions in the drawing to display in front of all other entities.

TfLoad and **TfSave** open and save handle, xsd, and strip data from DWT template files.

Time reports on the time spent in the drawing.

TInsert inserts blocks in the cells of tables.

Tolerance draws tolerances (datum indicators and basic dimension notation).

Toolbar and **-Toolbar** displays and hides toolbars.

ToolPalettes opens the Tool Palettes bar.

ToolPalettesClose closes the Tool Palettes bar.

-ToolPanel opens tool panels by name at the command bar.

Torus draws three-dimensional torrid solids.

TpNavigate opens tool palettes or group at the command bar.

Trace draws traces.

Transparency toggles the transparency of monotone images; has nothing do with the transparency property.

Trim trims entities at a cutting edge defined by other entities.

TxtExp explodes text into polyline segments.

U Commands

U reverses the most recent command.

Ucs creates and displays named UCSes through the command bar (short for "user-defined coordinate system").

UcsIcon toggles the display of the UCS icon.

Undefine disables built-in commands.

Undo restores deleted entities.

UndoEnt undoes property changes to selected entities.

Union creates composite regions or solids by addition.

UnisolateObjects makes entities visible again following the IsolateObjects and HideObjects commands.

Units and **-Units** sets coordinate and angle display formats and precision.

UpdateField forces the values of field text to update.

Url opens the default Web browser (short for "uniform resource locator").

V Commands

View and **-View** saves, restores, and manages user-defined model and sheet views, and presets views.

ViewHorizontal rotates the viewpoint to make z=0 (horizontal)

ViewLabel adds labels to views; available through the Sheet Set manager only.

ViewRes sets the view resolution and toggles fast-zoom mode (short for "view resolution").

VisualStyles and **-VisualStyles** creates and edits visual style definitions in the Drawing Explorer or at the command line.

VmlOut exports drawings in VML format embedded in Web pages (short for "vector markup language").

VpClip clips viewports in layouts (short for "view port clipping").

VpLayer changes the properties of layers in the current paper space viewport (short for "view port layer").

VpMax and **VpMin** maximize and minimize the current viewport in paper space.

VPoint Changes the 3D viewpoint through a dialog box.

VPorts and **-VPorts** create one or more viewports in model space (short for "viewports").

VSlide displays images saved as SLD or WMF files (short for "view slide").

VBA Commands

(Available in Pro and Platinum editions only; vba = Visual Basic for Applications)

VbaIde opens the BLADE editing window; short for "integrated development environment".

VbaLoad and **-VbaLoad** loads VBA projects.

VbaMan manages VBA projects; short for "manager".

VbaRun and **-VbaRun** runs, creates, edits, and deletes VBA macros.

VbaSecurity sets the security level for running VBA macros.

VbaUnload unloads VBA projects.

ViewBase Commands

(Available in Pro and Platinum editions only)

ViewBase generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.

ViewDetail creates a detail view of a portion of a standard generated drawing at a larger scale.

ViewDetailStyle specifies the visual format of detail views and detail symbols.

ViewEdit changes the scale and hidden line visibility of drawing views; works in paper space only.

ViewExport exports the content of drawing views to Model space or to a new drawing; operates in paper space only.

ViewProj generates additional projected views from an existing drawing view.

ViewSection creates cross section views based on standard drawing views generated by the ViewBase command in paper space layouts.

ViewSectionStyle specifies the visual format of section views and section lines.

ViewUpdate updates drawing views.

W Commands

WBlock and **-WBLock** export blocks, selected entities, or the entire drawing as a DWG file.

WCascade, **WClose**, **WCloseAll**, **WNext**, and **WPrev** cascade the windows, close the current window, close all windows, and switch to the next or previous windows.

Weblight places Web lights.

Wedge draws three-dimensional solids with a sloped face tapering along the X axis.

WhoHas reports the ownership of a drawing file.

WhTile, **WiArrange**, and **WvTile** tiles windows horizontally, in an overlapping manner, or vertically.

WipeOut creates blank areas in drawings.

WmfOut exports the drawing in WMF (WIndows meta file), EMF (enhanced meta file), or SLD (slide) format.

WorkSets creates and loads named sets of drawing files.

Workspace sets the current workspace; creates, modifies, and saves workspaces.

WsSaveAs saves the current user interface by name.

WsSettings opens the Customize dialog box at the Workspace tab.

X Commands

XAttach attaches externally-referenced drawings.

XClip clips externally-referenced drawings.

XEdges extracts edges from 3D solids as lines.

XLine draws infinitely long lines.

XmlSave prompts for handles to save in an XML file.

XOpen opens externally-referenced drawings in a new window.

Xplode explodes entities, and provides control over the resulting entities.

XRef and **-XRef** attaches DWG files to the current drawing through the Drawing Explorer or the command line.

Z Commands

Zcenter toggles the 3D center entity snap; snaps to the center of planar or curved 3D faces.

Zknot toggles the 3D knot entity snap; snaps to a knot on a spline.

Zmidpoint toggles the 3D midpoint snap; snaps to the midpoint of a face edge.

Znearest toggles the 3D nearest entity snap; snaps to a point on the face of a 3D entity that is nearest to the cursor.

Znone disables all 3D snap modes.

Zoom increases or decreases the visible part of the drawing.

Zperpendicular toggles the 3D perpendicular entity snap; snaps to a point perpendicular to a face.

Zvertex toggles the 3D vertex entity snap; snaps to the closest vertex of a 3D entity.

# Commands

? displays the Help window.

2dIntersection toggles apparent intersection entity snap; snaps to the intersections of entities, even when they only appear to intersect in 3D space.

3D draws 3D polygon mesh objects: boxes, cones, cylinders, dishes, domes, pyramids, spheres, tori, wedges, or meshes.

3DArray constructs 3D rectangular arrays and rotated polar arrays.

3DCompare compares the 3D content of two drawing files.

3DConvert converts 3D solids to polyface meshes.

3DFace draws 3D 4-edged faces with optional invisible edges.

3DIntersection toggles Intersection entity snap; snaps to the intersections of entities.

3DMesh draws 3D surface meshes.

3DOsnap and **-3DOsnap** sets the entity snap modes for 3D entities through the Settings dialog box.

3DPoly draws 3D polylines.

APPENDIX B

Concise Summary of Variables & Settings

BRICSCAD USES VARIABLES TO STORE AND REPORT SETTINGS AFFECTING THE PROGRAM and drawings. There are two types of variables: *system* variables that mimic the names and values from AutoCAD, and *preference* variables unique to BricsCAD. You access and change variables through a dialog box (**Settings** command) or directly on the command line (**SetVar** command).

This appendix lists over 1,000 variable names in alphabetical order.

UPPERCASE text indicates the name is also found in AutoCAD as a system variable

MixedCase text means the variable is a *preference*, and so is unique to BricsCAD

Blue text indicates that the variable is new in V20

~~**ikeThrough**~~ text indicates the variable was removed from BricsCAD

userid or *login* refers to your computer login name

When you see **Read-only** (r/o), it means that you cannot change the variable's value; the value has been set by BricsCAD or by the operating system.

A Variables

Variable Name	Read-only	Default Value
ACADLSPASDOC		0
ACADPREFIX	r/o	“C:\Users\userid\AppData\Roaming\Bricsys\BricsCAD\V20x64\en_US\Support\; C:\Program Files (x86)\Bricsys\BricsCAD V20x64**Support**\,; C:\Program Files (x86)\Bricsys\BricsCAD V20x64**Fonts**\; C:\Program Files (x86)\Bricsys\BricsCAD V20x64**Help**\en_US\”
ACADVER	r/o	“20.0 BricsCAD”
AcisHlrResolution		-1
ACISOUTVER		70
AcisSaveAsMode		0
AdaptiveGridStepSize		4.0000
AFLAGS		0
ALLOWBREAKLINECROSSINGS		“1”
ALLOWEDBENDANGLES		“1”
AllowTabExternalMove		1
AllowTabMove		1
AllowTabSplit		1
ANGBASE		0
ANGDIR		0
Anglesamplinginterval		“5”
ANNOALLVISIBLE		1
ANNOAUTOSCALE		-4
AnnoSelected	r/o	0
ANNOTATIVEDWG		0
AntiAliasRender		2
AntiAliasScreen		1
APBOX		0
ArcTessellation		"0.01"
APERTURE		10
AREA	r/o	0
AREAPREC		-1
AREAUNITS		“in ft mi µm mm cm m km”
ARRAYASSOCIATIVITY		1
ARRAYEDITSTATE	r/o	0
ARRAYTYPE		0
Associativity		“3”
ATTDIA		0
ATTMODE		1
AttractionDistance		3
ATTREQ		1
AUDITCTL		0
AuditErrorCount	r/o	0
AUNITS		0
AUPREC		0
AutoAdoptSizes		"1"
AUTOCOMPLETEDELAY		0.3
AUTOCOMPLETEMODE		47
AutoFlipQuarterTurn		"1"
AutomaticConnection		“1”
AutomaticTees		"0"
AUTOMENULOAD		1
AutoResetScales		0
AutosaveChecksOnlyFirstBitDBMOD		1

Variable Name	Read-only	Default Value
AUTOSNAP		119
AutoTrackingVecColor		171
AutoUpdateRooms		“1”
AutoVpFitting		1
AXISMODE		0
AXISUNIT		X= 0 Y= 0 Z= 0

B Variables

BACKGROUNDPLOT		2
BActionColor		"7"
BACKZ	r/o	0
BASEFILE		“Default-mm.dwt”
BDependencyHighlight		1
bimConnectCutType		“0”
bimMatchProp		"1"
BIMOSMODE		0
BINDTYPE		0
BKGCOLOR		7
BKGCOLORPS		7
BLIPMODE		0
BLOCKEDITLOCK		0
BLOCKEDITOR		0
BlocksPath		“C:\Users\userid\Documents\”
bmAutoUpdate		1
bmForceUpdateMode		0
bmFormTemplatePath		""
bmReportPanel		0
BMUPDATEMODE		0
BndLimit		1000
BoundaryColor		95
BParameterSize		12
BpTextHorizontal		1
BSysLibCopyOverwrite		0
BtMarkDisplay		1
BVMODE		0

C Variables

CACHELAYOUT		1
CAMERADISPLAY		0
CAMERAHEIGHT		0
CANNOSCALE		“1:1”
CANNOSCALEVALUE	r/o	1
CDATE	r/o	20160211.15522
CECOLOR		“ByLayer”
CELTSCALE		1
CELTYPE		“ByLayer”
CELWEIGHT		-1
CenterCrossGap		“0.05x”
CenterCrossSize		“0.1x”
CenterExe		0.1200
CenterLayer		“.”
CenterLtscale		1.0000
CenterLtype		“Center2”

Variable Name	Read-only	Default Value
CENTERLTYPEFILE		“Default.Lin”
CenterMarkExe		1
CETRANSPARECNY		“ByLayer”
CGEOCS	r/o	“”
CHAMFERA		0
CHAMFERB		0
CHAMFERC		0
CHAMFERD		0
CHAMMODE		0
~~*ChapooLog*~~		
~~*ChapooLogVerbose*~~		
~~*ChapooOnModified*~~		
~~*ChapooServer*~~		
~~*ChapooTempFolder*~~		
~~*ChapooUploadDependencies*~~		
~~*ChapooWebsite*~~		
CheckDwlPresence		0
CIRCLERAD		0
CLAYER		“0”
CLEANSCREENOPTIONS		15
CLEANSCREENSTATE	r/o	0
ClipBoardFormat		1
CLIPBOARDFORMATS		127
CliPromptLines		4
CLISTATE	r/o	1
CloseChecksOnlyFirstBitDBMOD		0
CloudDownloadPath		"c:\users\userid\documents\bricsys247\"
CloudLog		0
CloudLogVerbose		0
CloudOnModified		1
CloudServer		“https://my.bricsys247.com/”
CloudSessionPath		"c:\users\userid\appdata\local\bricsys\bricscad\"
CloudTempFolder		“C:\Users\userid\AppData\Local\Temp\Bricsys_24_7\”
CloudUploadDependencies		1
CMATERIAL		“ByLayer”
CMDACTIVE	r/o	1
CMDDIA		1
CMDECHO		1
CmdLineEditBgColor		“#fefefe”
CmdLineEditFgColor		“#202020”
CmdLineFontName		“Consolas”
CmdLineFontSize		10
CmdLineListBgColor		“#ecf1ff”
CmdLineListFgColor		“#000000”
CMDLNTEXT		“:”
CMDNAMES	r/o	“SETTINGS”
CMLEADERSTYLE		“Standard”
CMLJUST		0
CMLSCALE		1
CMLSTYLE		“Standard”
CMPCLRMISS		1
CMPCLRMOD1		253
CMPCLRMOD2		2
CMPCLRNEW		3

Variable Name	Read-only	Default Value
CMPDIFFLIMIT		1000
CmpFadeCtl		80
CmpLog		0
ColorTheme		0
COLORX		11
COLORY		112
COLORZ		150
COMAcadCompatibility		0
COMPASS		0
Componentspath		“C:\Users\userid\AppData\Roaming\Bricsys\BricsCAD\V20x64\en_US\Support\Bim\Components\”
CONSTRAINTBARDISPLAY		3
ContinuousMotion		0
ConvertToArrays		1
COORDS		1
COPYMODE		0
CPLOTSTYLE		“ByColor”
CPROFILE	r/o	“Default”
CreateThumbnailOnTheFly		1
CREATEVIEWPORTS		1
CrosshairDrawMode		2
CROSSINGAREACOLOR		91
CTAB		“Model”
CTABLESTYLE		“Standard”
Ctrl3DMouse		1
CtrlMButton		1
CTRLMOUSE		1
CURSORSIZE		3
CVPORT		2

D Variables

Variable Name	Read-only	Default Value
DataLinkNotify		2
DATE	r/o	2456335.6613464
DBCSTATE	r/o	0
DBLCLKEDIT		1
DBMOD	r/o	0
DCTCUST		“”
DCTMAIN		“en_US.dic”
ddBetweenKnots		2
ddFastMode		0
ddGridAspectRatio		0
ddMaxFacetEdgeLength		0
ddMaxNumGridLines		10000
ddNormalTol		15
ddPointsPerEdge		0
ddSurfaceTol		0
ddUseFacetRES		1
DefaultBlockName		0
DefaultInsPoint		0
DEFAULTLIGHTING		0
DefaultLightShadowBlur		8
DefaultNewSheetTemplate		“”
~~DefaultRoomHeight~~ replaced by RoomHeight		
DEFLPLSTYLE		“Normal”

Variable Name	Read-only	Default Value
DEFPLSTYLE		“ByColor”
DeleteInterference		1
DeleteTool		1
DELOBJ		1
DEMANDLOAD		3
DGNFRAME		2
dgnImp2dClosedBSplineCurveImportMode		0
dgnImp2dEllipseImportMode		0
dgnImp2dShapeImportMode		0
dgnImp3dClosedBSplineCurveImportMode		1
dgnImp3dEllipseImportMode		0
dgnImp3dObjectImportMode		0
dgnImp3dShapeImportMode		1
dgnImpBreakDimensionAssociation		0
dgnImpConvertDgnColorIndicesToTrueColors		0
dgnImpConvertEmptyDataFieldsToSpaces	1	
dgnImpEraseUnusedResources		0
dgnImpExplodeTextNodes		0
dgnImpImportActiveModelToModelSpace		1
dgnImpImportInvisibleElements		1
dgnImpImportPaperSpaceModels		1
dgnImpImportViewIndex		-1
dgnImpRecomputeDimensionsAfterImport		0
dgnImpSymbolResourceFiles		“”
dgnImpXRefImportMode		2
DGNOSNAP		1
DIASTAT	r/o	0
DisplayAxes		“0”
DisplayAxesForMep		“0”
DisplayScaling	r/o	125
DisplaySidesAndEnds		“1”
DisplaySnapMarkerInAllViews		0
DisplayTooltips		1
DISPPAPERBKG		1
DISPPAPERMARGINS		1
DISPSILH		0
DISTANCE	r/o	0
dmAuditLevel		1
DMAUTOUPDATE		1
dmConnectionCutType		0
dmExtrudeMode		0
dmPushPullSubtract		1
DMRECOGNIZE		0
DockPriority		1
DocTabPosition		0
DONUTID		0.5
DONUTOD		1
DRAGMODE		2
DragModeHide		0
DRAGMODEINTERRUPT		1
DRAGOPEN		1
DRAGP1		10
DRAGP2		25
DRAGSNAP		0

Variable Name	Read-only	Default Value
DrawingPath		“C:\Users\userid\Documents\”
DrawingViewPreset		“none”
DrawingViewPresetHidden		0
DrawingViewPresetScale		“”
DrawingViewpPesetTangent		0
DrawingViewPresetScale		""
DrawingViewPresetTangent		0
DRAWORDERCTL		3
DxfTextAdjustAlignment		0
DWFFRAME		2
DWFOSNAP		1
DwfVersion		2
DWGCHECK		0
DWGCODEPAGE	r/o	“ANSI_1252”
DWGNAME	r/o	“Drawing1.dwg”
DWGPREFIX	r/o	“C:\Program Files (x86)\Bricsys\BricsCAD V20x64\”
DWGTITLED	r/o	0
DXEVAL		12
DxfTextAdjustAlignment		0
DYNCONSTRAINTMODE		1
DYNDIGRIP		31
DynDimAperture		20
DynDimColorHot		142
DynDimColorHover		142
DynDimDistance		1
DynDimLineType		1
DYNDIVIS		1
DynInputTransparency		65
DYNMODE		3
DynPiCoords		0

Dimensions Variables

DIMADEC		0
DIMALT		0
DIMALTD		2
DIMALTF		25.4
DIMALTRND		0
DIMALTTD		2
DIMALTTZ		0
DIMALTU		2
DIMALTZ		0
DIMANNO	r/o	0
DIMAPOST		“”
DIMARCSYM		0
DIMASO		1
DIMASSOC		2
DIMASZ		0.18
DIMATFIT		3
DIMAUNIT		0
DIMAZIN		0
DIMBLK		“”
DIMBLK1		“”
DIMBLK2		“”
DIMCEN		0.09

Variable Name	Read-only	Default Value
DIMCLRD		0
DIMCLRE		0
DIMCLRT		0
DIMDEC		4
DIMDLE		0
DIMDLI		0.38
DIMDSEP		“0”
DIMEXE		0.18
DIMEXO		0.0625
DIMFIT		3
DIMFRAC		0
DIMFXL		1
DIMFXLON		0
DIMGAP		0.09
DIMJOGANG		0.7853981634
DIMJUST		0
Dimlayer		“.”
DIMLDRBLK		“”
DIMLFAC		1
DIMLIM		0
DIMLTEX1		“”
DIMLTEX2		“”
DIMLTYPE		“”
DIMLUNIT		2
DIMLWD		-2
DIMLWE		-2
DIMPOST		“”
DIMRND		0
DIMSAH		0
DIMSCALE		1
DIMSD1		0
DIMSD2		0
DIMSE1		0
DIMSE2		0
DIMSHO		1
DIMSOXD		0
DIMSTYLE	r/o	“Standard”
DIMTAD		0
DIMTDEC		4
DIMTFAC		1
DIMTFILL		0
DIMTFILLCLR		“BYBLOCK”
DIMTIH		1
DIMTIX		0
DIMTM		0
DIMTMOVE		0
DIMTOFL		0
DIMTOH		1
DIMTOL		0
DIMTOLJ		1
DIMTP		0
DIMTSZ		0
DIMTVP		0
DIMTXSTY		“Standard”

Variable Name	Read-only	Default Value
DIMTXT		0.18
DIMTXTDIRECTION		0
DIMTZIN		0
DIMUNIT		2
DIMUPT		0
DIMZIN		0

E Variables

EDGEMODE		0
ELEVATION		0
ElevationAtBreaklineCrossings		“0”
EnableAttraction		1
EnableBimBkUpdate		"0"
EnableHyperlinkMenu		1
EnableHyperlinkTooltip		0
ERRNO		0
EXPERT		0
ExpInsAlign		0
ExpInsAngle		0
ExpInsFixAngle		1
ExpInsFixScale		1
ExpInsScale		1
EXPLMODE		1
ExportAcisFormatVersion		"0"
ExportCatiav4FormatVersion		"0"
ExportCatiav5FormatVersion		"0"
ExportHiddenParts		“0”
EXPORTMODELSPACE		0
EXPORTPAGESETUP		0
EXPORTPAPERSPACE		0
ExportParasolidFormatVersion		"0"
ExportProductStructure		“1”
ExportStepFormatVersion		“1”
EXTMAX	r/o	-1.0000E+20,-1.0000E+20,-1.0000E+20
EXTMIN	r/o	1.0000E+20,1.0000E+20,1.0000E+20
EXTNAMES		1

F Variables

FACETRATIO		0
FACETRES		0.5
fbxExportCameras		1
fbxExportEntities		1
fbxExportEntitiesSelType		0
fbxExportLights		1
fbxExportMaterials		1
fbxExportTextures		0
fbxExportTexturesPath		"c:\program files\bricsys\bricscad v20 en_us\"
FEATURECOLORS		1
FIELDDISPLAY		1
FIELDEVAL		31
FILEDIA		1
FILLETRAD		0.5
FILLMODE		1

Variable Name	Read-only	Default Value
FittingRadiusType		“0”
FittingRadiusValue		“1.5”
FLATLAND		Off
FONTALT		“simplex.shx”
FONTMAP		“default.fmp”
FRAME		3
FRAMESELECTION		0
FRONTZ	r/o	0
FULLOPEN	r/o	1

G Variables

GDIOBJECTS	r/o	3768
GearteethNumber		“1”
GENERATEASSOCVIEWS		0
GEOLATLONGFORMAT		1
GEOMARKERVISIBILITY		1
GeomRelations		0
GetStarted		1
GfAng		0.0000
GfClr1		“5”
GfClr2		“7”
GfClrLum		1.0000
GfClrState		0
GfName		1
GfShift		0
GLSWAPMODE		2
GradientColorBottom		“#d2d2d2”
GradientColorMiddle		“#fafafa”
GradientColorTop		“#ffffff”
GradientMode		“0”
GRIDAXISCOLOR		252
GRIDDISPLAY		3
GRIDMAJOR		5
GRIDMAJORCOLOR		253
GRIDMINORCOLOR		254
GRIDMODE		0
GRIDSTYLE		0
GRIDUNIT		1/2”,1/2”
GRIDXYZT		1
GRIPBLOCK		0
GRIPCOLOR		72
GRIPDYNCOLOR		140
GRIPHOT		240
GRIPHOVER		150
GRIPOBJLIMIT		100
GRIPS		1
GRIPSIZE		4
GRIPTIPS		1
~~*GsDeviceType*~~		
GsDeviceType2D		0
GsDeviceType3D		1

Variable Name	Read-only	Default Value

H Variables

HALOGAP		0
HANDLES	r/o	1
HANDSEED		“64”
Headroom		"2000"
HIDEPRECISION		0
HideSystemPrinters		0
HIDETEXT		1
HIDEXREFSCALES		1
HIGHLIGHT		1
HIGHLIGHTCOLOR		142
HIGHLIGHTEFFECT		0
~~*HomeGradientColorBottom*~~ *replaced by HorizonBkg_SkyLow*		
~~*HomeGradientColorMiddle*~~ *replaced by HorizonBkg_SkyHorizon*		
~~*HomeGradientColorTop*~~ *replaced by HorizonBkg_SkyHigh*		
~~*HomeGradientMode*~~ *replaced by HorizonBkg_Enable*		
HorizonBkg_Enable		1
HorizonBkg_GroundHorizon		“#878787”
HorizonBkg_GroundOrigin		“#5F5F5F”
HorizonBkg_SkyHigh		“#239BFF”
HorizonBkg_SkyHorizon		“#FFFFFF”
HorizonBkg_SkyLow		“#FAFAFF”
HotkeyAssistant		1
HPANG		0
HPANNOTATIVE		0
HPASSOC		1
HpBackgroundColor		“.”
HpColor		“.”
HPBOUND		1
HPBOUNDRETAIN		0
HPDOUBLE		0
HPDRAWORDER		3
HPGAPTOL		0
HPLAYER		“.”
HPLINETYPE		0
HpIslandDetection		0
HPMAXAREAS		0
HPNAME		“”
HPOBJWARNING		10000
HPORIGIN		0”,0”
HPSCALE		1
HPSEPARATE		0
HPSPACE		1
~~HPSTYLE~~ *replaced by HpIslandDetection*		
HPTRANSPARENCY		“.”
HYPERLINKBASE		“”

I Variables

IfcExplodeExternalReferences		“0”
IfcExportBaseQuantities		“0”
IfcExportElementsOnOffAndFrozenLayer		“1”
IfcExportMultiplyElementsAsAggregated		“0”
IfcExportTesselation		"0"

Variable Name	Read-only	Default Value
IfcImportBimData		“1”
IfcImportBrepGeometryAsMeshes		"0"
IfcImportModelOrigin		"0"
IfcImportParametricComponents		“0”
ImportIfcProjectStructurAsXrefs		“0”
IfcImportSpaces		“0”
ImageCacheFolder		“C:\Users\userid\AppData\Local\Temp\ImageCache\”
ImageCacheMaxMemory		160
ImageDiskCache		1
IMAGEFRAME		1
IMAGEHLT		0
ImageNotify		0
ImportColors		“1”
ImportCreoAlternateSearchPaths		“”
ImportCuiFileExists		0
ImportHiddenparts		“0”
ImportIfcProjectStructureAsXrefs		“0”
ImportIgesSimplify		“1”
ImportIgesStitch		“1”
ImportInventorAlternateSearchPaths		“”
ImportNxAlternateSearchPaths		“”
ImportPmi		“1”
ImportProductStructure		“2”
ImportRepair		“0”
ImportSimplify		“0”
ImportSolidedgeAlternateSearchPaths		“”
ImportSolidworksAlternateSearchPaths		“”
ImportSolidworksRotateYz		“1”
ImportStepRotateYz		“0”
ImportStitch		“0”
IncludePlotStamp		1
INDEXCTL		0
INETLOCATION		“http://www.bricsys.com”
INSBASE		0”,0”,0”
INSNAME		“”
INSUNITS		1
INSUNITSDEFSOURCE		0
INSUNITSDEFTARGET		0
InsUnitsScaling		1
INTERFERECOLOR		“BYLAYER”
InterfereLayer		“Interference”
INTERFEREOBJVS		“”
INTERFEREVPVS		“”
InteriorElevationMinLength		“20”
InteriorElevationOffset		“2”
INTERSECTIONCOLOR		257
INTERSECTIONDISPLAY		0
ISAVEBAK		1
ISAVEPERCENT		50
ISOLINES		4

L Variables

LASTANGLE		0
LASTPO		0”,0”,0”

Variable Name	Read-only	Default Value
LASTPROMPT	r/o	“: SETTINGS”
LATITUDE		37.795
LayerFilterExcess		250
LAYERPMODE		1
LAYLOCKFADECTL		50
LAYOUTREGENCTL		2
LayoutTab		1
LegacyCodeSearch	r/o	0
LengthSamplingInterval		“40”
LENGTHUNITS		“”
LENSLENGTH		50
LevelOfDetail		"0"
LicExpDays		31
LICFLAGS		7
LICKEY	r/o	“7897-9999-0000-99999-0000”
LightGlyphColor		30
LIGHTGLYPHDISPLAY		1
LIGHTINGUNITS		0
LightWebGlyphColor		1
LIMCHECK		0
LIMMAX		1’,9”
LIMMIN		0”,0”
LINEARBRIGHTNESS		0
LINEARCONTRAST		0
LISPINIT		1
LoadMechanical2d		0
LOCALE		“en_US”
LocalRootFolder		“C:\Users\userid\AppData\Local\Bricsys\BricsCAD\V20x64\en_US\”
LOCALROOTPREFIX	r/o	“C:\Users\userid\AppData\Local\Bricsys\BricsCAD\V20x64\en_US\”
LOCKUI		0
LOFTANG1		1.5707963268
LOFTANG2		1.5707963268
LOFTMAG1		0
LOFTMAG2		0
LOFTNORMALS		1
LOFTPARAM		7
LOGFILEMODE		0
LOGFILENAME	r/o	“”
LOGFILEPATH	r/o	“C:\Users\userid\AppData\Local\Bricsys\BricsCAD\V20x64\en_US\”
LOGINNAME	r/o	“*userid*”
LONGITUDE		-122.394
LookFromDirectionMode		1
LookFromFeedback		1
LookFromZoomExtents		1
LTSCALE		1
LUNITS		4
LUPREC		4
LWDEFAULT		25
LWDISPLAY		0
LWDISPSCALE		0.55
LWUNITS		1

M Variables

Variable Name	Read-only	Default Value
M_axisDiameter		6
M_totalAaxisLength		130
MACROREC		0
MACROTRACE		0
MANIPULATOR		0
ManipulatorColorTheme		0
MANIPULATORDURATION		250
ManipulatorSize		1
MassPrec		-1
MassPropAccuracy		0.01
MASSUNITS		“oz lbs stone mg g kg tonne”
MaterialsPath		""
MAXACTVP		64
MAXHATCH		100000
MAXSORT		200
MAXTHREADS		0
MBSTATE	r/o	0
MBUTTONPAN		1
MEASUREINIT		0
MEASUREMENT		0
Mech2dSaveFormat		2013
MENUBAR		1
MENUCTL		1
MENUECHO		0
MENUNAME	r/o	“C:\Users\userid\AppData\Roaming\Bricsys\Bricscad\V20x64\en_US\Support\default.cui”
MESHTYPE		1
MiddleClickClose		1
MILLISECS	r/o	436750804
MIRRTEXT		1
MLEADERSCALE		1
MODEMACRO		“”
MSLTSCALE		1
MSOLESCALE		1
MTEXTCOLUMN		0
MTEXTDETECTSPACE		1
MTEXTED		“”
MTEXTFIXED		2
MTFLAGS		0
MultiSelectAngularTolerance		“3”
MyDocumentsFolder		“C:\Users\userid\Documents\”
MYDOCUMENTSPREFIX	r/o	“C:\Users\userid\Documents\”

N Variables

NEARESTDISTANCE		1
NAVVCUBEDISPLAY		3
NAVVCUBELOCATION		0
NAVVCUBEOPACITY		50
NAVVCUBEORIENT		1
NavVCubeSize		4
NFILELIST		10
NOMUTT		0

Variable Name	Read-only	Default Value
NORTHDIRECTION		0

O Variables

OBJECTISOLATIONMODE		0
OBSCUREDCOLOR		257
OBSCUREDLTYPE		0
OFFSETDIST		-1
OFFSETERASE		0
OFFSETGAPTYPE		0
OLEFRAME		2
OLEHIDE		0
OLEQUALITY		0
OLESTARTUP		0
OPMSTATE	r/o	1
OrbitAutoTarget		1
ORTHOMODE		0
OSMODE		4133
OSNAPCOORD		2
OSNAPZ		0
OSOPTIONS		1

P Variables

PanBuffer		1
PanelButtonSize	r/o	1
PAPERUPDATE		0
PARAMETERCOPYMODE		1
PdfCache		2
PdfEmbeddedTtf		1
PdfExportSolidHatchType		2
PDFFRAME		1
PdfHatchToBmpDpi		300
PdfImageAntiAlias		1
PdfImageCompression		1
PdfImageDPI		300
PdfImportApplyLineweight		1
PdfImportAsBlock		0
PdfImportConvertSolidsToHatches		0
PdfImportImagePath		"pdf images"
PdfImportJoinLineAndArcSegments		1
PdfImportLayersUseType		0
PdfImportRasterImages		0
PdfImportSolidFills		1
PdfImportTrueTypeText		1
PdfImportTrueTypeTextAsGeometry		0
PdfImportUseGeometryOptimization		1
PdfImportVectorGeometry		1
PdfLayersSetting		1
PdfLayoutsToExport		0
PdfMergeControl		0
PdfNotify		0
PDFOSNAP		1
PdfPaperHeight		297
PdfPaperSizeOverride		0

Variable Name	Read-only	Default Value
PdfPaperWidth		210
~~*PdfPRCCompression*~~		
~~*PdfPRCExport*~~		
~~*PdfPRCSingleViewMode*~~		
PdfRenderDPI		300
PdfShxTextAsGeometry		0
PdfSimpleGeomOptimization		1
~~*PdfTextIsSearchable*~~		
PdfTtfTextAsGeometry		0
PdfUsePlotStyles		1
PdfVectorResolutionDpi		2400
PdfZoomToExtentsMode		1
PDMODE		0
PDSIZE		0
PEDITACCEPT		0
PELLIPSE		0
PERIMETER		0
PERSPECTIVE		0
PFACEVMAX		4
PICKADD		1
PICKAUTO		3
PICKBOX		4
PICKDRAG		0
PICKFIRST		1
PICKSTYLE		1
PictureExportScale		1
_PKSER	r/o	“”
PlacesBarFolder1		0
PlacesBarFolder2		1
PlacesBarFolder3		3
PlacesBarFolder4		5
PLATFORM	r/o	“Microsoft Windows NT Version 6.2”
PLINECACHE		0
PLINECONVERTMODE		0
PLINEGEN		0
PLINETYPE		2
PLINEWID		0
PlotCfgPath		“C:\Users\userid\AppData\Roaming\Bricsys\BricsCAD\V20x64\en_US\PlotConfig\”
PLOTID		“”
PlotOutputPath		“”
PLOTROTMODE		2
PlotStylePath		“C:\Users\userid\AppData\Roaming\Bricsys\BricsCAD\V20x64\en_US\PlotStyles\”
PLOTTER		0
PLOTTRANSPARENCYOVERRIDE		1
PLQUIET		0
Pointcloud2dVsDisplay		1
PointcloudBoundary		1
PointcloudCacheFolder		"c:\users\userid\appdata\local\temp\pointcloudcache\"
PointcloudPointMax		4000000
PointcloudPointSize		2
POLARADDANG		“”
POLARANG		90
POLARDIST		0
POLARMODE		0

Variable Name	Read-only	Default Value
POLYSIDES		4
POPUPS	r/o	1
PpState	r/o	0
PreviewDelay		30
PREVIEWEFFECT		2
PREVIEWFILTER		5
~~*PreviewTopdown*~~		
PREVIEWTYPE		0
PreviewWndInOpenDlg		1
PrintFile		“.”
PrintPdfPreview		1
PRODUCT	r/o	“Bricscad”
PROGBAR		1
PROGRAM	r/o	“BRICSCAD”
PROJECTIONTYPE		0
PROJECTNAME		“”
ProjectSearchPaths		“”
PROJMODE		1
PROMPTMENU		3
PromptMenuFlags		1
PromptOptionFormat		2
PromptOptionTranslateKeywords		1
PropertyPreview		1
PropertyPreviewDelay		500
PropertyPreviewObjLimit		500
PropPrevTimeout		1
PROPUNITS		103
PropUnitsVersion		1
PROXYGRAPHICS		1
PROXYNOTICE		1
PROXYSHOW		1
PROXYWEBSEARCH		1
PSLTSCALE		1
PSOLHEIGHT		4
PSOLWIDTH		0.25
PSTYLEMODE	r/o	1
PSTYLEPOLICY		1
PSVPSCALE		0
PUBLISHALLSHEETS		1
PUCSBASE		“”

Q Variables

QAFLAGS		0
QTEXTMODE		0
~~*QuadAperture*~~		
QuadCommandLaunch		1
~~*QuadCommandSort*~~		
QuadDisplay		0
QuadExpandDelay		110
~~*QuadExpandGroup*~~		
QuadExpandTabDelay		50
QuadGoTransparent		0
QuadHideDelay		1000

Variable Name	Read-only	Default Value
QuadHideMargin		40
QuadIconSize		32
QuadIconSpace		1
QuadMostRecentItems		2
QuadPopupCorner		1
QuadShowDelay		150
_QuadTabFlags		12
QuadToolipDelay		1200
~~*QuadWarpPointer*~~		
QuadWidth		5

R Variables

R12SaveAccuracy		8
R12SaveDeviation		0
RASTERPREVIEW		1
RE_INIT	r/o	0
RealTimeSpeedUp		5
REALWORLDSCALE		1
RecentPath		“C:\Users\userid\Documents\”
Redhilite_ducslocked_face_alpha		25
Redhilite_ducslocked_face_color		"#007aff"
RedHiliteFull_Edge_Alpha		100
RedHiliteFull_Edge_Color		“#007AFF”
RedHiliteFull_Edge_ShowHidden		0
RedHiliteFull_Edge_Smoothing		1
RedHiliteFull_Edge_Thickness		2
RedHiliteFull_Face_Alpha		10
RedHiliteFull_Face_Color		“#007AFF”
Redhilite_hiddenedge_smoothing		1
Redhilite_hiddenedge_thickness		1
RedHilitePartial_SelectedEdgeGlow_Alpha		75
RedHilitePartial_SelectedEdgeGlow_Color		“#FFFFFF”
RedHilitePartial_SelectedEdgeGlow_Smoothing		1
RedHilitePartial_SelectedEdgeGlow_Thickness		3
RedHilitePartial_SelectedEdge_Alpha		100
RedHilitePartial_SelectedEdge_Color		“#007AFF”
RedHilitePartial_SelectedEdge_ShowGlow		1
RedHilitePartial_SelectedEdge_Smoothing		1
RedHilitePartial_SelectedEdge_Thickness		2
RedHilitePartial_SelectedFace_Alpha		10
RedHilitePartial_SelectedFace_Color		“#007AFF”
~~RedHilitePartial_UnselectedEdge_Alpha~~		~~20~~
~~RedHilitePartial_UnselectedEdge_Color~~		~~“#007AFF”~~
RedHilitePartial_UnselectedEdge_ShowHidden		1
~~RedHilitePartial_UnselectedEdge_Smoothing~~		~~1~~
~~RedHilitePartial_UnselectedEdge_Thickness~~		~~1~~
RedHilite_HiddenEdge_Alpha		50
RedHilite_HiddenEdge_Color		“#FFFFFF”
RedHilite_HiddenEdge_Smoothing		1
RedHilite_HiddenEdge_Thickness		1
RedSdkLineSmoothing		0
ReduceLengthType		“0”
ReduceLengthValue		“0.5”
RefeditLockNotInWorkset		0

Variable Name	Read-only	Default Value
REFEDITNAME	r/o	“”
REGENMODE		1
RegExpand		1
REMEMBERFOLDERS		1
RenderMaterialDownload		1
RenderMaterialPath		“C:\ProgramData\...”
RenderMaterialStaticPath		“C:\Program Files\...”
RenderUsingHardware		1
ReportPanel		0
ReportPanelMode		0
RestoreConnections		"1"
RevCloudArcStyle		0
REVCLOUDCREATEMODE		1
REVCLOUDGRIPS		1
RevCloudMaxArcLength		0.375
RevCloudMinArcLength		0.375
RhinoVersion		60
RIBBONDOCKEDHEIGHT		120
RIBBONSTATE	r/o	0
Riserheight		"170"
RoamableRootFolder	r/o	“c:\users...”
ROAMABLEROOTPREFIX	r/o	“C:\Users\userid\AppData\Roaming\Bricsys\BricsCAD\V20x64\en_US\”
ROLLOVEROPACITY		100
ROLLOVERTIPS		1
RolloverSelectionSet		1
Roomheight		"3000"
RTDISPLAY		1
RTRotationSpeedFactor		1
~~*RTWalkSpeedFactor*~~ *replaced by the StepSize variable*		
RubberbandColor		40
RubberbandStyle		1
RunAsLevel		2

S Variables

Safemode	r/o	0
SaveChangeToLayout		1
SAVEFIDELITY		1
SAVEFILE	r/o	“”
SAVEFILEPATH		“C:\Users\userid\AppData\Local\Temp\”
SaveFormat		1
SaveLayerSnapshot		1
SAVENAME	r/o	“”
SaveOnDocSwitch		0
SAVEROUNDTRIP		1
SAVETIME		60
SCREENBOXES	r/o	26
SCREENMODE	r/o	1
SCREENSIZE	r/o	145’-8”,73’-3”
SCRLHIST		256
SDI		0
SearchAll		0
SectionScale		“0.02”
SectionSettingsSearchPath		""
SectionSheetsetTemplateImperial		“”

Variable Name	Read-only	Default Value
SectionSheetsetTemplateMetric		“”
SecureLoad	r/o	0
SELECTIONANNODISPLAY		1
SELECTIONAREA		1
SELECTIONAREAOPACITY		25
SelectionModes		0
SELECTIONPREVIEW		3
SELECTSIMILARMODE		130
SHADEDGE		3
SHADEDIF		70
SheetNumberLeadingZeroes		1
SheetSetAutoBackup		1
SheetSetTemplatePath		“C:\Users\userid\AppData\Local\Bricsys\BricsCAD\V20x64\en_US\Templates\Sheet Sets\”
SHORTCUTMENU		18
SHORTCUTMENUDURATION		250
ShowDocTabs		1
ShowFullPathInTitle		0
SHOWLAYERUSAGE		0
ShowScrollButtons		1
ShowTabCloseButton		0
ShowTabCloseButtonActive		0
ShowTabCloseButtonAll		1
ShowTabControls		1
ShowWindowListButton		1
SHPNAME		“”
SingletonMode		0
SKETCHINC		0.1
SKPOLY		0
SkpStitch		1
SKYSTATUS		0
SlabThickness		"250"
smAssemblyExportReportPathType		"0"
smAssemblyExportSolidTypesInReports		"1"
smAttributesLayerColor		“7”
smAttributesLayertextheight		“0.01”
smAttributesLayerTextHeightType		“0”
smBendAnnotationsLayerColor		“5”
smBendAnnotationsLayerTextHeight		“0.01”
smBendAnnotationsLayerTextHeightType		“0”
smBendlinesDownlayerColor		“1”
smBendlinesDownlayerLinetype		“Continuous”
smBendlinesDownlayerLineweight		“-3”
smBendlinesUplayerColor		“1”
smBendlinesUplayerLinetype		“Continuous”
smBendlinesUplayerLineweight		“-3”
SMCOLORBEND		“#FFDC50”
SMCOLORBENDRELIEF		“#64D296”
SMCOLORCORNERRELIEF		“#64D296”
SMCOLORFLANGE		“#90A4AE”
smcolorflangereferenceside		“#68a4ae”
Smcolorform		“#8791e1”
SMCOLORJUNCTION		“#FF6E40”
SMCOLORLOFTEDBEND		“#A0DCFA”

Variable Name	Read-only	Default Value
smColorMiter		“#af46d8”
smColorTab		""
smColorWrongBend		“#ff3300”
smContoursLayerColor		“7”
smContoursLayerLinetype		“continuous”
smContoursLayerLineweight		“30”
smConvertPreferFormFeatures		“0”
smConvertRecognizeHoles		“0”
smConvertRecognizeRibControlCurves		“0”
smConvertWrongFeatureThicknessDeviationType		"0"
smConvertWrongFeatureThicknessDeviationValue		"0.2"
smdefaultbendlineextenttype		“0”
smdefaultbendlineextentvalue		“0.25”
smdefaultbendradiustype		“2”
smdefaultbendradiusvalue		“1”
smdefaultbendreliefwidthtype		“0”
smdefaultbendreliefwidthvalue		“0.5”
smdefaultcornerreliefdiametervalue		“-1”
smdefaultflangesplitextensiontype		“0”
smdefaultflangesplitextensionvalue		“0.1”
smdefaultflangesplitgaptype		“0”
smdefaultflangesplitgapvalue		“0.1”
smdefaultformfeatureunfoldmode		“4”
smDefaultHemGapType		"1"
smDefaultHemGapValue		"0.005"
smdefaultjunctionalignmenttorelief		“0”
smdefaultjunctiongaptype		“0”
smdefaultjunctiongapvalue		“0.001”
smdefaultkfactor		“0.27324”
smDefaultLoftedBendNumberSamples		"10"
smdefaultreliefextensiontype		“0”
smdefaultreliefextensionvalue		“0.1”
smdefaultribfilletradiustype		“0”
smdefaultribfilletradiusvalue		“5”
smdefaultribprofileradiustype		“0”
smdefaultribprofileradiusvalue		“2”
smdefaultribroundradiustype		“0”
smdefaultribroundradiusvalue		“1”
smdefaultsharpbendradiuslimitratio		“5”
smDefaultTabChamferDistanceType		"0"
smDefaultTabChamferDistanceValue		"0.1"
smDefaultTabClearanceType		"0"
smDefaultTabClearanceCalue		"0.1"
smDefaultTabDistanceType		"0"
smDefaultTabDistanceValue		"20"
smDefaultTabEdgeType		"0"
smDefaultTabFilletRadiusType		"0"
smDefaultTabFilletRadiusValue		"0.1"
smDefaultTabHeightType		"0"
smDefaultTabHeightValue		"1"
smDefaultTabLengthType		"0"
smDefaultTabLengthValue		"4"
smDefaultTabSlotNumber		"3"
smdefaultthickness		“0.078740157480315”

Variable Name	Read-only	Default Value
smexportosmapproximationaccuracy		“0.000393701”
smexportosmminimaledgelength		“0.001968505”
smformfeaturesdowncolor		“6”
smformfeaturesdownlayerlinetype		“continuous”
smformfeaturesdownlayerlineweight		“-3”
smformfeaturesupcolor		“6”
smformfeaturesuplayerlinetype		“continuous”
smformfeaturesuplayerlineweight		“-3”
smjunctioncreatehealcoincident		“0”
smoverallannotationslayercolor		“3”
smoverallannotationslayerlinetype		“continuous”
smoverallannotationslayerlineweight		“-3”
smparametrizeholesparametrization		“3”
smrepairloftedbendmerge		“0”
smsmartfeatures		“3”
smSplitAmbiguousInput		"0"
smsplitconvertbendtojunction		“1”
smsplithealcoincident		“0”
smsplitorthogonalbendsplit		“0”
SMTARGETCAM		“”
SNAPANG		0
SNAPBASE		0”,0”
SNAPISOPAIR		0
SnapMarkerColor		20
SnapMarkerSize		6
SnapMarkerThickness		2
SNAPMODE		0
SNAPSTYL		0
SNAPTYPE		0
SNAPUNIT		1/2”,1/2”
SOLIDCHECK		1
SORTENTS		127
spaAdjustMode		0
spaCheckLevel		10
spaGridAspectRatio		0
spaGridMode		1
spaMaxFacetEdgeLength		0
spaMaxNumGridLines		512
spaMinUGridLines		0
spaMinVGridLines		0
spaNormalTol		15
spaSurfaceTol		-1
spaTriangMode		1
spaUseFacetRES		1
SPLFRAME		0
SPLINESEGS		8
SPLINETYPE		6
SRCHPATH		“C:\Users\userid\AppData\Roaming\Bricsys\BricsCAD\V20x64\en_US\Support\; C:\Program Files (x86)\Bricsys\BricsCAD V20x64**Support**\; C:\Program Files (x86)\Bricsys\BricsCAD V20x64**Fonts**\; C:\Program Files (x86)\Bricsys\BricsCAD V20x64**Help**\en_US\”
SSFOUND		“”
SSLOCATE		1
SSMAUTOOPEN		1

Variable Name	Read-only	Default Value
SSMPOLLTIME		15
SSMSHEETSTATUS		2
SSMSTATE		0
StackPanelType	r/o	0
Stairwidth		"1000"
StampFontSize		0.2
StampFontStyle		“Arial”
StampFooter		“”
StampFooterOffsetX		0
StampFooterOffsetY		0
StampHeader		“”
StampHeaderOffsetX		0
StampHeaderOffsetY		0
StampUnits		0
Statusbar		1
STARTUP		1
STEPSIZE		6
StlPositiveQuadrant		1
STEPSPERSEC		2
StructureTreeConfig		“mechanical.cst”
SURFTAB1		6
SURFTAB2		6
SURFTYPE		6
SURFU		6
SURFV		6
SvgBlendedGradients		0
SvgDefaultImageExtension		“.png”
SvgGenericFontFamily		0
~~*SvgHiddenLineRemoving*~~		
SvgImageBase		“”
SvgImageUrl		“”
SvgLineWeightScale		1
SvgOutputHeight		768
SvgOutputWidth		1024
SvgPrecision		6
SvgScaleFactor		0
SYSCODEPAGE	r/o	“ANSI_1252”

T Variables

TabControlHeight		25
TABMODE		0
TabsFixedWidth		0
TangentLengthType		“0”
TangentLengthValue		“0”
TARGET		0”,0”,0”
TDCREATE	r/o	2456335.5399919
TDINDWG	r/o	0.121354456
TDUCREATE	r/o	2456335.8733252
TDUPDATE	r/o	2456335.5399919
TDUSRTIMER	r/o	0.121354456
TDUUPDATE	r/o	2456335.8733252
TeeTangentLengthType		"0"
TeeTangentLengthValue		"0.5"
TemplatePath		“C:\Users\userid\AppData\Local\Bricsys\BricsCAD\V20x64\en_US\Templates\”

Variable Name	Read-only	Default Value
TEMPPREFIX		“”
TestFlags		0
TestGsFlags		0
TEXTANGLE		0
TEXTED		2
TEXTEDITMODE		0
TEXTEVAL		0
TEXTFILL		1
TEXTQLTY		50
TEXTSIZE		0.2
TEXTSTYLE		“Standard”
TextureMapPath		“C:\Program Files (x86)\Bricsys\BricsCAD V20x64\Textures\1\”
THICKNESS		0
ThreadDisplay		“0”
THUMBSIZE		1
TILEMODE		1
TILEMODELIGHTSYNCH		1
TIMEZONE		-8000
ToolbarMargin	r/o	0
ToolbuttonSize	r/o	0
TooliconPadding	r/o	0
Tips		1
~~*ToolbarIconSize*~~		
TOOLPALETTEPATH		“C:\Users\userid\AppData\Roaming\Bricsys\BricsCAD\V20x64\en_US\Support\ToolPalettes\”
TOOLTIPS		1
TPSTATE	r/o	0
TRACEWID		0.05
TRACKPATH		0
TRANSPARENCYDISPLAY		1
TrayIcons		1
TrayNotify		1
TrayTimeout		0
TreadLength		"290"
TREEDEPTH		3020
TREEMAX		10000000
TRIMMODE		1
TrustedPaths	r/o	""
TSPACEFAC		1
TSPACETYPE		1
TSTACKALIGN		1
TSTACKSIZE		70
TTFASTEXT		3

U Variables

UCSAXISANG		90
UCSBASE		“”
UCSDETECT		0
UCSFOLLOW		0
UCSICON		3
UCSICONPOS		0
UCSNAME	r/o	“”
UCSORG	r/o	0”,0”,0”
UCSORTHO		1

Variable Name	Read-only	Default Value
UCSVIEW		1
UCSVP		1
UCSXDIR	r/o	1”,0”,0”
UCSYDIR	r/o	0”,1”,0”
UNDOCTL	r/o	5
UNDOMARKS	r/o	0
UNITMODE		0
~~*UseBIM*~~		
UseCommunicator		1
~~*UseMechanical*~~		
USERI1 *thru* USERI5		0
USERR1 *thru* USERR5		0
USERS1 *thru* USERS5		“”
~~*UseSheetMetal*~~		
UseStandardOpenFileDialog		0

V Variables

Variable Name	Read-only	Default Value
VbaMacros		1
VENDORNAME	r/o	“Bricsys”
VerboseBimSectionUpdate		“1”
_VERNUM	r/o	“19.1.06 (UNICODE)”
VersionCustomizableFiles	r/o	“344”
VIEWCTR	r/o	10 7/16”,4 1/2”,0”
VIEWDIR	r/o	0”,0”,1”
VIEWMODE	r/o	0
VIEWSIZE	r/o	297
VIEWTWIST	r/o	0
VIEWUPDATEAUTO		1
VISRETAIN		1
VOLUMEPREC		-1
VOLUMEUNITS		“in ft mi µm mm cm m km”
VPROTATEASSOC		1
VSMAX	r/o	-1.0000E+20,-1.0000E+20,-1.0000E+20
VpMaximizedState	r/o	0
VSMIN	r/o	1.0000E+20,1.0000E+20,1.0000E+20
VTDURATION		750
VTENABLE		3
VTFPS		7

W Variables

Variable Name	Read-only	Default Value
Wallwidth		"250"
WarningMessages		65535
WHIPARC		1
WHIPTHREAD		0
WINDOWAREACOLOR		150
WIPEOUTFRAME		1
WMFBKGND		0
WMFFOREGND		0
WNDLMAIN		2
WNDLSCRL		0
WNDLSTAT		1
~~*WNDLTABS*~~		
WNDLTEXT		1

Variable Name	Read-only	Default Value
WNDPMAIN		0”,0”
WNDPTEXT		3’-4”,3’-4”
WNDSMAIN		101’-2”,66’-11”
WNDSTEXT		118’-4”,86’
~~*WorkspaceSecurity*~~		
WORLDUCS		1
WORLDVIEW		1
WRITESTAT	r/o	1
WSAUTOSAVE		1
WSCURRENT		“2D Drafting”

X Variables

XCLIPFRAME		2
XDwgFadeCtl		70
XEDIT		1
XFADECTL		50
XLOADCTL		1
XLOADPATH		“C:\Users\userid\Documents\”
XNotifyTime		5
XREFCTL		0
XRefNotify		1
XREFOVERRIDE		0

Z Variables

ZOOMFACTOR		60
ZOOMWHEEL		0

# Variables

3DCOMPAREMODE		3
3DOSMODE		11
3dSnapMarkerColor		5

APPENDIX C

What's New in BricsCAD V20

BRICSYS UPDATES BRICSCAD SEVERAL TIMES A YEAR, WITH A MAJOR UPDATE EACH FALL.

This appendix lists BricsCAD's new and changed functions in V20, and is compiled from version 20.1.05. Changes are highlighted throughout this book, but be aware that information on these pages is not comprehensive. For information on functions added since this book was published, please see https://www.bricsys.com/common/releasenotes.jsp.

New command and variable names are shown here in boldface **blue**, and updated ones in boldface **black**. They are listed in alphabetical order in the following sections:

- New Entities
- Command Name Changes
- User Interface, File Operations, and Printing
- 2D Drawing and Editing
- Blocks, Tables, and Data Extraction
- Dimensioning
- 3D
- Point Clouds
- BIM
- Civil
- Mechanical
- Sheet Metal
- Communicator
- Customization and Security

BricsCAD V20 installs and runs independently from previous BricsCAD versions.

Shape is no longer supplied in a separate program, and is installed with BricsCAD V20. It can be started from the new Launcher dialog box.

New Entities

BricsCAD V20 displays surfaces containing t-spline sub-surfaces, NURBS surfaces where control points termination in the shape of a T.

New entities and commands for doing civil engineering work in BricsCAD V20 Platinum. The new entities are:

- TIN surfaces
- Grading entities
- Alignment entities

The commands associated with civil engineering are Alignment, AlignmentEdit, AlignmentView, Grading, Tin, TinEdit, TinModify, TinMerge, TinVolume, and TinExtract.

COMMAND NAME CHANGES

Drag is the new name for the **BimDrag** command.

bmInsert and **DesignTable** commands are now available at the BricsCAD Pro license level.

Parameters panel is renamed Parameters and Constraints panel as it supports non-parametric 3D constraints.

Plot command now displays the Print command's dialog box, but reverts to command line prompts in unattended mode.

WNDLSTAT variable is renamed **STATUSBAR**, and the **WNDLTABS** variable is renamed **LAYOUTTAB**.

USER INTERFACE, FILE OPERATIONS, AND PRINTING

BricsCAD V20 starts with a new Launcher dialog and then displays a new Start page; these replace the GetStarted dialog box.

Launcher displays workspaces to choose from.

Start page lists recently opened drawings, and starts new ones from scratch or from templates. BricsCAD

GotoStart command opens the new start tab.

ColorTheme variables toggles between the traditional light theme (1) and the new dark theme (0; default).

CommandLine now allows a foreground color of solid black and a custom background color. When AutoComplete is in use, pressing the backspace key removes one typed character each time it is pressed.

DgnImport command now imports 3D polylines from .DGN files.

DgnImportOptions opens the Settings dialog box at the DgnImport section.

DstConvert converts sheetset DST files to XML format.

DynPiCoords variable displays dynamic coordinate input with relative (0) or absolute (1) distances and angles.

UcsDetect variable is expanded so that dynamic UCS supports entities other than faces of 3D solids:

3 — enable for 3D faces and 2D entities
2 — enable for 2D entities only
1 — enable for 3D faces only
0 — disable
negative — disable, but remember the previous setting

DUCS status bar control toggles what dynamic UCS detects:

- Faces of 3D solids (default)
- Other entity types: point, line, polyline, 2D polyline, 3D polyline, ray, xline, arc, circle, ellipse, spline, text, mtext, solid, 3Dface, trace, block insert, viewport, mline, leader, mleader, hatch, helix, camera, light, section, shape, PDF underlay, and image.

Explorer command alerts you when a referenced text style or dimension style is removed

Grip selection process is more efficient. Grips, pickbox, and other glyphs are now scaled automatically for high resolution (4K) displays.

HideSystemPrinters variable allows users to hide Windows system printers in the Print dialog, keeping just the .pc3 printer configurations.

ImageAttach command now loads compressed TIFF files with more than half a billion pixels; this limitation had not affected images in Big-TIFF format which use 64 bit offsets.

IME (input method editor- composition window no longer closes unexpectedly when the mouse is moved over the drawing view.

Layer, **PDF**, **Image**, and **XRef** commands now bring up modeless, dockable panels instead of the Drawing Explorer. Use the ExpLayers, ExpPdf, ExpImage And ExpXrefs commands bring up the Drawing Explorer for these items.

LookFrom widget now remains active during commands, and no longer disappears when entering clean screen mode. But it is no longer active while other application windows are in the foreground.

Manipulator has an improved initial location of the Manipulator for associative arrays. The MIRROR and SCALE options support the C key (hidden Copy option) to make multiple copies with the Manipulator.

NearestDistance variable now controls the types of dimensions displayed: current values, or individual X, Y and Z components of nearest distances. Scaling of arrowheads displayed during zooming is added.

New and **Open** commands that launch another instance of BricsCAD with the \t (template) option create new drawings based on appropriate template, instead of opening the .dwt template file itself. Drawings with errors in AcDbAssocAction records can now be opened as corrupted records are ignored.

Ortho is properly reported in the status bar.

PdfImport command displays an alert message when no objects are imported, or when errors occurs during the import process. The amount of "grainy dust" in the display of PDF underlays of scanned image is reduced, when the PDF display color is set to match the background color.

PeditExt command (for editing polyline vertices and segments) now works with 2D classic and 3D polylines, in addition to lightweight polylines.

Print command is improved when printing layouts with many viewports. **Plot** command now displays the print dialog box in the interactive mode, and behaves as **-Plot** in the unattended mode. When running in unattended or scripted mode, **-Plot** overwrites files silently when **Expert** variable is > 1. Users can choose to use multiple folder paths for the **PlotCfgPath** and **PlotStylePath** variables.

Properties panel Coordinate pickbutton behavior now makes: dynamic dimensions available.

Polar snap (**Snaptype**=1) is now supported with the **PolarDist**, **PolarAng**, **SnapMode**, and **AutoSnap** variables.

Quad now opens correctly on a hot-plugged secondary display. When the quad is turned off, a short-right-click suffices to display the context menu for selected entities. Polyline segments can now be selected as entity filters for quad buttons. The LwPolyline_Edge entity is added to the list displayed in Add Entity Alias dialog box.

Redraw command is improved for drawings with many inserts with attributes located at the drawing origin and when silhouette display is on (DispSilh = 1).

RrRot command improves the calculation of the center of rotation when **OribtAutoTarget** variable is 0 and the selection point is not on any object or surface.

SaveAs command keeps the old name of a not-yet-saved drawing when the SaveAs operation fails.

Screenshot takes a screen grab of the current space, excluding all UI elements

Shell allows non-ASCII characters in paths.

ShowFulPpathInTitle variable displays the full document path in the title of the main BricsCAD window, when set to **1**. The title bar displays the name of the lower license level when **RunAsLevel** is applied to run BricsCAD V20 at a reduced level.

Snap previously gave precedence to the point with higher priority when snap points with different snap priorities (e.g. end point and intersection) were inside the snap aperture box; now the winning snap point is the one closest to the cursor. Previously, the distance to the cursor was computed only in the case where the snap points had equal priority.

StatusBar changes the appearance of the Cursor Coordinate values, depending on whether its enabled or disabled.

Structure command gains the option to deselect entities when hiding them via the panel; . "on" by default.

Toolbar command now shows previously hidden toolbars when it is repositioned. **Tooltips** variable is now functional and shown in Settings dialog. Toolbars are shown in sub-menus by menu group, and similar named toolbars are further grouped into nested sub-menus.

Toolpalettes command gain support for palette separators.

ViewEdit gains the option to change the selection set of several views, as long as they are the same for all views.

ViewHorizontal rotates the viewpoint to make z=0 (horizontal)

VpMax and **VpMin** commands maximize and restore current viewport. Paper space viewport(s) in model space can now be edited, copied, moved, and erased. VPLOCK status bar button allows users to see and set the display lock status of the active layout viewport.

WndlStat variable is renamed **StatusBar**, and the **WndlTabs** variable is renamed **LayouTtab**.

2D Drawing and Editing

Arc and **Pline** commands change the direction arcs are drawn to clockwise when you hold down the **Ctrl** key.

CopyGuided copies entities along guidelines.

CopyToLayer command copies entities to layers.

-Gradient command is the command-line version of the **Gradient** command's dialog box.

Hatch command no longer creates undesired associations between hatches and externally adjoining boundaries (e.g. for externally adjoining rectangles).The **-Hatch** command now fills gradient patterns. **HatchGripEdit** adds and removes grips from hatches and gradients.

-ImageAttach command has improved text prompts.

InsertGuided and **-InsertGuided** inserts blocks along guide curves

LAYER command that works on the command-line now allows the locking and unlocking of xref-dependent layers. **LayerP** command now properly reverts changes made by sequential calls to **LayIso**.

The Layer Filter Panel now supports Layer Group filters. Entity selection from locked layers is now available for the **LayOff** and **LayFrz** commands for any block selection setting.

MPolygon objects now support grips.

MText command gains a Paragraph Spacing dialog box. The on-screen mtext editor now offers better support for high resolution displays.

MLeader command no longer allows annotative blocks to be used with block content. MLeader command Creating an annotative mleader in a custom UCS adds an MText copy when mtext content s used.

Number command gains a new parameter, Tolerance: it considers two solids equal when their centroids are within a given tolerance.

ParametricBlock creates a parametric block from entities in the drawing; useful for BEdit.

Project Browser gains the Add Sheet, Add Subset, and Remove Subset options, and the Properties option to Sheet/View/Subset nodes.

Sections can now be updated in the background, enabled by checking the Enable Background Update item in the Project Browser menu. Tree controls are now user-resizable and will always fit in the browser window, and configurable margins are added for viewport placement.

You can now display Sheet/View labels as “number-name”. Also implemented is drag and drop of schedules onto a sheet/layout; you can also drag & drop a Sheet/Subset to another subset. You can extend an existing sheetset from the Project Browser, and there’s a new project setup dialog, too.

Propagate command gains support for features other than windows and doors. This command is faster at the propagation of window features, and section movement.

Its Box select behavior now aligns with BricsCAD’s regular box select behavior.

Two types of propagated details can be parameterized: connections between linear solids and connections between planar solids. This parameterization allows users to apply a detail to target configurations with different geometric parameters, such as layer thicknesses, profiles and angles.

Properties panel can now use Start offset and End offset properties to edit path arrays.

QuickDraw command now draws the projected grid on top of a slab when the user drags the blue quickdraw cursor from the outside (of a building) inwards. It also now copies stairs across multiple floors automatically.

Style command no longer applies the vertical flag for fonts that do not support vertical orientation (all TTF fonts and some SHX fonts).

Text command now displays text using its own color, instead of the color of the current layer, during editing.

UndoEnt undoes property changes to selected entities.

BLOCKS, TABLES, AND DATA EXTRACTION

2D constraints now can be applied to block insertion points, and to entities inside blocks. It is now possible to add flip states to a parametric block by defining flip lines.

AttDef command adds the Select Property context menu item to select properties from the list. The positioning of multi-line attribute definitions is improved. The fields specified in the Default value of single line attribute definitions are preserved after closing the Define Attribute dialog box.

AttSync command now allows wild-card patterns for block names.

BEdit (block editor) and **RefEdit** (reference editor) commands now edit blocks with proxy entities, if they are cloneable, transformable, eraseable, and able to change visibility.

Blockify command now gives users the option to specify the names of the blocks to be created. The input set to the command now supports wipeout entities. Block transformation matrices are now simplified after Blockify executes.

DataExtraction command adds the 'table column total' option. Tables generated from data extractions have improved formatting. data extraction now offers block views,. which are graphical thumbnails that are inserted into tables containing the results from data extraction operations. Changed dragging initiation keypress on the Properties wizard page to **ctrl+right-click drag**, to support context menus.

DataLink command makes tables with attached data links that are compatible with AutoCAD.

Dynamic blocks now sport grips for lookup parameters.

Table command adds the datalink update tool to the quad. Tables with breaks have improved selection and editing. Improved table highlighting when changing "Enable breaks" from true to false. Now you can edit repeating label rows in a sub-table. The command's Datalink update now removes or clears rows when a datalink range shrinks.

Dimensioning

Improved the application of dimension text height settings, and text style changes made in the MText editor.

CenterDisassociate disassociates center lines and marks from circles and arcs

DimLayer, **HpLayer**, and **CenterLayer** variables do not accept xref layers as targets.

DimTEdit now behaves properly when dragging an annotative dimension text entity.

DimGap variable now allows only positive values now; a zero value is considered as positive so that there will be no frame around dimension text.

General 3D

While hovering the vertex of a 3D solid, holding down the **Ctrl**-key displays the coordinates in the roll-over properties panel.

QuickDraw command assigns roof slabs to new floors in buildings.

Sweep command gains an interactive version to allows users to sweep profiles along a sequence of curves (not just a single curve), or just along a portion of the curve(s).

Radial grids are now supported in section views.

POINT CLOUDS

-PointCloudAttach command attach point cloud files at the command line.

PointCloudPreprocess command pre-processes additional file formats, including ReCAP project and source files to produce highly efficient .vrm (Virtual Reality Model) files that attach directly to drawings. You specify the units for the point cloud and a progress indicator shows the status as the pre-processor runs. -PointCloudPreprocess is the command-line version.

PointCloudCrop crops the extents of the current point cloud, while PointCloudUncrop removes cropping from point clouds.

BIM

(BUILDING INFORMATION MODELING)

AutoRoom and **Bimify** commands find more rooms with walls of a non-standard shape.

BimDrag command has been renamed to **Drag**, and is available in BricsCAD Pro, Platinum, BIM and Mechanical.

BCF (OpenBIM Collaboration Format) panel now allows users to add comments and change properties in the BCF panel. It allows users to view and manage model-based issues based on the BCF standard.

BimApplyProfile command applies profiles to flow fittings and flow fitting faces. When "AutomaticConnection" is on, applying a profile to a flow element causes adjacent elements to adapt. Setting "RestoreConnection" now controls the restoration of structural connections after BimApplyProfile command runs.

BimAttachComposition command now supports alignment options, and extends ply slicing accommodate building elements containing a cylindrical reference face.

BimClassify command gains a new BIM element type called Annotation; it is accompanied by a default template (used in BIMTAG).

BimCurtainWall command now provides connection-type nodes.

BimDecompose command now decomposes composition-based solids into their separate plies.

BimFlip command now works with asymmetric profile-elements, both structural and MEP.

BimFlowConnect command creates tee flow fittings in all T-shaped connections when the 'AutomaticTees' option is switched on. Connecting solids with invalid axes now behaves as expected. The command allows generation of a flow layout from an array of flow terminals and a designated main terminal.

BimGrid command now supports anonymous grid blocks in section views, and offer improved snaps. Previews of a BIM grid, and the resulting grid, now use the current "No Selection" properties.

Bimify command now works on selection sets, and you can now specify a project type (Architectural/Structural/MEP) during Bimify.

BimRoom command now labels invalid rooms with an 'invalid room' stamp.

BimSection command now changes the clipping state when you press the **Ctrl** key. The default of "on", which corresponds to dynamic clipping; when off, you can position the section by snapping to entity snap points, when entity snap mode is on.

The command now displays all hatch patterns of the same material at the same scale in the same 2D drawing when working with different scales. It now applies hatch patterns to generated elevations, and section indicators for detail sections and interior elevations on plans.

BimSectionUpdate command now creates room lines, and its performance was improved for non-associative views.

BimSetLoadBearingDirection sets load bearing directions for slabs.

BimSpatilAllocations command adds building and story definitions at the command line.

BimStair command creates straight stairs as a rectangular parametric array, optionally fixed to a wall. You can attach selectable dimensions at the extents of the stair.

BimTag command now allows you to place tags on nested entities manually, and offers a new Composition option. The command now highlights related entities during manual tagging mode.

Mleader styles for BIM objects are now supported.

Pressing the **Ctrl** key during manual tag placement switches between different tags.

The format of the _TagTypeToStyle.csv file has been changed to .xml. New styles can be added, based on the styles used for automatically placed tags.

BmInsert command now detects if a selected .rfa file has been previously imported, and creates an instance of the existing component to reduce file size.

BIM components have been updated with correct naming and classifications. Windows and Door components use index colors instead of true colors.

DisplayAxesForMEP variable now displays the axes of flow bends, reducers and tees, when switched on.

Profiles of flow fittings are now shown in, and can be changed from the property panel.

IfcExport command now supports the IFC4 Standard, and you can select the IFC format (2X3 or 4).

It exports block attributes and component parameters, has improved performance when exporting polyface meshes, and exports profiles as parameterized IFC profiles. It also now exports analysis models.

IfcImport command supports the IFC4 standard and offers a progress meter shows a more accurate indication of the IFC import process.

Invalid symbols in the names of spatial structure elements are replaced to get valid file names for external references.

Windows and doors on a different storey than their containing walls are now correctly positioned when the project structure is imported as a set of xrefs.

Manipulator preserves connections between flow elements and structural elements during rotation.

MatchProp command now offers the option to match BIM properties between BIM objects. The new **BimMatchProp** variable controls its behavior.

Properties of a linear element's axis (start/end points, delta, length) are shown in the property panel and can be edited.

Ribbon in the BIM workspace was re-designed to contains all of BricsCAD V20's new BIM features.

Schedules now generate elevation and plan symbols for windows and doors. A dialog box to create Schedule definitions is now available in the Project Browser.

SectionSettingsSearchPath variable can be configured to specify alternative locations for files containing section plane settings, section, room and grid callout symbols, tag mappings, story indicators and 2D section result layers.

Separate command now retains BIM data as expected.

SetLevelOfDetail command displays render materials and composition plies when set to high..

CIVIL

Alignment creates horizontal, vertical, or 3D alignments on TIN surfaces

AlignmentEdit edits horizontal and vertical alignments

AlignmentView views alignment along TIN surface

AlignmentVInitial creates vertical alignments

LandXmlEexport exports the drawing in LandXML format

Grading interactively modifies TIN surfaces to create grading effects, such as for roads and foundations.

LandXmlImport iimports LandXML files into the current drawing

Tin (short for triangulated irregular network) imports data from TIN files to create land surfaces, and converts Civil 3D surfaces to BricsCAD format.

TinEdit adds and removes points, break lines, and boundaries in TIN surfaces.

TinExtract creates a mesh or 3D solid between TIN surfaces or between a TIN surface and elevation or vertical offset.

TinMerge merges two or more TIN surfaces into a single one.

TinModify deforms or smooths TIN surfaces.

TinVolume creates a TIN volume surface between a base and TIN surfaces or an elevation

MECHANICAL

(BM = BRICSCAD MECHANICAL; DM = DIRECT MODELING)

The 3D constraints gain a new **Path** constraint moves a point along a curve specified by an expression and the curve's parameters.

Constraints with numeric parameters can now be geometry-driven, which means their values aren't imposed upon the model; instead, they are derived from the model as other constraints are solved. Geometry-driven constraints can be used as inputs for expressions, including variables, other constraints, associative arrays and component parameters.

The vertices of a 3D solid can now be selected for 3D constraints with the **SelectionModes** variable set to 15.

A 3D constraint icon is now shown when a constrained line, circle or xline is selected.

bmBom command now creates BOM tables with new capabilities, including extraction of built-in properties of components; adjusting units for properties; configuring the sorting rules; adding formula columns and footers; and saving and loading of BOM templates.

bmExplode command adds the option to automatically create an exploded view for a given assembly.

bmExternalize command now forces saving of the current drawing prior to calling this command in batch mode, without asking for file names.

bmMech command now allows selection of blocks and external references to be converted into components.

DesignTable command improves the diagnostics for design table import from a CSV file. Several diagnostic messages were added, including 'invalid column names' and 'unknown error'.

dmFrozenGroup command defines groups of entities that can be frozen by setting the group parameter to a non-zero value. This is equivalent to putting the group on a frozen layer. The group parameter can be also controlled by an expression.

Mechanical Browser now allows group features to be edited via a right-click on the feature's icon. It also allows description editing for nested local definitions, and the animation range works with smaller values.

-Parameterts command now supports string values.

Parameters panel is renamed 'Parameters and Constraints' panel as it supports non-parametric 3D constraints. Unsatisfied constraints are displayed in the color purple. "Show driven", "Show driving" and "Clear unused variables" options were added to the context menu.

SHEET METAL

(SM = SHEET METAL)

Settings for Sheet Metal are now categorized and grouped by feature type.

smAssemblyExport command adds the sheet metal type "Standard part" classification HTML and JSON reports. The "Solid types in reports" setting determines what types of solids will be included in reports, 'Sheet Metal' and/or 'Poor Sheet Metal'.

The new "Report path type" setting determines which file path types will be used in reports: relative (default) or absolute.

The JSON file is generated with a list of reachable components, their solids with unfolding information, and the assembly tree structure (starting at the root).

The HTML report uses relative paths to embedded images.

smExplode explodes bends. forms, hem, junctions, and tabs

smFlangeContour command automatically creates a flange, linked with an existing SM model via a bend, from a provided, closed contour.

smHemCreate command creates different types of hem features.

smJunctionCreate command now creates junction features on curved hard edges between two bends(or lofts).

smSelect command gains a new option, Flat edge', that suggests the selection of co-planar free edges, similar to the smFlangeEdge and smHemCreate commands.

smSplit command automatically finds the face to split, based on underlying geometry for input line or point. In case of ambiguity, the UCS associated with the required face is used. The Propagate option automatically suggests multiple splits of a similar type.

smTabCreate command creates a tab connection between two flanges.

smUnfold command has a new option to place the unfolded view (with all necessary annotations) into a Paperspace view. Bend table information is added to the attributes layer of the un-folded part's .dxf file. The unfolding of models with coincident geometry cases is substantially improved.

ViewBase and **smUnfold** commands place unfolded views of sheet metal parts in paper space using new options.

COMMUNICATOR

Communicator now supports the import of McNeel's Rhinoceros (.3DM) files. Importable data types are: B-Reps (Solids, Surfaces, Regions), Curves, Meshes, Text annotations, Blocks and Block references, Materals and Layers.

Customization and Security

BricsCAD V20 is compiled with Visual Studio 2017. C++ extension DLLs (Dynamic Link Libraries) need to be compiled with the same platform toolset in order to be compatible with BricsCAD V20.

BOM (bills of material) status of a component can now be queried via the components API.

RibbonPanelMargin workspace property specifies the margin, in pixels, along the sides of ribbon panels.

Select a workspace on the Workspace tab, and that workspace's property pane can be expanded by dragging the splitter bar.

LegacyCodeSearch, **TrustedPaths**, and **SecureLoad** variables are read-only but whose values are changed by users with administrator privileges.