Practical Autodesk AutoCAD 2021 and AutoCAD LT 2021

A no-nonsense, beginner's guide to drafting and 3D modeling with Autodesk AutoCAD



Yasser Shoukry and Jaiprakash Pandey

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Yasser Shoukry
Jaiprakash Pandey

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BIRMINGHAM - MUMBAI

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Contributors

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> *Most of all, I would like to thank my loving and patient family and friends for their continued support, patience, and encouragement throughout the long process of writing this book.*

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> *I would like to thank my parents for their support and also my sister Kalpana for her encouragement throughout the process of writing this book.*

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Preface

The book is about AutoCAD, which is a computer-aided 2D and 3D drafting software developed by Autodesk. If you are an engineering student, a drafter, or a design engineer, or if you just have great ideas and need a powerful drafting tool to bring them to life, then AutoCAD was designed for you. AutoCAD was the first CAD computer program ever created and is still the best 2D drafting tool on the market. In addition, it has a comprehensive and intuitive set of 3D modeling tools.

This book is written as a guide for you to explore the capabilities of AutoCAD, starting from scratch. We will look at the basic drawing tools and learn about the basic concepts. From there, we'll be moving gradually up to advanced drawing management, external references for working in groups, and 3D modeling.

Who this book is for

This book is for absolute beginners when it comes to AutoCAD who want to learn 2D and 3D AutoCAD right from scratch. The book can be used for AutoCAD and AutoCAD LT, and although it uses the 2021 version of AutoCAD, students using older versions from 2016 onward will be able to follow the book without any issues. This book is not recommended for those using AutoCAD on a Mac, as the user interface will be different.

For installing AutoCAD some of the basic hardware requirements are following:

- 8 GB Minimum and 16 GB recommended.
- 2.5-2.9 GHz processor Minimum and 3+ GHz recommended.
- 64 Bit windows operating system with a minimum disk space of 7 GB.
- Display 1 GB basic with 4 GB recommended.

For a detailed list of hardware, specification check this page on Autodesk knowledge network `https://autode.sk/3cFj7d3`

What this book covers

`Chapter 1`, *An Introduction to AutoCAD*, teaches you about units, the user interface, and the basic navigation tools of AutoCAD.

`Chapter 2`, *Basic Drawing Tools*, is where you will do your first set of drawings with basic drawing commands, such as Line, Circle, and Arc.

`Chapter 3`, *Learning about Modify Commands*, is where you will learn how to modify basic geometries using tools such as fillet, trim, scale, and mirror.

`Chapter 4`, *Working with Arrays and Reusable Objects*, goes into how to make patterns with arrays and symbols using blocks and attributes.

`Chapter 5`, *Managing Drawings with Layers and Properties*, explores managing objects in your drawings using layers and changing properties such as color, linetype, and lineweight.

`Chapter 6`, *Working with Hatches, Text, and Dimensions*, is all about adding annotations such as dimensions, text, and multileaders to your drawings.

`Chapter 7`, *Tables, Isometric and Parametric Drawings*, delves into making your own tables in AutoCAD or using tables from Excel. Also, you will learn how to do parametric and isometric drawings.

`Chapter 8`, *Customization Tools*, looks at making custom linetypes and hatches, as well as customizing the user interface of AutoCAD.

`Chapter 9`, *External References and Dynamic Blocks*, details the use of external references in your drawing, as well as how to make and modify dynamic blocks.

`Chapter 10`, *Introduction to 3D Modeling*, covers how to optimize the working environment in AutoCAD for 3D modeling.

`Chapter 11`, *Creating Primitive 3D Shapes*, will teach you how to create simple primitive 3D shapes.

`Chapter 12`, *Conversion between 2D and 3D*, aims to show you how to convert 2D shapes to 3D objects, and how to create shots and sections from 3D models.

`Chapter 13`, *Modifying 3D Objects*, focuses on how to use conventional modification tools with 3D bodies as well as specialized 3D modification tools.

`Chapter 14`, *Surfaces and Mesh Modeling*, dives into surface modeling and mesh modeling in AutoCAD.

`Chapter 15`, *Paper Space Layouts and Printing*, shows how to prepare your drawings and print them in a professional way.

`Chapter 16`, *Rendering and Presentation*, gives you an overview of how to set up a scene and produce high-quality renders for your 3D models.

To get the most out of this book

You need to have AutoCAD or AutoCAD LT installed in order to follow the book. This book is made with the Windows operating system version of AutoCAD in mind. If you are using AutoCAD on Mac, then you will find a different user interface.

Software/hardware covered in the book	**Operating system requirements**
AutoCAD 2021	Windows 7 or 10

You can get the free 30-day trial version of AutoCAD from the Autodesk website and, if eligible, you can also get the free student version of AutoCAD from the Autodesk student portal.

Download the color images

We also provide a PDF file that has color images of the screenshots/diagrams used in this book. You can download it here: `https://static.packt-cdn.com/downloads/9781789809152_ColorImages.pdf`

Conventions used

There are a number of text conventions used throughout this book.

`CodeInText`: Indicates code words in text, database table names, folder names, filenames, file extensions, pathnames, dummy URLs, user input, and Twitter handles. Here is an example: "To use this tool, click on its icon or use its command alias, `LAYISO`."

Bold: Indicates a new term, an important word, or words that you see onscreen. For example, words in menus or dialog boxes appear in the text like this. Here is an example: "Open a blank drawing and start the **Line** command by clicking on the **Line** tool in the **Draw** panel."



Warnings or important notes appear like this.



Tips and tricks appear like this.

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1
An Introduction to AutoCAD

Welcome to this book about *Practical Autodesk AutoCAD 2021 and AutoCAD LT 2021*. This book covers the essential tools, commands, and features that you need to know to get up and running with Autodesk AutoCAD software.

Autodesk—the parent company that makes AutoCAD releases a new version of AutoCAD every year, and with the new release, comes an improved feature set, bug fixes, and new commands. AutoCAD is also the flagship product of Autodesk; it's been the core drafting software for years and it still has a dominant position in the CAD industry as a design and drafting tool.

With AutoCAD, you can make 2D drawings and 3D models and create photorealistic renderings for a presentation. Though popular in the industry as a drafting tool, AutoCAD is much more than that. It is feature-packed with 2D drafting, 3D modeling, drawing management tools, drawing collaboration tools, 3D rendering features, and most recently, cloud-based features have been added to the core AutoCAD package.

The most recent *One AutoCAD* update now allows you to access other AutoCAD verticals, such as AutoCAD Electrical, Civil, and Architecture with your subscription, and with that, you now have access to a big library of blocks and tools that were only available in the respective vertical product.

In this book, we will use AutoCAD 2021 to explain the tools and commands, but you can use this book even if you are using older or newer versions of AutoCAD. Along with the usual commands and toolsets, this book also covers the new features of AutoCAD 2021.

We will cover the following topics in this chapter:

- AutoCAD LT and AutoCAD for Mac
- Understanding the AutoCAD user interface

- Navigating in AutoCAD
- Units and limits

So, let's get started with the technical requirements and the PC specification you need for the smooth functioning of Autodesk AutoCAD software.

AutoCAD LT and AutoCAD for Macintosh

Autodesk AutoCAD is primarily made for the Windows operating system, but it has a **Macintosh** (**Mac**) version as well. The Mac version has a different user interface and is somewhat limited in features compared to AutoCAD for Windows.

So, if you are a Mac user, you will find the user interface a little different but still, it's the same software and functionally, the commands will work the same. The major differences between the Mac version and the Windows version of AutoCAD are the following:

- Mac has a classical AutoCAD-type user interface whereas the Windows interface is modern and ribbon-based.
- Some tools will work on the command bar instead of the palette in the Mac version.
- Many Windows AutoCAD tools are either reworked or completely missing in the Mac version.

So, in a nutshell, it is recommended to use the Windows version of AutoCAD and this book has also been written for the Windows version of AutoCAD, so Mac users may find it difficult to follow along due to the previously listed differences. However, it is still workable and, with a little bit of trial and error, you will be able to use this book with the Mac version of AutoCAD, too.

Another type of AutoCAD that is currently available is AutoCAD LT and it is a sort of lighter version of AutoCAD. The LT version, though cheaper, does not support most of the 3D tools and LISP.net customizations. Despite these missing features, AutoCAD LT still supports AutoCAD files and you can seamlessly transfer files between AutoCAD and AutoCAD LT.

This book is fully compatible with AutoCAD LT for Windows and you can use this book if you are an AutoCAD LT user. You will, however, find the AutoCAD-only tools missing.

Now that we have looked at the differences between AutoCAD for Windows and Mac and AutoCAD LT, let's explore the user interface of the program.

Understanding the user interface

AutoCAD 2021, just like its predecessors, has a ribbon view with panels and tabs. You can launch the AutoCAD software using its icon on the desktop or from the `AutoCAD 2021 – English` folder in the start menu. When the software starts up, you will see an interface like this:

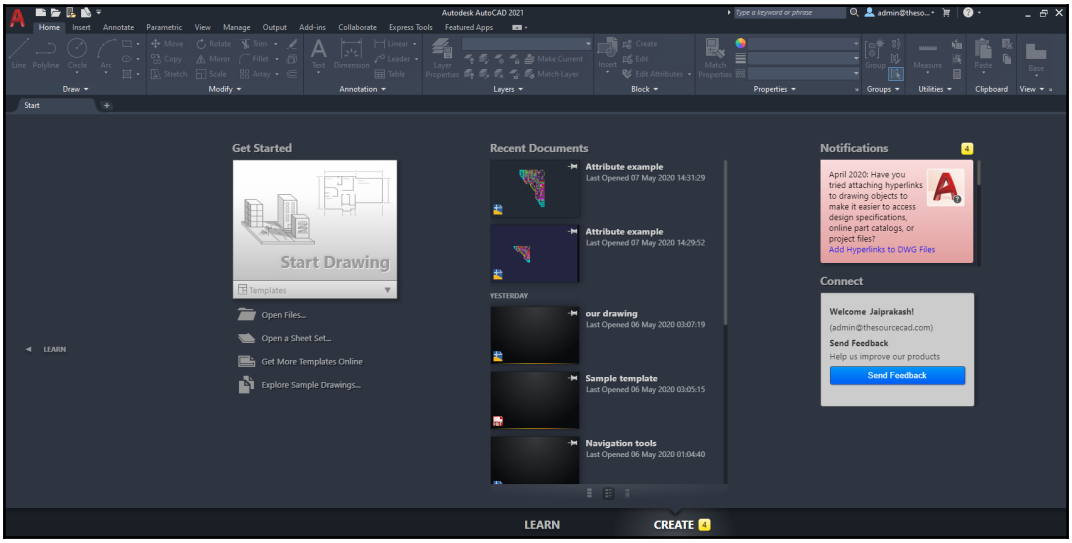


Figure 1.1: The start screen of AutoCAD

Here, we have three columns: **Get Started**, **Recent Documents**, and **Notifications**. **Get Started** has all the options you will need to start a new drawing or open an existing one. The **Recent Documents** column shows a list of all the recently opened drawings and you can use this tab to directly open any drawing. The **Notifications** column has information related to your account, such as an update notification and A360 account notifications. A360 is the cloud-based storage provided by Autodesk with an AutoCAD subscription and apart from cloud storage, you also get many other features, such as collaboration, markups, and rendering, on the A360 cloud account.

To start a blank drawing template, click on the **Start Drawing** box in the **Start** tab. Clicking on this box launches the default template listed under the **Templates** list. By default, a `acad.dwt` or `acadiso.dwt` template will be used to open the drawing. These templates contain settings such as units, limits, and precision and at this point, selecting any of these templates won't make a difference as we will learn about making templates and using them in drawings later in this book:

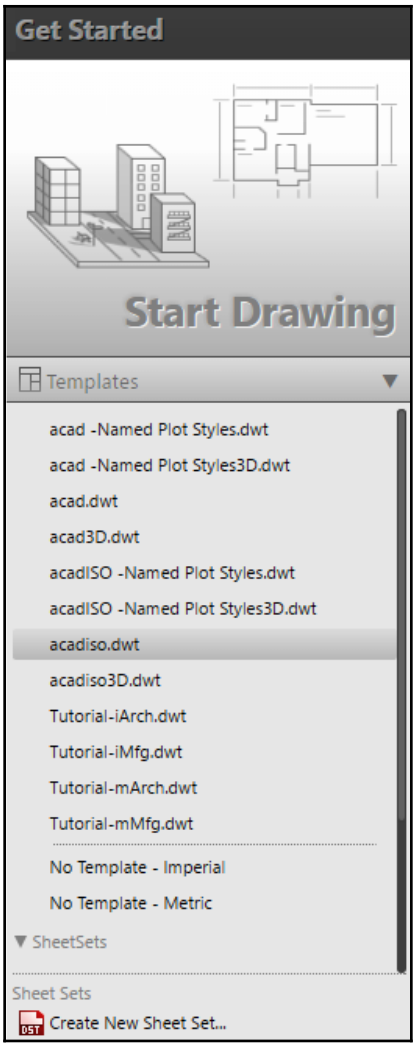


Figure 1.2: The Templates list in the Start Drawing tab

So, when the blank drawing loads up, your interface will look similar to the start screen, but this time with an active ribbon and blank drawing area. Let's talk about all the components in this workspace:

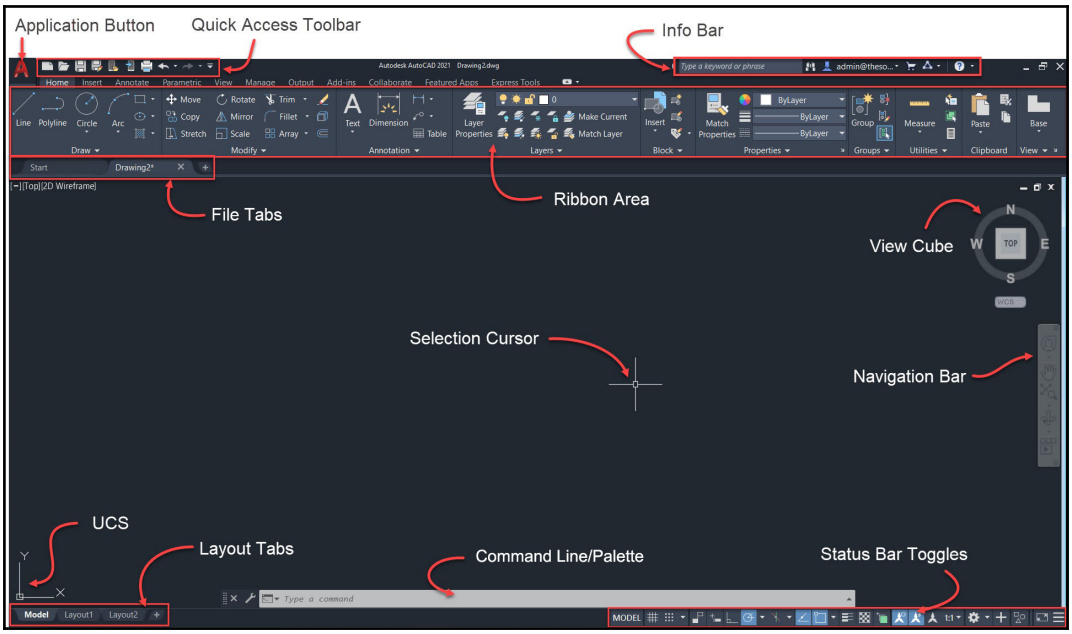


Figure 1.3: The AutoCAD 2021 user interface

The user interface parts are labeled in *figure 1.3*, and here is a short description of each of the parts mentioned in this screenshot.

The application button

Clicking on this big **A** icon will open a menu with options to **open an existing drawing**; **open a new template**; **save**, **print**, or **export drawings**; and other common options.

The quick access toolbar

This toolbar contains most of the frequently used tools from the application button. To add or remove an option from the quick access toolbar, simply click on the arrow pointing downward at the end of the toolbar and check or uncheck the option you want to keep or hide:

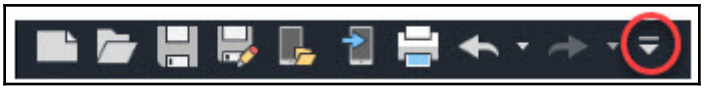


Figure 1.4: The quick access toolbar

The quick access toolbar contains tools such as **open**, **save**, **plot**, **layer**, **workspace**, **sheet set manager**, **properties**, and the menu bar. Some of these tools are also available in the application menu of AutoCAD.

The info bar

In the input field of the info bar, which reads **Type a keyword or phrase**, you can type the name of any command or tool to get more information about it. Simply type the name of the command and hit the *Enter* key to get the help file related to the command. You need to be connected to the internet in order to use this feature. This bar also contains other features, such as your **sign-in account**, **access to the Autodesk app store**, and more help options.

The ribbon area

The ribbon area contains all the tools and commands properly arranged into different tabs and panels. **Home**, **Insert**, **Annotate**, **Parametric**, and so on are the tabs in which different panels are placed in a meaningful order. For example, the **Home** tab contains the **Draw** and **Modify** panels, which in turn contain most of the frequently used commands:

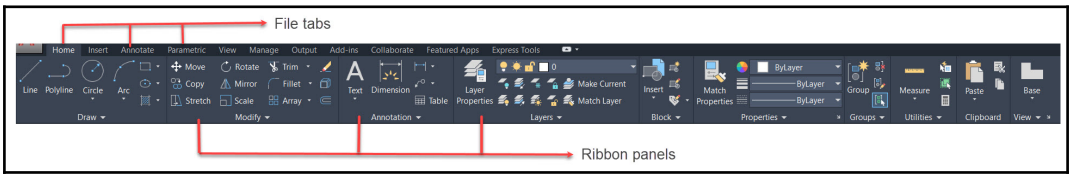


Figure 1.5: Tabs and panels in the AutoCAD ribbon area

Similarly, other less frequently used panels and subsequently, the commands, are in other tabs, such as the **Parametric** or **Express Tools** tabs, which are other tabs found in the ribbon area.

The panels and tabs are not permanent and you can hide or reveal them by simply right-clicking on any panel. To hide or show any tab, just right-click on any panel and go to the **Show Tabs** option. Then, check the tab that you want to show; or similarly, to hide any tab, repeat the same process and uncheck the tab from the right-hand side context menu:

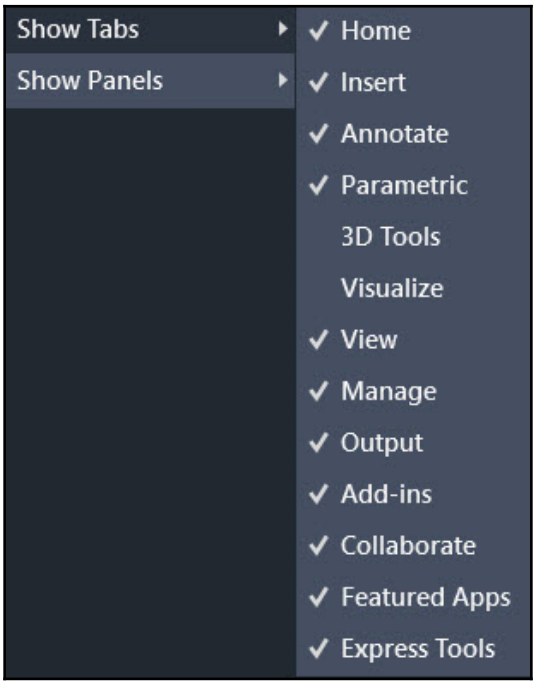


Figure 1.6: The Show Tabs and Panels options

To hide or show any panel of a tab, just go to the tab and right-click on any of its panels, then right-click and go to the Show Panels option and check or uncheck the panels that you want to show or hide.

The file tabs

The file tabs show the drawing files that are open in the current session of AutoCAD. You can open as many file tabs as you want and to open any new blank drawing, you can click on the **+** icon right next to the last file tab:

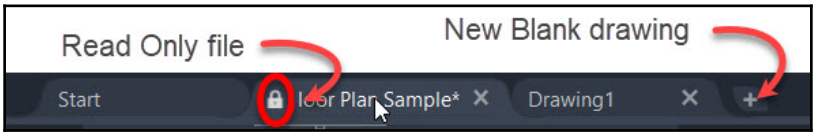


Figure 1.7: The file tab options

The padlock icon in a file tab indicates that the file is read-only and you can only view that kind of file and not modify it. Hovering the cursor over any open tab also shows the thumbnail preview of the drawing. To open the drawing of the selected tab, simply click on the thumbnail that shows up on the cursor.

The ViewCube and navigation bar

The ViewCube is an AutoCAD-only tool and is not available in AutoCAD LT, whereas the navigation bar is available in both AutoCAD and LT, but in LT, it is somewhat limited in features. If you don't see the ViewCube, simply go to the top left of the display area and click on the **–** sign and select **ViewCube** and **Navigation Bar**:

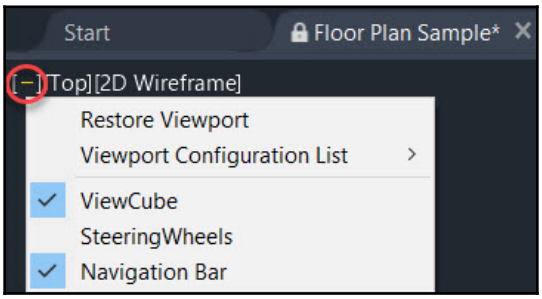


Figure 1.8: The ViewCube and Navigation Bar options

This **—** sign won't show up on AutoCAD LT, but you can use the `NAVBAR` command to turn it on or off.

The selection cursor

This is the cursor that shows up on the AutoCAD screen it is basically two perpendicular lines with a small cube at the center of the intersection of the lines. This is the default state of the cursor and depending on the selection, your cursor might look different and sometimes additional icons will also show next to the cursor. These are the three most basic cursor modes; we will discuss the other cursor modes in this book as and when needed:

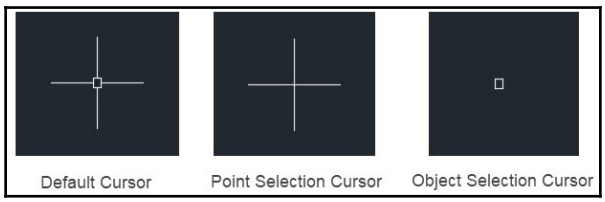


Figure 1.9: The different cursor modes

The Default cursor type appears when none of the commands are active, so essentially, the first cursor type is visible in the default state of AutoCAD. When you activate a command where you need to select a precise point, the second cursor—Point Selection—appears, which is just two mutually perpendicular lines intersecting at the midpoints. The Object Selection cursor will be visible when a command where object selection is required is active.

No matter which of the cursors shown in the preceding figure is active, if you want to return to the default state of the cursor, then simply press the Esc key on your keyboard. You can press the Esc key multiple times and it won't affect the default state of your cursor.

The command line/palette

Just at the bottom of the drawing area, you will see the command line or command palette. In this book, I will refer to it as the command line. All your typed commands will show up in this command line; you don't need to click on the command line to start typing any AutoCAD command, just start typing without selecting anything and the input will be added to the command line directly.

In its default state, the command line will show the **Type a command** message and when you select any command, it will show the name of the command and, next, a set of instructions in the command line, as in the following screenshot:

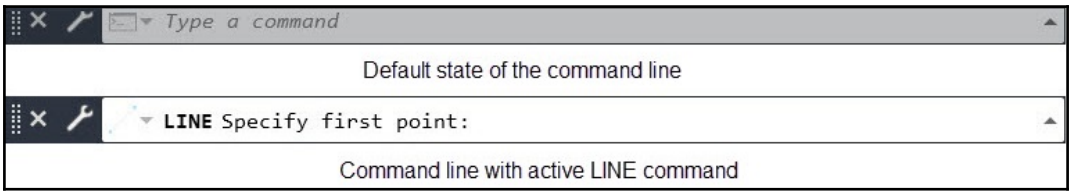


Figure 1.10: The different states of the command line

If for some reason you don't see the command line, then press the *Ctrl + 9* keys to turn it on; you can press the *Ctrl + 9* keys again to hide the command line. This command line is like a floating palette and you can even resize it or move it around in the drawing area. You can see a list of previously entered commands in the command line by pressing the *F2* function key.

The user coordinate system

The **user coordinate system** (**UCS**) shows the **X**, **Y**, and **Z** directions of the Cartesian coordinate system in the drawing area. Usually, the UCS will be visible at the bottom-left corner of the drawing area:

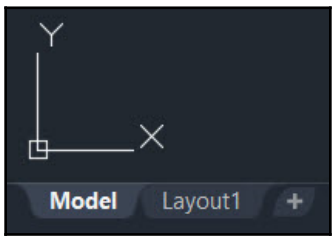


Figure 1.11: The UCS with an *X* and *Y* axis

In the default state of the 2D workspace, the UCS will only show the *X* and *Y* axis; the *Z* axis, which points outward from the AutoCAD screen, will not be visible. The point of intersection of the *X* and *Y* coordinates in this UCS is the origin, or `0,0`, point. We will discuss more about using the Cartesian coordinate system later in this chapter.

The layout tabs

The layout tabs are like the sheets of your drawing on which you can arrange different views of the drawing and finally plot multiple sheets. For example, if you have a house plan in a model space where the floor plan, elevations, section, and detail drawings are all made in the model space, then you can use the layout tabs to place them separately on different sheets, such as a layout for plan view, another for all elevations, and another for all detail drawings. By default, the **Model** tab will be active and for the most part of this book, we will work inside the **Model** tab. We will learn about layouts in `Chapter 15`, *Paper Space Layouts and Printing*, of this book.

The status bar toggles

The options in the status bar help in making precise drawings. The tools in the status bar, such as **Object Snap**, help you to select precise points, such as the midpoint and endpoints of any geometry, and tools such as polar and ortho mode help you to make lines that are inclined at any angle or horizontal and vertical lines. The toggles on the status bar can be turned on by clicking on them once, and similarly, clicking on them again turns them off. Most of the status bar tools have their function key shortcuts as well and you can see the function key shortcut of any status bar option by hovering your cursor over it:

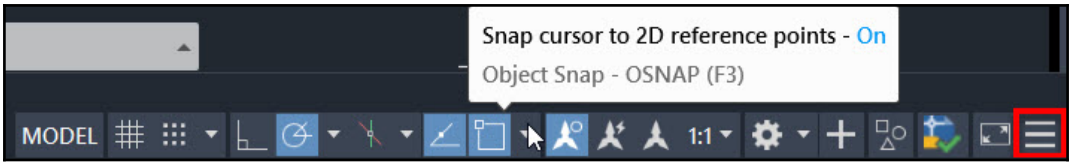


Figure 1.12: The status bar toggles with customization option

Not all of the status bar options are visible in the status bar and to make the options visible or hide them, use the customization option, which looks like three lines at the end of the status bar. Just click on the three lines, shown in the red circle highlighted in figure 1.12, and check the options you want to show on the status bar and uncheck the ones you want to hide. We will learn more about the status bar options later in this book.

So, now that you are familiar with the user interface and the way the ribbon area works, let's move on to the navigation tools. In the next section, we will talk about the navigation tools and how these tools can be accessed using the mouse or from the AutoCAD interface.

Navigating in AutoCAD

To navigate in AutoCAD, a three-button mouse with left-click, right-click, and the middle scroll wheel is recommended. The laptop touchpad can be used but with a few limitations.

If you have access to lesson files, then open `drawing 1.1` in AutoCAD. If you don't have lesson files, you can open any drawing file to follow along. This is a sample drawing by Autodesk AutoCAD that is available as a read-only file when you launch the software for the first time.

Selecting and panning

To select any object in the drawing, hover your cursor over it and left-click. The object will be selected and blue rectangular dots, called multi-function grip, will show up. The selected object will also get highlighted:

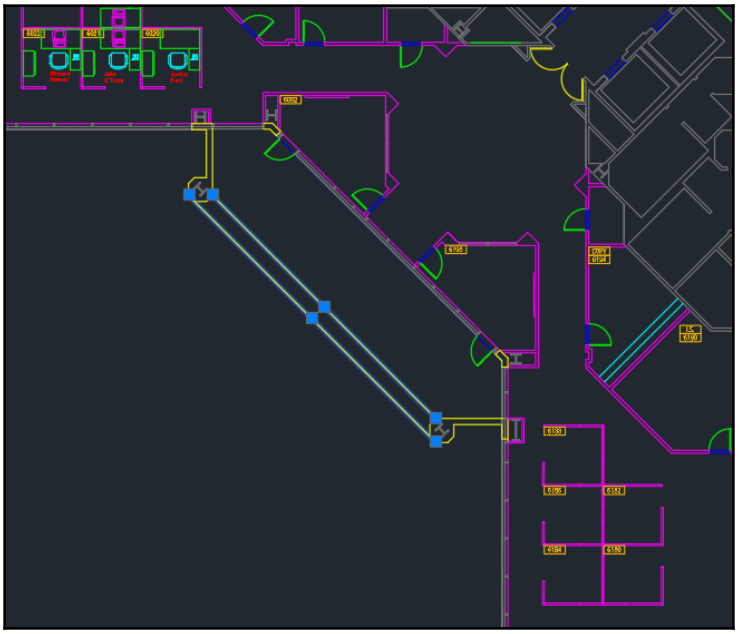


Figure 1.13: Selected lines highlighted in the drawing

To remove the object from your selection, simply press the *Esc* key on your keyboard. If the selection is active even after pressing the *Esc* key once, you can press the *Esc* key multiple times as well.

To select multiple objects, you can click on them one by one; you don't need to press and hold the *Shift* key to make multiple selections. If you want to remove all the objects from your selection, you can press the *Esc* key, but if you only want to remove selected objects from your selection, then press and hold the *Shift* key and then click on the objects you want to remove from the selection. Using this method, you will be able to keep the complete selection and only selected objects will be removed from the selected group.

To pan the complete drawing in the drawing area, press and hold your middle mouse wheel so that the cursor changes into a hand icon, and then move your mouse. The complete drawing will pan in the drawing area and it will follow the movement of the mouse:

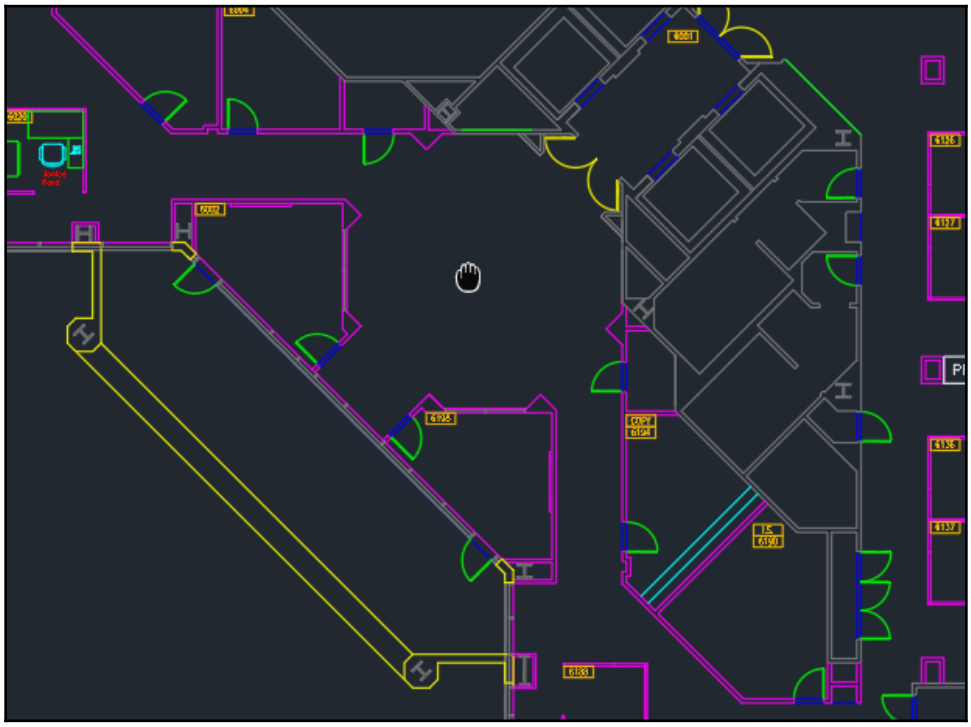


Figure 1.14: The hand type icon of the pan tool

You can also select the pan tool from the navigation bar and then use left-click to pan the drawing.

Just like selecting and panning, zooming is also pretty easy and there are many ways we can zoom our drawing area. We will discuss this in the next section.

Zooming in and out

To zoom in and out, you can rotate the mouse wheel. Take your mouse pointer to the part of the drawing that you want to zoom in or out of and then rotate the wheel. Rotating it in one direction will zoom in and in the other direction will zoom out. The point where you place your cursor will become the center of zoom.

If your drawing is very large or very small and even off-screen, then simply zooming in or out may take time and a lot of zooming and panning will be required to fit the drawing to the visible space. To overcome this issue, you can use the zoom extents option, which fits all the objects in the drawing to the visible space. To use zoom extents, double-click the mouse wheel and everything in your drawing will fit into the available space.

The zoom tool does not affect the actual size of the drawing; it only affects the scale. A line of a 1 mm length will remain equal to 1 mm, no matter how big or small you make it using the zoom tool.

You can also use the navigation bar to use the zoom tools. To use the zoom tools from the navigation bar, click on the arrow underneath the zoom icon in the navigation bar and you will see a list of zoom options, as shown here:

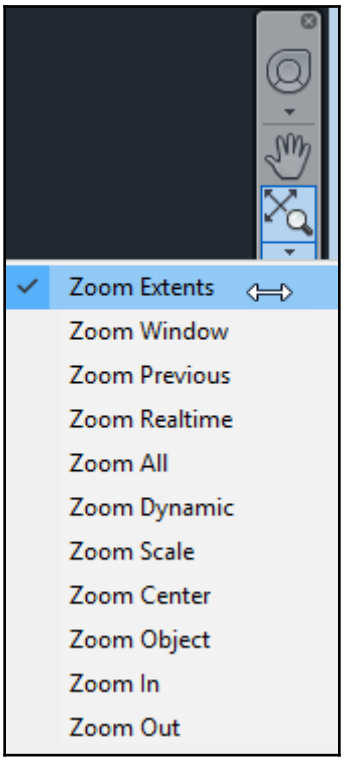


Figure 1.15: The zoom options in the navigation bar

Here is an explanation of some of the options available in the zoom menu:

- **Zoom Extents**: This option fits all the available objects in the visible drawing area. It can be used by double-clicking the scroll wheel.
- **Zoom Window**: This option lets you zoom a part of a drawing by making a window. To use this feature, click at a point in the drawing area then let go of your cursor. Then, click again at another point to complete the window. The drawing will be zoomed to fit the selected window.
- **Zoom Previous**: This option will take you to the previous zoomed state of your drawing. It's like an undo option for the zoom command.

- **Zoom Realtime**: This option lets you zoom your entire drawing by moving your cursor. When you select the option, your cursor will look like a magnifying glass. Now, click anywhere in the drawing area and hold your cursor, then move up to zoom in and move down to zoom out.
- **Zoom All**: This option fits the drawing as per the limits set in the drawing. We will discuss more about limits later in this book.
- **Zoom Object**: This option allows you to select the object(s) from the drawing and then only your selected object(s) is zoomed to fit the screen.

So, these are the zoom options available in AutoCAD. Although we have discussed lots of zoom tools here, you will primarily require the mouse wheel zoom in and out and the **Zoom Extents** tool, which is available by double-clicking the mouse wheel. Other zoom tools are not very frequently required but whenever you need them, you now know where to find them. In the next section, we will start learning about making selections using different selection sets available in AutoCAD.

Making selections

There are many ways of making selections in an AutoCAD drawing and in this section, we will discuss most of these selection features. A drawing may contain lots of objects and making a precise selection will be key to quickly adding properties to the correct set of objects and making modifications to them. So, the selection tools are essential to quickly and efficiently designing and drafting a workflow in AutoCAD.

A simple selection

To make a simple selection, click on any object in a drawing and it will be highlighted, indicating that it is selected. To include multiple objects in this selection set, simply click on more objects and they will be added to your selection set.

If you want to disable the multiple selection feature, you can use the `PICKADD` system variable. Type `PICKADD` then press *Enter* and then type `0` and press *Enter* again.

Now, AutoCAD will allow you to make only one selection at a time. If you select another object, the previously selected one will be removed from the selection. To add multiple objects to the selection in this condition, you need to press and hold the *Shift* key and then click on the objects to add them to the selection set.

If you have added multiple objects to your selection and you want to remove some of them, press and hold the *Shift* key and click on the selected object again and it will be removed from the selection set.

For this book, we will keep the `PICKADD` system variable value to `2`, so type `PICKADD` and press *Enter*. Type `2` and press *Enter* again to change the settings to the default.

The selection window

Click anywhere in the drawing area then let go of your cursor and move your mouse to the right. You will see a window with a solid boundary that is, by default, blue in color. This window is the selection window and all the objects that are completely inside this window will be added to the selection set. To finish making the selection, simply click again and the objects will be selected:

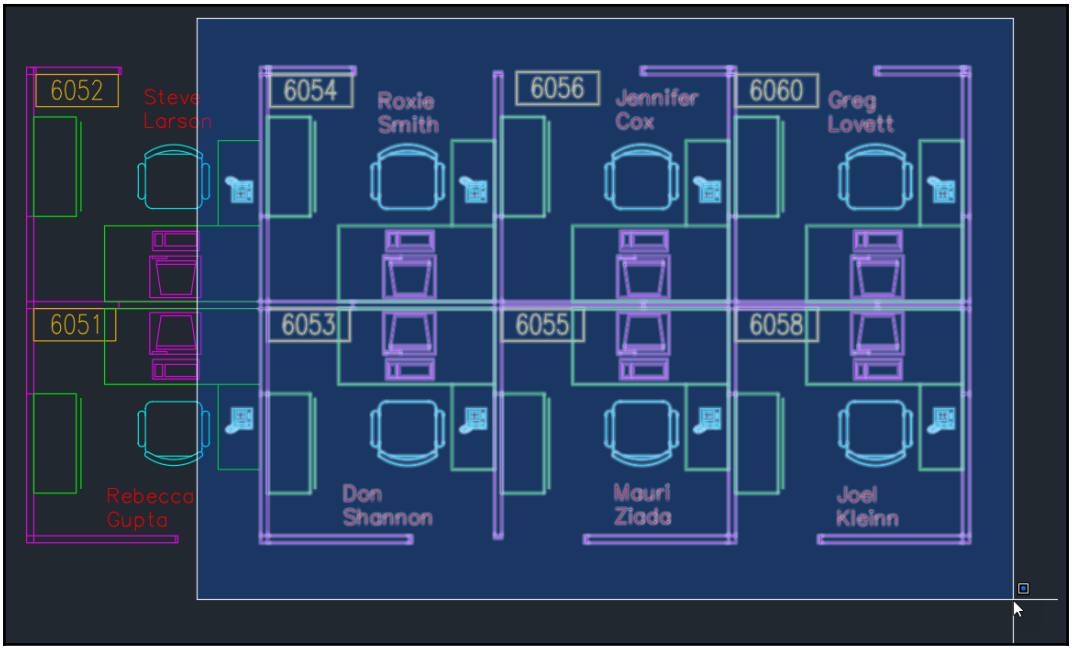


Figure 1.16: Objects selected with a selection window

In *figure 1.16*, all objects that are completely inside the selection window are highlighted and will be selected, whereas objects that are partially inside and partially outside will not be included in the selection set.

The crossing window

Click anywhere in the drawing area then let go of your cursor and move it to the left. A window with a dotted boundary will be made, which is called the crossing window.

All the objects that are inside this window, as well as objects that even touch the boundary of the window, will be included in the selection set. In *figure 1.17*, the same drawing and the same window have made a different selection. As you can see, the chair, desk, and so on of the **6052** and **6051** cabins are also selected, along with all the objects that are completely inside the crossing window, even though they are partially outside the window:

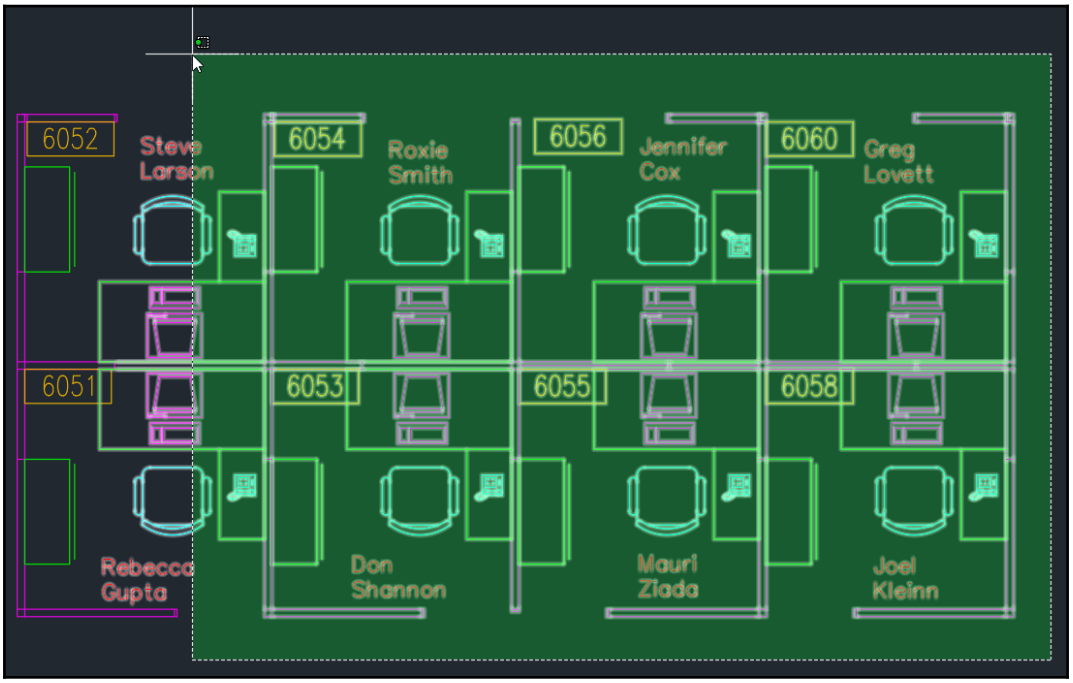


Figure 1.17: Objects selected with the crossing window

To make the selection, click again and the objects will be included in the section set.

The "window lasso" selection

This one is a relatively new selection set. To make a lasso selection, click anywhere in the drawing area then hold down your cursor and move to the right. You will get a freehand selection area with a solid boundary and all the objects completely inside this selection area will be selected:

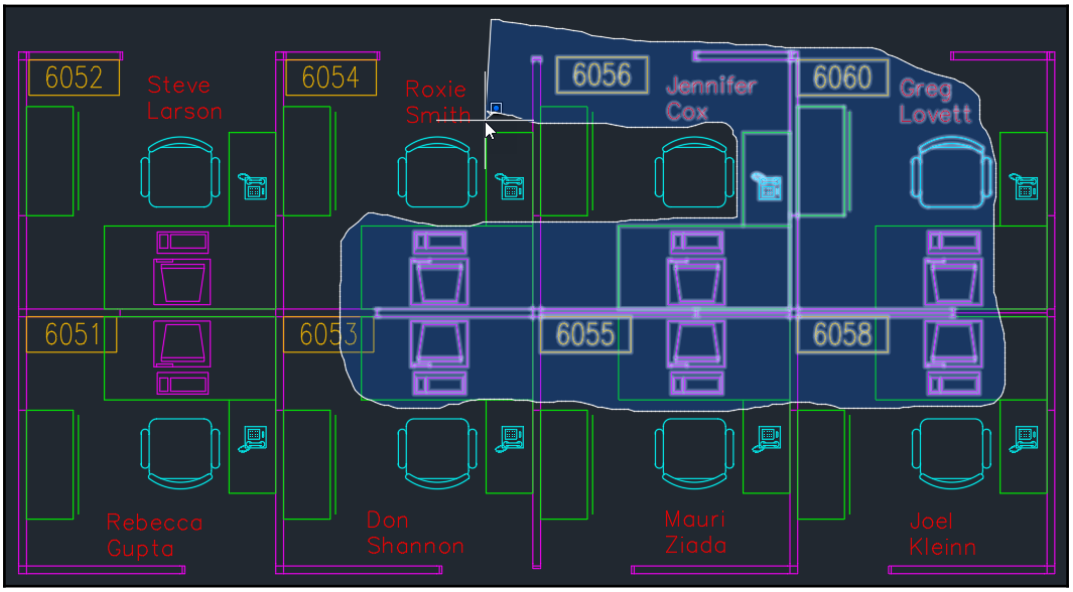


Figure 1.18: The "window lasso" selection

This selection set is like the selection window but in this case, instead of a rectangular window, you have the option of making a freehand selection, giving you more control.

The "crossing lasso" selection

To make this selection, click anywhere in the drawing area and hold down your cursor, then move your mouse to the left. You will get a freehand selection area with a dotted boundary. This section set is called the crossing lasso selection and, in this selection, all objects that are completely inside the selected area, as well as objects on the boundary, will be selected:

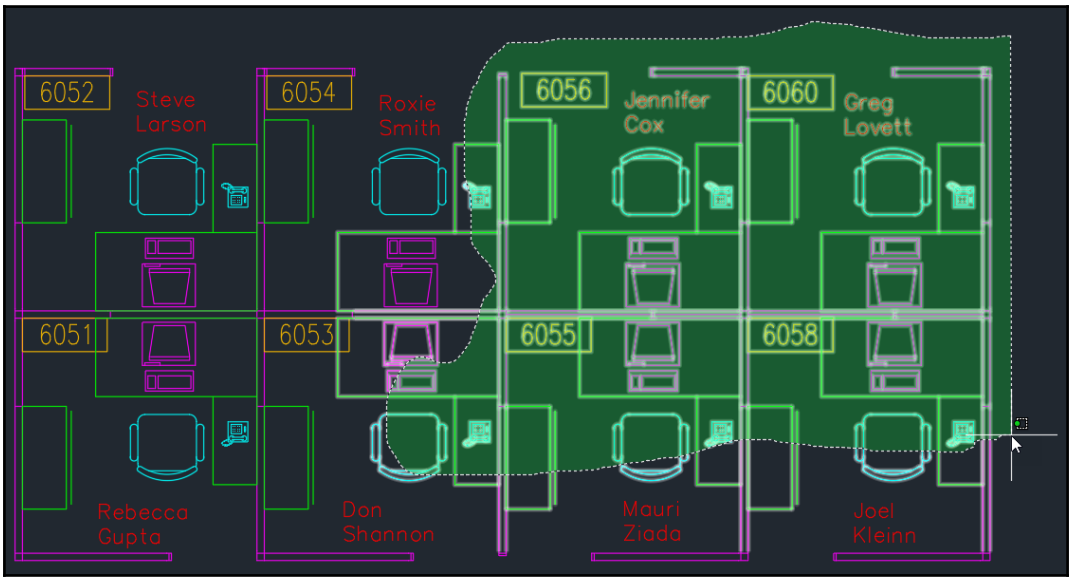


Figure 1.19: The "crossing lasso" selection

So, this was all about making selections in AutoCAD. In the next section, we will learn about setting the units and limits, which is essential to any drawing.

Setting units and limits

When you start a new drawing, its template has settings for units and limits that are used by default. You can use the settings used in the template or make your own settings for units and limits. In this example, we will open a blank drawing then set the units and limits in it and then we will save the drawing as a template file. To start a new drawing in AutoCAD, you can click on the start panel of the **Start** tab or select one of the readymade templates under the start panel.

The list of ready made templates will show up underneath the **Start Drawing** panel when you click the arrow pointing downward in the panel:

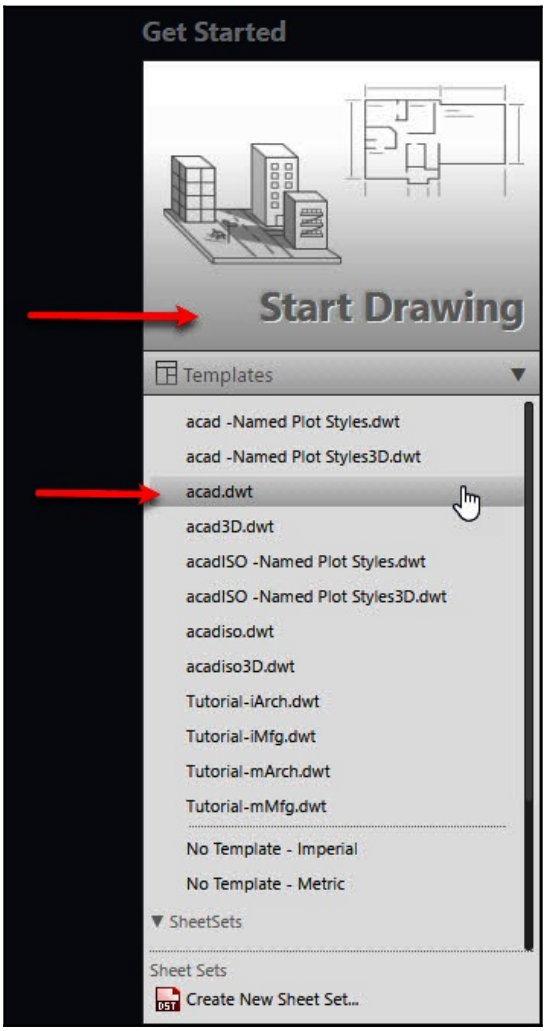


Figure 1.20: The Start Drawing panel and the Templates list in the Start tab

Figure 1.20 shows the **Start** tab with the start panel and the list of readymade templates. **Templates** are just a collection of settings and predefined objects that you can save to reuse with a new drawing. Here, we will start by setting some settings, such as units and limits, and then we will convert the settings into a template file.

Setting units

When you start a drawing with any template shown in *figure 1.20*, you will have some predefined unit settings, which we will modify now.

For this example, select the `acad.dwt` template from the start panel and a blank drawing will open up.

Type `UN` and press *Enter* to open the unit window. Alternatively, you can also click on the application button and select the **Drawing Utilities** option and then **Units** from the next menu:

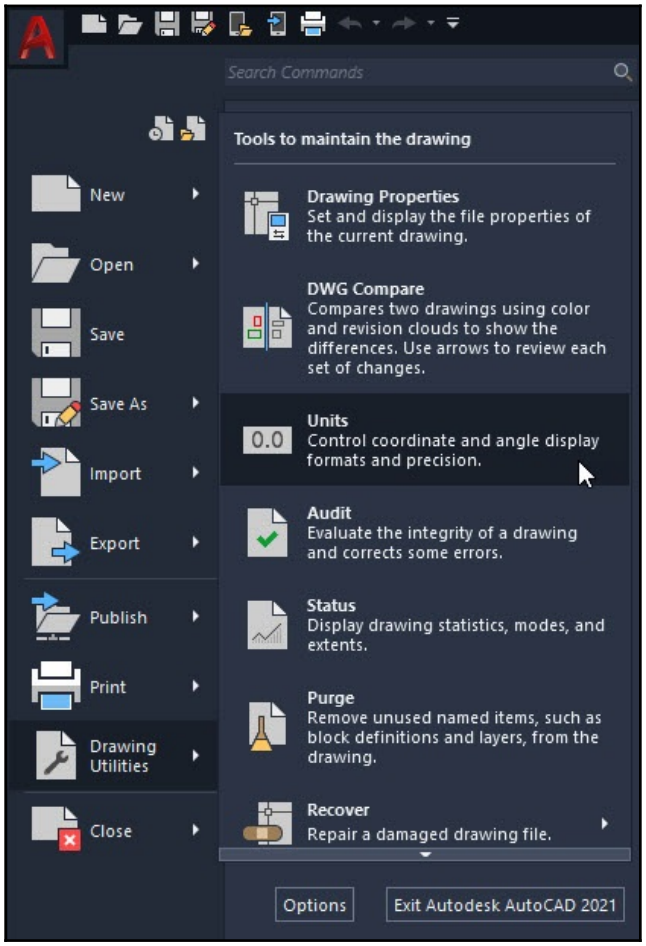


Figure 1.21: The Units option in the application menu

The **Drawing Units** window is divided into several panels, such as **Length**, **Angle**, **Insertion scale**, **Sample Output**, and **Lighting**. Let's talk about these options in the **Drawing Units** window:

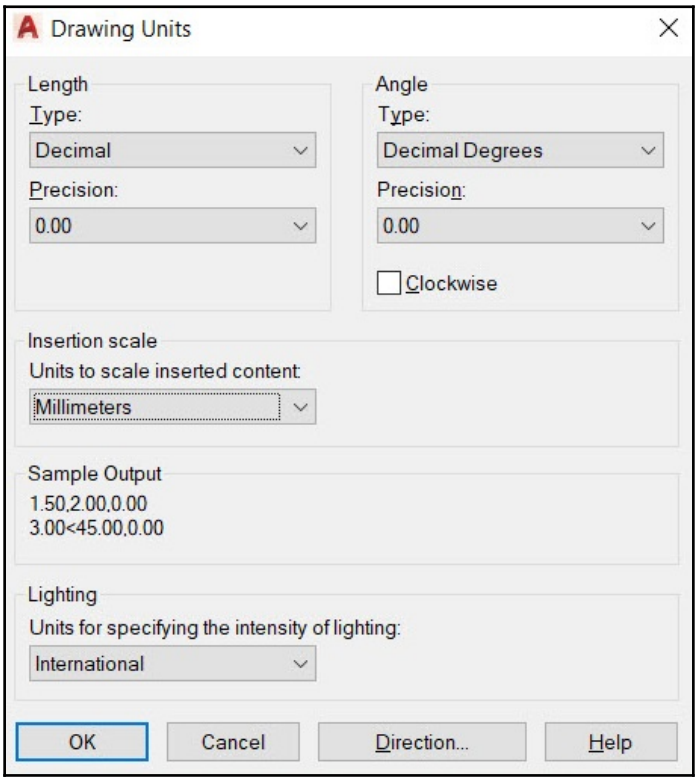


Figure 1.22: The Drawing Units window

In the **Length** panel, the drop-down window has many different length types, such as **Architectural**, **Decimal**, **Engineering**, **Fractional**, and **Scientific**. You can select **Decimal** from these options if you want a simple decimal type output, such as `12.2345` or `245.01289`.

The **Architectural** length type will show output in the form of feet and inches, such as `12' 6"`, where `12` is in feet (shown with the single apostrophe sign) and `6` is in inches (shown with the double apostrophe sign). The **Architectural** format can be used when you want to enter the length in terms of feet and inches, and it's mostly useful for civil and architectural drawings.

Other formats will have their respective representation types and you can select any from the list, as per your drawing and project requirements.

The next option of **Precision** controls the number of decimal places that will show in the drawing. This precision is not the precision with which AutoCAD calculates the dimensions. Instead, it's the precision that is displayed on the drawing. AutoCAD will always measure your drawing with maximum possible precision, no matter what precision you select in the **Drawing Units** window.

Just like the length type, the settings for the angle can be managed from the **Angle** panel. Here, you can select between **Decimal Degrees**, **Degree/Minute/Second**, **Grads**, **Radians**, and **Surveyor's Unit**. The precision is again the number of decimal places for the angle that will be displayed on the drawing.

In AutoCAD, the angle is measured in an anticlockwise direction and hence, anticlockwise is the positive angle. If, however, you want to make a clockwise direction the positive angle, click on the checkbox that says **Clockwise** in the **Angle** panel:

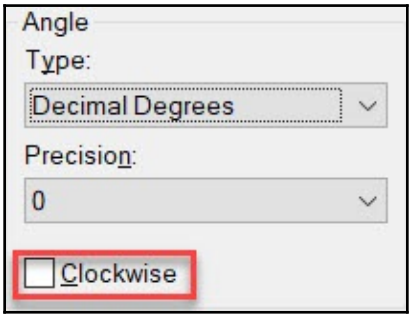


Figure 1.23: The Clockwise checkbox in the Angle panel

From the **Insertion scale** panel, you can change the unit type of the current drawing. All the drawings inserted from external files will be scaled to this selected unit no matter what the unit of the original drawing is.

There are lots of units in this list and you can even select the **Unitless** option from the list if you don't want to use any unit in the drawing.

Finally, the **Lighting** panel has two options: **American** and **International**. You can select one of the options from this list. The **Lighting** option will be used in the rendering section of chapter 16 and it has no application in the 2D drawing environment.

So, after setting the settings, click **OK** and the settings will be applied to the current drawing.

In this case, I am selecting the **Decimal Length** type with a `0.00` length precision and the **Decimal Degrees** angle type with a `0.00` precision. I will leave the **Clockwise** checkbox unchecked. In the **Insertion scale** panel, I will select **Millimeters** and **International** in the **Lighting** section. These settings are also shown in *figure 1.22*.

So, now we are done setting the unit settings, let's move on to setting the limits of our drawing area. We will discuss this in the next section.

Setting limits

In AutoCAD, we make our drawing on a scale of **1:1**, which means we are drawing to the actual scale of the project. For example, a square plate of 10 mm by 10 mm will be exactly the size in AutoCAD as well. But the drawing area can make the 10mm by 10mm square very big or small, depending on the zoomed state of the screen. If your drawing is zoomed in, the square may appear bigger and if it's zoomed out, the drawing may appear tiny.

So, depending on the size of the drawing that you want to make in your AutoCAD file, you can set the limits and then these settings can be used wherever needed. In the following example, we will add a drawing limit of `60, 40` units in our drawing:

1. To set the limits, type the `LIMITS` command and press *Enter*.
2. The command line will now prompt you to specify the lower-left corner of the screen from where you want the limit to start. By default, `0,0`, or the origin, is selected. You can specify any other point as well if you want, then press *Enter*.

3. Now, the command line will prompt you to specify the upper-right corner of the screen for the limit. Type the coordinates of the point in the form of *x*, *y* and press *Enter*:

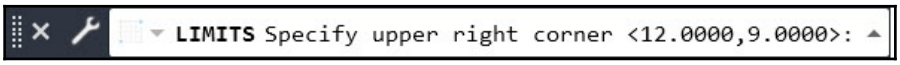


Figure 1.24: The limits command with the upper-right corner prompt

4. Select `0,0` as the lower-left corner and `60,40` as the upper-right corner. After setting the limits, type `Z` and press *Enter*, then type `A` and press *Enter* again in the command line.
5. The screen will reset to adjust according to the newly specified limits.

In our example, the height of the screen will be `40` and the length of the screen will be `60`, or any other that will be proportional as per the screen size. For example, in a square screen, the length will now be `40` as well, but for a rectangular screen, the length will depend on the aspect ratio of the monitor.

Setting the limit will just let you adjust a reference for working on the drawing. This property will not affect your drawings in any way, and you can freely zoom in and out to the change the drawing area and make your drawing even go outside this limit.

Saving settings as a template

So by now, we have set the unit settings and made changes to the drawing limits. It is not always practical to make these changes whenever we want to start with the same set of settings in a blank drawing. You can save these settings as a template file and then these templates can be loaded to start a new drawing with the same set of settings.

In the previous examples, we set the unit and template related settings, and now we will learn how to save these settings as a template file in the following example:

1. Click on the application button in the top left of the AutoCAD window and select **Save** as the option from the menu. Then, go to the **Drawing Template** option.

2. The default folder location for the templates will open up. You can save your template here or choose another location from this window. Select the location, give your template file a name, and hit the **Save** button:

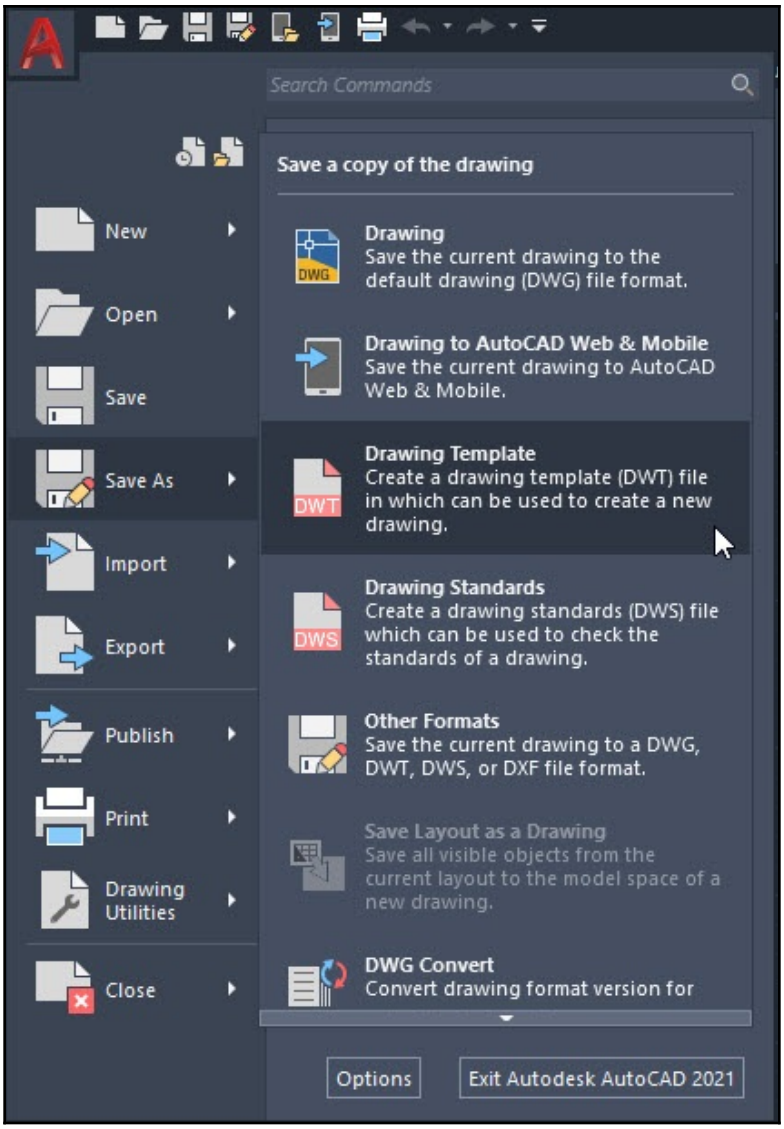


Figure 1.25: Saving template files

3. Your file will be saved with a `DWT` file extension, which is the extension for AutoCAD template files.
4. To load the template settings, simply double-click the next time you want to open a new drawing with the same settings as defined in the template.
5. After opening a template once, if you open a new drawing from the **Start** tab or by clicking the **+** icon in the file tabs, AutoCAD will load the last-used template, which in this case is the template saved by you.
6. To change the template to some other ready-made template, you need to once again open another template from the **Template** drop-down menu in the **Start** tab or click on the **+** icon in the quick access toolbar and select a template from the list of templates to open a new drawing.

When you simply save the drawing file by clicking the **Save** icon in the quick access toolbar or by clicking the **Save** icon in the application menu, the file will be saved in a `DWG` format. `DWG` format is the native drawing format with which AutoCAD drawing files are saved.

So, this is how you can save settings and other customizations as a template file in AutoCAD. The template can not only save the settings related to unit and limits, but it can also save other settings, such as status bar toggles and system variables, which we will learn about later in this book.

Now, let's talk about saving the drawing in its native `DWG` format.

Saving a drawing file as DWG

To save a drawing, simply click the **Save** button in the quick access bar or use the `SAVE` command. The **Save Drawing As** window will open up. Specify a location, give your file a name, and save it.

If you save your file on a cloud account, such as OneDrive or Dropbox, AutoCAD will save the drawing history as well. This drawing history is a new feature that you will find in the AutoCAD 2021 version only.

When you open a drawing saved in OneDrive, Dropbox, or Box, the **Version History** palette will open up, which will show a list of all drawing revisions, as in the following screenshot:

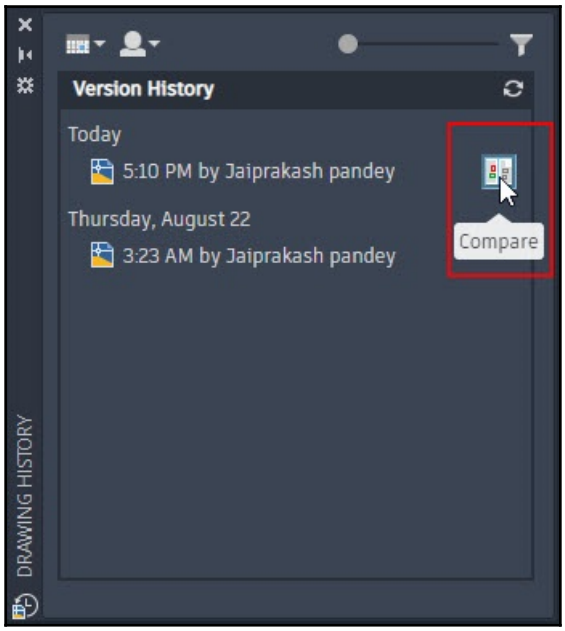


Figure 1.26: The drawing history palette with drawing revisions

To open any older revision of the drawing, hover your cursor over the drawing and you will see a drawing **Compare** icon, as in *figure 1.26*.

Click the drawing **Compare** icon and older revisions of the drawing will open up in a window, showing new and older drawings compared using a feature called drawing compare, which will be discussed later in this book.

Once you are done checking the compared drawing, click the green checkmark in the **Version Compare** bar, as shown here:

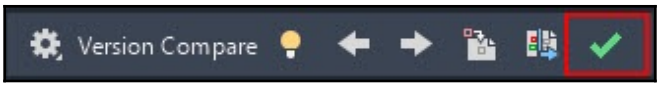


Figure 1.27: The Version Compare bar

If your drawing was saved by many other members, then the drawing history will show the name of the person who saved the drawing and you can use the name filter to select the drawings saved by any project member. You can also filter the drawing versions with respect to their dates. To open any drawing version, double-click on it from the drawing history palette and that version will open up in the drawing area.

So, this was all about the basic setup of AutoCAD and opening and saving drawing files. Before we move on, let's summarize what we have learned in this chapter.

Summary

Congratulations on finishing your first chapter! In this chapter, we have learned about the user interface of AutoCAD and how it works for different versions and operating systems. We also learned how to navigate in AutoCAD using our mouse and on-screen navigation tools. We then learned about the command line and selection sets. Finally, we learned how to add units and limits to our drawing and also how to save these settings as a template file that can be used repeatedly for new drawings. This chapter was essential to building your foundation for the software and for further learning.

In the next chapter, we will start to learn about the basic drawing and modification tools and using these basic tools you can start making your first set of AutoCAD drawings.

2
Basic Drawing Tools

As you are now familiar with the user interface of AutoCAD and the basic navigation tools, we can move on to learning about the basic draw tools. In this chapter, we will learn how to make simple drawings using the basic draw and modify tools available in AutoCAD. Using these tools, you will be able to make and modify the simplest kind of drawings in AutoCAD. These tools also happen to be the most frequently used ones in AutoCAD.

We will cover the following topics in this chapter:

- Understanding the coordinate system
- Using the `Line` command
- Basic status bar modes
- Making circles and arcs
- Making rectangles and polygons
- The `Move` and `Copy` commands
- The `Rotate` and `Fillet` commands
- The `Trim` and `Extend` commands

So, let's begin by understanding the **user coordinate system** (**UCS**) in AutoCAD.

Understanding the coordinate system

Understanding the coordinate system is essential to understanding the way AutoCAD works. In AutoCAD, you can assign length and angles, as well as coordinate values, to make drawings, but to do all this, knowledge of the coordinate system is essential.

Primarily, these are two types of coordinate systems that we will use to make geometries in AutoCAD, and they are Cartesian and polar coordinates. First let's have a look at what Cartesian coordinates are.

Cartesian coordinates

AutoCAD follows the Cartesian coordinate system, which is a graphical method of assigning coordinates to a point in space. The simple three-dimensional space has three coordinates, namely ***X***, ***Y***, and ***Z***, which are mutually perpendicular to each other, as in the following diagram. The point of intersection of the three mutually perpendicular axes is the origin, which is represented as `(0,0,0)`:

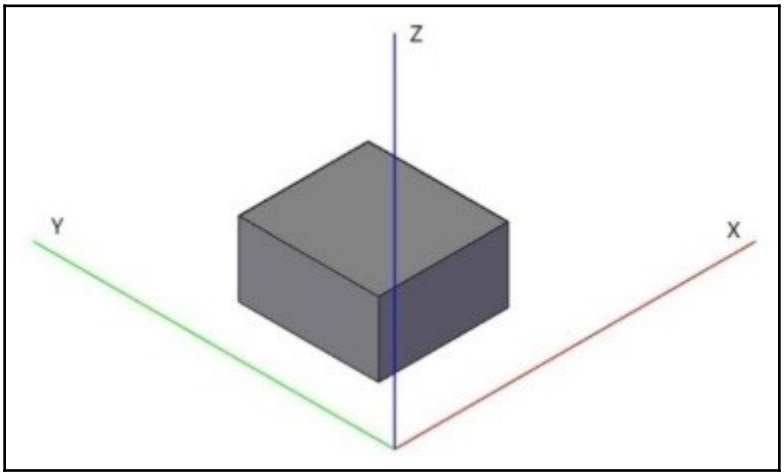


Figure 2.1: Mutually perpendicular coordinates

The position of any point in a three-dimensional space can be specified using these three axes, which are represented by the *X*, *Y*, and *Z* axes in the preceding diagram. But for a two-dimensional space, we only need to use the *X* and *Y* axes to define the position of any point.

In a two-dimensional space, the simple `(X,Y)` coordinate system is used and any point in a two-dimensional space can be defined using these two coordinates only. Take the example of the following graph. Here, the origin is mentioned as `(0,0)`, which is also the point of intersection of the *X* and *Y* axes, represented by horizontal and vertical lines, respectively:

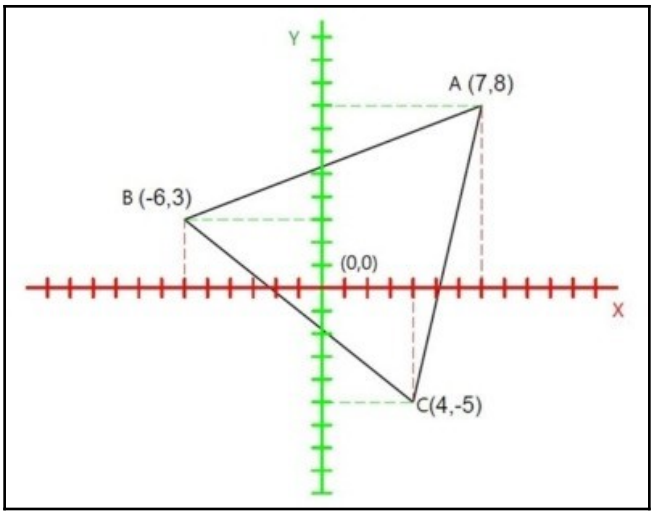


Figure 2.2: Cartesian coordinates

The **A (7,8)** point is at `7` units from the origin along the *X* axis and at `8` units along the *Y* axis. Similarly, the **B (-6,3)** point is at `6` units along the negative side of the *X* axis and at `3` units along the positive side of the *Y* axis. In the case of the **C (4,-5)** point, the distance from the positive side of the *X* axis is `4` units, and its distance along the negative side of the *Y* axis is `5` units.

The *X* axis points to the right of the origin are positive and the points to the left of the origin are negative. Similarly, on the *Y* axis, the points on top of the origin are positive and the points below the origin are negative.

Polar coordinates

Using polar coordinates, we can also represent points in a two-dimensional space. In this case, one polar distance and an angle with respect to the *X* axis are required instead of the *X* and *Y* coordinate values. To understand this clearly, have a look at the following graph:

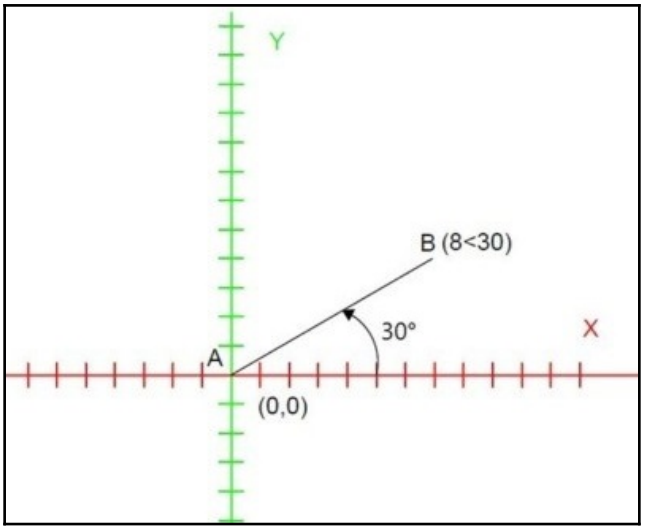


Figure 2.3: Polar coordinates

In this case, the **B** point is represented by **(8<30)**, where `8` is the distance between the **A** and **B** points. Here, **A** is the origin and `30` is the angle between line `AB` and the positive *X* axis in an anticlockwise direction.

This type of coordinate representation, where a point in space is represented by an angle with respect to the positive *X* axis and the distance from the origin, is known as a polar coordinate system.

Throughout this book, we will use both methods of coordinates to make our drawing. Drawings in AutoCAD are not essentially made only with coordinate values. For most of the cases, we use a general approach of direct distance entry and we use coordinates only in specific situations.

In the next section, we will start making our first drawing with the **Line** command using direct distance entry as well as different coordinate values.

Using the Line command

You can select the **Line** command from the command bar, using its command alias, or you can also select the command from the ribbon panel. The **Line** command will make a straight-line segment of any specified length. This is the most basic of the draw tools and one of the most frequently used ones in the drawing workflow.

The **Line** command is in the **Draw** panel of the **Home** tab. To start the command, you need to left-click on its icon once or you can use its command alias, `L`. In this case, let's start the **Line** command from the ribbon panel:



Figure 2.4: The Line command

Once your command is active, you will notice that the cursor will change into a point selection cursor that looks like two perpendicular intersecting lines and the command line will also show the name of the command, along with the prompt, as shown in the following screenshot:

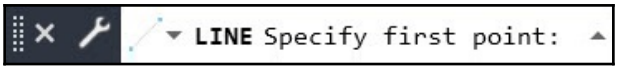


Figure 2.5: The Line command with the command name and instruction



Where is the command line?
In case you are not able to see the command line/bar at the bottom of the drawing area, press the *Ctrl + 9* keys to make it visible or to hide it.

Now, AutoCAD is ready for your input and you can start making the line. To do that, perform the following steps:

1. Click on any point in the drawing area and the line will start from that point. Move your cursor and you will notice that the line will follow the movement of the cursor and it will stretch with the cursor, too. This line is also called a rubber bending line, which follows your cursor.
2. Click on a second point in the drawing area and the fixed-length segment of the line will be made and the rubber bending line will again follow from the last point where you clicked. Repeat the process to make additional lines, and when you are done making the geometry, press the *Enter* or *Esc* key to exit the command.

This is the general workflow for making a random curve in AutoCAD but, as you have noticed, this method lacks precision as the distance was not specified for the line. To make drawings with precise distances, you need to use the direct distance entry method, which is explained in the next section.

Making lines with direct distance entry

In AutoCAD, direct distance entry is the most obvious way of making lines of precise length. This method is fairly easy, too. To explain this method, I will make a rectangle with a length of `7` units and a width of `5` units using the direct distance entry method and a line tool, as in the following example:

1. Select the **Line** command from the **Draw** panel of the **Home** tab, as in figure 2.4, or type `L` and press *Enter* to start the command using its command alias.
2. The command line will prompt you to specify the first point, as in figure 2.5. Click anywhere in the drawing area to add the first point of the line and let go of your mouse cursor and move your mouse elsewhere. The rubber bending line will be formed starting from the first point.
3. Move your cursor in a horizontal direction and type a distance value. In this case, type `7` and press the *Enter* key again.
4. The line will be formed in a horizontal direction with a length of `7` units.
5. Now, move your cursor in an upward direction and again type another value. In this case, type `5` and press *Enter*. Another line in a vertical direction with a length of `5` units will be formed.

6. Move your cursor again to the left, type `7`, and press *Enter* again to make another horizontal line.
7. Move your cursor down, type `5`, and press *Enter* again. The line will return back to the starting point.
8. The command will still continue and you will have a rubber bending effect of the line. Press the *Enter* or *Esc* key to exit from the **Line** command.

Once you are done with all the previous steps, you will have a rectangle that looks like the following screenshot. In this rectangle, the length is `7` units and the width, or height, is `5` units:



Figure 2.6: A rectangle using the Line command

This direct distance method is generally used to make drawings in AutoCAD and, as you have noticed, it allows you to make precise drawings as well. A similar workflow can also be used to make other geometries.

There are other methods of making drawings in AutoCAD and we will discuss these in the next section.

Making lines using absolute coordinates

Let's take the example of this triangle shown here. In this case, all three coordinates of this triangle are labeled as point **A**, **B**, and **C**:

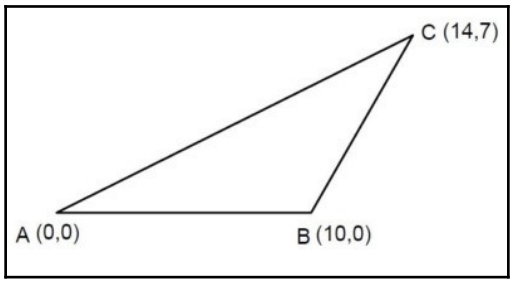


Figure 2.7: A triangle using the Line command

We will make this triangle in the following example using the **Line** command, but instead of direct distances, we will use coordinate values:

1. Select the **Line** tool from the **Draw** panel or use its command alias, `L`, to start the command.
2. The command line will prompt you to specify the first point for the line, as in figure 2.5. Type `0,0` for the first coordinate point, which is also the **A** point of the triangle, and press *Enter*.
3. Now, we need to specify the coordinates of the second point, **B**. Type `10,0` and press *Enter* again. The cursor will move to the **B** point of the triangle.
4. Once again, we need to specify the coordinates of the next point, **C**. So, type `14,7` and press *Enter*.
5. Now, our cursor is at the **C** point and you can simply type `0,0` and press *Enter* to return to the first point, which is the origin. To exit the command, press the *Enter* or *Esc* key once.

As you have noticed, we only required the coordinates to make this triangle, but the coordinates are rarely used for making drawings and, in most cases, the length and angle values are predominantly used. In real-world drawings too, we will use direct distance entry primarily, but there will be a few instances where coordinates will also be used to make drawings.

In the next example, we will learn how to make a simple drawing using the polar coordinate system.

Using polar coordinates

Using polar coordinates, you can add distance and angle values directly to the command line and they need not be entered separately. To explain this, I will use the following diagram:

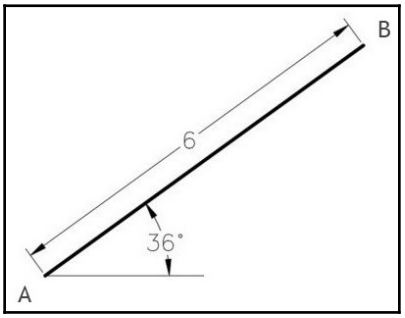


Figure 2.8: Adding a distance and angle

In this case, we need to draw a line that is inclined at an angle of `36` degrees with respect to the positive side of the *X* axis and has a length of `6` units. The following is the workflow for making this line:

1. Type `L` and press *Enter* to start the line command.
2. Type `0,0` and press *Enter* to start the line from the origin.
3. Type `6<36` in the command line and press *Enter*.
4. The line with a length of `6` units and a `36` degree angle with respect to the *X* axis will be made.

In this case, we started the line from the origin, and the distance from point **A** to point **B** is `6` units and the angle this line makes with respect to the positive side of the *X* axis is `36` degrees. So, you can add both of these values in the polar coordinate to form a `DIS<ANG` format, where `DIS` is the distance and `ANG` is the angle.

If, however, you start the line not from the origin, but from a random point in the drawing area and still want the same result, then you need to add an `@` sign before the polar coordinates. This method is known as the relative coordinate system, which is explained next.

Using relative coordinates

To explain the relative coordinate system, I will once again use the same diagram that we used in the previous section, but in this case, the line will not start from the origin. Rather, we will start it randomly from any point in the drawing area, as in the following diagram:

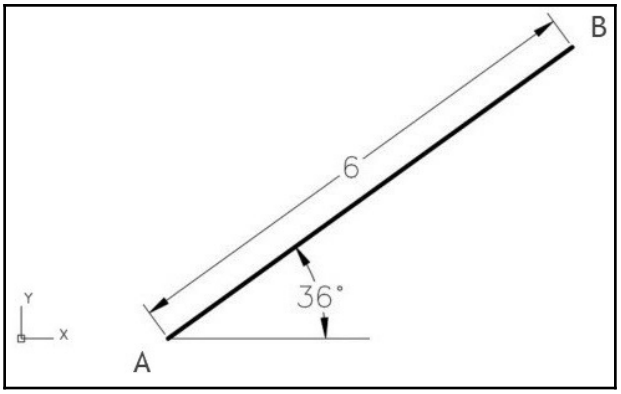


Figure 2.9: Making a line using the relative coordinate system

So, the same line with a length of `6` units that is not starting from the origin and has an inclined angle of `36` degrees can be made using the following workflow:

1. Type `L` and press *Enter* to start the line command.
2. Click on any point in the drawing area to start the line.
3. Type `@6<36` in the command line and press *Enter*.
4. The line with a length of `6` units and a `36` degree angle with respect to the *X* axis will be made.

This `@` sign represents the relative coordinate system, which allowed us to make a line from a point that is not on the origin. Here is another example of **relative coordinates**.

As explained in the previous example, relative coordinates are helpful when you are making a drawing from a point that isn't the absolute origin and still want to use the selected point as a reference for adding the coordinates. In this example, I will create the following triangle using relative coordinates:

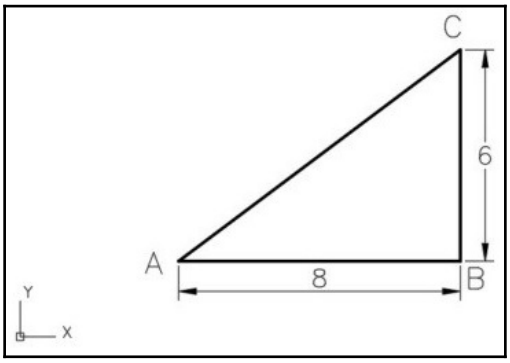


Figure 2.10: Triangle to be made using the relative coordinate system

Here, you can start the drawing at point **A**, which is not on the origin, and then progressively make your drawing by entering distances. Instead of direct distances, however, we will use relative coordinates to make this triangle in the following example:

1. Start by selecting the **Line** command and then click on a point in the drawing area, making sure that the point is not at the origin. Let's call it point **A**. As the **A** point is chosen randomly, we can't specify an exact value of the `(X,Y)` coordinate for the **B** point with respect to the **A** point. So, in this case, we can use relative coordinates to specify a coordinate value of the **B** point with respect to the **A** point. Relative coordinates assume that the last point you clicked or selected is the origin and then make all measurements from that last point as if that point were the origin.
2. So, if point **A** were the origin, then point **B** should be `8,0`, with respect to point **A**. To make the **AB** horizontal line, type `@8,0` and press *Enter*. The line will end up at point **B**, as in figure 2.10. Note the `@` sign before the coordinate value. This `@` sign is added to indicate that the next coordinates are "relative" with respect to the point that we previously clicked, which is the **A** point, in this case, and it will assume the **A** point as the origin instead of the absolute origin of the drawing.
3. Once you have reached point **B**, don't exit the line command, but type `@0,6` and press *Enter*. You will notice that AutoCAD will reach point **C**, as in figure 2.10, and in this case, the **B** point will also be treated as the origin and the coordinate value of the **C** point with respect to the **B** point is `0,6`, which is shown with the `@` sign.
4. You can complete the triangle by clicking again on the **A** point and then pressing *Enter* to exit the line command.

So, now that we have seen different methods of making the drawing in AutoCAD, let's use a combination of these methods to make a simple drawing.

Making a drawing without coordinate values

So far, we have used different coordinate systems to make a drawing, but it is generally not the ideal way of making drawings in AutoCAD. Generally, we would use direct distances and angles instead. In this section, we will learn how to use this method to make drawings. To explain this example, I will use this triangle:

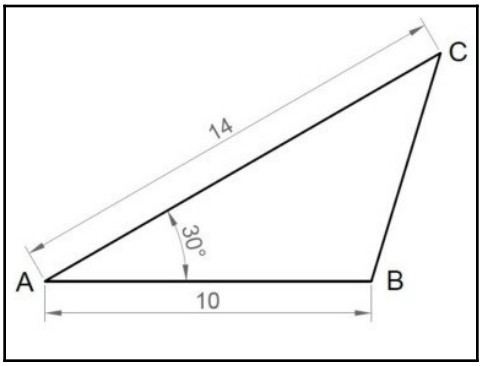


Figure 2.11: A triangle to be made using the direct distance entry method

In this drawing, there is no coordinate information provided and we will use only the dimension values, such as the length and angle, provided here to make it. We will make this triangle performing the following steps:

1. Open a blank drawing and start the line command by clicking on the **Line** tool in the **Draw** panel, or by using the `L` command.
2. Click at a point in the drawing area to start the rubber bending line and move your cursor toward the right side. Type `10` in the command line and press *Enter*.
3. Press *Enter* again to exit the command.

4. Click again at the starting point of the line (point **A**) and type `<30`, and then press *Enter*. Notice the < angle sign before `30`. In this case, entering the angle sign before `30` will tell AutoCAD to take the numeric value as an angle and not a distance. Once you press *Enter*, you will notice that the line will be locked at an angle of `30` degrees with respect to the positive side of the *X* axis.
5. Move your cursor in the direction of the `30` degrees line and type `14`, and then press *Enter* again. This will make a line at an angle of `30` degrees with a length of `14` units.
6. Click on the **B** point, as in the preceding diagram, and press *Enter* again to exit the command.

In this case, you saw that geometry can also be made by entering values of the distance and angle directly in the command line. This method is relatively easy when compared to the coordinate entry method. This is also the most common way of making drawings in AutoCAD. There are also some status bar modes that help you to generate references that can be used to make precise drawings. These modes are Dynamic Input, ortho, and polar tracking, and we will discuss them in the next section.

The status bar modes

The status bar modes help you make precise drawings in AutoCAD. The tools in the status bar can be toggled on and off by clicking on their icons or by using their function keys. Not all status bar icons are visible by default and you can toggle the visibility of the icons as per your requirements:



Figure 2.12: The status bar toggles

To change the visibility of icons in the status bar, click on the customization icon, shown as three dashes on the far-right side of the status bar, and click on the name of the icon that you want to show on the status bar. When visible, a checkmark will show next to the icon's name. In the following screenshot, you can see **Dynamic Input** and other status bar toggles checked in the customization menu:

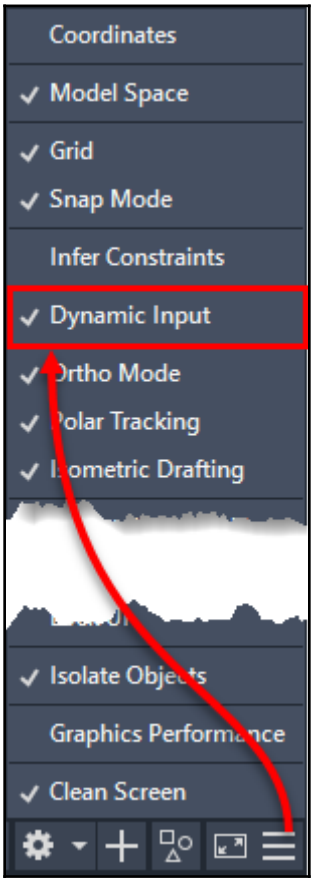


Figure 2.13: The customization menu and checked toggles

In this section, we will discuss some of the most basic status bar toggles that we need in order to render drawings precisely.

Using Dynamic Input

Dynamic Input allows you to enter the distances, angles, and other values right inside a drawing with a visual reference. Using the Dynamic Input tool, you can bypass the command line and enter the details directly in the drawing.

To explain the Dynamic Input tool, I will again use the drawing of an inclined line shown in the following diagram:

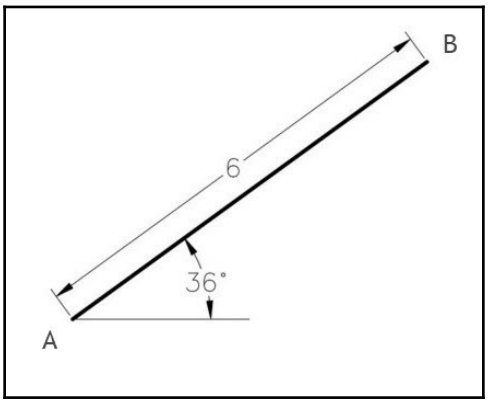


Figure 2.14: A line inclined at an angle

Here, we will make our inclined line with a length of `6` units and an angle of `36` degrees, with respect to the positive side of the *X* axis, performing the following steps:

1. Click on the Dynamic Input icon in the status bar and it will turn blue when active. You can also type `DYNMODE`, then press *Enter*, and then type `3` and press *Enter* again to activate the Dynamic Input mode. The default value of the `DYNMODE` system variable is `-3`. You can also activate or deactivate the Dynamic Input mode using the *F12* function key:



Figure 2.15: The Dynamic Input icon

2. When the Dynamic Input mode is active, select the **Line** command and you will now notice a tooltip on the cursor with the value of the *X* and *Y* coordinates of the point, as in the following screenshot:



Figure 2.16: The *X* and *Y* coordinate values on the tooltip cursor

3. Now, click at a point in the drawing area and your tooltip on the cursor will change to represent a length and an angle field, as in the following screenshot:

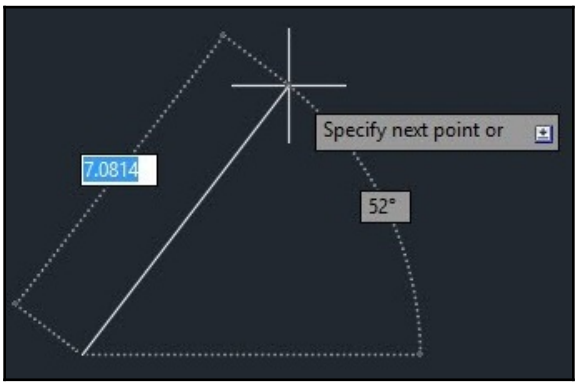


Figure 2.17: The length and angle fields in the Dynamic Input mode

4. Here, the length field is active and ready for your input. Type the length of the line in this field and press the *Tab* key on your keyboard.
5. The line will be locked at a length of `6` units and a lock sign will also appear next to the length field of the tooltip. Also, the angle field will be highlighted. Now, enter the angle value without any angle sign, which in this case will be `36`, and press *Enter*.

We now have our required line with a length of `6` inclined to an angle of `36` degrees, with respect to the *X* axis.

So, as you have noticed, this is a quick and effective way of making geometries in a drawing, which you can use to bypass the coordinate or direct distance entry methods. If, for any reason, you don't want to use this Dynamic Input tool, then click on the Dynamic Input icon again on the status bar to deactivate it.

So, now you know that Dynamic Input makes it easy to add distances and angles, let's move on to learning about another status bar mode called ortho mode, which makes rendering horizontal and vertical lines a breeze. We will discuss this status bar mode in the next section.

Using ortho mode

So far, we have used the line command to make geometries in arbitrary directions. But if you want to restrict the direction of your lines to horizontal and vertical directions, then you can use ortho mode. Ortho mode restricts the movement of the cursor to a horizontal or vertical direction only, so with ortho mode active, you will be able to make lines only in a horizontal or vertical direction. To activate ortho mode, you can use the following workflow:

1. Click on the ortho mode icon in the status bar, as in the following screenshot, or press the *F8* function key on your keyboard:



Figure 2.18: The ortho mode icon in the status bar

2. Select the **Line** tool from the **Draw** panel or use its command, `L`, and click on a point in the drawing area.
3. Now, move your cursor around and you will see that the line will remain restricted to a horizontal or vertical direction, depending on the movement of your cursor.

You can click on different points to make the geometry but the lines will always remain horizontal or vertical. To deactivate ortho mode, simply click on its icon in the status bar again or press the *F8* function key on your keyboard. Just like ortho mode, there is another mode in the status bar that lets you make lines on any angle you specify. This mode is called polar tracking and we will discuss it in the next section.

Using polar tracking

Polar tracking allows you to make geometries at any angle you want. To activate polar tracking, click on its icon in the status bar, as in the following screenshot. You can also activate polar tracking by pressing the *F10* function key on your keyboard:



Figure 2.19: Polar tracking mode in the status bar

When polar tracking is active, it will automatically deactivate ortho mode.

1. Start the **Line** command again by selecting its command, `L`
2. Click at a point in the drawing area and move your cursor in the horizontal or vertical direction.
3. You will notice a green tracking vector when the cursor is horizontal or vertical and this will help you to restrict the line to a horizontal and vertical direction:

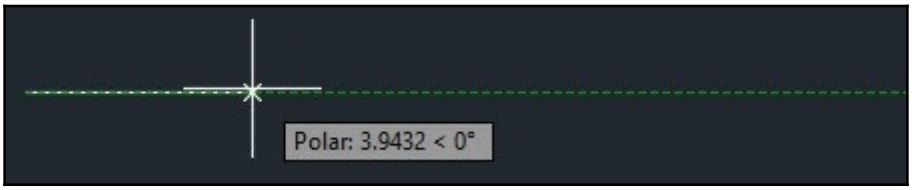


Figure 2.20: The green tracking vector when the line is horizontal

But this is not all. In this case, you can assign a different angle to polar tracking mode and it will start restricting your lines to those angles.

1. To change the angle of polar tracking, click on the small arrow right beside the polar tracking icon.
2. When the angle menu shows up select the angle that you want to choose.
3. By default, `90` and its multiples will be selected, but we will change this to `30` and its multiples, as shown in the following screenshot:



Figure 2.21: The different angle options in polar tracking

4. You can select any other angle value as well, if you want to, from the list.

After making your selection, move your cursor again to the drawing area and now you will find a green tracking vector after an interval of every `30` degrees or at every angle that is a multiple of `30` degrees as shown in the following figure:

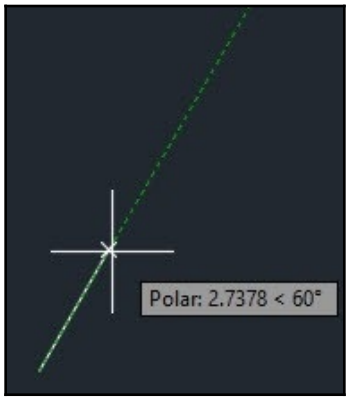


Figure 2.22: The green tracking vector along a 60-degree angle

These status bar options allow you to make geometries. To make drawings, we will use a combination of these status bar tools as and when needed.

So, now that we know the methods for making simple line drawings with precision in AutoCAD, we will move on to learning about other geometries, such as circles, arcs, and polygons.

Making a circle

You can select the **Circle** command from the **Draw** panel in the **Home** tab, or you can also use its command, `C`. When you click the **Circle** flyout, you will find six different methods for making a circle in AutoCAD:

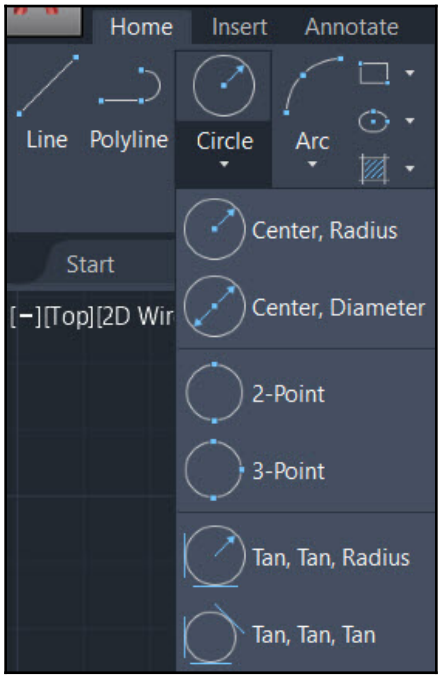


Figure 2.23: The six draw tools in the Circle flyout

We will talk about all of these methods in the following sections.

Center, radius, and diameter

The first option in the **Circle** flyout will let you make a circle with center and radius values. The following is the workflow for making a circle using a center and radius:

1. Select the first option from the **Circle** flyout. The command line will now prompt you to specify the center of the circle:

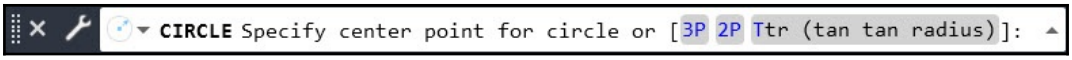


Figure 2.24: The circle command in the command line

2. Click on any point in the drawing area and this will be selected as the center of the circle.
3. Now, the command line will prompt you to specify the radius of the circle. Type the radius value in the command line and press *Enter* to make the circle, or you can also move your cursor in the drawing area and click on a point to make your circle.
4. The circle will be made with a specified radius and center point.

Let's repeat the preceding example again to make a circle with a diameter value this time:

1. Type `C` and press *Enter* to start the **Circle** command, or use the **Circle** tool from the **Circle** flyout.
2. Click on a point to specify the radius of the circle and then the command line will prompt you to specify its radius:

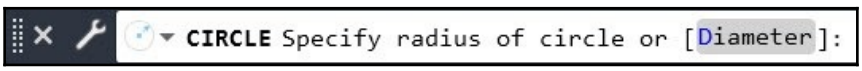


Figure 2.25: The radius prompt in the circle command line

3. Don't add the radius value at this point. In the command line, you will also see that after **or**, **Diameter** is highlighted with `D` in uppercase. Using this option, you can convert the radius input of the command line to the diameter. So, instead of radius, type `D` and then press *Enter*.

4. The command line will change again and this time it will prompt you to specify the diameter instead of the radius. You can also click on the highlighted **Diameter** option in the command line to select it instead of typing `D` and pressing the *Enter* key:

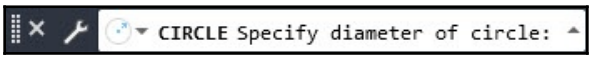


Figure 2.26: The diameter prompt in the circle command line

5. Now, type the value of the diameter in the command line and press *Enter* and your circle will be rendered with the required diameter. In the **Circle** flyout, the second option, **Center, Diameter**, will also let you make a circle with a center and diameter.

So, these two methods are nearly identical, the only difference being the radius or diameter value that you need to make the circle. The methods shown in the next section are, however, completely different and you don't even need the radius or diameter value for them.

2-Point and 3-Point

The next circle option, **2-Point**, requires you to specify two points for making the circle. For this, I will use the triangle shown here. Here, **AB** is `10` units and **AC** is `8` units in length:

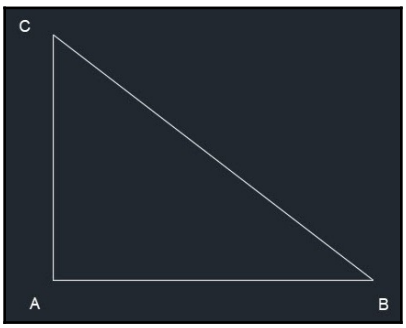


Figure 2.27: A triangle with 10 and 8 unit side lengths

In the following example, I will make a 2-point circle on the AC line of our sample triangle:

1. Select the **2-Point** option from the **Circle** flyout.
2. Click on the **A** point, then click on the **C** point, and you will end up with a circle that looks like the one in the following diagram.
3. In this case, the **AC** line is on the diameter of the circle and the length of the **AC** line will be equal to the circle's diameter:

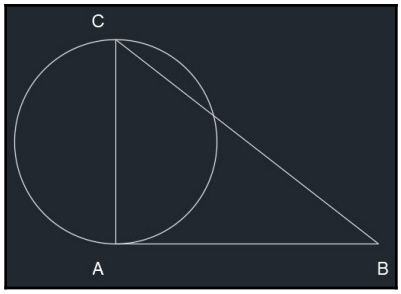


Figure 2.28: A 2-point circle on the AC line

The next example, a **3-Point** circle, will need three points to make the circle:

1. Select the **3-Point** circle command from the **Circle** flyout and then click on the **A** point, then the **B** point, and finally, on the **C** point of our example triangle.
2. The circle will be made by connecting all three points of the triangle, as in the following diagram:

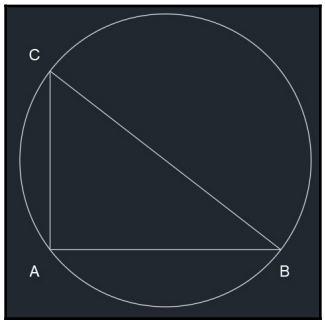


Figure 2.29: A 3-point circle on triangle ABC

The next set of options in the **Circle** flyout will let you make the circle with the tangent and radius as references. We will discuss this next.

Tan, Tan, Radius and Tan, Tan, Tan

The next example will require you to specify two geometries on which the circle will be tangent and then the radius of the circle:

1. Select the **Tan, Tan, Radius** circle option from the flyout.
2. Click anywhere on the **AB** line and then click anywhere on the **AC** line. Now, the command line will prompt you to select the radius of the circle, as in the following screenshot.
3. Type `2` and press *Enter* and you will have your circle that is at a tangent to the **AB** and **AC** lines. The radius of the circle will be `2` units, as demonstrated in the following diagram:

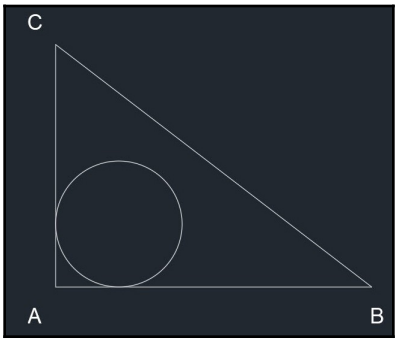


Figure 2.30: A circle tangent to two lines

In the last circle option, **Tan, Tan, Tan**, you need to only specify three geometries on which the circle will be tangent:

1. Select the **Tan, Tan, Tan** circle tool from the **Circle** flyout.
2. Click on the **AB** line, then click on the **BC** line, and finally, click on the **AC** line.

3. You will get a circle that is tangent to all three lines and it will look like this:

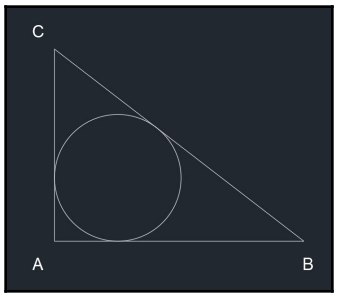


Figure 2.31: A circle tangent to three lines

So, in this way, you can make circles using different methods and the selection of the method depends on the dimensions provided and the type of geometry you are required to make. In the next section, we will explore the arc tool, which is basically a part of the circle, but the workflow of making an arc is very different from that of a circle.

Making an arc

An arc is a segment of a circle and there are lots of ways that you can make one in AutoCAD. The method that you use to make the arc depends on the type of geometry that you want to make. In this case, I will explain some of the most frequently used methods of making an arc and I will use this right-angled triangle with the **A**, **B**, and **C** vertices in all of the following examples:

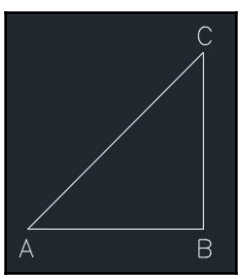


Figure 2.32: A right-angled triangle, ABC

Select the **Arc** tool from the **Draw** panel in the **Home** tab. You can also use its command: ARC:



Figure 2.33: The arc tools in the Arc flyout of the Draw panel

The command line will now prompt you to select the first point of the arc. Click on the **A** point of the triangle, then specify the second point as the **B** point, and the third point as the **C** point. An arc connecting the **A**, **B**, and **C** points will be formed, as shown:

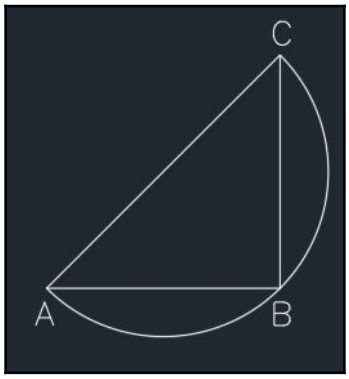


Figure 2.34: An arc connecting all three points

This was the most basic and obvious way of making an arc in AutoCAD, but there are lots of other ways of making arcs as well. Let's select the **Start, Center, End** option from the **Arc** dropdown and then, gradually, we will look at the other frequently used options.

Start, Center, End

As the name of the command suggests, you need to specify the start point first, then the center point, and lastly, the end point. Click on the **B** point to specify the start point, then click on the midpoint of the **BC** line, which will be taken as the center point of the arc, and lastly, click on the **C** point.

An arc that starts from the **B** point with the center on the midpoint of the **BC** line and ends on the **C** point will be formed, as shown:

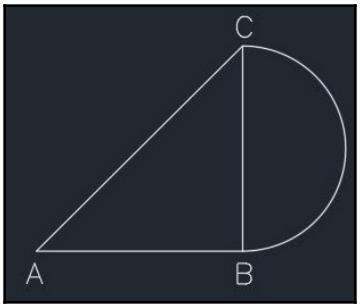


Figure 2.35: An arc made on the BC line with a start point on B and an end point on C

In this case, the arc is formed outside the triangle because the arc will be formed in an anticlockwise direction with respect to the start point, which is the **B** point. If you select **C** as the start point and **B** as the end point, you will get your arc on the opposite side. Alternatively, you can also press and hold the *Ctrl* key while making the arc to change the direction of the arc, irrespective of the start and end points.

Start, End, Radius

The next arc tool that I will tell you about is **Start, End, Radius**. For this arc tool, you need to specify the start point, end point, and radius value. The direction of the arc will be determined by the order in which you select the start and end points.

To make this arc, I will select the option from the **Arc** drop-down menu of the **Draw** panel and click on the **B** point, and then on the **A** point. Now, the command line will prompt you to specify the radius of the arc. Enter the radius value in the command line and press *Enter*. The final arc will look like this:

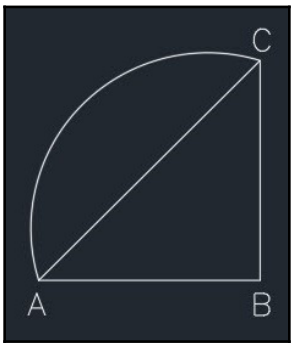


Figure 2.36: An arc made with the Start, End, Radius option

The diameter of the arc, in this case, should always be greater than the length of the **AC** line, otherwise it will not be possible to construct the arc. For our example, I have used a diameter of `16` units (or a radius of `8` units) and the length of the **AC** line is `14`.

In this case, you too can change the order of the start and end points to reverse the direction of the arc, or you can press and hold the *Ctrl* key while making the arc to reverse the direction of the arc.

Center, Start, End

The last arc in this series of arc options that I will explain here is **Center, Start, End**. By now, you must have understood the workflow of making the arc. In this case, you need to click on the center point of the arc, then the start point, and lastly, the end point.

Select the tool from the **Arc** drop-down menu in the **Draw** panel and click on the midpoint of the **AB** line as the center point. Now, click on the **A** point and then click on the **B** point. The arc shown here will be made:

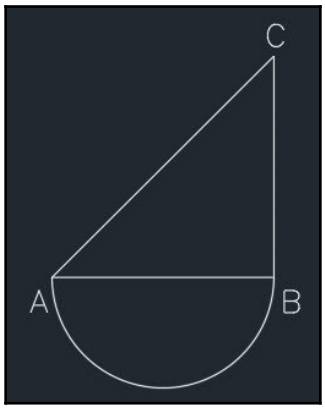


Figure 2.37: An arc made with the Center, Start, End option

Here, we can also reverse the order of selection of the start and end points to reverse the direction of the arc. For example, you can select the midpoint of the **AB** line as the center of the arc, then **B** as the start point, and then **A** as the end point, and the arc will be formed inside the triangle. You can also press and hold the *Ctrl* key to reverse the direction of the arc while making it. In this case, the **AB** line will be the diameter of the arc.

So, these are the most frequently used arc tools from the list, but there are also other tools in the list and I encourage you to explore the remaining arc tools yourself. After arcs, we will explore the **Rectangle** command, which is another frequently used draw tool. You can make a rectangle or square using lines, but we have this direct tool as well, which enables us to make rectangles very easily with fewer clicks.

Making a rectangle

To make a rectangle, use the `REC` command or use the **Rectangle** tool from the **Draw** panel in the **Home** tab. The rectangle command will help you make a four-sided rectangle or square. There are a few different ways of making a rectangle using the rectangle tool. We will first learn how to make a rectangle using coordinates and later, we will also see the method of making a rectangle using Dynamic Input. Let's begin by using coordinates to make our rectangle.

Making a rectangle using absolute coordinates

To make a rectangle using coordinates, you need to deactivate the Dynamic Input option from the status bar. To turn it off, click on its icon in the status bar or press the *F12* function key:



Figure 2.38: Dynamic Input mode in the status bar

When Dynamic Input is on, a tooltip will show next to your cursor and when it is off, you won't see a tooltip, as in the following screenshot:

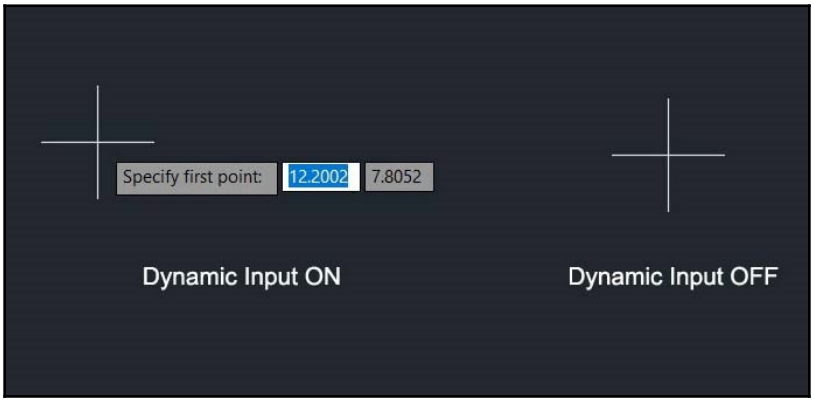


Figure 2.39: Cursor mode when Dynamic Input is on and off

Here, I will make a rectangle that starts from the origin and has a length of `8` units and a width of `3` units:

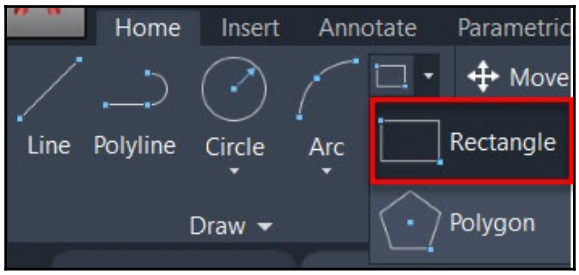


Figure 2.40: The Rectangle command in the Draw panel of the Home tab

The following is the workflow for making this 8 x 3 rectangle:

1. Click on the **Rectangle** tool in the **Draw** panel or type the `REC` command and press *Enter*.
2. The command line will prompt you to specify the first point. Type `0,0` and press *Enter*.
3. Then, type `8,3` with the next prompt and press *Enter* again.
4. The final rectangle will be made with a length of `8` units and a width of `3` units.

In this case, the first point was the lower-left point of the rectangle, shown as **1** in the following diagram, which is also the origin, and the second point was the upper-right vertex with coordinates `8,3`, shown as **2** in the diagram:



Figure 2.41: A rectangle with the lower-left corner on the origin and the upper-right corner on point 8,3

For point **2**, we added `8,3`, where `8` is the length as well as the *X* coordinate value and `3` is the height of the rectangle as well as the *Y* coordinate.

Making a rectangle using relative coordinates

In the previous example, the rectangle started from the origin; hence, the coordinates of point **2** also represented the length and width. However, if you don't want the rectangle to start from the origin and you instead want it to start from any other point, then you need to use the relative coordinates, which are explained here:

1. Start the **Rectangle** command from the **Draw** panel or use its `REC` command.
2. Click on any point in the drawing area to specify the first point of the rectangle.
3. Type `@8,3` and press *Enter* to specify the next point of the rectangle.
4. The rectangle will be made with a length of `8` units and a width of `3` units.

In this case, we have used the `@` sign before the point **2** coordinates because the first point was chosen randomly from the drawing area and it was not on the origin. So, adding the `@` sign makes point **1** the origin for this particular case and the values of point **2** will be measured with respect to the first point.

If you start the **Rectangle** command from any random point and add the second point as `8,3`, then the second point of the rectangle will end up on the absolute `8,3` point, with respect to the absolute coordinate system, and the length and width of the rectangle, in this case, won't be `8` and `3`, respectively.

Making a rectangle with Dynamic Input

Using Dynamic Input skips all these issues that we have with coordinates and lets you directly add the length and width of the rectangle, so you will have your rectangle with those dimensions. Before we make a rectangle using the Dynamic Input status bar option, we need to first activate it:

1. Activate the Dynamic Input status bar option by clicking on its icon or using its function key, *F12*.
2. Start the **Rectangle** command from the **Draw** panel of the **Home** tab or use its `REC` command.

3. You will notice a tooltip next to your cursor with coordinate values in real time. Click on any point to specify the first point of the rectangle:

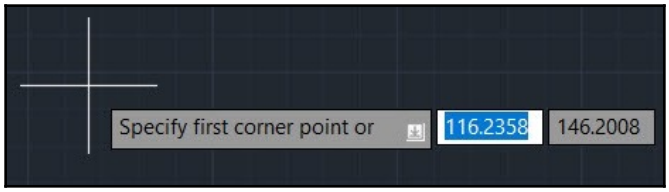


Figure 2.42: The first point option of the Rectangle command on the cursor tooltip when Dynamic Input is active

4. Type the length of the rectangle along the *X* axis and press the *Tab* key. The length of the rectangle will be locked to a `10` units length and the *Y* axis width field will become active:

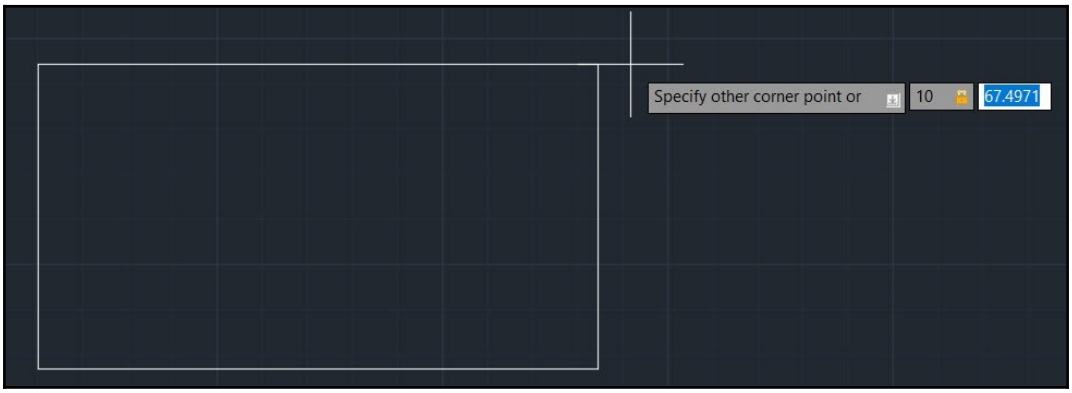


Figure 2.43: The length value locked on the tooltip in the rectangle command

5. Type the width value along the *Y* axis of the rectangle and press *Enter* again. In this case, I am using a width of `5` units.

The rectangle will be made with a length of `10` units and a width of `5` units. In this case, we used a positive value of *X* and *Y* that is, `10` and `5`, but you can use negative values as well to make the rectangle in different quadrants with respect to the first point. For example, the `-10` and `5` units will make the rectangle in the second quadrant, `-10` and `-5` will make it in the third, and `10` and `-5` will make the rectangle in the fourth quadrant. The following illustration will clarify this point further:

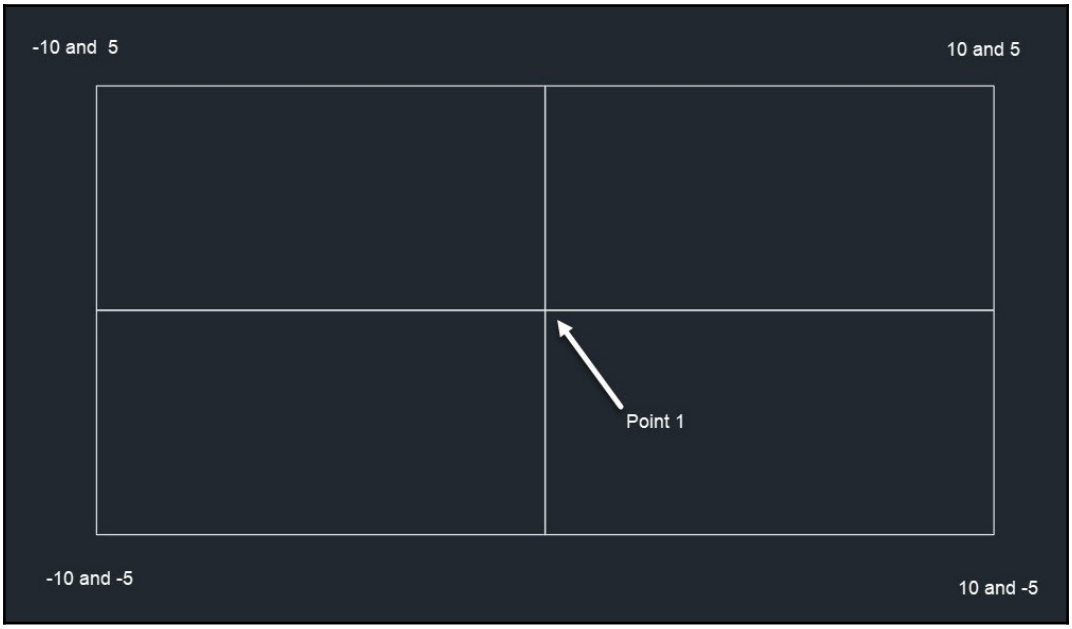


Figure 2.44: The four coordinate points of the rectangles with respect to the common starting point of the rectangle

The rectangle is a four-sided polygon but, in AutoCAD, you can make other polygons, such as a pentagon, hexagon, and heptagon as well. You can even make a polygon with tens or hundreds of sides. You can do all of this using the polygon command, which we will explore next.

Making a polygon

Polygons are closed geometries made with three or more sides. The smallest polygon is a triangle and the largest polygon is a circle, which is made up of an infinite number of sides.

In AutoCAD, you can make a polygon with a minimum of 3 and a maximum of 1,024 sides:

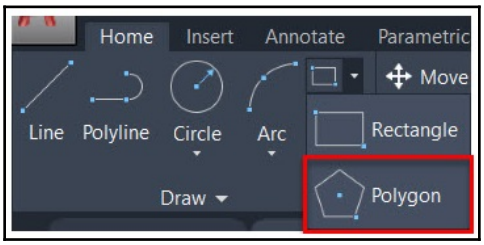


Figure 2.45: The Polygon command in the Draw panel

The **Polygon** tool is in the **Draw** panel of the **Home** tab in the expanded rectangle flyout, as in the preceding screenshot. Before we start making a polygon, we need to learn about the two types of polygon options, namely inscribed, and circumscribed, in AutoCAD.

Inscribed and circumscribed polygons

When you use the polygon command, you are presented with the inscribed and circumscribed options, so before we dig deeper into the polygon tool, let's understand what inscribed and circumscribed geometries are.

In the following diagram, the first polygon is inscribed in a circle with a radius equal to the length of the green line. In this case, the vertices of the polygon are touching the circumference of the circle.

In the second case, the polygon is circumscribed about the circle, which has a radius equal to the length of the green line and in this case, the midpoints of the sides of the polygon are touching the circumference of the circle:

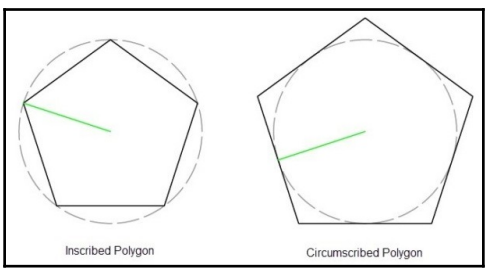


Figure 2.46: The polygon inscribed and circumscribed about a circle

When making polygons, AutoCAD will prompt you to specify the radius of the polygon, which is essentially the radius of the inscribed or circumscribed polygon. The type of polygon that you need to make depends on the dimensions provided in the drawing.

Making polygons

So, now that you know what an inscribed and a circumscribed polygon are, we are ready to make our first polygon. In this case, I will make a pentagon, which is a polygon with five sides, and I will use a polygon inscribed in a circle:

1. Select the **Polygon** tool from the **Draw** panel of the **Home** tab, or type `POL` and press *Enter*.
2. The command line will now prompt you to specify the number of sides of the polygon. Type `5` and press *Enter*.
3. Now, the command line will prompt you to specify the center of the circle. Click on a point in the drawing to specify the center.
4. Now, the command line will prompt you to specify the type of polygon: **Inscribed in circle** or **Circumscribed about circle**:



Figure 2.47: The inscribed and circumscribed options in the Polygon command

5. Select **Inscribed in circle** for this case.
6. Now, the command line will prompt you to specify the radius of the circle. Enter `5` as the radius and press the *Enter* key.

You will notice that an inscribed polygon with a radius of `5` units will be rendered. In this case, the radius of the polygon is the radius of the inscribed circle that this polygon is made in.

You can use a similar process to make a polygon that is circumscribed about the circle. Although these two options may seem like the only options for making a polygon in AutoCAD, its not always possible to have the radius of inscribed or circumscribed circles. If you only have the side length of the polygon, then you can use this next method to make a polygon using the side length.

In this example, I will make a hexagon with the length of its sides as `6` units:

1. Start the **Polygon** command from the **Draw** panel or use its `POL` command.
2. Now, the command line will prompt you to specify the number of sides. Type `6` and press *Enter*.
3. Now, the command line will prompt you to select the center of the polygon, but in this case, select the `Edge` option from the command line:



Figure 2.48: The Edge option in the polygon command line

4. Now, specify the first point of the edge by clicking anywhere in the drawing area.
5. You will notice that the cursor will now follow the edge of your polygon. Simply click on the second point to make a polygon of a required length, or specify the length in the command line and press *Enter*:

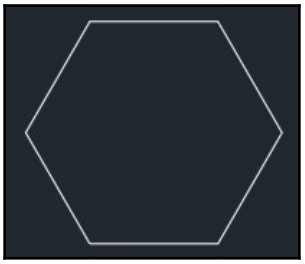


Figure 2.49: A hexagon made with the Polygon command

A polygon with the required edge length will be made. In this case, you were not required to specify the inscribed or circumscribed circle as the reference.

Now that we know about some of the basic draw tools that make a drawing, let's move on to learning about a few of the basic modify tools.

The Move and Copy commands

To move drawings in AutoCAD, you can use the **Move** command from the **Modify** panel of the **Home** tab, or you can use its command, `M`:

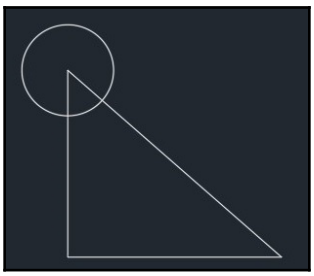


Figure 2.50: A sample drawing to be used for the Copy command

In this case, we will use the **Move** tool to move the circle from one of the vertices of the triangle shown in the previous diagram down to the other:

1. Select the **Move** tool from the **Modify** panel and then click on the circle from the drawing area and press *Enter*.
2. Now, click on the center of the circle. This center will become the base point from where the circle will be picked.
3. Now, click on the lower-right vertex of the triangle. The circle will be moved to its new location.

To copy the circle on all three vertices, you can use the **Copy** tool from the **Modify** panel of the **Home** tab, or you can use its command, `CO`:

1. Click on the **Copy** command from the **Modify** panel and then click on the circle that you want to copy from the drawing area and press *Enter*.
2. Now, once again, click on the center of the circle and that will become the base point of your selection.
3. Click on the other two vertices of the triangle to paste the circle there.

To end the command, press *Enter* again. The final drawing after copying the circles on all three vertices will look like this:

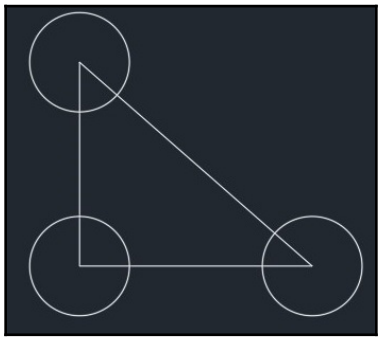


Figure 2.51: A circle copied on all the vertices of the triangle

So, now that you know about the basic drawing modification tools, such as Move and Copy, let's explore some other modification tools. We will discuss the Rotate tool next.

The Rotate command

As the name suggests, the **Rotate** command can be used to rotate an object about a point. In this case, I will use a door symbol to explain the command, which is shown here:

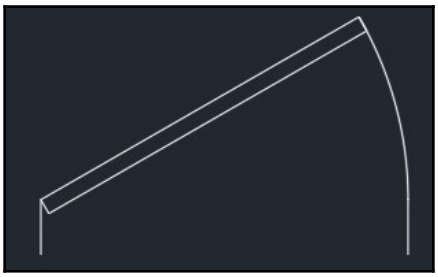


Figure 2.52: A door symbol used for the Rotate command

The **Rotate** command rotates the selected object from its base point and you can rotate the object from its original angle or also by using a reference angle. In the following sections, we will discuss all the methods for using **Rotate** commands. Let's start with the simple rotate feature.

Simple Rotate

Currently, the door is horizontal and we can rotate it to change its rotation angle with respect to its current angle, which is 0 degrees, as the door is horizontal:

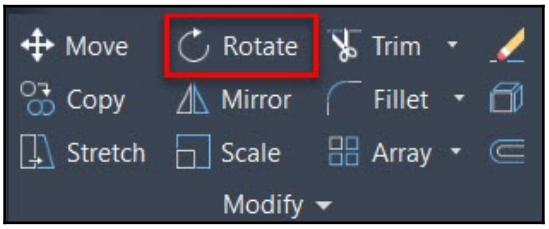


Figure 2.53: The Rotate command in the Modify panel

To rotate this door with respect to its original angle, we will perform the following steps:

1. Select the **Rotate** command from the **Modify** panel or use its command, `ROTATE`.
2. The command line will prompt you to select the objects. Click on the door to select it and press *Enter*.
3. Now, the command line will prompt you to specify the base point. Click on the lower-left corner of the door as the base point. The base point will become the pivot point of the rotation and it will remain fixed; the door will rotate about that point.
4. Now, the command line will prompt you to specify the angle of rotation. Type `30` and press *Enter*.

The door will rotate to an angle of `30` degrees, with respect to the current angle of 0 degrees, and the final door should look like this:



Figure 2.54: The door symbol rotated to an angle of 30 degrees

In a similar way, you can specify different base points and rotation angles to get different results. The **Copy** option, which shows up when you select the base point in the rotate command, will let you rotate a copy of the original drawing:

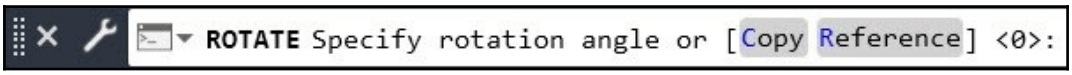


Figure 2.55: The Copy subcommand in the Rotate command line

To use the **Copy** option, type `C` when the command line appears as in the previous screenshot and press *Enter*. This will select the `Copy` subcommand from the command line. You can also click on the highlighted `Copy` text from the command line to select this option.

Now, if you rotate the door, you will get a copy of the original door and the original door will also remain in its place. After using the **Copy** option in the preceding example, and a rotation angle of `90` degrees, we ended up with two doors that look like this:

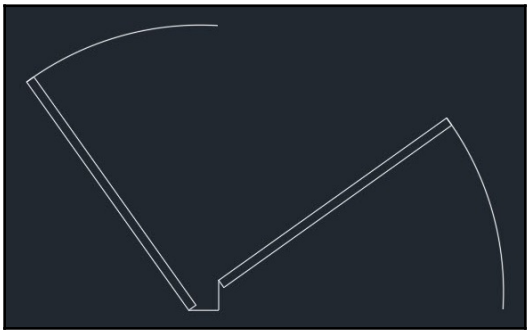


Figure 2.56: The door symbol rotated by 90 degrees with the Copy subcommand selected

When an object is at any known angle, it is easy to rotate it to any other angle, but if your object is at an unknown angle and you still want to rotate the object to any known final angle, then you can use **Rotate** with the **Reference** option, which we will discuss next.

Rotate with Reference

In the previous case, the door symbol was at an angle of `0` degrees (perfectly horizontal) and hence, its rotation angle can be specified pretty easily. But let's now take the case of this window symbol. It is inclined to an unknown angle and so, to rotate this to any specific angle, we will use the rotate with reference option:

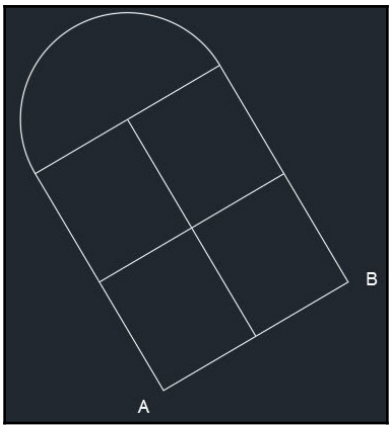


Figure 2.57: A window symbol inclined to an unknown angle

Here, the window symbol is inclined to a random angle with respect to the horizontal axis. Now, if you want to rotate this window symbol so that it becomes perfectly horizontal, then you need to use the **Reference** subcommand of the **Rotate** command.

To do this, I will once again start with the **Rotate** command:

1. Type `RO` and press *Enter* to start the **Rotate** command or click its icon in the **Modify** panel of the **Home** tab.
2. Select all the objects that make the window and press *Enter*.
3. Click on the **A** point to specify the base point.
4. Select the **Reference** option from the command line, or type `R` and press *Enter* to select the **Reference** option.
5. Now, you need to specify the reference angle and in this case, we will specify it by clicking on **A** and then the **B** point.
6. Now, the command line will prompt you to specify the angle. Type `0` and press *Enter*.

You will notice that the window will now become horizontal; that is, its angle will now be 0 degrees, as in the following diagram:

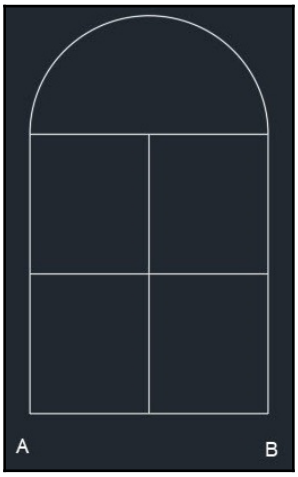


Figure 2.58: The window symbol after rotating it with respect to the reference

In this case, by specifying the reference angle, you can make AutoCAD rotate the drawing to any specific angle with respect to the positive side of the *X* axis, even when the angle to which the object is inclined is unknown.

The next modify command that we will discuss is **Fillet** and this command lets you add rounded corners. It may look like a tool that can make subtle changes to the drawing, but you will find it pretty useful as it not only makes rounded edges, but it also has other sets of useful features, which we will discuss in the next section.

The Fillet command

The **Fillet** command can be used to add round corners to the sharp edges of the drawing. For example, in this case, the fillets are added to the vertices of the **A** diagram, shown here, to make it look rounded in the corners, as shown in the **B** diagram:

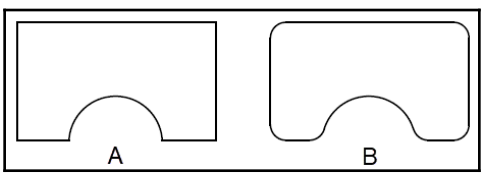


Figure 2.59: A diagram before and after applying fillets to vertices

To use the **Fillet** command, select it from the **Modify** panel in the **Home** tab, as in the following screenshot, or use its command, `F`:

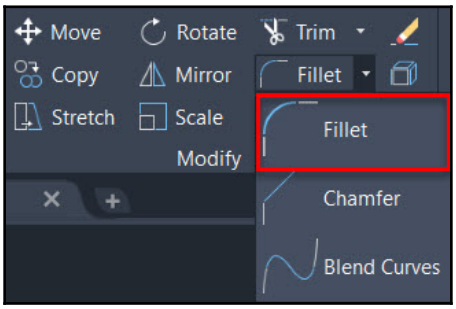


Figure 2.60: The Fillet command in the Modify panel of the Home tab

Let's take the example of the following diagram. This diagram has been made with the **Line** command:

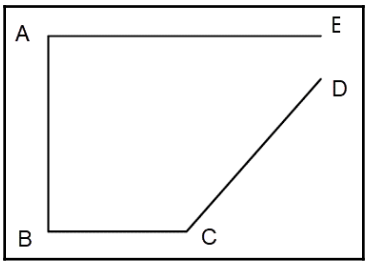


Figure 2.61: A sample diagram with an open vertex

Using the following steps, we will add a fillet to the **A** vertex of this diagram:

1. Select the **Line** command from the **Modify** panel or type `F` and press *Enter* to start the command.
2. Now, select the **Radius** option from the command line or type `R` and press *Enter* to select the radius option. Then, type the value of the radius that you want to apply on the vertex and press *Enter*.
3. Now, the command line will prompt you to select the objects to fillet.
4. Click on the **AB** line, then on the **AE** line, and the fillet will be applied to the **A** vertex and the final diagram will look like this:

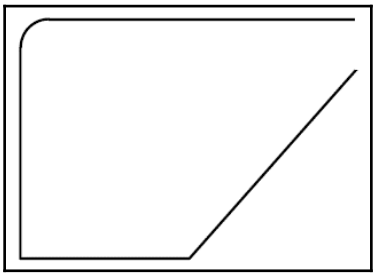


Figure 2.62: A diagram with fillet applied on the A vertex

You can also apply fillets on the open edges of the drawing. In the preceding example, the **ED** edge is open and we can close it with a rounded fillet or with a sharp vertex using the **Fillet** command:

1. Type `F` and press *Enter* to start the **Fillet** command or click on its icon on the **Modify** panel of the **Home** tab.
2. Click on the **Radius** option of the command line or type `R` and press *Enter* to start the option.
3. Type `0` and press *Enter*.
4. Click on the **AE** line, then the **CD** line.
5. The lines will merge at the point of intersection and you will get a closed drawing, as shown here:

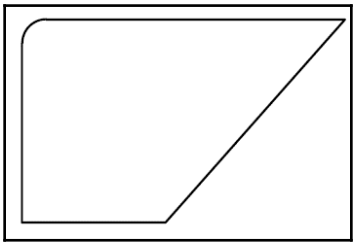


Figure 2.63: A diagram with a fillet of 0 radii applied to the open vertex

In this case, you can also use a radius value at the **ED** vertex and then, instead of merging at a point, a fillet with a specified radius will be made.

While making the fillet, if you select the **Polyline** option from the command line, you can apply fillets on the multiple vertices of the drawing made with a polyline. As an example, if you want to apply a fillet on all four vertices of a rectangle with a length of `10` and a width of `5` units, then you can use this workflow:

1. Start the **Fillet** command from the **Modify** panel or using its command, `F`.
2. Type `R` to select the **Radius** option from the command line or click on the **Radius** option to select it.
3. Type `1` for the radius and press *Enter*.
4. Now, click on the **Polyline** option from the command line or type `P` and press *Enter* to select the option:

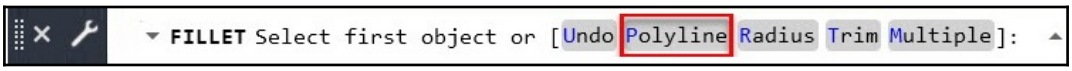


Figure 2.64: A polyline subcommand in the fillet command line

5. Click on any line segment of the rectangle and you will notice that the fillet will be applied to all four vertices of the rectangle.

So, now you know about the features of the fillet tool and how it can be used to add not only rounded corners but also other modifications to a drawing. Let's move on to the **Modify** command and explore one of the most frequently used modify tools, called **Trim**. This command lets you delete part of a drawing. We will explore its features in the next section.

The Trim command

Using the **Trim** command, you can remove parts of a drawing up to its intersection point or vertex. To explain this command properly, I will use the diagram shown here:

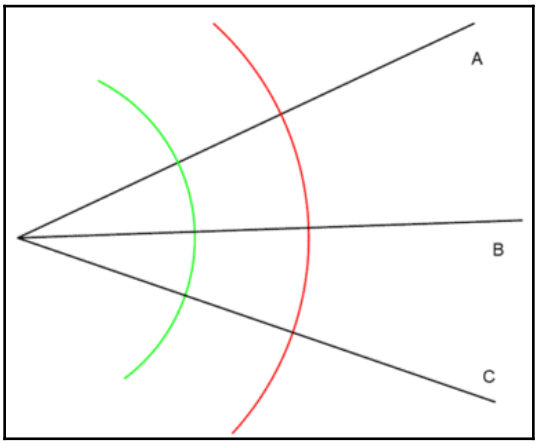


Figure 2.65: A sample drawing for the Trim command

Here, we have three lines, **A**, **B**, and **C**, intersected by two arcs. We will trim the lines and arcs with respect to one another in the following examples:

1. Select the **Trim** command from the **Modify** panel or use its command, `TR`.
2. Hover your cursor over the **A**, **B**, or **C** lines on the right side of the red arc. You will notice that the color of the line will fade up to the red arc, indicating the part that will be trimmed.
3. Click the line and it will be trimmed up to the red arc.

In this case, if you click another line or arc, it will be trimmed up to the next available boundaries. This is the default way that the trim command works in the 2021 version of AutoCAD.

However, if you are using older versions of AutoCAD, then the workflow will be slightly different. Here is the trim command's workflow for the older versions of AutoCAD:

1. Select the **Trim** command from the **Modify** panel or use its command, `TR`.
2. Now, the command line will prompt you to select the objects. Click on the green arc and press *Enter*.
3. Now, click on the **A**, **B**, or **C** lines near the text and you will notice that the line will be trimmed up to the green arc, even though you have a red arc crossing the line before the green one.

In this case, you have selected the green arc as the boundary. So, it will trim the line with respect to the selected boundary only.

To include everything as a trimming boundary in older AutoCAD versions, select the **Trim** command again and press the *Enter* key directly, without making any specific selection from the drawing. The `select all` option is selected in the angle bracket and hence, pressing the *Enter* key selects all the objects in the drawing as trimming boundaries, as shown here:

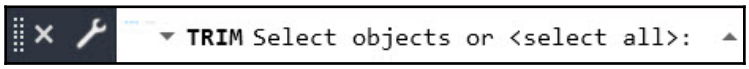


Figure 2.66: The select all option in the trim command line

This will select everything in the drawing area as a trimming boundary because the `select all` option is selected in the angle bracket, as in figure 2.66, and now if you click on any object, it will be trimmed up to the next boundary.

So, if you click on the **A** line somewhere near the **A** text, then the line will be trimmed up to the next boundary, which is the red arc, in this case. Selecting the same line again will trim it to the next boundary, which is a green arc, and so on. The last segment of the line will not be trimmed as there is no further trimming boundary.

Similar to the **Trim** tool, we have a tool that does the opposite of trim, that is, it will extend the object to the selected boundary. The workflow of the **Extend** tool is also similar to the **Trim** tool and we will discuss that in the next section.

The Extend command

The **Extend** command works in a completely opposite way. It extends the drawing up to the selected boundary. The workflow of the extend command is also the same and in this case, you also need a boundary that another geometry will extend to:

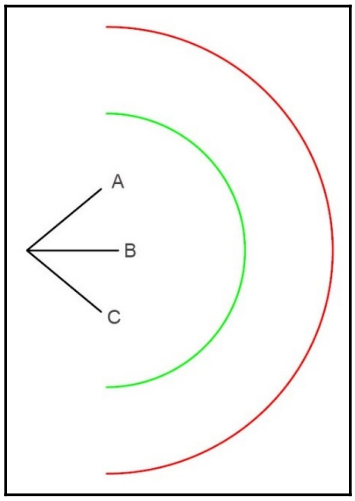


Figure 2.67: A sample diagram for the extend command

To explain the extend command, I will use this diagram shown in the preceding figure. Here, we will extend the lines up to the next boundaries using different extend options:

1. Type `EX` and press *Enter*, or click on the **Extend** tool from the **Modify** panel to start the extend command.
2. Simply click on the line you want to extend. In this example, click on the **A** line and it will extend up to the green arc.
3. Click on the **A** line again and it will again extend up to the red arc this time. If you click on the **A** line a third time, it will have no effect as there is no further boundary to extend the line.

This is the workflow for extending a command in the 2021 version of AutoCAD, but if you are using older versions of AutoCAD, the workflow once again will be slightly different. Here is the workflow of the **Extend** command from the older versions:

1. Type `EX` and press *Enter* or click on the **Extend** tool from the **Modify** panel to start the **Extend** command.
2. Now, the command line will prompt you to select the object to extend to. Click the red arc and press the *Enter* key.
3. Now, click on the **A** line and it will extend up to the red arc, ignoring the green arc that is before it.

If you want to include all the objects to be included as a boundary for the **Extend** command, then follow this workflow:

1. Start the **Extend** command again by using the `EX` command or click on the extend icon from the **Modify** panel of the **Home** tab.
2. When the command line prompts you to select the objects, press the *Enter* key without selecting anything. This will ensure all objects in the drawing area are selected as the extend boundary:

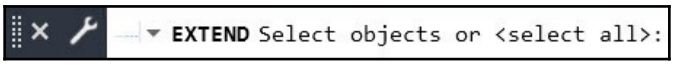


Figure 2.68: The select all option in the command line of the extend command

3. Now, click on the **A** line and it will extend to the green arc. Click on the **A** line again and it will extend further up to red arc.

Similarly, if you click on other lines, they will also extend up the first boundary, that is, the green arc, and then clicking on them again will extend it further to the red arc.

Let's consider this situation where we have a boundary on both the sides of the **AB** line:

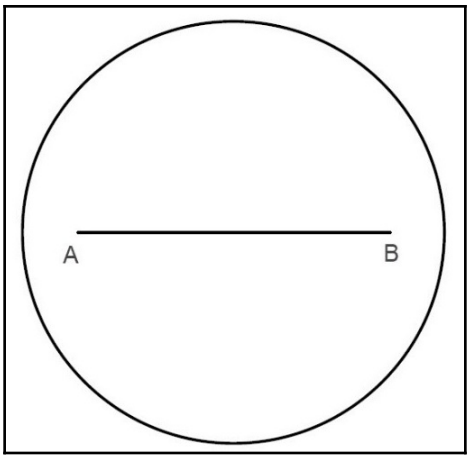


Figure 2.69: The AB line with boundaries on both the ends

In this case, if you use **Extend** on the **AB** line, the line will extend either to the left or the right side to touch the circle. The side to which the line extends will depend on the point where you click on the line. If you click close to the **A** point, the line will extend to the left and similarly, if you click close to the **B** point, then the line will extend to the right. It's obviously not possible to click exactly at the center.

If you are not happy with the way the trim and extend commands work in the 2021 version of AutoCAD and want to switch back to the legacy behavior, then follow this workflow:

1. Select the **Trim** command from the **Modify** panel or use its command, `TR`.
2. Select the **Mode** option from the command line.
3. Change the option from **Quick** to **Standard** and you will have the standard behavior of the trim tool restored.

You can follow the same workflow for restoring the **Extend** command's behavior, too.

So, now we are equipped with the basic drawing tools that are required to make simple geometries in AutoCAD. I recommend you try making a simple drawing yourself with the tools learned about in this chapter.

Summary

In this chapter, we learned about different coordinate systems and input methods in AutoCAD. We also learned about the most basic draw command—Line—and different methods of making a line in AutoCAD. We then learned about other draw tools, such as Circle, Arc, Rectangle, and Polygon, which are the building blocks of drawings. Then, we also learned different methods of manipulating these drawing building blocks using modify commands, such as **Move**, **Copy**, **Rotate**, **Fillet**, **Trim**, and **Extend**.

These drawing and modifying tools are the building block of any AutoCAD drawing and you will find yourself using these tools in almost every drawing. In the next chapter, we will focus more on making complex drawings using advanced draw tools, status bar options such as object snap, and using modify tools such as offset, scale, and stretch. We will also discuss the hatch and gradient tools that let you fill closed boundaries with patterns and colors. So, I will see you in the next chapter.

3
Learning about Modify Commands

So, now that the basics have been covered, we will step into the advanced draw and modify tools and explore more of the AutoCAD features. The tools and commands that we will discuss in this chapter include Object Snap toggles that help in making precise drawings and advanced drawing tools such as construction lines and ellipses. We will also learn about complex shapes, such as splines, and modify tools such as offset, stretch, and chamfer.

At the end of this section, you will be able to work with complex drawings and you will also be equipped with the tools required to create precise drawings in AutoCAD.

The following are the topics covered in this chapter:

- Using Object Snap, Object Snap override, and Object Snap tracking
- Ellipses and construction lines
- Revision clouds
- Region, Point, and Point style
- Working with Spline
- More Modify commands

Before we begin with more draw commands, it is necessary for us to understand Object Snap and its application in a drawing.

Using Object Snaps

Making precise drawings in AutoCAD is not possible without Object Snaps, also called `OSNAPS`. Object Snap is an option on the status bar with a bunch of control points. These control points are the geometric points, such as the endpoint, center, midpoint, intersection, and so on, which if activated become available for selection in the drawing.

The Object Snap status bar option looks like a box with a tiny box in the top left of it, as in the following screenshot:



Figure 3.1: The Object Snap status bar toggle

You can activate Object Snap by clicking on the icon once or by using its function key, *F3*. When active, the Object Snap icon looks blue and when deactivated, it looks gray, just like the other icons in the status bar.

When you click on the tiny arrow pointing downward next to the Object Snap icon, you will see a list of snap points that AutoCAD will recognize when the Object Snap option is activated. The snap point menu is shown in the following screenshot:

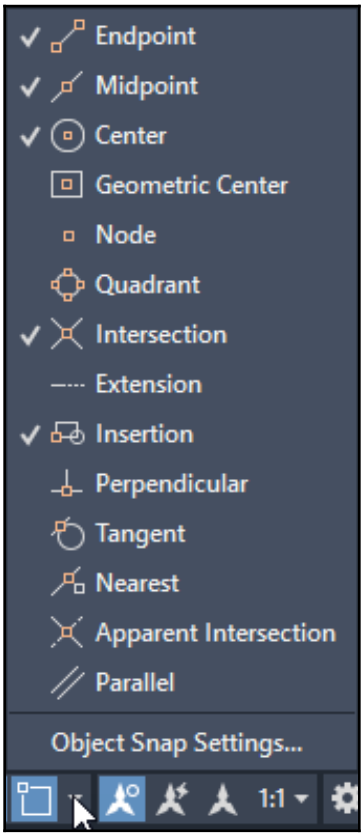


Figure 3.2: The AutoCAD snap points in the Object Snap menu

By default, not all these snap points are active, and you can activate or deactivate them by clicking on them once. The active snap points have a checkmark next to it.

In the next section, we will discuss almost all of these Object Snaps, except **Insertion** and **Apparent Intersection**, as these require knowledge of blocks and 3D tools, which we will learn about later in this book.

The Endpoint and Midpoint snaps

I will start explaining the **Object Snap** tool with the Endpoint and Midpoint snaps. For that, activate the Endpoint and Midpoint snaps from the **Object Snap** menu. Also, make sure the Object Snap icon is active on the status bar. When the Object Snap is active, it will appear blue in color.

In the following example, I will show you Endpoint and Midpoint Object Snaps in action:

1. Start the **Line** command and make a horizontal line. Now, don't exit the command and make a vertical line, as shown:

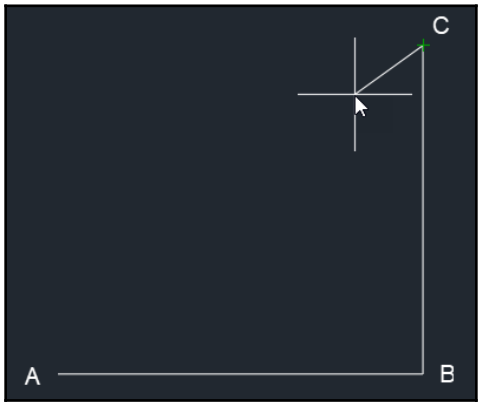


Figure 3.3: Horizontal and vertical lines

2. Now, move your cursor close to the **A** point and you will notice a green box with **Endpoint** highlighted on the tooltip. Also, when you click, the cursor will snap to the **A** point, which is the starting point of the line, even though you clicked at some distance from the exact **A** point.

3. In the following figure, you can see that the Endpoint snap is highlighted even though we are clicking at some distance from the exact **A** point. As long as we have the Endpoint snap visible, the cursor will snap exactly to the endpoint or the **A** point:

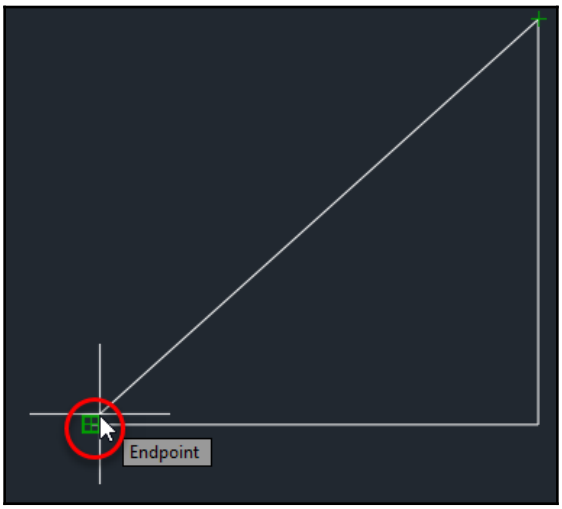


Figure 3.4: The endpoint snap, which looks like a green square, is highlighted

4. This green marker is the Endpoint Object Snap and it helps you snap exactly to the endpoint of the line. If you deactivate the Object Snap and then try to do the same exercise, you won't see the green endpoint Object Snap and it will be impossible to snap exactly to the endpoint of the line.
5. After snapping to the endpoint, press *Enter* to finish the Line commands.
6. Now, press the *Enter* key again to start the Line commands and click on the **B** point of the triangle and this time again, the line will snap to the endpoint of the lines.

7. Now, move your cursor close to the midpoint of the **AC** line and this time you will notice a green triangular snap that lets you snap exactly to the midpoint of the **AC** line, as shown:

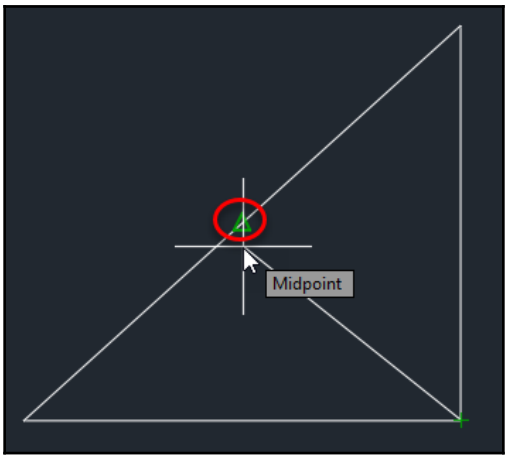


Figure 3.5: The Midpoint snap highlighted as a green triangle

8. Once again, this is the **Midpoint** Object Snap that we activated from the Object Snap status bar menu and this lets you snap to the midpoint of a line or arc.

In a similar way, other Object Snap options will let you snap to the corresponding geometric points and you will be able to do so only when the required Object Snap is active in the Object Snap menu.

The Center snap

This Object Snap lets you snap to the center of a circle, arc, or ellipse, and this option is shown here:

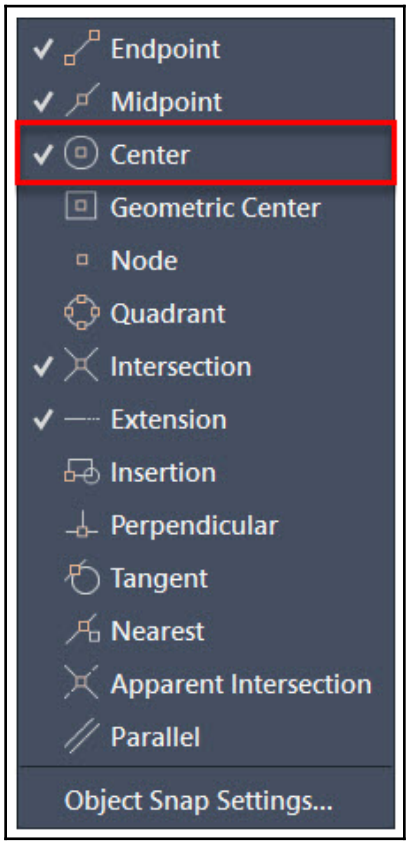


Figure 3.6: The Center snap option in the Object Snap menu

To use this snap, activate it from the **Object Snap** menu and then start any **draw** command and move your cursor close to the center of any circle, arc, or ellipse and the center Object Snap will show up. If for some reason you don't see the **Center** snap, then move your cursor close to the boundary of the circle, ellipse, or arc and the **Center** snap will show up.

The Geometric Center snap

This Object Snap option lets you snap to the Geometric Center, which is also called the centroid of drawings.

To use this Object Snap, make sure it is active in the **Object Snap** menu and then make a closed drawing made with a single curve, such as a polyline or spline. The **Geometric Center** snap is highlighted in the following screenshot:

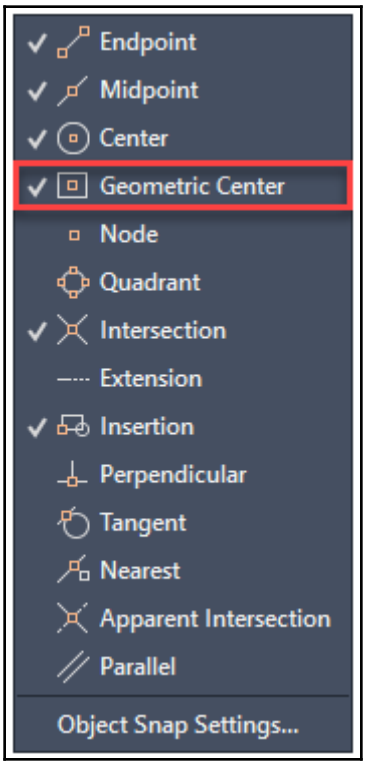


Figure 3.7: The Geometric Center snap in the Object Snap menu

The **Geometric Center** snap will not work for open drawings or drawings made with curves, such as lines. So, if you want to use the Geometric Center snap for closed curves made with lines or spline, then you need to first join them into a single unit using the join command.

To show the **Geometric Center** snap, I am using a pentagon and a random polyline curve, as in the following figure:

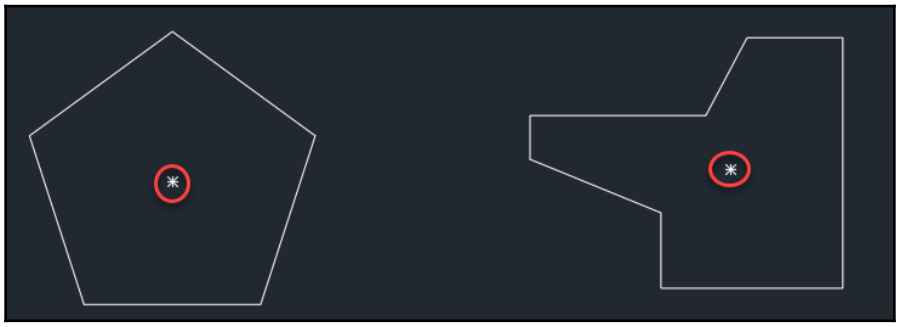


Figure 3.8: Geometric centers highlighted for two closed polyline geometries

When you select a **draw** command and move your cursor close to the boundary of the drawing, its geometric center will highlight and then you can snap to it. In the preceding figure, the geometric snap is highlighted in the red circles.

The Node snap

This Object Snap is the next one on the list of Object Snaps and it lets you select the point objects. We will discuss more about point objects and nodes later in this chapter.

The Quadrant snap

This Object Snap option lets you select the quadrant of a circle, arc, or ellipse. Quadrants are like the four "sides" of these circular geometries. The quadrants of these geometries are shown here:

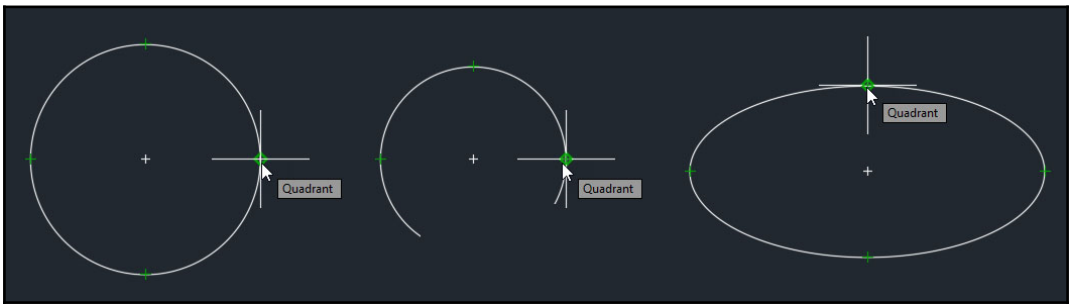


Figure 3.9: The Quadrant snap points highlighted in green for a circle, arc, and ellipse

The quadrants may not always be required and so it is recommended to keep them off to avoid conflict with other snap points, such as a tangent. You can activate the quadrant only when required and once you are done selecting it, you can deactivate it again.

The Intersection snap

As the Object Snap name **Intersection** suggests, this Object Snap will let you snap to the intersection point of any two curves. This snap is very useful and it is recommended that you keep this snap active most of the time. To use this snap, select it from the Object Snap menu then select a **draw** command and move your cursor close to the intersection point of two curves. A green cross indicating the **Intersection** snap will show up, as shown:

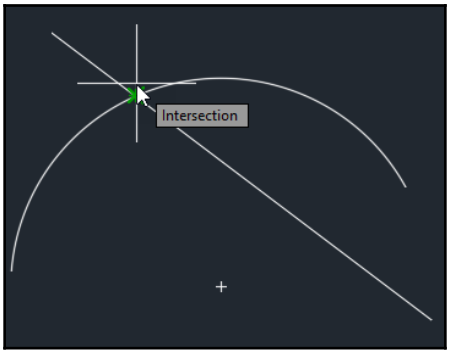


Figure 3.10: The Intersection snap point highlighted at the point of intersection of the line and arc

The **Intersection** snap will show for almost every kind of intersecting drawing. The Intersection snap also shows up for curves that are apparently intersecting on a 2D plane:

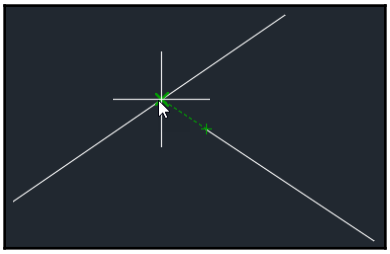


Figure 3.11: An Intersection snap for apparently intersecting lines

In the preceding example, the two lines are not actually intersecting, but when we track a line toward the second one, the point of apparent intersection is highlighted and you can snap to that point.

The Extension snap

This one is a unique snap as it won't show any existing snap points, rather it will let you extend an existing curve. To use this Object Snap, select it from the **Object Snap** menu:

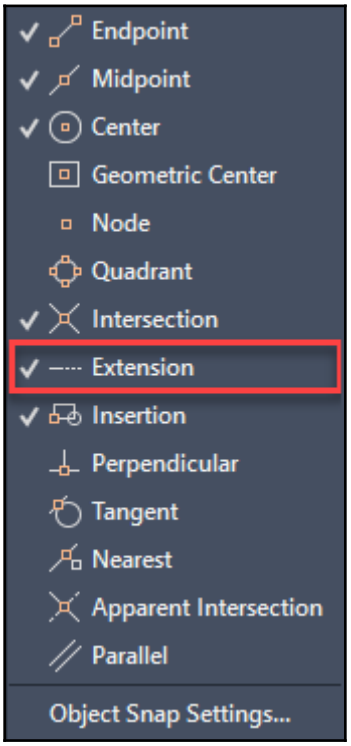


Figure 3.12: The Extension snap option in the Object Snap menu

Now, make a line, inclined to some angle, and an arc, as shown:

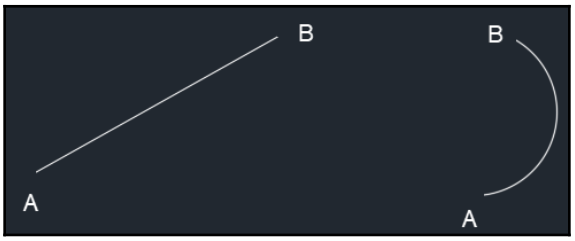


Figure 3.13: A line inclined to an angle and an arc

Now, activate the **Line commands** and click on the **B** point of the line in the preceding figure. Move your cursor in the direction of the line and you will notice that the line will follow the angle of inclination of the line and it will "extend" the line in its direction. This is possible because of the **Extension** Object Snap tool.

A similar extension effect can be achieved for the arc as well. In this case, select the **Arc command** and then click on either the **A** or **B** point and then move your cursor along the circumference of the arc and you will notice that it will extend the arc in its direction, as shown:

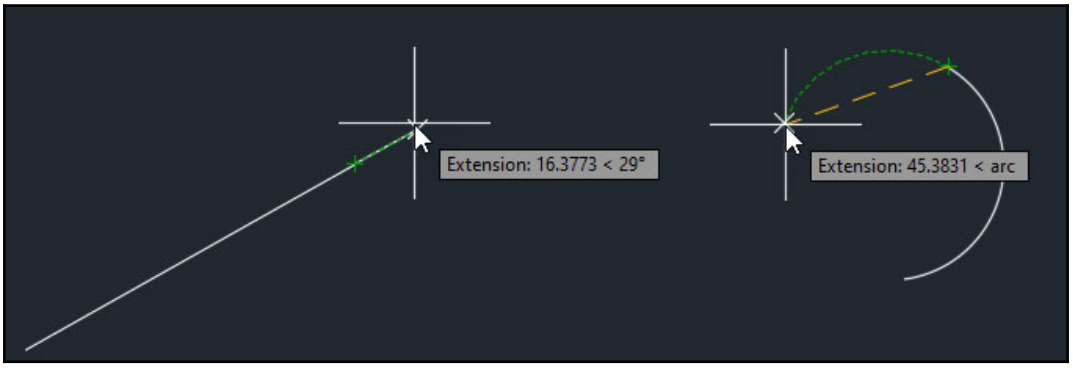


Figure 3.14: Extension options for a line and arc

The **Extension** snap option can be very handy in situations where you need to track a geometry without actually changing the drawing. You can keep this Object Snap active for most of the time, too, as it gives you access to snaps where geometry does not actually exist.

The Perpendicular snap

This Object Snap option lets you make lines perpendicular to an existing line. To use this Object Snap, activate it from the **Object Snap** menu. I will use the inclined line, as in the following figure, to show you how this snap works:



Figure 3.15: A line inclined to an unknown angle+633

Make this inclined line and then select the **Line** tool again and click anywhere above the line. Now, move your cursor along the line and at a point where the new line is perpendicular to the existing line, a **Perpendicular** snap will show up, as shown:

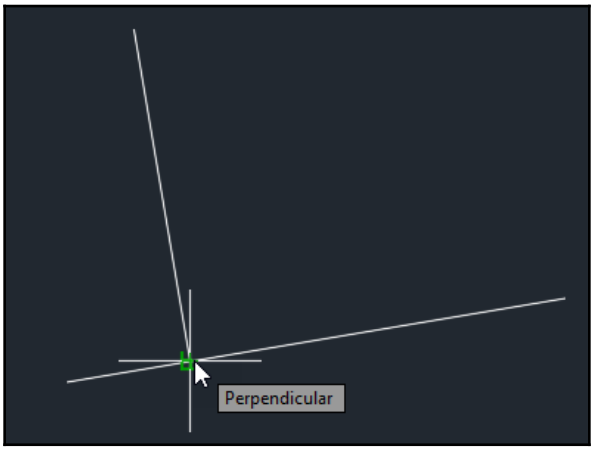


Figure 3.16: A Perpendicular snap making a line perpendicular to the inclined line

A **Perpendicular** snap is not often required and so it is recommended to keep it off most of the time and activate it only when required.

The Tangent snap

The **Tangent** Object Snap lets you snap a line at a tangent point on another curve. The tangent is the point where two curves touch at exactly one point, and this is what the **Tangent** Object Snap does; it lets you snap to the tangent points of two curves. To use this Object Snap, once again make sure it is active in the status bar:

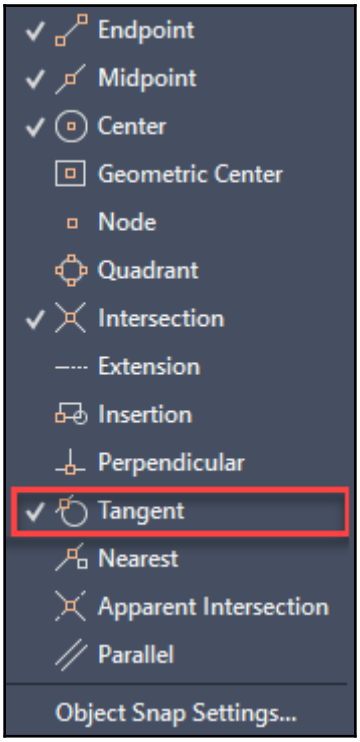


Figure 3.17: The Tangent snap option in the Object Snap menu

Make a circle and select the **Line** command and click on any point near the circle and then move your cursor close to the circle. You will notice a green **Tangent** Object Snap icon, like the one shown here:

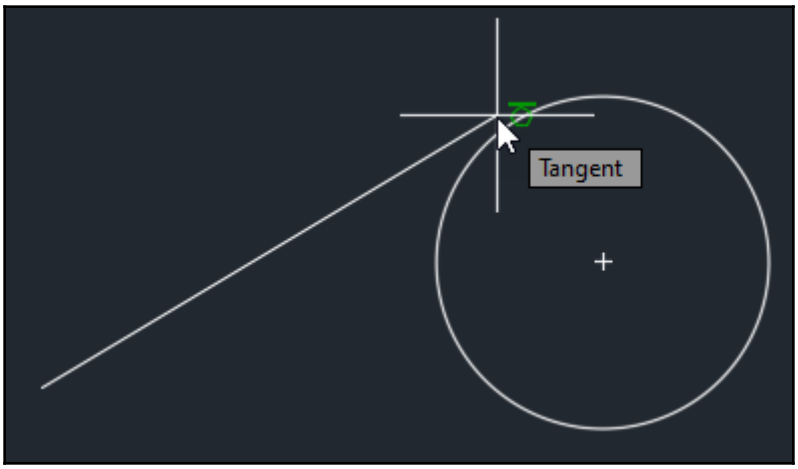


Figure 3.18: A Tangent snap shown in green formed between a line and a circle

Click to snap the line at the tangent point of the circle and you will have a line that starts at a random point but ends at a tangent point on the circle.

The Nearest snap

This Object Snap lets you snap to a point on the curve that is nearest to your cursor. You can also assume that this Object Snap lets you snap to any point on the perimeter of the curve.

To use this snap, once again make sure it is active in the **Object Snap** menu, as shown:

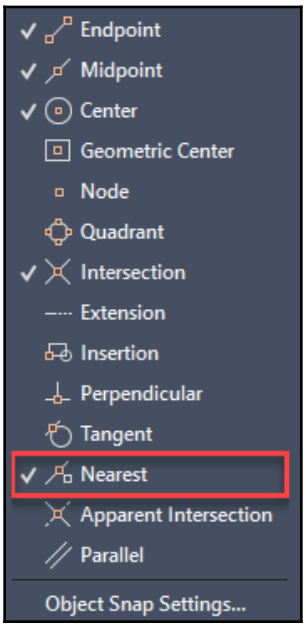


Figure 3.19: The Nearest snap option in the Object Snap menu

To explain this Object Snap option, I will once again make a circle. Now, select the **Line commands**, or if you prefer, any other **draw** command, and then move your cursor close to the circumference of the circle. You will notice that a **Nearest** snap that follows the circumference of the circle will show up and it will let you snap to any point on the circumference where you click:

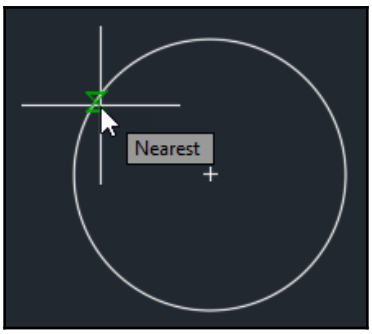


Figure 3.20: A Nearest snap snapping the cursor to the circumference of the circle

As you can see, this Object Snap is a dynamic type; that is, it has no specific point on the curve and it will always snap your drawing at the point closest to the cursor and the perimeter of the curve. Because of its dynamic nature, the **Nearest** Object Snap can be a source of error too as it might snap to points that are too close. So, it is recommended to keep this Object Snap deactivated all the time and activate it only when it is required.

The Parallel snap

As the name indicates, this Object Snap will let you make lines parallel to another line. To use this snap, make sure it is active in the status bar, as shown:

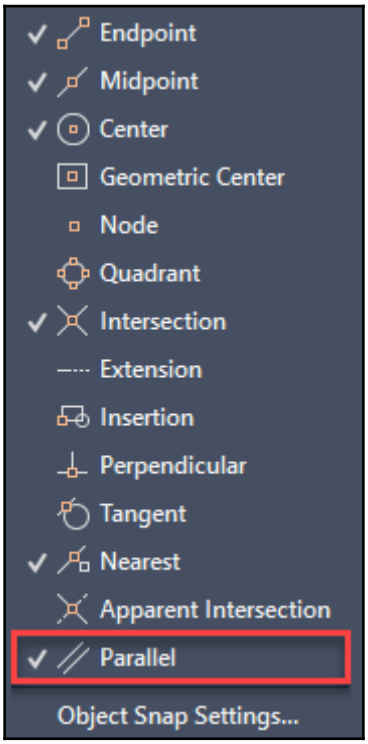


Figure 3.21: The Parallel snap point in the Object Snap menu

In the following example, I will make a line parallel to the existing inclined line using the **Parallel** Object Snap option:

1. Make a line that is inclined to any angle, as shown:

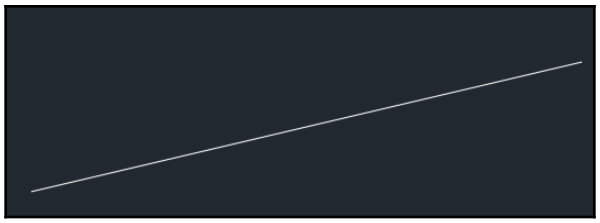


Figure 3.22: A line inclined to an unknown angle

2. Now, select the **Line** command again and click at any point to start the line. Now, move your cursor close to the existing line and then move it out so that it becomes parallel.
3. Once your line becomes exactly parallel to the existing line, a **Parallel** Object Snap, as shown in the following figure, will show up. You will also notice a green tracking line to guide you to make a parallel line:

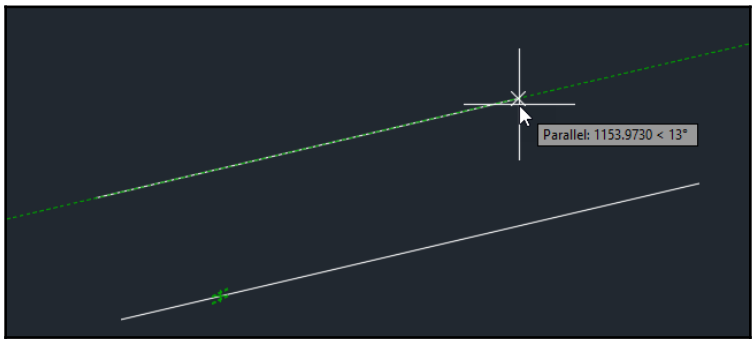


Figure 3.23: A parallel snap making a green parallel tracking vector

4. Just click at any point along this green tracking line to make a line that is parallel to the existing line. This Object Snap option is also not used often and it is recommended to keep it off when not required.

So, these were the Object Snaps that help make precise drawings. Though you may be tempted to keep all of these Object Snap options active, I would still strongly recommend against it and suggest you keep only the most often used Object Snaps active as it will avoid conflict between many snap points, thereby avoiding errors in your drawing.

There is also another way of using these Object Snap points—with the help of Object Snap overrides—which we will discuss in the next section.

The Object Snap override

Sometimes, it can be overwhelming to work with all these Object Snaps, and in drawings where there are lots of snap points close to each other, it can really get difficult to snap to any specific point. In those situations, you can use the Object Snap override to snap to the point that you need, overriding all other points even when they are active on the **Object Snap** menu.

To use the Object Snap override, select any **draw** command, preferably the **Line commands**, and then press and hold the *Shift* key until you see the override icon next to the cursor, as shown:

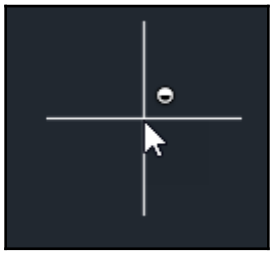


Figure 3.24: The Object Snap override icon next to the cursor

When you see the override icon, right-click on your mouse and you will get an **Object Snap** menu. Select the Object Snap point that you want to use for that command and then move your cursor close to the drawing. You will notice that now, AutoCAD will only snap to the selected Object Snap point and it will not highlight any other Object Snap, even though they are still active in the Object Snap menu.

One of the practical applications of this Object Snap override is in a situation as in the following figure. In the figure, we have two circles and we want to make a line that is at a tangent to both of the circles:



Figure 3.25: Two circles made with different radii

Here, if you start with the **Line** command, then you need to start directly at the tangent point on the circumference of one of the circles, but you can't do that without actually making the line and when making the line, you need to specify a fixed point, which you can't do on the circumference of the circle.

In this situation, select the **Line** command then press and hold the *Shift* key and then right-click to get the Object Snap override menu. Select **Tangent** from this override menu and then click on the circumference of the bigger circle and now you will notice that you will end up with a line that follows the circumference of the circle in the tangent direction, just like the Nearest snap. Repeat the Object Snap override and then select **Tangent** again and this time click on the smaller circle and you will end up with a line that is at a tangent to both of the circles.

In this case, you can't use Nearest to snap to the first circle as the nearest snap will only let you snap to the circumference of the circle and it will not let you make the tangent line.

Using this method, you can make four tangent lines, as in the following figure, which otherwise is difficult, if not impossible, to do using direct Object Snaps:

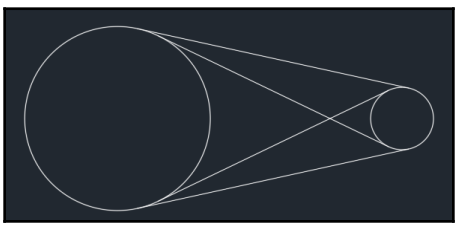


Figure 3.26: Four tangent lines made with the Object Snap override option between two circles

So, as you have noticed, the Object Snap override is a great way of suppressing all the Object Snaps except the selected one in the drawing. If you want to exclusively use an Object Snap, then instead of hiding all other Object Snaps from the Object Snap menu, use the Object Snap override and you will have the same result.

As you have noticed, Object Snap is an indispensable tool when it comes to creating a precise drawing. Just like Object Snap, there is another status bar option called **Object Snap tracking**, which makes the process of creating precise drawings easier. In the next section, we will learn about using Object Snap tracking, too.

Object Snap tracking

Object Snap tracking is a great tool for making geometries with reference to existing points in a drawing. To use this tool, activate it from the status bar, as in the following screenshot. It's the tool right next to the Object Snap icon:



Figure 3.27: The Object Snap tracking option in the status bar

Now, let's consider a scenario where we want to make a circle at point **P**, which is the intersection of the **AB** and **ED** lines if extended further along the dashed line, as shown:

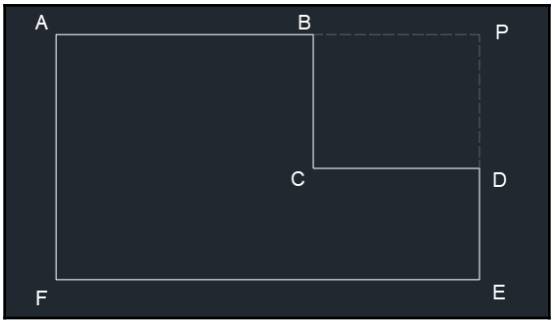


Figure 3.28: A sample drawing for the Object Snap tracking option

In this case, if we don't make the **BP** and **PD** lines, then we won't get the intersection point of these lines, which should be at the center of the circle. So, to get a reference like this, which does not actually exist in the drawing but exists only when existing drawings are extended further, we can use Object Snap tracking.

Now, to make the required circle on the **P** point, start the **Circle** command and then move your cursor to the **B** point but don't click. Now, move your cursor along the **BP** line so that a green tracking line shows up and when it does, move to the **D** point and once again, don't click and just track the **DP** line. You will once again get a green tracking line. Keep moving till you reach the **P** point and an intersection marker will show up, as shown:

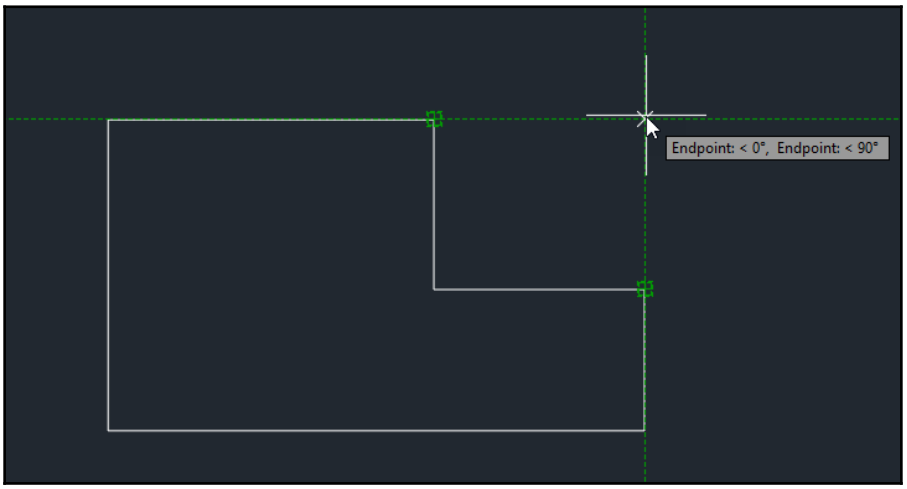


Figure 3.29: The Object Snap tracking vectors are shown in green

Click to specify the center of the circle and then add its radius to make the required circle on the **P** point. In a similar way, you can use Object Snap tracking to track existing points of a drawing to get the references that don't exist in the drawing and make drawings using those references.

So, now we know all about creating precise drawings using Object Snaps and status bar options, it's time to dig deeper into the draw tools. In the next section, we will do just that with the ellipse tool, which happens to be a conic curve.

Making an ellipse

Conic curves are formed when a cone is intersected by a plane. As shown in the following diagram, when a plane not parallel to the base of a cone cuts it in a specific way, an ellipse is made. An example of an ellipse in the real world is the path of planets around the sun:

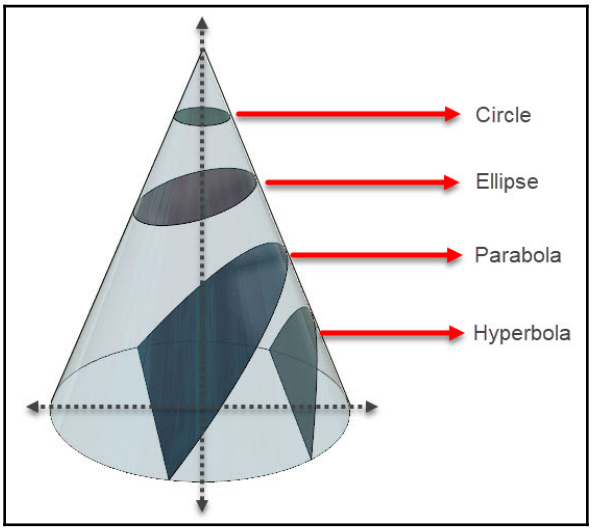


Figure 3.30: The conic curves made by intersecting cones along different planes

There are two ways of making an ellipse in AutoCAD. You can make it with a center and an endpoint or with endpoints of the axes. Both the ellipse commands are in the **Draw** panel in the **Home** tab under the ellipse flyout, as shown:

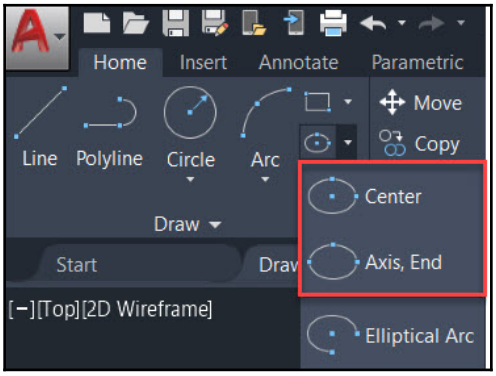


Figure 3.31: The Center and Axis, End ellipse options in the Draw panel

To make a center ellipse, select the **Center** option from the ellipse flyout in the **Draw** panel:

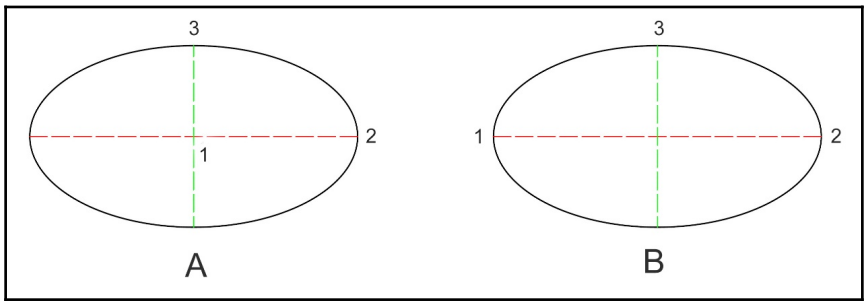


Figure 3.32: The A ellipse is made with the Center ellipse option and the B ellipse is made with the Axis, End option

The command line will now prompt you to specify the center point of the ellipse. Click on a point and that point will become the center, shown as **1** in the **A** ellipse in figure 3.32.

Now, click on the endpoint of one of the axes, click on **2**, then click on **3** to make the ellipse.

To make the ellipse using the **Axis, End** option, you need to first click on the axis end, shown as **1** in the **B** ellipse, then the second point of the axis, shown as **2**, and then one of the points of another axis, shown as **3**.

Though there are tools for making an ellipse, we unfortunately don't have a direct tool for making other conic curves in AutoCAD. So, now let's move on to another type of draw tool, called **construction line**. This tool is more to help create geometry rather than the actual drawing itself; we will discuss it in the next section.

The Construction Line command

This command is available in the expanded **Draw** panel in the **Home** tab, as shown:

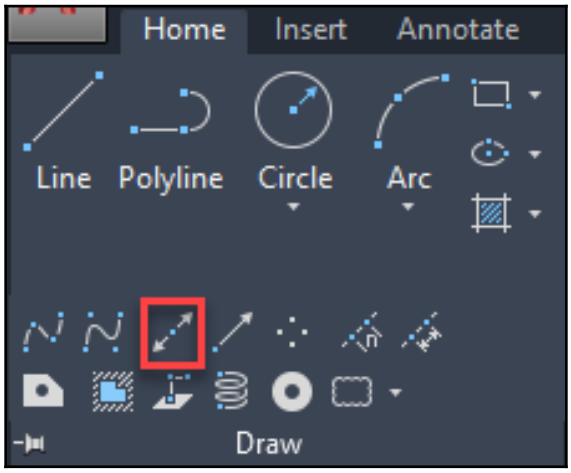


Figure 3.33: The Construction Line option in the expanded Draw panel

Construction Lines are infinite lines that are helpful when you make lines for reference. An example of a reference line is a construction line made using the reference of a floor plan to make the elevation. In the following figure, the red lines are Construction Lines:

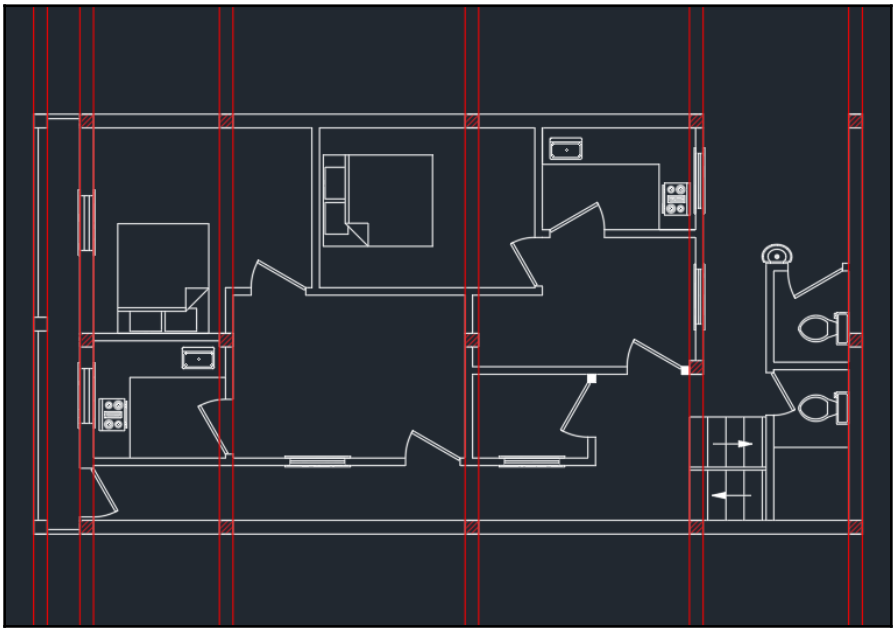


Figure 3.34: Red lines made for reference using the construction line tool

Using these red lines, you can make another drawing that exactly aligns with the floor plan shown here. Make a note here that I have used a red color for the construction line to make them look distinct but in general, the construction lines take the default drawing color.

In the following example, I will show you the steps to make a construction line:

1. To make a construction line, select the tool from the expanded **Draw** panel of the **Home** tab or use its command, `XL`.
2. Now, click on a point to specify the first point of the line and then click on another point in the drawing area to specify the direction of the construction line.
3. As mentioned earlier, construction lines are infinite lines and no matter how much you zoom out, they will always appear to extend on both sides. You can, however, trim your construction line to make it of finite length and in doing so, the construction line turns into a simple line.

In the following figure, I have trimmed the construction line using the circle as a reference and the resulting object is a line:

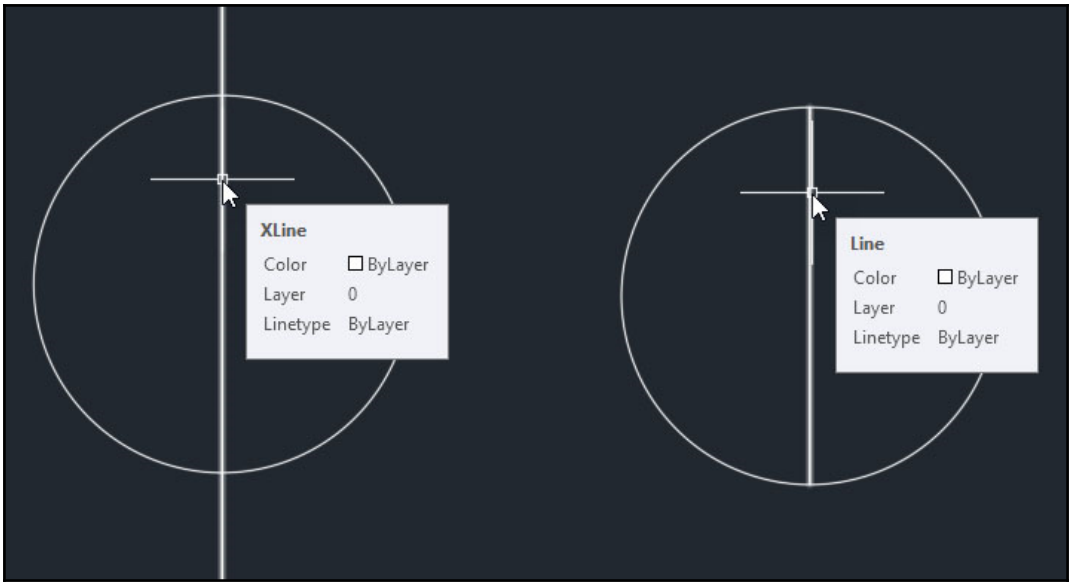


Figure 3.35: A construction line turning into a line when reduced to a finite length

As you may have noticed, you can use construction Lines to aid the drawing workflow but when these construction lines are converted to a finite length, they become normal lines and so can be used directly in the drawing, too. The next draw tool that we will discuss in the next section is unlike all the other draw tools that we have discussed so far. This tool is called **revision cloud** and it is only used as markup in a drawing and it is usually removed from a drawing when the drawing markups are addressed.

The revision cloud tool

Revision cloud is a markup tool that is not generally used to make things in AutoCAD; rather it is used to show areas in a drawing that are subjected to further inspection or to draw attention to some parts of the drawing.

The revision cloud tool is also in the expanded **Draw** panel in the **Home** tab, as shown:

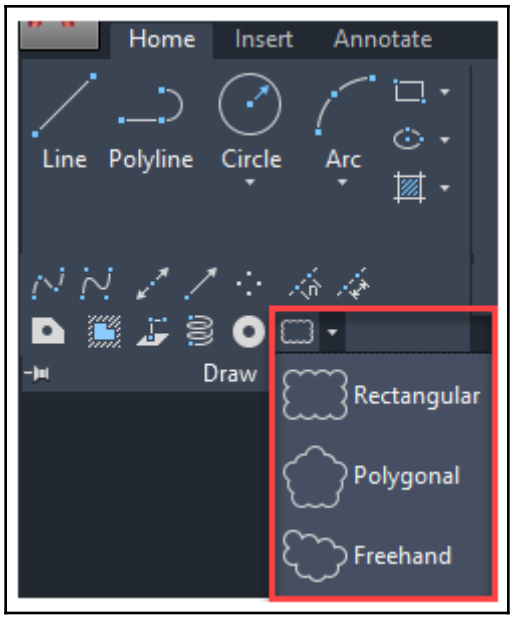


Figure 3.36: The revision cloud options in the expanded Draw panel

As you can see, there are three different ways of making a revision cloud in AutoCAD: **Rectangular**, **Polygonal**, and **Freehand**.

The Rectangular revision cloud tool

The Rectangular revision cloud tool will make a revision cloud in the shape of a rectangle. In the following example, we will learn how to make this type of revision cloud in our drawing:

1. Select the **Rectangular** tool from the revision cloud flyout or use its command, `REVCLOUD`.
2. Now, the command line will prompt you to specify the first point of the revision cloud. Click on the point from where you want to start it.
3. Then, the command line will prompt you to specify the opposite corner of this Rectangular revision cloud. Click on another point so that it includes the area you want to highlight using the revision cloud.

In the following figure, I have highlighted the bed blocks using revision cloud as their size is too big for the room and needs revision:



Revision Clouds are not green in color. Here, I have used a green color to highlight them clearly.

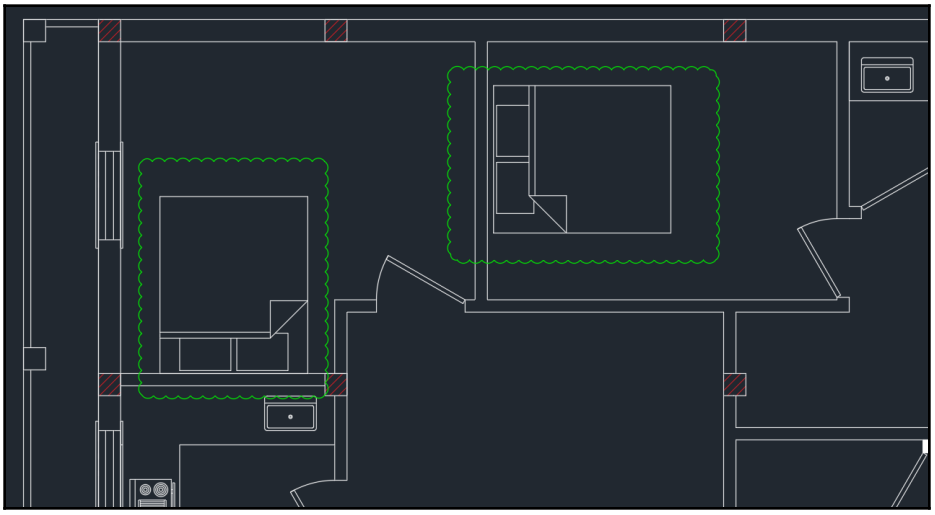


Figure 3.37: Bed blocks highlighted with revision clouds

Revision Clouds generally have a standard arc length that will be automatically adjusted as per the scale of the current drawing. So, when you make your first revision cloud, it will be made with the proper arc lengths. This is, however, a new feature added in the 2021 version of AutoCAD and if you are using an older version of AutoCAD, then you may need to adjust the arc length of the revision cloud as per the drawing scale:

1. To adjust the arc length, start the **revision cloud** command.
2. Click the **arc length** option from the command line.
3. Type in the approximate arc length and press *Enter*. If you are using a version older than 2021, then you will be prompted to specify the small arc length and the large arc length.

Once you are done setting the arc length, make revision cloud and it will be made with specified properties. Revision cloud is a different object type called `Revcloud`, and when you hover your cursor over it will show it as such.

So, as you have noticed, adding Rectangular revision cloud is similar to adding a rectangle to a drawing and just like a rectangle, this revision cloud can also be modified after its added to the drawing using the multifunction grips.

The Polygonal revision cloud tool

To include a Polygonal area with multiple sides, you can use the next revision cloud method—Polygonal:

1. Select **Polygonal** from the revision cloud flyout and once again click on a point to start it.
2. Then, click on another point and keep on clicking to enclose the area that you want to highlight using revision cloud.

3. When you are done making the revision cloud, press *Enter* to exit the command:

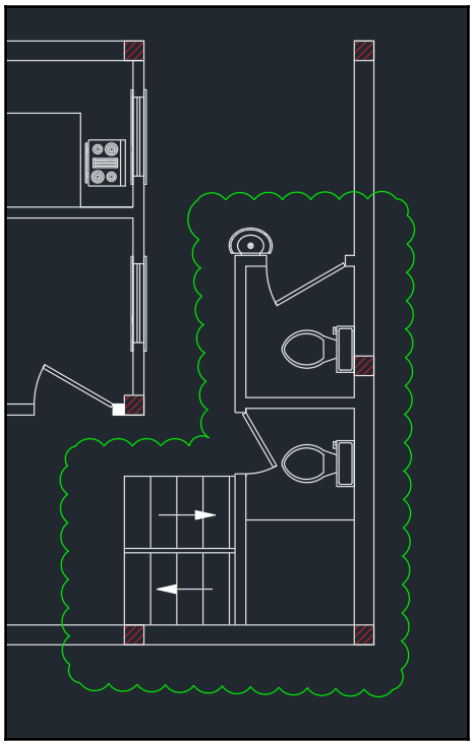


Figure 3.38: Revision cloud made with the Polygonal option

In the preceding figure, I have made a Polygonal type revision cloud that includes a part of the drawing. Although in this case the polygon contains six vertices, you can make Polygonal revision clouds with as many sides or vertices as you want.

The Freehand revision cloud tool

As the name suggests, you can make a revision cloud with this method using Freehand mouse movement, as shown in the following example:

1. To use this option, select the **Freehand** option from the **revision cloud** flyout.
2. Now, click on a point and let go of your cursor.

3. Start moving your mouse. You will notice that a Freehand revision cloud that tracks your mouse movement will be made and when you bring it close to the starting point of the revision cloud, the command will automatically exit:

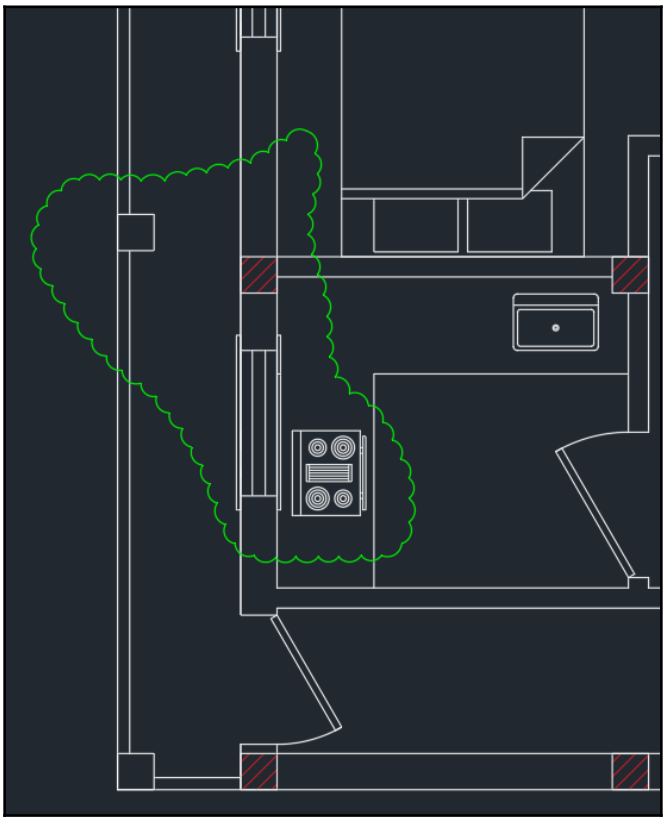


Figure 3.39: A Freehand revision cloud in the drawing

In the preceding figure, we have a Freehand revision cloud that has no vertices. You can make this type of revision cloud where you don't need the precision of Polygonal or Rectangular revision clouds and just want to enclose a patch of drawing with a revision cloud.

Working with regions

A region is a different kind of 2D object on which you can perform Boolean operations, such as Union, Subtract, and Intersect. You can convert closed 2D objects into a region using the `REGION` command or using the region tool from the expanded **Draw** panel:

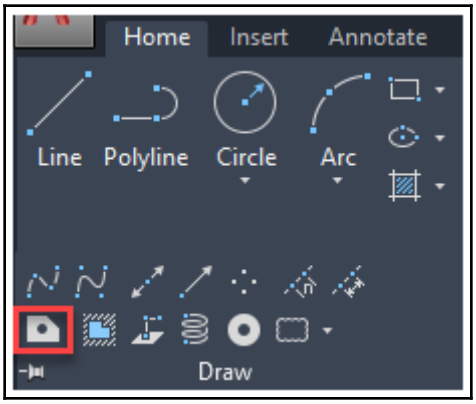


Figure 3.40: The region tool in the expanded Draw panel

Let's make two simple regions and then we will perform Boolean operations on them.

Making regions

We will learn how to make regions using two closed geometries, that is, a circle and a triangle, as shown:



Figure 3.41: A circle and a triangle

In the following example, we will first make these closed drawings and then we will learn how to convert them into regions:

1. Make the circle using the circle tool from the **Draw** panel and use any radius value. Also, make the triangle with any dimensions using the line of polyline tool.
2. Now, select the region tool from expanded Draw panel or use its command, `REGION`.
3. The command line will now prompt you to select the objects. Select the circle and triangle and press *Enter.*
4. Visibly, the drawings will still look the same but they are now converted into regions and a message confirming this will show up above the command line, as shown:

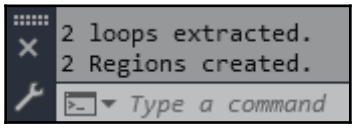


Figure 3.42: The Command Prompt when objects are converted to regions

So, now the circle and triangle are converted into two separate regions, which are basically rigid 2D shapes. So, after the region tool, let's learn about the point tool, which is also called **nodes**, in AutoCAD.

Point and point style

Points, or nodes as they are called in AutoCAD, are single-pixel objects that are generally used as a reference. The point command is also in the expanded **Draw** panel of the **Home** tab, as shown:

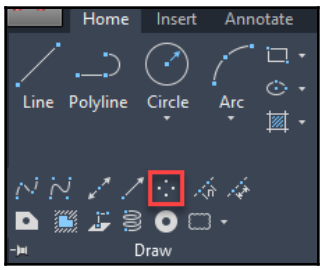


Figure 3.43: Multiple point tools in the expanded Draw panel

To make the points, select the point command from the **Draw** panel of the **Home** tab or use its command, `PO`. Click on different places in the drawing area to make the points. Once you are finished making the points, press *Enter* to escape the point command.

As you will notice, the points are too tiny to notice and though they may be visible in the blank drawing area, they will simply disappear when overlapped with other geometries:



Figure 3.44: Points created with the default point style

To make the points prominently visible, you can change the point style. To change the point style, select the **Point Style...** tool from the expanded **Utilities** panel in the **Home** tab, as shown in the following figure. Alternatively, you can also use its command, `PTYPE`:

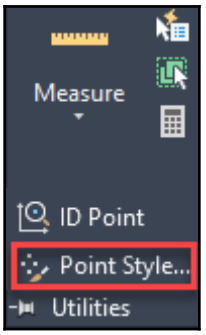


Figure 3.50: The Point Style... option in the expanded Utilities panel

The **Point Style** window has a list of point styles that you can choose from. The first point style is selected by default, which is the single-pixel point type. The second point style is blank and will make your points disappear. The remaining point styles can be selected to make points look more prominent:

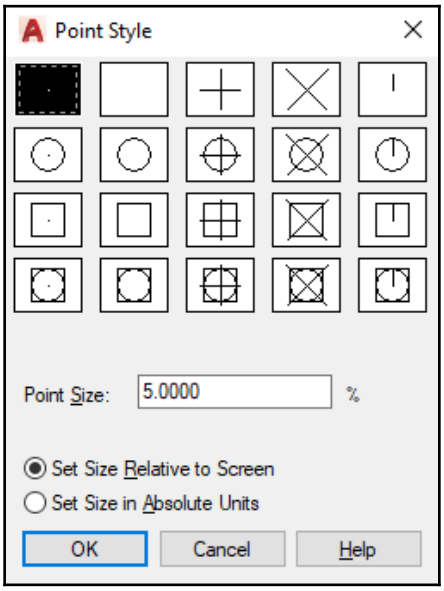


Figure 3.51: Different point styles in the Point Style window

The point size is relative to the drawing area and the size of the point will change when you zoom in or out. But if you want a fixed size or the absolute size for the points, then select the second radio button, **Set Size in Absolute Units**, from the **Point Style** window and then specify the point size in the **Point Size** field.

This is how the points we made earlier will look with the changed point style:



Figure 3.52: Points replaced with different and more prominent looking point style

The points, or nodes, can only be selected if the node Object Snap is active in the Object Snap menu. If the node Object Snap is not active, then you won't be able to snap to these points.

Working with Spline

Splines are mathematical curves called **nonuniform rational B-splines** (**NURBS**) that can be used to make organic shapes in AutoCAD. An example of a drawing made with a spline is this top view of a mouse. Making this shape with regular draw tools, such as Circle, Arc, and Line, will be difficult, as shown:

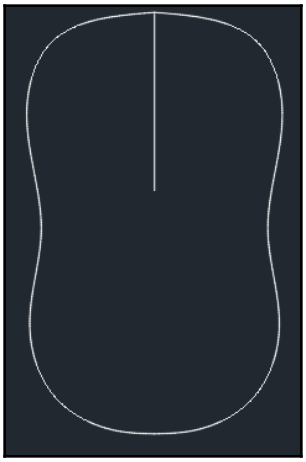


Figure 3.53: A simple top view of a computer mouse made with the spline tool

In AutoCAD, there are two ways of making spline. They are spline fit, which is defined by the fit points, and spline **control vertice** (**CV**), which is defined by the CVs or control vertices that act as a frame for the curve.

Spline fit

In the expanded **Draw** panel in the **Home** tab, you will find the spline tool, as shown in the following figure. Alternatively, you can use its command alias, `SPL`:

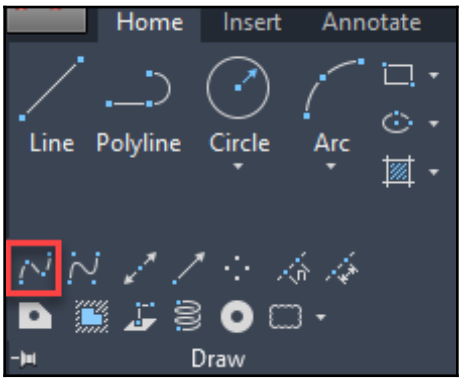


Figure 3.54: The spline fit option in the expanded Draw panel

In the next example, we will make a simple spline and then we will modify it using fit points:

1. Start the Spline command and then click on a point in the drawing area.
2. The command line will prompt you to specify the next point. Click on the next point and then keep clicking to add multiple points.
3. Once you have finished making the spline, press *Enter* to exit the command.
4. After making the spline, select it, and its fit points shown with blue square dots will highlight.
5. You can modify this spline by selecting these blue dots. Select any one of the fit points and then move it in the drawing area and the complete spline will change.

As you may have noticed, in the case of splines, you can primarily specify points by clicking in the drawing area and, though this method won't make drawing with precise dimensions easier, it is still useful in situations where you only need to draw the outline of objects:

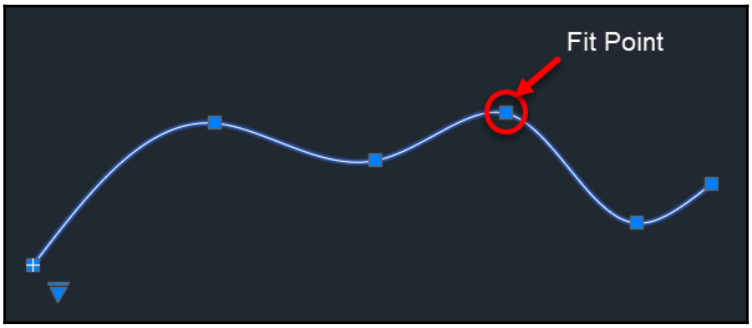


Figure 3.55: Fit point grip in the spline fit

The spline fit offers more curvature and you can use it for many drawings but when you need more control over the flow of the spline curve, then you can use another kind of spline, called **Spline CV**, with control vertices, which we will discuss next.

Spline CV

This method is also like the spline fit method and the workflow for making spline using this method is also the same. The only difference is in the way control vertices are made, in this case:

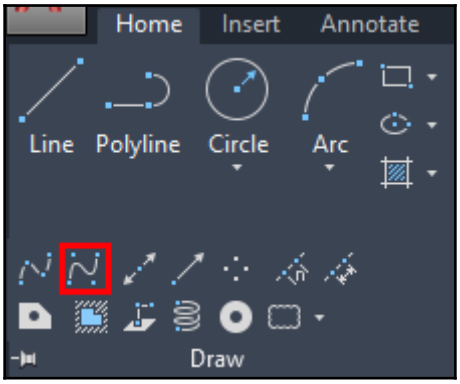


Figure 3.56: The Spline CV option in the expanded Draw panel of the Home tab

In the following example, we will make a simple drawing using Spline CV and we will also modify it using its grips:

1. To make Spline CV, select the tool from the expanded **Draw** panel of the **Home** tab, as shown in the preceding screenshot.
2. Click on a point in the drawing area to start the spline and then keep clicking on other points to make the spline.
3. Once you have finished making it, press the *Enter* key to exit the spline command.
4. Select the spline you just made, and in this case, you will notice the control points or CV, as shown in the following figure. You can further modify the spline shape by moving these CVs:

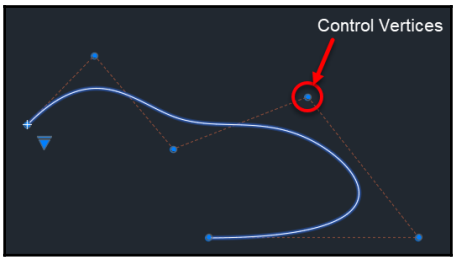


Figure 3.57: Control vertices highlighted in the Spline CV

The Spline fit and CV can be changed from one form to another using the blue arrow grip, as in the following figure. Click on the grip and select the spline type you want from the menu:

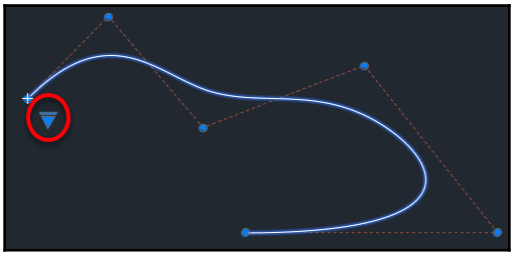


Figure 3.58: The grip for changing the spline type highlighted in the red oval

Splines are great for making curves where precision is not needed and also where you just want to trace the outline of an underlying image. Even in splines, you can use "fit point" or "control vertices" to control the degree of curvature control and make a loose or a tightly controlled spline. In the next section, we will start discussing tools that help to fill closed areas with patterns or colors. We will start with the hatch command.

More Modify commands

So, now you know about making simple drawings with most of the draw commands, including hatches, it's time to dig deeper into the realm of Modify commands, which will further help us to improve our drawing workflow.

Let's start with one of the simplest Modify commands: **Mirror**.

The Mirror command

Using the **Mirror** command, you can make a mirror image of a drawing object in AutoCAD. To explain this command, I will use the drawing shown here:

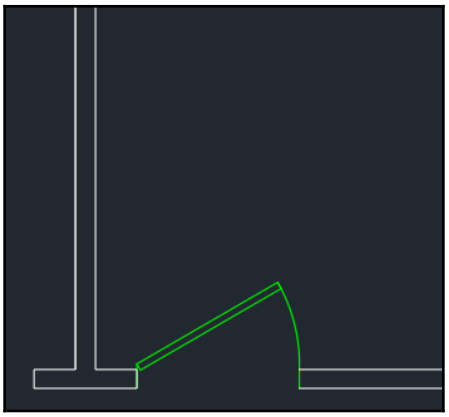


Figure 3.59: Drawing to be used for the Mirror command

Here, we have a door on the right side of the wall and we need to add another door on the left side of the wall with an exact distance. In the following example, we will mirror the existing door about mid-wall to get the second door using the Mirror command:

1. Select the **Mirror** command from the **Modify** panel or use its command alias, `MI`.

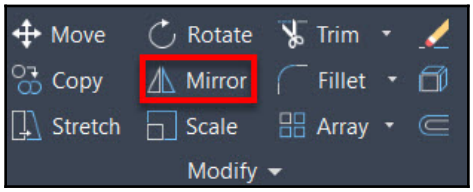


Figure 3.60: The Mirror command in the Modify panel

2. Now, the command line will prompt you to select the object. Click on the green door symbol to select it and press *enter*.
3. Now, the command line will prompt you to specify the mirroring line. This will be a reference line that the mirror image will be made around. Click on the midpoint of the wall and then again on another point right above it, as shown:

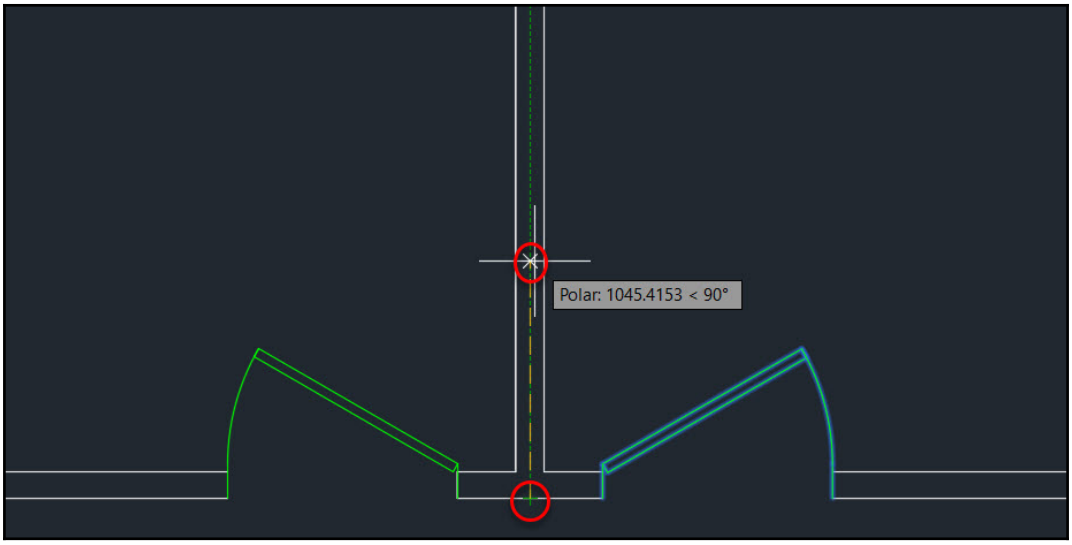


Figure 3.61: Mirroring the door on the right using the Mirror command and the mirror line, highlighted by red ovals

Now, the command line will prompt you to specify whether you want to erase the source object. The source object is the door symbol we used to make the mirror image. If you select `Yes`, the original door symbol will be erased but its mirrored copy will be retained. If you select `No`, both the door symbols will be retained. The default option, in this case, is `No`, which is shown in the angle bracket. Simply press *Enter*; the default value will be selected and you will get a mirror image of the door symbol:



Figure 3.62: The erase source objects option of the Mirror command

The Mirror tool treats text and drawing objects differently and so when you mirror text, you won't get the expected result. To mirror text just like a drawing object, change the `MIRRTEXT` system variable to `1`. The default value of the `MIRRTEXT` system variable is `0`. After, the mirror tool lets you look at another modify tool, called **offset**, which helps you make a copy of a drawing at an offset distance.

The Offset command

The Offset command lets you make an offset copy of an object at the distance you specify:



Figure 3.63: A drawing made with a polyline for the Offset command

In the following example, I will explain this command using the preceding drawing. This drawing is made with a single polyline and there are no breaks in its boundary. In the following example, we will offset the polyline to make another copy at the offset distance:

1. Select the Offset command from the **Modify** panel, as in the following screenshot, or use its command alias, `O`:

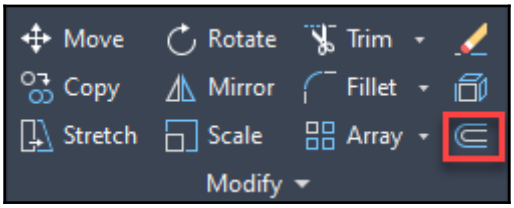


Figure 3.64: The Offset command in the Modify panel

2. Now, the command line will prompt you to specify the offset distance. Type the distance that you want and press *Enter*. In this case, I am using an offset distance of `5` units.
3. Now, click on the boundary of the polyline to select it and you will notice that an offset copy will appear on your cursor. If you hover your cursor outside the shape, the offset copy will be made outside and if you click inside, the offset will be added inside the drawing:

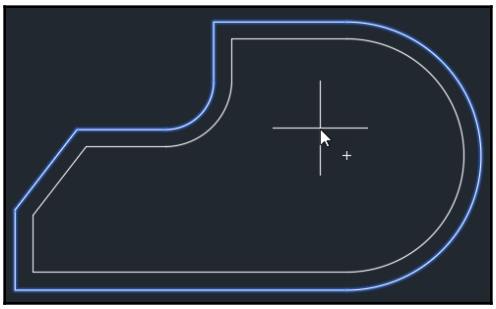


Figure 3.65: A polyline offset inside at a distance of 5 units

4. Even after making the offset, the command will remain active and you can select other objects to make the offset at the specified distance. To exit the command, press the *Esc* key on your keyboard.

In this case, the offset has been made on a polyline, but if you try making an offset on a similar drawing that is made with separate lines and arcs, then the result will be very different, as shown:

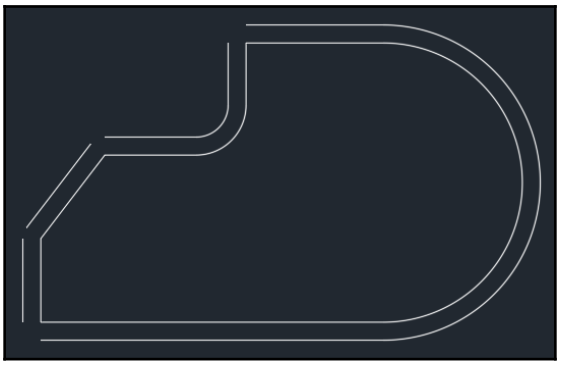


Figure 3.66: An Offset made outside the drawing when the object was made with separate lines and arc

Here, all the line segments and arcs are not connected to each other and so you need to select them one by one and make offsets separately, which will result in the offset shown in figure 3.66.

The Scale command

This command lets you scale a drawing up or down or, in other words, you can use the **Scale** command to make a drawing larger or smaller. You can select the **Scale** command from the **Modify** panel, as in the following screenshot, or you can also use its command alias, `SC`:

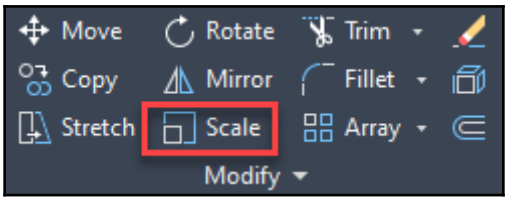


Figure 3.67: The Scale command in the Modify panel

For the **Scale** command, I will use this window symbol, where the length of the **AB** line is `10` units:

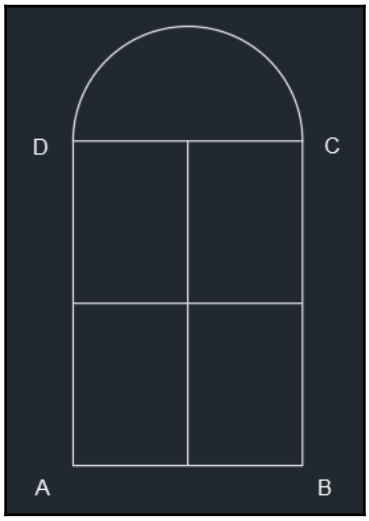


Figure 3.68: A sample drawing for the Scale command where the AB line is 10 units in length

In the following example, we will learn how to change the scale of the window symbol shown in the preceding figure using different scale factors:

1. Start the **Scale** command, then select the complete drawing that you want to scale, and press the *Enter* key.
2. AutoCAD will now prompt you to select the base point. While scaling an object, the base point remains fixed, so in this case, let's click on the **A** point of the window symbol for the base point.
3. Now, you need to specify the scale factor that will ultimately decide the final size of the selected drawing. A scale factor between `0` and `1` will decrease the size of the object and a scale factor greater than `1` will increase the size of the object. As an example, a scale factor of `0.5` will make the drawing half the size and a scale factor of `2` will make the drawing twice the size. Similarly, for decreasing the size by one fourth, you need to use a scale factor of `0.25` and for making it four times larger, the scale factor will be `4`.
4. In this case, let's type `2` for the scale factor and then press *Enter*.

A new window symbol twice the size of the original one will be made, and the **A** point will remain where it was before the Scale command as it was selected as the base point.

In this case, the original object was scaled up in size but if you want to keep the original object in the drawing and want to scale a copy of it, then you can use the `Copy` subcommand of the Scale command, which is explained in the next section.

Making a copy of the scaled object

If you want to scale the drawing and want to keep the original object too, this can be done using the **Copy** subcommand of the Scale command. To do this, once again start the Scale command and then select the window symbol and press *Enter*. Now, click on a point to specify the base point, in this case, I am once again selecting **A** as the base point. Now, the command line will prompt you to specify the scale factor and, in the subcommands, you will notice the `Copy` and `Reference` options. Click on the `Copy` option from the command line or type *C* and press *Enter* to select the `Copy` subcommand:

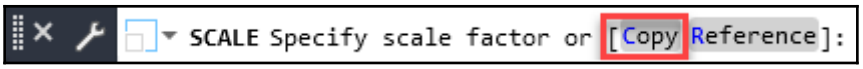


Figure 3.69: The Copy subcommand in the Scale Command Prompt

Now, type the scale factor. I am using a scale factor of `0.5`, in this case. Then, press *Enter* and you will have two window symbols now: an original one and another one that is half the size of the original one as shown:

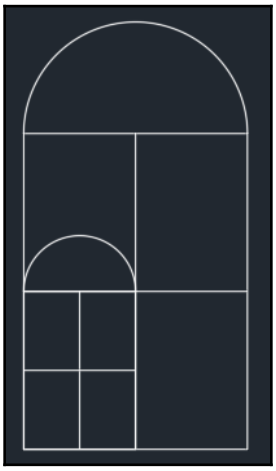


Figure 3.70: A drawing scaled down with a factor of 0.5 and with the Copy subcommand selected

After making a scaled copy of original objects, let's talk about changing the scale of the object with respect to a reference.

Scale with a reference

Let's consider a situation where we have the window symbol that we have used so far with an unknown length of the **AB** baseline and we want the length of the **AB** line to become `16` units and the rest of the window should scale proportionally. In this case, you can use the reference option to scale the drawings with reference to any specific length and you can avoid using a scale factor altogether. This method of scaling drawings using a reference is shown in the following example:

1. To use this feature, once again start with the scale tool, select the window symbol that you want to scale, and press the *Enter* key.
2. Now, the command line will prompt you to specify the base point. Select the `A` point once again and now, from the command line options, select `Reference` or type `R` and press *Enter* to select the `Reference` subcommand, as shown:

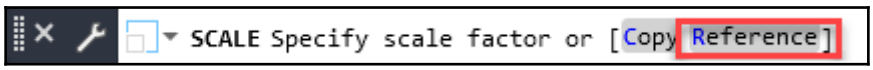


Figure 3.71: The Reference subcommand of the Scale command

3. Now, the command line will prompt you to specify the reference length. In this case, the reference is the **AB** line of the window and if you know its length, type it in and press *Enter*. Otherwise, click on the point and then the **B** point and AutoCAD will take that length as the reference length.
4. Now, the command line will prompt you to specify a new length. Type the new length of **AB** that you want; in this case, type `16` and press *Enter*.

You will notice that the window will scale so that the length of the **AB** line on the window becomes `16` units and other lines and arcs of the window will change in size, proportionally.

So, this was all about using Scale command in different situations. In the next section, we will learn to make chamfers on the edges or vertices of different geometries.

The Chamfer command

Using the **Chamfer** command, you can add chamfers or slant lines along the edges of intersecting lines. An example of a chamfer is shown in the following figure. Here, a chamfer is applied to the **C** vertex of the rectangle:



Figure 3.72: A chamfer applied to the C vertex of the rectangle

You can start the **Chamfer** command from the expanded **Chamfer** flyout in the **Modify** panel of the **Home** tab, as in the following screenshot, or you can also use its command alias, `CHA`:

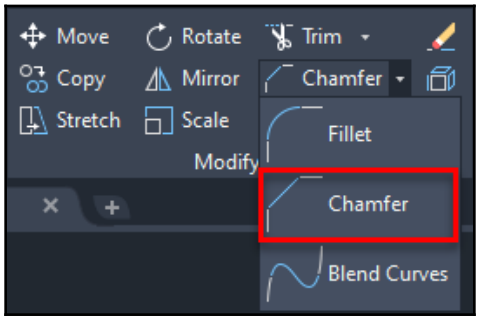


Figure 3.73: The Chamfer option in the Modify panel

To explain the **Chamfer** command, once again I will use a simple rectangle with a length of `10` units and a height of `5` units made with a polyline. In the following example, we will learn how to create chamfers on the vertices of this rectangle:

1. Start the **Chamfer** command from the **Modify** panel or using its command alias, `CHA`. Now, the command line will prompt you with many options, as shown:

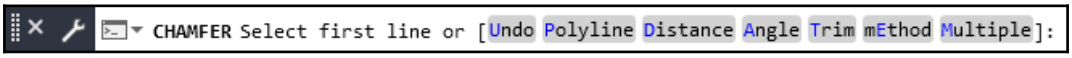


Figure 3.74: The Chamfer command with all its subcommands on the command line

2. You need to specify the chamfer distance before you can apply it to the vertices of the rectangle. To specify the chamfer distance, select the `Distance` subcommand from the command line or type `D` and press *Enter* to select the `Distance` subcommand.
3. Now, type the first distance value—in this case, I am using `2` units—and then press *Enter*. Now, type the second distance value—I am using `1` unit—and then press *Enter* again.
4. Now, the command line will prompt you to select the first line for the chamfer. Click on the longer line of the rectangle and then click on the adjacent shorter line of the rectangle. A chamfer, as in the following figure, will be added:



Figure 3.75: A chamfer applied to the C vertex of the rectangle with the first distance as 2 and the second distance as 1

5. In this case, the red dotted line on the **C** vertex is the first chamfer distance of `2` units and the green line is the second chamfer distance of `1` unit.

While making the chamfer, we selected the **CD** line first and then **BC**, so the first distance became `2` units and the second distance `1` unit, but you could have clicked on **BC** first to make it the first distance of `2` units and **CD** next to make it the second distance. So, the order of selection of the chamfer lines decides the chamfer distance.

Here, we selected unequal chamfer distances, but you can also select equal chamfer distances to make uniform chamfers or you can use a distance and angle value for the chamfer, which is explained in the next section.

Making a Chamfer using an angle

Adding a distance is not the only method of making chamfers and in the next example, we will use a combination of a distance and an angle to make a chamfer:

1. Once again, start the **Chamfer** command from the **Modify** panel or use the command alias, `CHA`. Now, select the `Angle` option from the subcommands of the `Chamfer` command, as shown:



Figure 3.76: The angle subcommand of the chamfer command in the command line

2. Now, the command line will once again prompt you to specify the chamfer length of the first line. In this case, I am using a length of `2` units, so type `2` and press *Enter*.
3. Now, the command line will prompt you to add an angle value with respect to the first line (in this case, you won't be prompted for the second distance). Type `30` and press *Enter*.
4. Now, we will have a chamfer with a length of `2` units and inclined to an angle of `30` degrees, with respect to the 2-unit line. Select the **CD** line and then the **BC** line and we will have our chamfer, as shown:

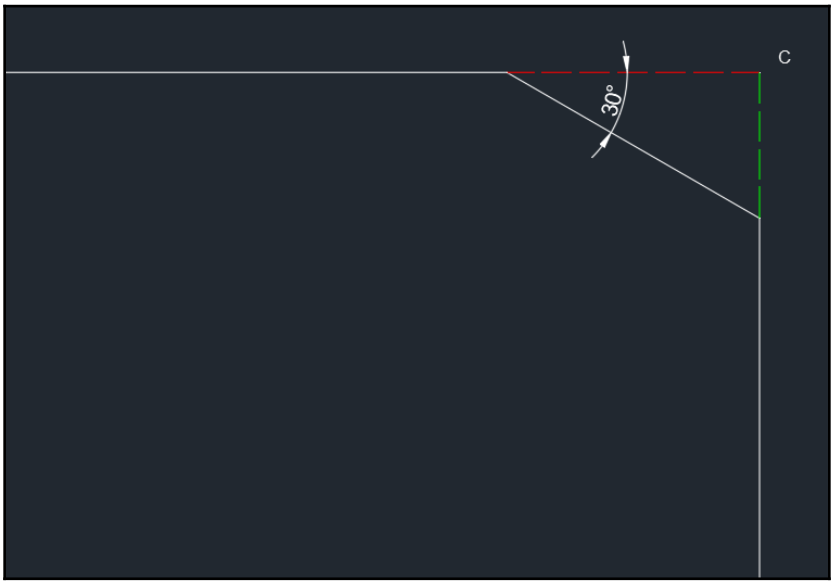


Figure 3.77: The chamfer made with the angle option on the C vertex of the rectangle with a 30-degree angle and 2-unit length

5. As you can see in the preceding figure, the line shown in red is the first chamfer distance of `2` units and the 30-degree angle is between the line and the chamfer itself.

In this case, the **Chamfer** command exits once you have finished making the chamfer but if you want the **Chamfer** command to continue even after you have made the chamfer, then select the `Multiple` option from the chamfer subcommands, as shown:

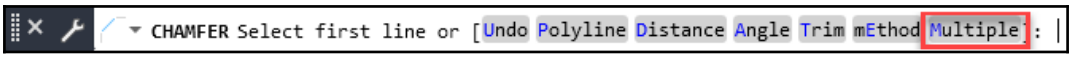


Figure 3.78: The multiple subcommand of the chamfer command

After that, follow the normal steps for making a chamfer and the command will continue, even after making the chamfers. To exit the chamfer command, press the *Esc* key on your keyboard. In the preceding example, we made a chamfer using the `Multiple` option, but we needed to select all the vertices. We also have the option of making multiple chamfers using a single click, which is explained in the next section.

Making Chamfers on multiple vertices

In the case of polylines, you can apply chamfers on multiple vertices directly using the polyline subcommand. To explain this once again, I will use the rectangle we have used so far for the chamfers and apply chamfers on its multiple vertices in the following example:

1. Select the **Chamfer** command and click on the `Polyline` subcommand from the command line or type `P` and press *enter*.
2. Now, select the `Distance` subcommand from the command line and type the first and second distance for the chamfer. In this case, I am using a chamfer distance of `1` unit for both the `1` and `2` distances.
3. Now, click on any line of the rectangle and you will notice that the chamfer will be applied to all four vertices of the rectangle. You will see the following shape:

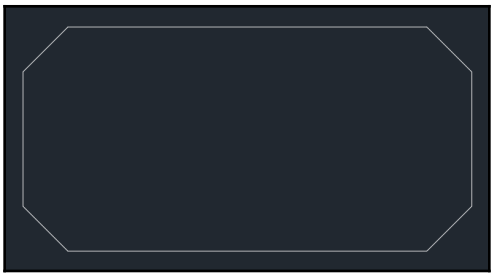


Figure 3.79: Chamfers made on all four vertices with the polyline subcommand

In this case, the rectangle was made with polyline, so the chamfer was applied to all the vertices simply by clicking on any side of the rectangle. In the case of open curves, which are also made with polyline, you can use this method to make chamfers on multiple vertices.

After the **Chamfer** command, its time to discuss a command that will join broken geometries into a polyline, arc, line, or spline. This command is called **Join** and I have explained it in the next section.

The Join command

Using the Join command, you can join different curves that are collinear or connected end-to-end into a single curve. To explain the join command, I will use the three drawings shown here:

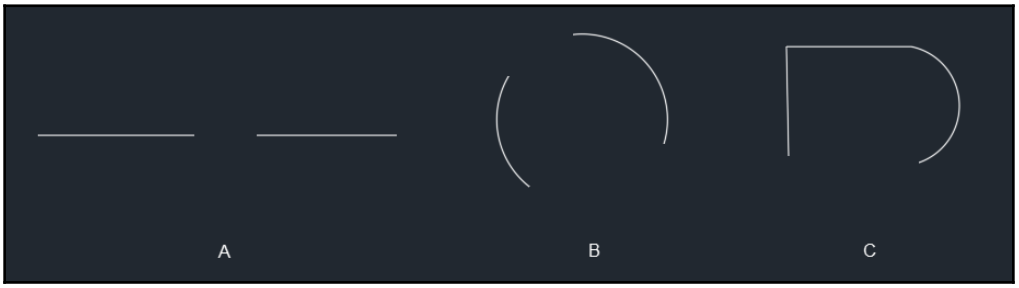


Figure 3.80: Sample drawings used for the join command

In the following example, we will join these drawings together and we will also merge the gaps that exist in these drawings:

1. Start the Join command from the expanded **Modify** panel in the **Home** tab or use its command alias, `J`:

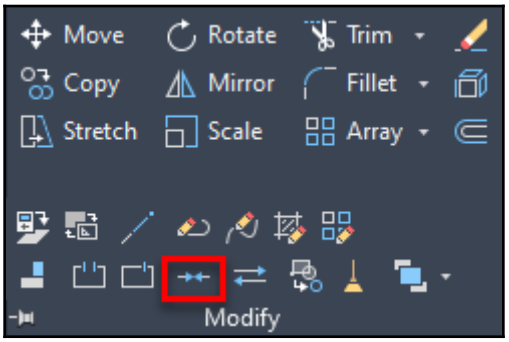


Figure 3.81: The join command in the expanded Modify panel

2. Now, the command line will prompt you to select the source object or multiple objects that you want to join. In this case, select two lines, as the ones labeled **A** in figure 3.80, and press the *Enter* key.

3. You will notice that not only do the lines join to fill the gap, but they also merge to become a single line, too.
4. Now, we will merge the gap in the arcs labeled **B** in figure 3.80. Once again, select the join tool and then click on the arc on the left and then the arc on the right and press *Enter*.
5. You will notice that now the arc gap is merged in an anticlockwise direction, starting from the arc selected first.

In this case, the gap at the bottom will be merged in an anticlockwise direction and moves from the arc to the left to the arc to the right. If, however, you select the arc to the right first and then the arc to the left next, then the gap at the top will be merged:

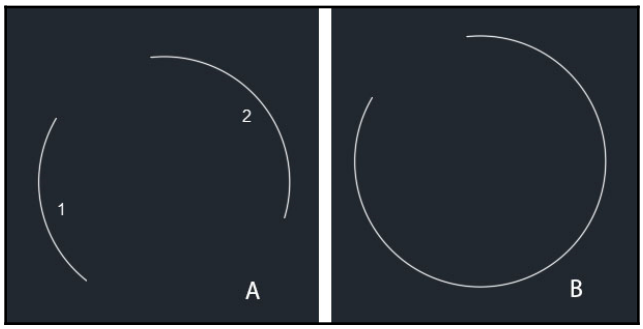


Figure 3.82: The join command merging two arcs

The **C** shape in figure 3.80 is made of three connected curves—two lines and one arc—which are connected from end to end. If you use the join command on these three curves, you will end up with a single polyline. So, you can use the join tool to connect the curves into a polyline or spline.

The Explode command

The Explode command is, in a way, the opposite of the join command and using this command, you can convert a polyline into separate lines and arcs. To explain this command, I will use this drawing made with the Polyline command:

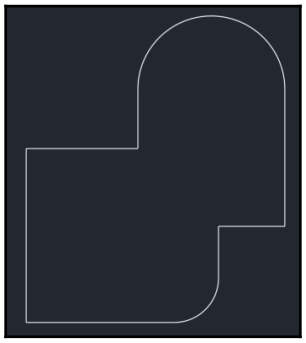


Figure 3.83: A sample drawing made with a polyline for the explode command

If you select any one of the segments of this drawing, the complete drawing will be selected because it is made with a polyline. Now, to break this drawing into separate lines and arcs, we will use the explode command:

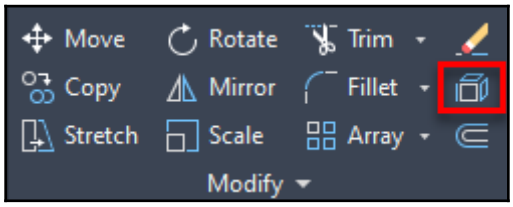


Figure 3.84: The explode command in the Modify panel

Start the explode command from the **Modify** panel in the **Home** tab or use its command alias, `X`. Now, click on the polyline and press *Enter*.

The drawing is now exploded and if you select it now, all its lines and arcs will be selected separately:

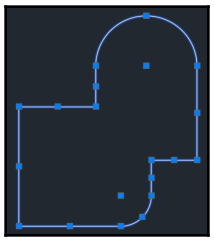


Figure 3.85: A drawing after it's exploded. All the lines and arcs are separately selectable here

Although in this case we used the explode command just to explode a polyline, it can be used in a variety of ways. For example, you can explode a hatch pattern and then all the lines of the hatch pattern will become separately selectable. Needless to say, the hatch will no longer work like a hatch pattern and will be converted to a simple object too. You can also explode other objects in your drawing, such as blocks, which we will discuss in `chapter 5`, *Managing Drawings with Layers and Properties*, and exploding them will convert them back to simple draw objects.

The Stretch command

I saved this command for last as it is a little different from all the commands we have discussed so far. Using this command, you can stretch the drawing in any direction without changing the rest of the drawing. You can start this command from the **Modify** panel of the **Home** tab or you can also use its command alias, `STRETCH`:

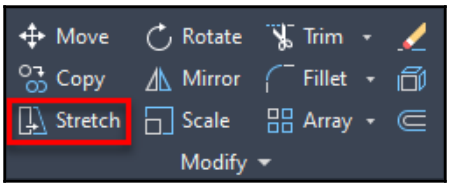


Figure 3.86: The stretch command in the Modify panel

I will explain this command using the drawing shown in the following figure. This drawing is made with a simple line and arc tool, but you can also make it with a polyline and it won't affect the workflow in any way:

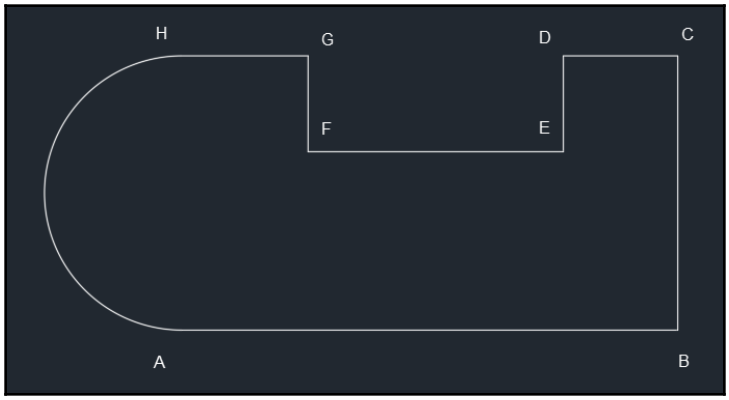


Figure 3.87: A sample drawing to be used for the stretch command

Now, let's consider a scenario where we want to increase the gap made by points **D**, **E**, **F**, and **G**. To make this kind of very specific change, you can use the stretch command, as in the following example:

1. Start the **Stretch** command and now the command line will prompt you to select objects.
2. In this case, you don't simply select the objects. Rather, you need to do it in a very specific way. You can see a specific instruction about selecting the object by expanding the command line.
3. When you expand it, you will notice that the command line prompts you to select the object using the crossing window or crossing polygon. So, in this case, we will select the object using the crossing window:

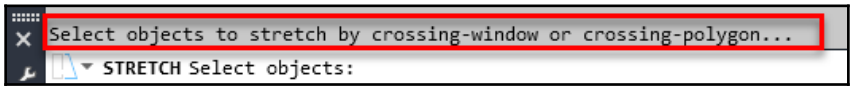


Figure 3.88: The selection option highlighted in the stretch command

4. To make the crossing window, click above the **C** point and let go of your cursor. Then, move your mouse to the left so that you get a green window with dotted boundaries, which is the crossing window, and make the selection as shown:

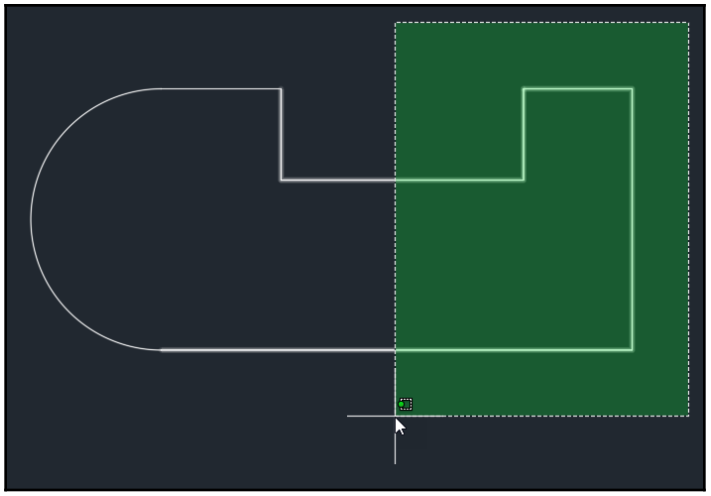


Figure 3.89: The crossing window made to stretch the object so that only the gap in the center is effected

5. As you can see, we are not selecting the entire object. Rather, we are only making the crossing window up to the gap that we want to stretch. Once you have this kind of window, click again to make the selection and then press the *Enter* key.
6. Now, the command line will prompt you to specify a base point. Click on **B** and now move your cursor to the right. You will notice that now the gap will increase along with your cursor.
7. Click again on another point or type the gap length up to which you want to increase the gap and you will have a new drawing where only the specified gap is changed, as shown:

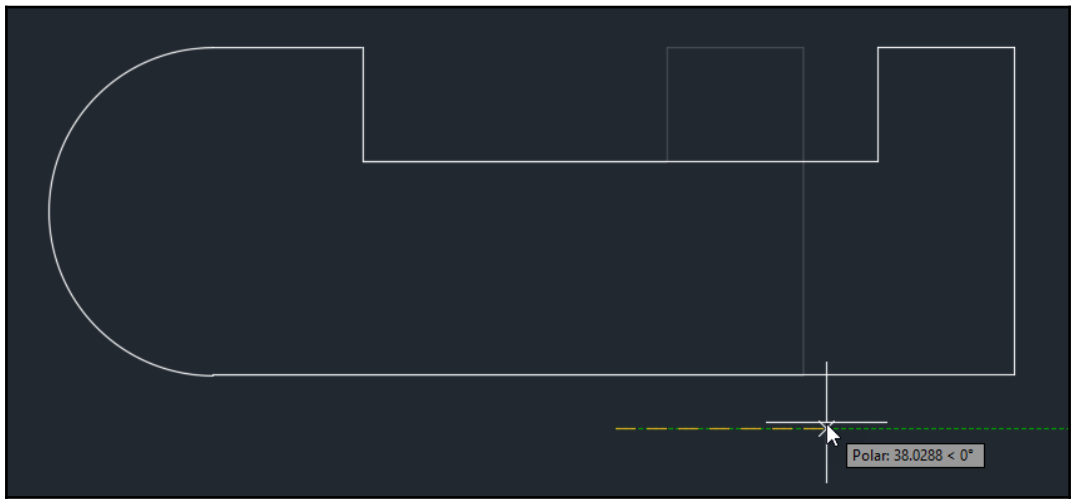


Figure 3.90: Object stretching increasing the centre gap of the drawing

In a similar way, if you want to stretch the drawing so that the part belonging to the **DC** line changes in length, then your selection window and result will look like this:

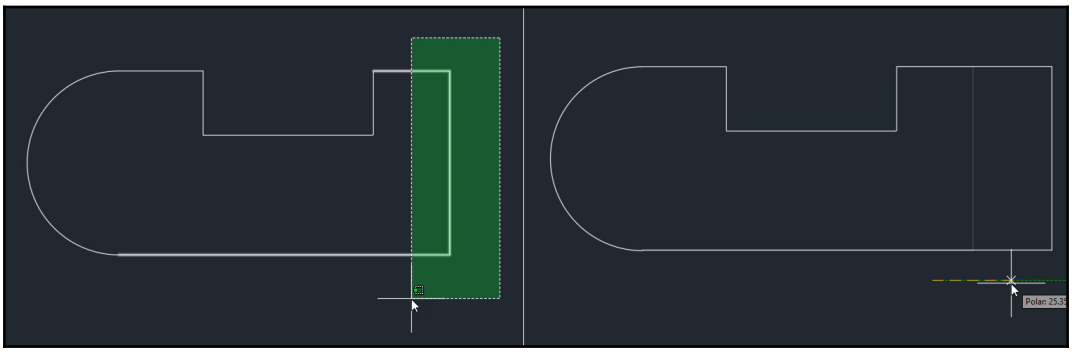


Figure3.91: A crossing window made in the drawing and the drawing stretched as a result

The stretch command has an unusual kind of workflow but it is very useful and used quite often. In the previous examples, we used the stretch command only on straight lines, but you can use it on arcs and ellipses, too.

Summary

With this chapter, you are now equipped with all the draw and Modify commands and other toolsets needed to make almost any kind of drawing. In this chapter, we learned how to make drawings with precision using Object Snaps. We also learned how to make modifications in our drawings using modify tools such as scale, offset, and mirror. We also learned how to add markups using revision cloud and finally, we learned how to make advanced modifications in our drawing using tools such as stretch.

In the next chapter, we will learn how to make a repeated set of objects using an array tool and we will also learn how to make the reusable symbols in AutoCAD popularly known as blocks. We will also learn how to make advanced types of blocks with attributes that enhance the overall functionality of the blocks.

4
Working with Arrays and Reusable Objects

In most of your drawings, you will find the same set of objects repeated over and over again in patterns. You will also notice that in many drawings, some of the objects can be reused, such as furniture, doors, or windows. Typically, we convert these objects into named objects in AutoCAD called blocks, and then we use them wherever required. This chapter is all about making these patterns and reusable objects. We will begin this chapter with some advanced status bar modes, such as grid and snap, and then we will move on to learning about Arrays, as well as making and modifying blocks. Later in this chapter, we will also learn how to create groups and blocks containing attributes.

Here is what we will cover in this chapter:

- Advanced status bar modes
- Making Arrays
- Working with blocks
- Learning how to use groups
- Working with attributes

So, let's begin by looking at grid mode in the status bar.

Using advanced status bar modes

In this section, we will learn about the grid, snap, and selection cycling status bar modes. The first two modes help us make drawings using a background grid as a reference, whereas the last mode, selection cycling, helps us make a selection of overlapping objects in an existing drawing. We will begin this topic by looking at the grid mode status bar option.

Grid mode

In the status bar, you will notice that the first icon looks like a grid, as in the following screenshot:



Figure 4.1: The grid status bar toggle

This option toggles the visibility of the grid lines in the drawing area. You can also use the *F7* function key to toggle its visibility.

The grid lines look like lines on a graph paper and they merely represent lines and you can't actually "snap" to them. The grid lines are made with two types of lines: major lines and minor lines. Major lines are the bold lines that make up big blocks and minor lines are the lighter ones that further divide these major blocks into smaller segments, as shown:

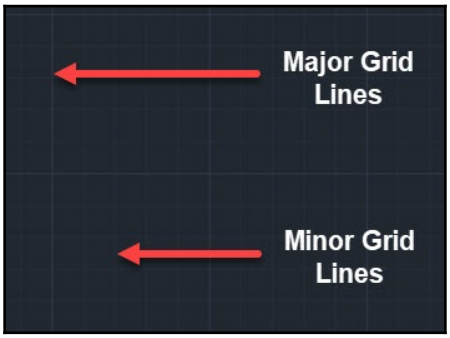


Figure 4.2: Major and minor grid lines in the drawing area

In the following example, we will learn how to modify the spacing of the default grid:

1. To adjust the spacing between these grid lines, right-click on the grid mode icon in the status bar and click on **Grid Settings**.
2. The **Drafting Settings** window will open up with **Snap and Grid** as the active tab. In this tab, you will find all the grid-related settings in the panel to the right, as shown:

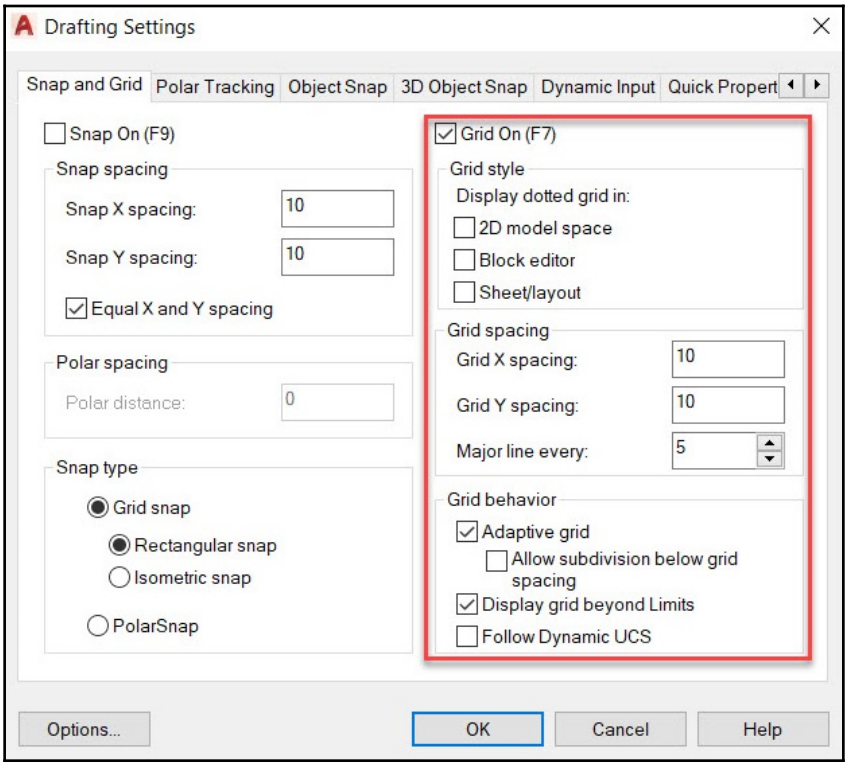


Figure 4.3: The grid settings panel in the Drafting Settings window

3. To change the spacing of the grid lines, go to the **Grid spacing** panel and change the **Grid X spacing:** and **Grid Y spacing:** values to any other value. In this case, I will change the *X* and *Y* spacing values to `5` units.
4. The next option, **Major line every:**, will control the number of smaller blocks that a major line will be added after. I will change this value to `10` units.
5. After making these changes, click on **OK** to close the **Drafting Settings** window to apply the changes. The grid with its updated parameters will look as in the following screenshot:

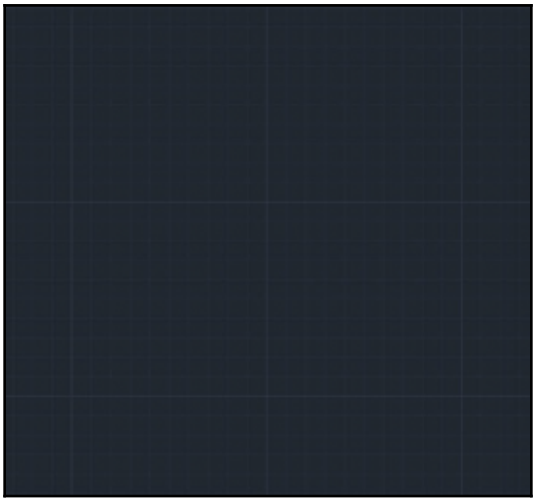


Figure 4.4: A grid with updated parameters

6. Here, you can clearly see that the major lines are now formed after every 10 grid blocks in both the *X* and *Y* directions and the distance between the major lines in both the *X* and *Y* directions is `5` units.
7. I will once again go to the **Grid spacing** option in the **Drafting Settings** window and I will change the *X* and *Y* spacing to `1` unit and **Major line every:** to `10`, as shown:

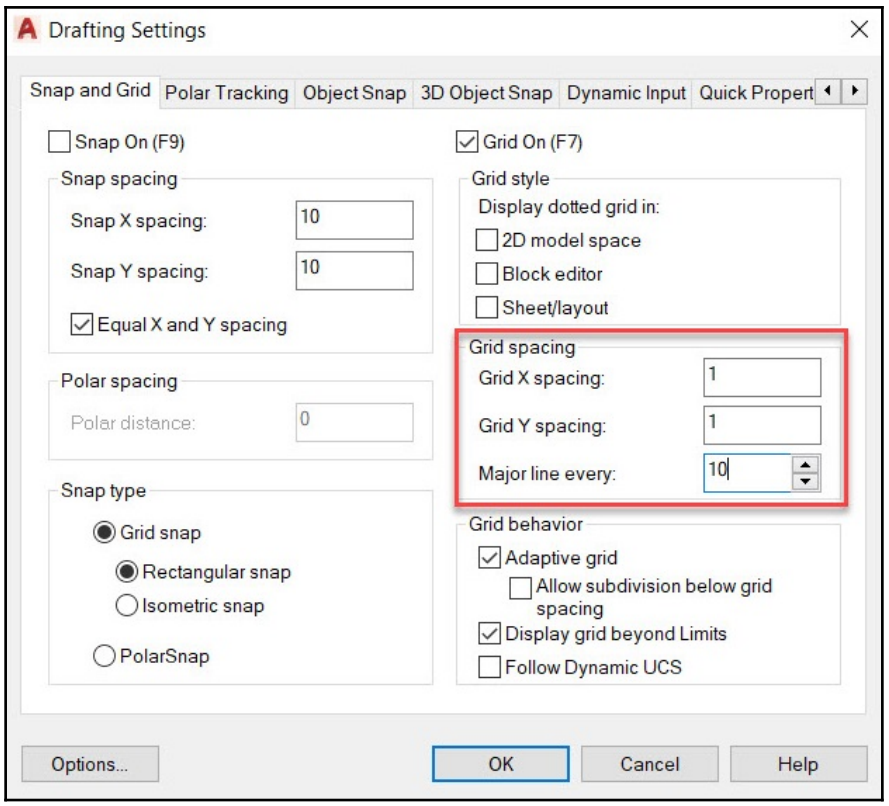


Figure 4.5: The grid spacing options in the Drafting Settings window

8. Click **OK** to close the **Drafting Settings** window and save these changes. Now, the grid lines are spaced as follows:

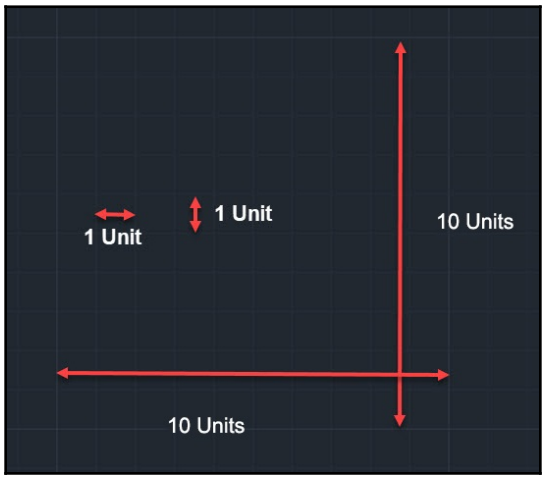


Figure 4.6: The grid line spacings shown with red arrows

The grid lines are dynamic and can be changed as per the scale of the drawing. In this case, apart from an approximate visual reference, the grid line has nothing much to offer; but, when used in conjunction with snap mode, this becomes a really useful tool for making drawings. We will discuss the snap mode feature later in this chapter but before we get to that, let's learn more about grid spacing and the way the grid looks in the next section.

What if the grid spacing does not match the settings?

If after changing these parameters you are still not getting grids with the proper spacing, then try zooming in until you no longer see any more grid lines and you will see grids of the proper size.

There may be situations where your grids look like dotted points, as in the following screenshot, and not like graph lines:

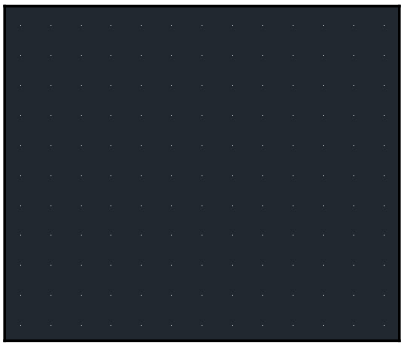


Figure 4.7: A grid that looks like dotted points

If that's the case, then go to **Drafting Settings** and uncheck all the options in the **Display dotted grid in:** section in the **Grid style** panel, as shown:

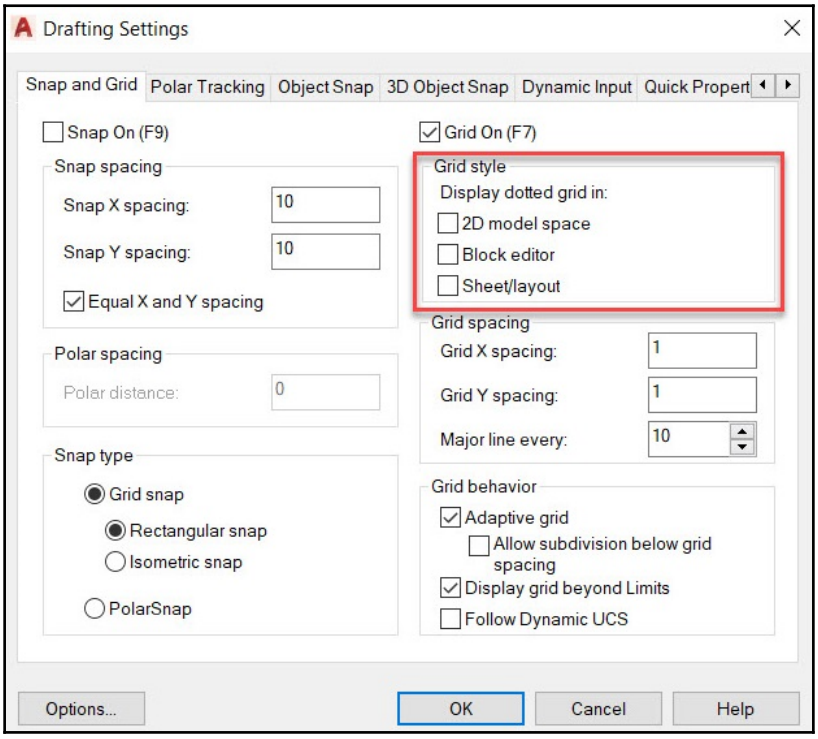


Figure 4.8: The dotted grid options in the Snap and Grid tab

Once you have unchecked all the dotted grid checkboxes, the grid will once again return back to the normal line type view, with horizontal and vertical grid lines.

The next option that we will now discuss is snap mode, which—if used in conjunction with grid mode—can help make drawing quicker by using the grids as references.

Snap mode

The snap mode option, which is next to the grid mode option in the status bar, lets you actually snap to the grid lines and to other points in the drawing area that are placed at equal distances. You can toggle snap mode using its icon in the status bar or by using its function key, *F9*:



Figure 4.9: The snap mode toggle in the status bar

In the following example, we will learn how to set the snap spacing equal to the grid line spacing so that our cursor snaps exactly to the grid intersection points:

1. To change the snap settings, right-click on the snap icon and select **Snap settings** from the menu. The **Drafting Settings** window will open up with **Snap and Grid** as the active tab.
2. Change the value of **Snap X spacing:** and **Snap Y spacing:** to `1` each. In the previous example, we set the grid spacing to `1` unit, so here we will also set the snap spacing to `1` unit to keep the grid spacing equal to the snap spacing.
3. Once you have made the changes shown in the following screenshot, click **OK** to apply these changes and exit the command:

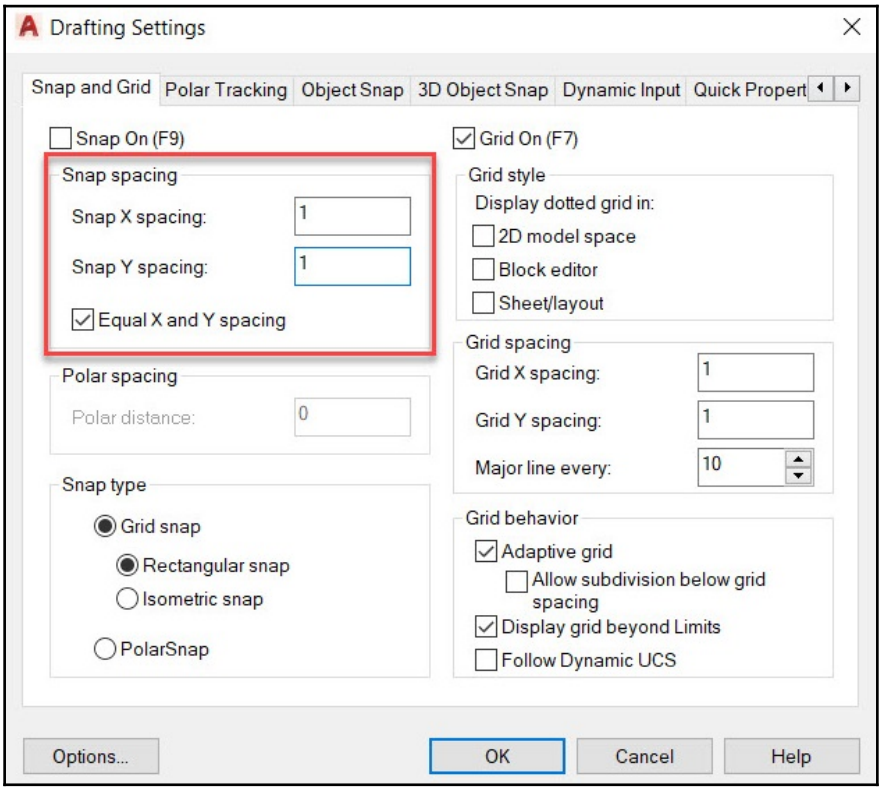


Figure 4.10: The snap spacing fields in the Snap and Grid tab

4. Now, make sure the grid mode and snap mode options are active in the status bar and then select a draw command.

5. Move your cursor in the drawing area and this time, you will notice that the cursor will only snap to the intersecting points of the grid:

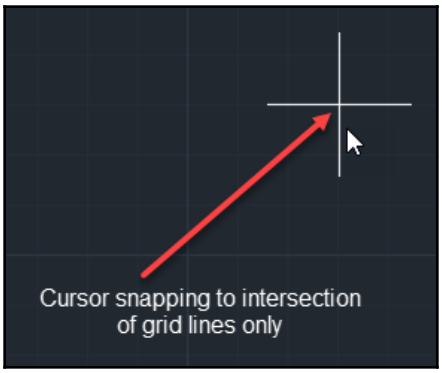


Figure 4.11: When snap mode is active, the cursor will only snap to defined snap points

You can change the snap distance to some other value and then the cursor will snap at that interval instead of the intersection of grid lines. The snap and grid modes are helpful in situations where you want to take the reference of these grid lines to make drawings.

As you might have noticed in this case, you can count the number of grid lines to work out the distance in both the *X* and *Y* directions. For example, if you want to make a rectangle with a length of `10` units and a width of `5` units, then all you need to do is just count **10 blocks** along *X* to make one line and **5 blocks** along *Y* to make another line for the rectangle:

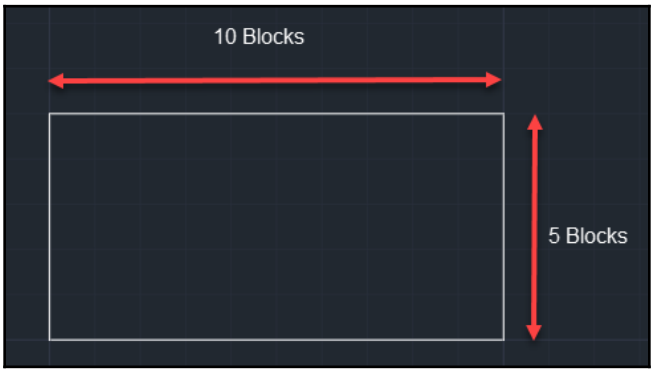


Figure 4.12: A rectangle made with the snap and grid mode toggles

Once you are done with snap mode, make sure you deactivate it in the status bar, or else the cursor will keep snapping only to the grid intersection points and won't let you make accurate drawings. To deactivate snap mode, click on the snap mode status bar option in the status bar or press the *F9* function key.

Next up in this series of status bar options is the selection cycling option, which lets you cycle a selection through overlapping objects. The selection cycling toggle makes it easier to select required objects, even when they are not directly visible. We will discuss this status bar toggle in detail in the next section.

Selection cycling

Selection cycling helps us make selections of overlapping objects or objects that are too close to make an accurate selection in AutoCAD. To use this tool, you need to activate it first from the status bar. Click on the two overlapping squares, shown in the following screenshot:



Figure 4.13: The selection cycling toggle in the status bar

In the following example, I will show you how selection cycling can help make selections when different objects overlap:

1. If you don't see the selection cycling icon in the status bar, click on the three dash lines, also called the customization option, at the end of the status bar and select **Selection Cycling** from the list of available status bar tools, shown in the following screenshot.

2. Once selected, the selection cycling toggle option will show up on the status bar and then you can click on it to activate it:

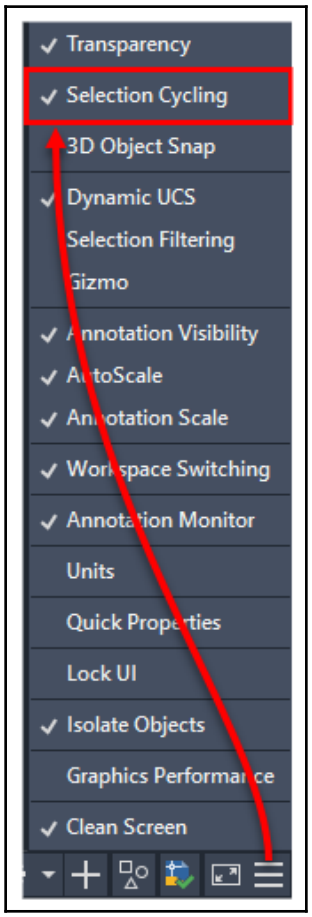


Figure 4.14: The Selection Cycling option in the customization menu

3. For this example, I will use the following drawing. In this drawing, the text is overlapping hatches and other patterns, which makes it difficult to select:

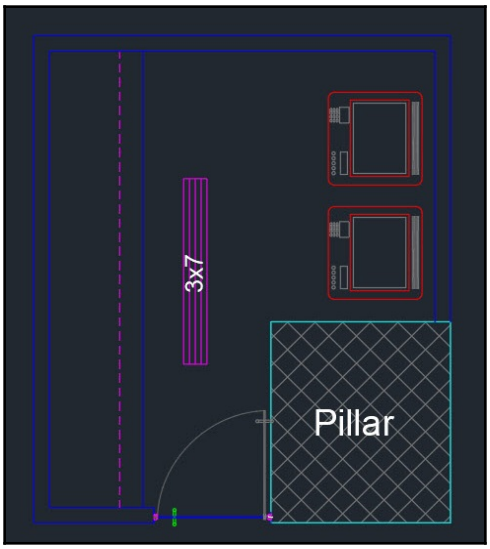


Figure 4.15: Text overlapping hatches in the sample drawing

4. Now, hover your cursor over the **Pillar** text and you will see the selection cycling icon, which looks like two overlapping squares:



Figure 4.16: The selection cycling icon highlighting multiple overlapping objects

5. Click to select the text and you will see a menu with a list of the overlapping objects. In this case, the list contains **Hatch** and **MText**:

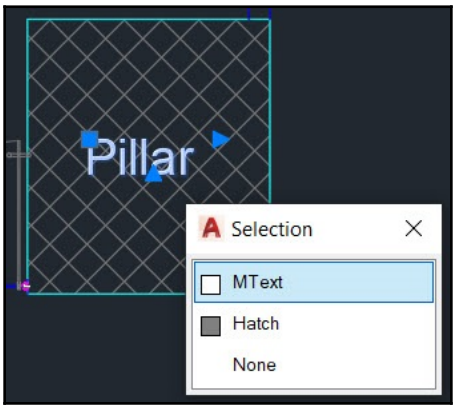


Figure 4.17: The selection cycling menu with a list of the overlapping objects

6. Click the object from the list that you want to select and it will be selected from the drawing area.

As you can see, this tool is really helpful for these kinds of situations, but it is not recommended to keep it on all the time as it may add a few extra steps to your workflow. So, try to keep selection cycling off and only activate it when needed.

With the selection cycling option covered, we have now finished learning about the required status bar options and now we will move on to learn more about "modify" tools. The next tool is **Array**, which lets us make patterns along different paths, such as rectangular, circular, or any other selected path.

Making Arrays

The Array command helps us make patterns along a path. In AutoCAD, you can make an Array along a circular path (called a Polar Array), a rectangular path (called a Rectangular Array), and along any open path (called a Path Array).

We will start the next section with the Rectangular Array and then move on to the other Array tools.

The Rectangular Array

Here, we have a drawing of a window on a simple wall and we want to make two rows and five columns of windows on this wall, separated at an equal distance. For this, we can use the Rectangular Array:

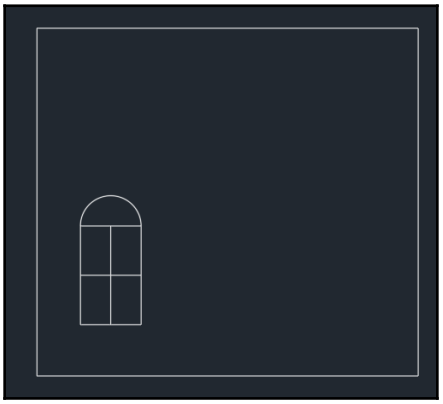


Figure 4.18: A sample drawing to use for the Rectangular Array

In the following example, we will learn how to make multiple copies of the window using the Rectangular Array command:

1. Click on the arrow next to the **Array** icon in the **Modify** panel of the **Home** tab and select **Rectangular Array** from the expanded flyout:

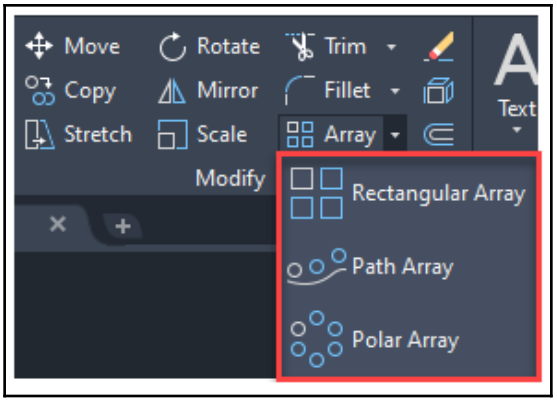


Figure 4.19: All the Array options in the expanded Array flyout of the Modify panel

2. Now, select the complete window inside the rectangular wall and press the *Enter* key. An **Array Creation** tab will show up and an Array with three rows and four columns will show up:

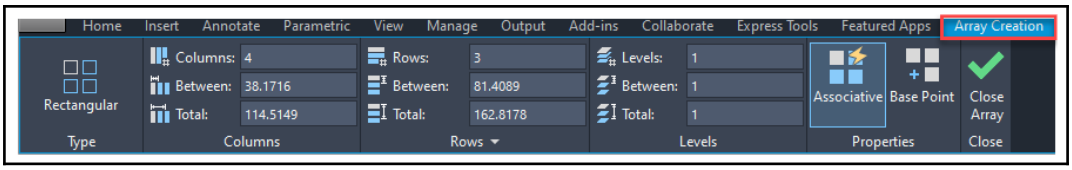


Figure 4.20: The Array Creation tab for the Rectangular Array command

3. You can further modify this Array using the blue grips on the Array itself or you can also use the **Array Creation** tab. Let's start with the grips:

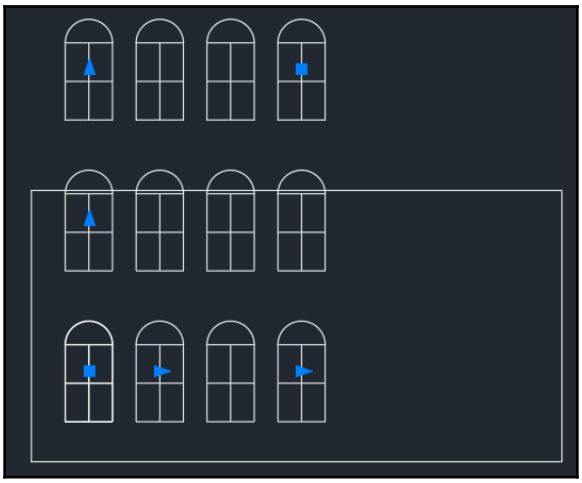


Figure 4.21: The default Array with three rows and four columns, made with the Rectangular Array command

4. Click on the arrow grip on the bottom-right of the Array and move it to the right. You will notice that more columns will be added and moving it to the left will decrease the number of columns. Similarly, if you select the arrow grip along the bottom row toward the left, you will be able to increase or decrease the gap of the Array. The arrow grips in the vertical direction can be used to increase the number of rows and the gap. In the following screenshot, I have labeled the arrow grips with their specific functions:

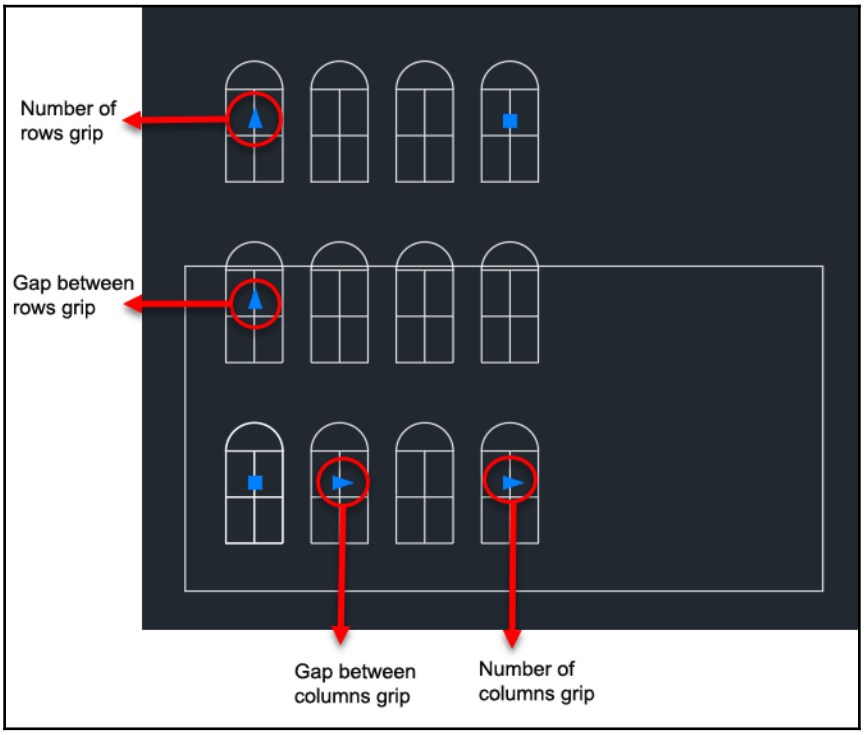


Figure 4.22: Different grips and their function on the Array

As you have noticed, this method of dynamically moving the grips is not very precise when it comes to the distance between row and column objects. So, to make Arrays with more precise dimensions, you can use the **Array Creation** tab, which will be explained here:

1. In the **Array Creation** tab, you will see the **Columns**, **Rows**, and **Levels** panels. You can change the parameters related to columns, rows, and levels in the respective panels. Start with the **Columns** panel. Here, we want to add four columns, so in the **Columns** field, I will type `5`. Press the *Enter* key after entering the value and the number of columns will change to five.
2. Go to the **Between** field and type the gap that you want between objects of the column. In this case, I used a gap of `56` units. After adding the gap, press *Enter* again and the Array will update with the new gap value.

3. You will also notice that now, the **Total** field in the **Columns** panel will show the total gap between the first and last column of the Array and this value will update automatically. If you change the total gap, then the **Between** value will update accordingly:

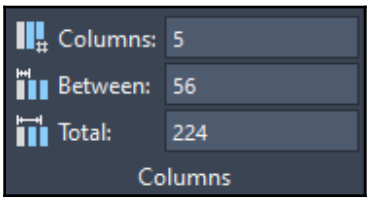


Figure 4.23: The Columns panel of the Array Creation tab

4. In a similar way, you can change the number of rows and the gap between rows in the **Rows** panel.
5. The **Levels** panel will change the number of Array objects along the *Z* axis, but since we are working only along the *X* and *Y* axis, the changes made along the *Z* axis won't show up as it will simply overlap over existing objects.

Here are the values that I used for my Array:

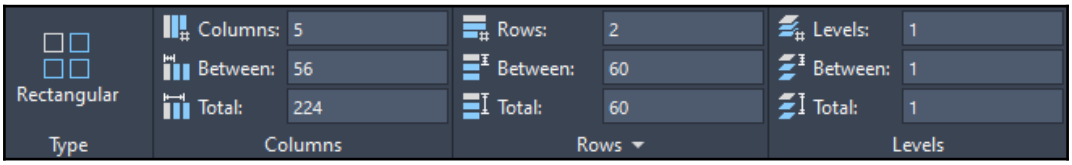


Figure 4.24: The final set of values used for the sample Array

Once you are finished making an Array, click on the **Close Array** green checkmark at the end of the **Array Creation** tab. In the next example, we will discuss the **Associative** option in the **Array Creation** tab, which has a special significance when it comes to making Arrays.

The Associative option

In the **Array Creation** tab, you will see the **Associative** option in the **Properties** panel. This option has a special significance to the Array tool:

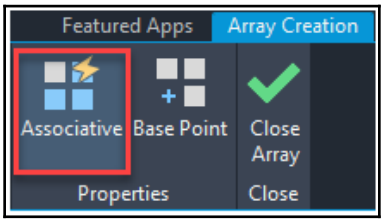


Figure 4.25: The Associative option in the Array Creation tab

While making an Array, if you keep the **Associative** option checked, you will get an associative Array, which works as a composite object, and selecting any one object in the Array will select the whole Array, as shown:

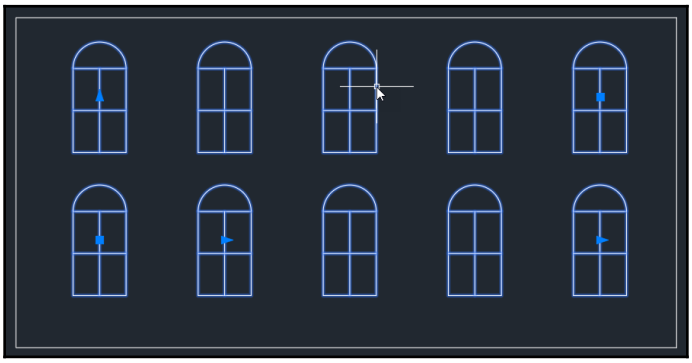


Figure 4.26: All the objects in the Array are selected as the Array is associative

However, if you uncheck this option at the time of making the Array, then all of the objects in the Array will be independent of each other. So, in a non-associative Array, if you select any object from the Array, then only that object will be selected.

An associative Array also lets you modify the Array even after you have finished making it. If you select an Array that is associative, you will get a new **Array** tab, which can be used to again modify the existing Array. The same thing will not happen with a non-associative Array:

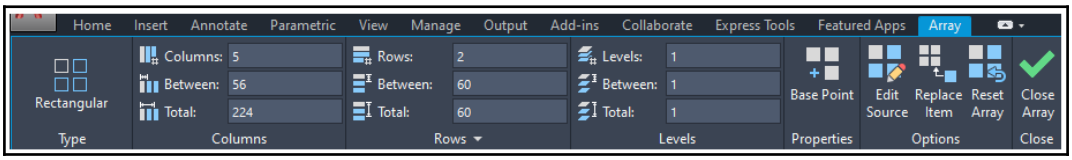


Figure 4.27: The Array tab, which shows up when an associative Array is selected

If you have an associative Array—where selecting any object selects all the objects of the Array—and you want to convert this type of Array into a simple Array where all objects are separate, then you can do so using the explode tool. Select the associative Array from your drawing area then type *X* and press *Enter*. Here, `X` is the command alias of the explode command. You can also use the explode command from the **Modify** panel of the **Home** tab.

Now, the Array that you will get will have all objects independent of the Array. This type of Array will also lose its ability to be modified.

The Polar Array tool

Using the Polar Array tool, you can make an Array along a circular path. To explain the Polar Array tool, I will use this drawing:

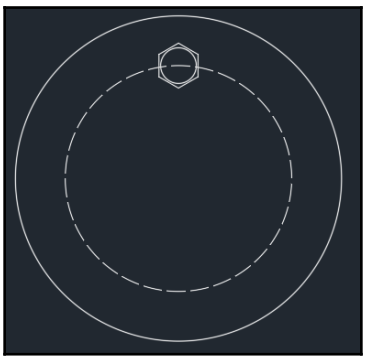


Figure 4.28: A sample drawing used for the Polar Array tool

In this case, we need to make five copies of the hex bolt along the circular path, shown by the dashed circle in the previous figure. For this, you need to start with the Polar Array tool, as instructed here:

1. Select the **Polar Array** tool from the **Array** flyout in the **Modify** panel of the **Home** tab.
2. Select the hex bolt from the drawing area and press the *Enter* key.
3. Now, the command line will prompt you to specify the center of the Array; in this case, the center of the Array is the center of the dashed circle. Click on the center of the dashed circle to specify the center of the Array.

4. Now, you can change the Array parameters using the blue grips on the Array or you can change it from the **Array Creation** tab as well. Click the blue arrow grip that shows up on the circular Array and move it along the circular path. You will notice that two grips will now show up, as shown:

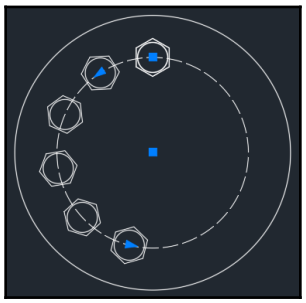


Figure 4.29: The two arrow type grips on the Polar Array

5. If you click the arrow type grip close to the first hex bolt on the top of the circle, it will let you change the gap between the Array objects. The last arrow grip will change the overall distance that the Array is applied to along the dashed circle.

You can also change distances and the number of objects in the Array, along with other parameters, from the **Array Creation** tab, which we will do next:

1. The first panel in the **Array Creation** tab is **Items**, where you can type in the number of objects that you want in the Array. Type in the total number of Array copies you want in the **Items** field. The default value in this field is `6`, but I will change it to `5` for this example.
2. The next field, **Between**, is the angle between objects in the Polar Array. The **Fill** angle is the total angle along which the Array is applied. I will change the **Fill** value to `360` degrees and the **Between** angle will change automatically to `72`, which is the angle between corresponding objects in the Array:

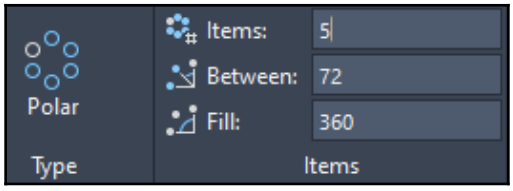


Figure 4.30: The Items, Between, and Fill fields in the Items panel of the Array Creation tab

3. The next panel is for rows but adding rows here won't be practical as we only want a single row; but to show you the effect of the **Rows** field, I will type `3` in the **Rows** field and set the distance between rows to `50`. The resulting Array will look like this:

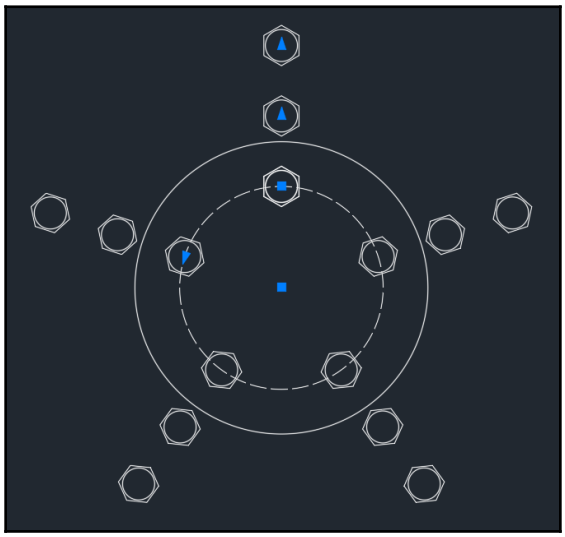


Figure 4.31: A Polar Array with three rows spaced at 50 unit gaps

4. As you can see in this case, the Array contains three rows of hex bolts with a gap of 50 units between the rows. So, once again, I will change the number of rows back to `1` and now the Array will look normal again.
5. The next panel, **Levels**, will add Array objects along the *Z* axis and as we are working on the *X* and *Y* axes only, changing its parameters won't make any visible difference to our drawing.
6. In the **Properties** panel, you will once again notice the **Associative** option, which lets you make Arrays that can be edited later. In the **Properties** panel, there is also a new option, Rotate Items, which we will discuss here.
7. When you uncheck this option, the Arrayed copies don't rotate, but when you keep this option checked, the Arrayed items rotate along the circular path, as shown:

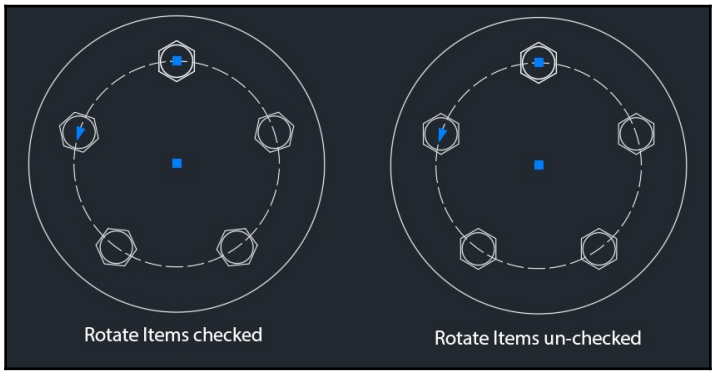


Figure 4.32: A Polar Array created with Rotate Items checked on the left and unchecked on the right

8. You will notice—in the first Array in the previous figure—that the hex bolt is rotating with respect to the dashed circle but in the second Array, the hex bolt is always in an upright position or it won't rotate at all. It also takes the same orientation as the original object in all of its Array copies.
9. Once again, when you have finished making the Array, click on the **Close Array** green checkmark and the Array will be made along the dashed circular path.

This was the Polar Array and, as the preceding examples show, it is great for situations where you want to make multiple copies of an object, such as a blot or nut along a circular plate. The next Array tool is the Path Array, which allows us to make Array copies along any selected path.

The Path Array

Using the Path Array, you can make a pattern of any object along your selected path. In this case, I have part of a road made with a spline, with a divider spline in the middle. Here, we need to make a pattern of solid rectangles along the mid-spline of the road:

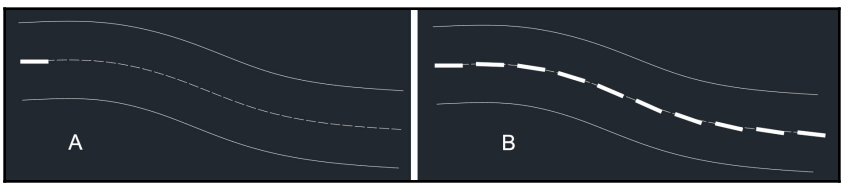


Figure 4.33: A sample drawing, A, for the Path Array and the end result of the Path Array in drawing B

In the following example, we will use the Path Array to make patterns of the first rectangle shown in drawing **A** along the dotted spline so that we end up with an Array as in drawing **B**:

1. Select the Path Array tool from the **Array** flyout in the **Modify** panel of the **Home** tab, or use its command alias, `ArrayPATH`.
2. The command line will prompt you to select the objects. Click on the rectangle with the solid hatch at the beginning of the dotted spline in the center of the road and then press the *Enter* key.
3. Now, the command line will prompt you to select the path of the Array. Click on the dotted spline and you will get an Array along the spline path that looks like this:

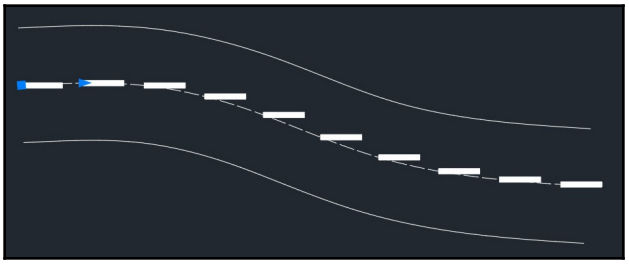


Figure 4.34: The default Array made with the Path Array tool

4. In this case, we will change a few parameters to make the Array look the way we want. For this, we will start with the number of objects. To do this, go to the **Items** panel in the **Array Creation** tab.
5. You will notice that the **Items** field is grayed out and so you won't be able to change the number of objects in it, but you will be able to modify the **Between** value—which is the gap between objects—which will eventually change the number of objects:

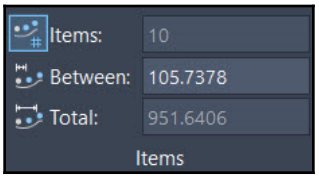


Figure 4.35: The Between option is the only one active in the Items panel of the Array Creation tab

6. Increase the gap between objects to decrease the number of objects and decrease the gap to further increase the number of objects.
7. If, however, you want to change the number of objects and not the gap, then go to the **Properties** panel of the **Array Creation** tab and change the **Measure** option to **Divide**, as in the following screenshot. Now, you will be able to change the number of objects in the **Items** field of the **Items** panel:

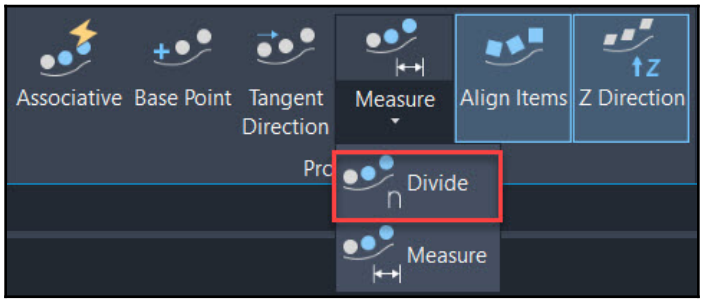


Figure 4.36: Changing the Measure type to Divide in the Properties panel of the Array Creation tab

8. In this case, I will change the number of objects to `10` and then the gap will change automatically in the **Between** field.
9. In the next panel, **Rows**, you can change the number of rows of this Path Array and also the gap between these rows.
10. The next panel, **Levels**, will let you change the number of levels along the *Z* axis. Changing any parameter in this panel will not affect the drawing in a visible way as the objects will overlap each other, which won't show up in the two-dimensional plane.
11. Now, let's change some options in the **Properties** panel of the **Array Creation** tab. The first one, **Associative**, will make an associative Array that can be modified even after finishing the Array command. You can keep this option checked if you want to make an associative Array or uncheck it if you don't.

12. Another option that is worth mentioning here is **Align Items**. When the **Align Items** option is checked, the Array objects will align with respect to the Array path and if you uncheck Align Items, then all the Array copies will look exactly like the original object and their alignment will not change with respect to the path. In the following screenshot, the Path Array is made with the **Align Items** option unchecked:

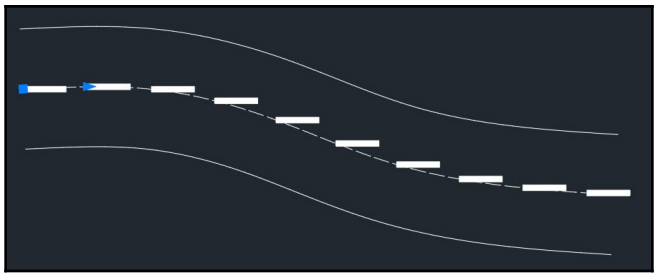


Figure 4.37: A Path Array created with the Align Items option unchecked

13. In this case, I will keep **Align Items** checked to keep the Array objects aligned with the Array path. Once you are finished making the Array, click on **Close Array** and you will have your Path Array as per the requirements.

Here is another example where the Path Array can be used to make the hex bolts along the dotted polyline of the plate:

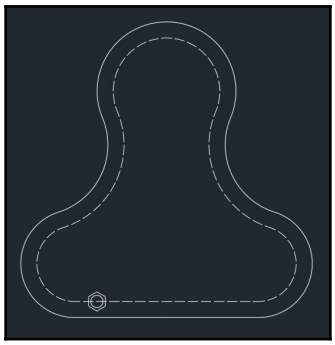


Figure 4.38: A sample drawing for the Path Array

To make an Array along this dotted polyline path, we will have to do the following:

1. Once again, start with the Path Array command from the **Modify** panel of the **Home** tab or use its command alias, `ArrayPATH`.
2. Now, select the hex bolt, which is represented by a hexagon and circle at the bottom-left of the drawing. After selecting it, press the *Enter* key.
3. Now, for the path, click on the dotted polyline shape and you will get a random Path Array, which may look completely out of place, depending on the settings you have selected in the **Array Creation** tab. In my case, the Array looked completely out of place, as shown:

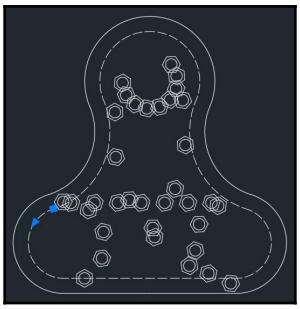


Figure 4.39: A Path Array created along the dotted path of the drawing

4. To fix this Array and to make it look normal along the path, I will change some options in the **Properties** panel. The first option I will change here is the **Align Items** option. I will uncheck it and now the Array looks manageable, but it is still not along the intended path:

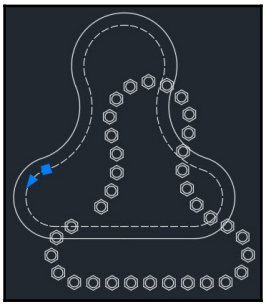


Figure 4.40: An Array with the Align Items option unchecked

5. In this case, to move the Array along the path, we need to change its base point. The point, in this case, is shown by the blue square grip, as in the previous figure. This grip should be at the center of the circle, as in the following figure:

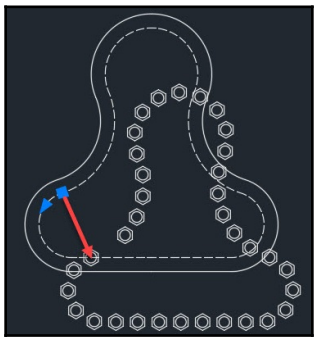


Figure 4.41: The original base point of the Array, shown by a blue grip, and the new point where it should be moved

6. So, to move the base point, I will select the **Base Point** option from the **Properties** panel and click on the center of the circle, shown in the preceding figure.
7. Now, the Array will move exactly along the path and it will look the way a Path Array should look.
8. In this case, you can change the number of objects from the **Items** field of the **Items** panel in the **Array Creation** tab. I will change the number of items to `18`. The final Array will look like this:

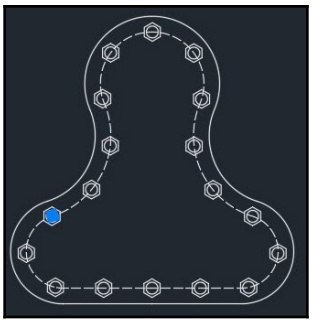


Figure 4.42: A Path Array with a new base point

So, as you have noticed, we can make very interesting patterns using different Array tools and this tool helps in eliminating the repetitive copy-pasting of an object about a pattern. The next tool that we will discuss is equally interesting and also helps eliminate repetitive tasks. This tool is **blocks** and by using it, you can add symbols to drawings and use the same symbol in multiple drawings, too.

Working with blocks

Blocks are reusable content that can be used again and again in a drawing (and in other drawings). You can make these blocks and then save them for use in other drawings. An example of a block is shown here:

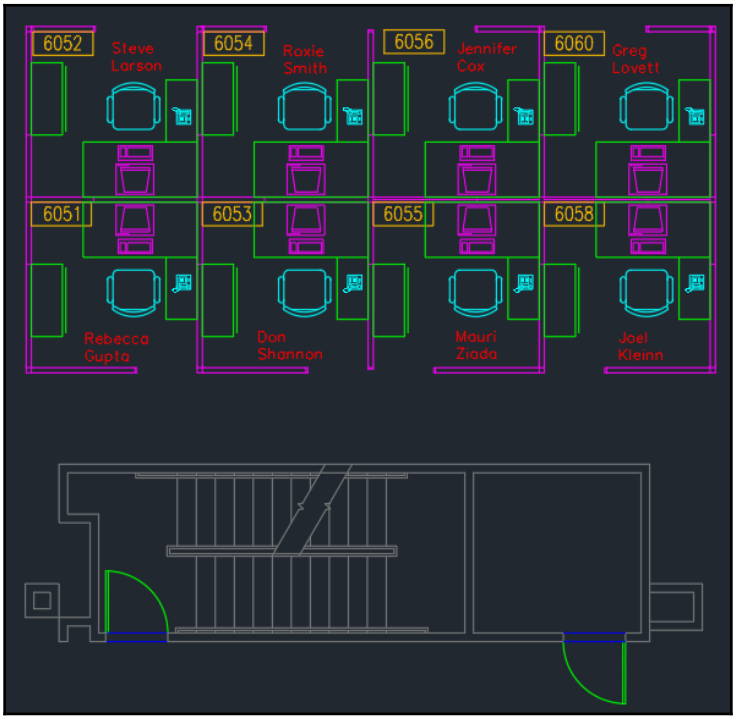


Figure 4.43: A sample drawing with blocks in it

In this drawing, the chair, doors, desk, computer, and telephone are blocks that have been made once and used repeatedly wherever needed. Though there are many advantages to using a block over simply copy-pasting the original symbol, the biggest advantage is the ability to modify all references of the block by modifying only one block. In the preceding drawing, if you happen to modify the chair, all the instances where the same chair block is used will modify automatically to reflect the changes made by you in the first block. In this section, we will start by making a simple door block and then move on to use and modify it.

Making blocks

Make a simple door symbol using the rectangle and arc commands, as shown:

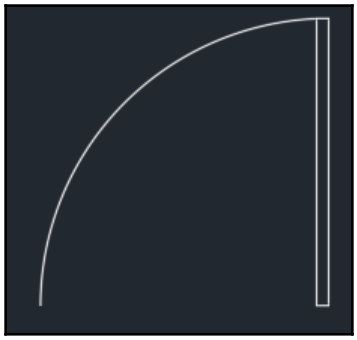


Figure 4.44: A simple door symbol made with the rectangle and arc commands

In the following example, we will learn how to convert this door symbol into a block:

1. Click on the **Create** option from the **Block** panel of the **Home** tab, shown in the following figure, or use the command alias, `B`:

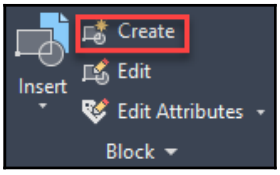


Figure 4.45: The Create option in the Block panel of the Home tab

2. The **Block Definition** window will pop up. Give this block a name in the **Name** field. I am naming it `Sample Door`.
3. In the **Base point** panel, click on the box that says **Pick point**. You will notice that the cursor will change to pick a point and the **Block Definition** window will also disappear:

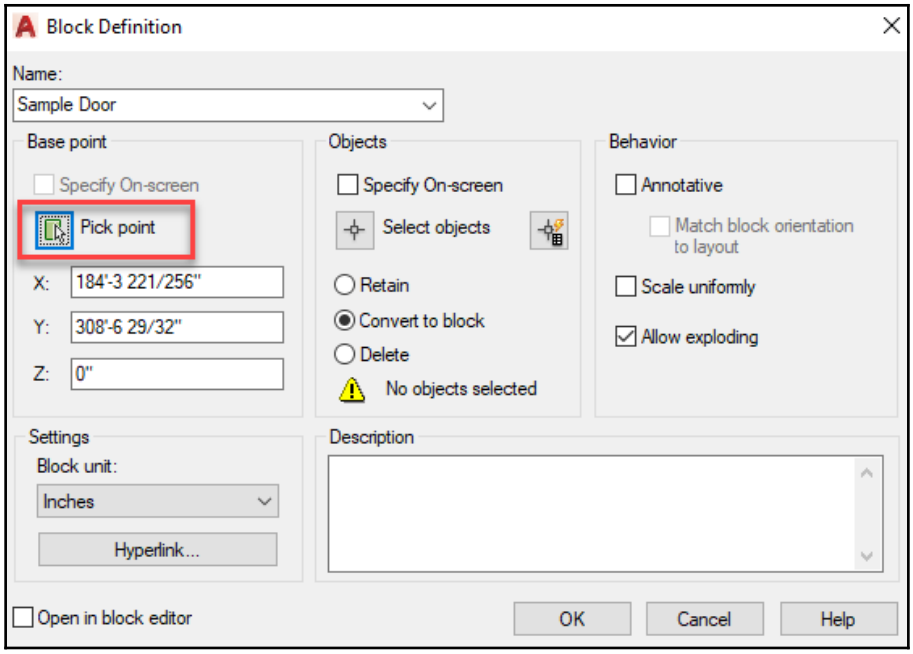


Figure 4.46: The Pick point option in the Block Definition window

4. Click on the lower-right corner of the door symbol to specify the base point and the **Block Definition** window will pop back up.
5. Click on the **Select objects** box in the **Objects** panel, then select all the rectangles and arcs that make the door symbol and press the *Enter* key.

6. In the **Objects** section, select the **Convert to block** radio button, as shown:

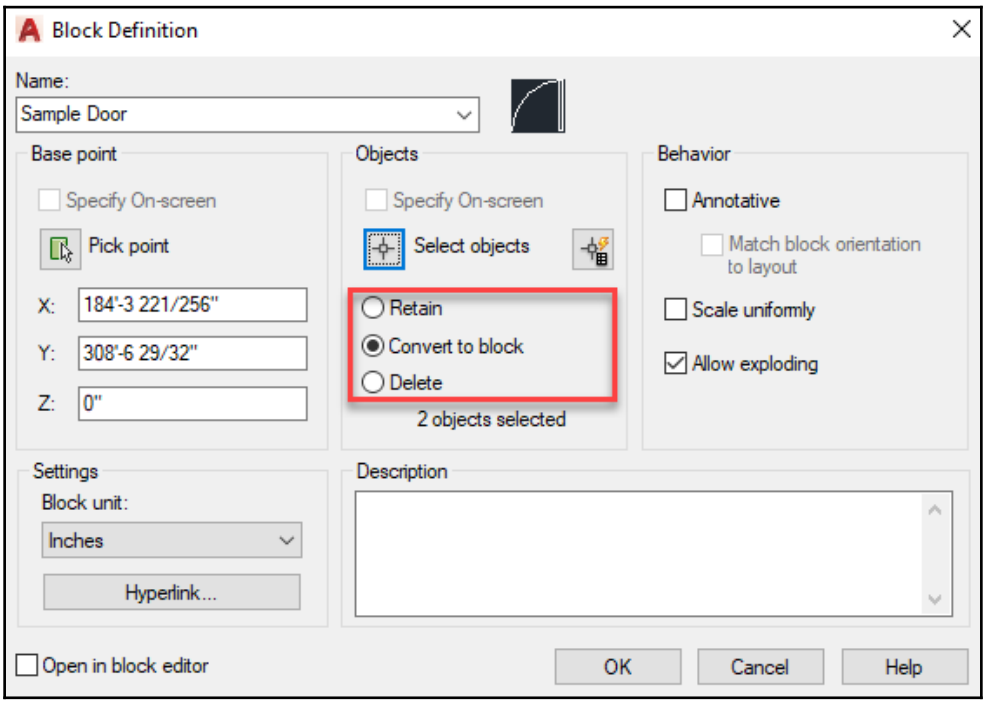


Figure 4.47: The Convert to block radio button, along with other options, in the Objects panel

7. From the **Behavior** section, check the **Allow exploding** checkbox and keep the other checkboxes unchecked. Also, make sure the **Open in block editor** checkbox is unchecked in the lower-left corner of the **Block Definition** window.
8. Once you are done with these settings, click **OK** and the door symbol will now be converted into a block.

Understanding the **Retain**, **Convert to block**, and **Delete** options from the **Objects** panel of the **Block Definition** window.
When making a block, I asked you to select the **Convert to block** option in the **Block Definition** window, shown in figure 4.47. If, however, you select any of the remaining options, namely **Retain** and **Delete**, you will still end up with a block. So what is the difference between these three different options?



These options control the behavior of objects that you use to create the block; that is, the rectangle and arc in the drawing area.

The **Retain** option leaves the rectangle and arc the way they are initially and even after making the block, if you hover your cursor over the objects you used to create blocks, the objects will show up as a polyline and arc, respectively, and not as a block.

The **Convert to block** option converts the on-screen object into a block reference, too and the **Delete** option deletes the on-screen objects, which are the polyline and arc, in this case.

Now, you have a door block in your drawing, which you can use anywhere in your current drawing. You can also use this block in other drawings if you want to. So, let's talk about the options we have for inserting blocks in the drawing.

Inserting blocks

To use this block in the drawing area, follow these instructions:

1. Click the **Insert** option in the **Block** panel and there you will see a preview of the block you just created.
2. Click on the preview of the block and the block will now follow your cursor, with the base point of block exactly at the cursor pointer.

3. Click on any point in the drawing area to insert the block, as shown:

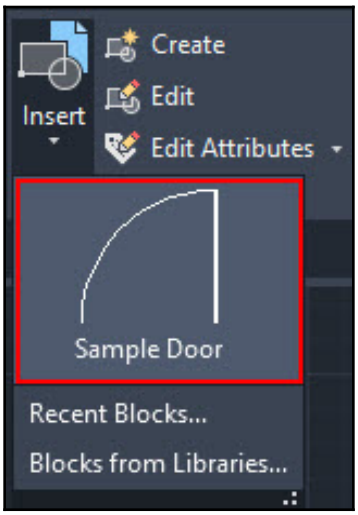


Figure 4.48: The Sample Door block in the Insert flyout of the Block panel

You can also make copies of existing blocks from your drawing and you will still end up with references of the block. If you make more blocks, they will be added to the block library, just like the door blocks, and you can use them by clicking the **Insert** option of the **Block** panel.

To see a list of recently used blocks and to access more options while inserting blocks, click the **Recent Blocks...** option in the **Insert** panel. The **Blocks** panel, with the three **Current Drawing**, **Recent**, and **Libraries** tabs, will show up, as shown:

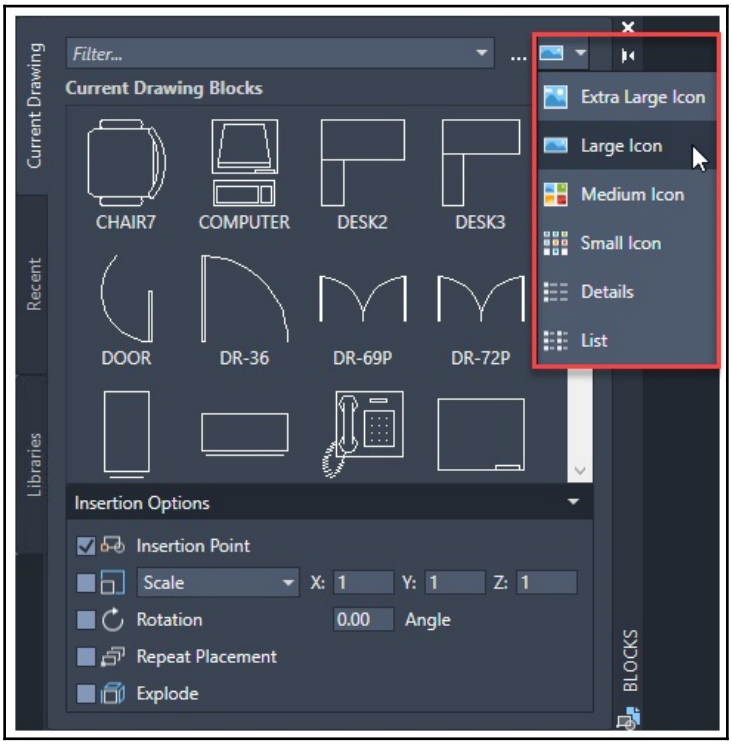


Figure 4.49: The block layout options in the Blocks panel

Click on the **Current Drawing** tab and the panel on the right will show all the blocks in the current drawing. You can change the way the blocks are visible in the panel using the **preview** option, as in the previous figure.

To insert a block from this panel, just click on it and then click in the drawing area to insert the block. The block will be added with its base point on the cursor. You can also drag and drop blocks from the preview area to the drawing area.

All the recently created, modified, or inserted blocks will show up in the **Recent** tab and you can use this as a filter to sort through the big list of blocks and access the blocks that you have recently used.

Insertion Options

Right underneath the block preview panel, you will find **Insertion Options**, which basically lets you change the way you insert blocks to your drawing. Here is an explanation of every option from the panel:

- **Insertion Point**: You can keep this option checked if you want to specify the insertion point on the screen. If, however, you don't want to specify the insertion point of the block, uncheck this option and specify the coordinate values of the insertion point in the **X**, **Y**, and **Z** fields.
- **Scale**: If you want to specify the scale of the block on the screen, then keep this option checked. Otherwise, uncheck this option and specify the scale using the **X**, **Y**, and **Z** fields. Changing the scale in different axes will change the block sizes separately along the **X**, **Y**, and **Z** axes:

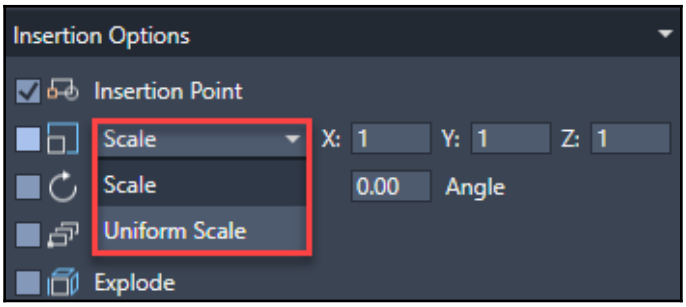


Figure 4.50: The Scale options in the Insertion Options panel

If you want to keep the size of objects uniform in all the scales, then click the **Scale** option and select **Uniform Scale** from the menu and then specify the uniform scale value in the next field.

- **Rotation**: This option lets you rotate the block. Keep this checkbox unchecked and specify the rotation angle in the **Angle** field, or check this option if you want to specify the rotation value in the drawing area.
- **Repeat Placement**: If you insert a block with the **Repeat Placement** option unchecked, the block insert command will terminate as soon as the block is added in the drawing area. But if you keep this option checked, the block insert command will remain active, with the base point of the selected block on the cursor, and you can keep clicking to insert as many references of the block as you want. To exit repeat placement, press the *Esc* key.
- **Explode**: If you want to insert a block as a simple drawing entity and not as a block reference, then keep the **Explode** option checked. This will ensure that the block is exploded after it is inserted in the drawing. Exploding a block changes it into its simple constituents.

Inserting blocks from other drawings

So, far we have seen methods of creating blocks and then inserting them in the same drawing. But if you want to transfer blocks from one drawing to another, you can use the **Blocks** panel for that, too. You can also create your own library of blocks using the **Libraries** tab of the **Blocks** palette. In the following example, I will show you how to create a block library and insert blocks from other drawings:

1. Make a folder in any location and give it a name. I am making a folder on the desktop and naming it `Sample Blocks`.
2. Place all the drawings that you want to use for your library in this folder.

3. Click the **Libraries** tab in the **Blocks** palette and then click the book stack icon in the top-right corner, as shown:

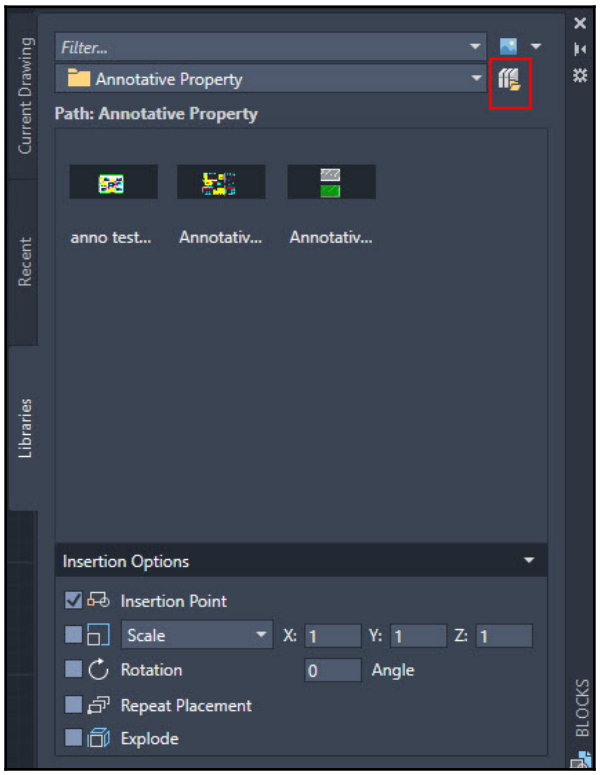


Figure 4.51: The add folder icon in the Libraries tab of the Blocks panel

4. Select the `Sample Blocks` folder and click **Open**.
5. All the drawings in that folder will load and they will show up in the **Libraries** tab. To insert these drawings as blocks, simply drag and drop them in the drawing area.
6. To insert the blocks in these drawings, double-click any drawing in the **Libraries** tab and all the blocks in that drawing will show up. Once again, drag and drop the block in the drawing area to insert it.

The base point, in the case of a complete drawing, is the origin of the original drawing and for blocks, it's the base point of the block:

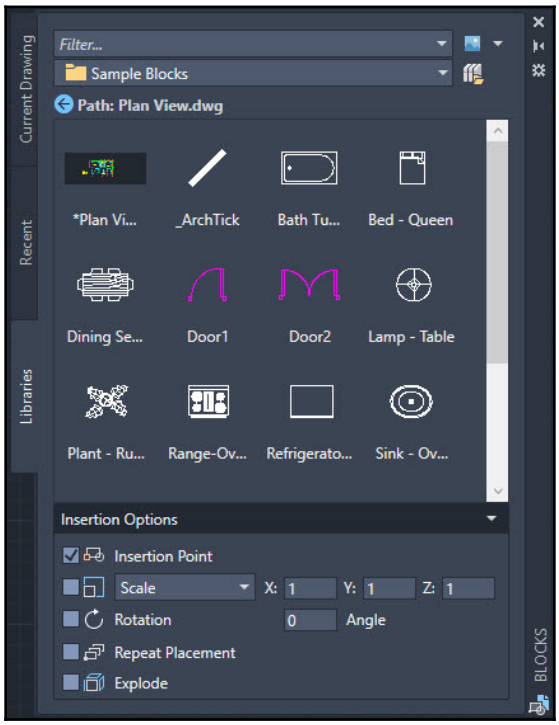


Figure 4.52: The blocks in a drawing in the Libraries tab of the Blocks palette

Once you have added a block from another drawing, it gets added to the **Current Drawing** list as well and then you can insert more references of that block from the **Current Drawing** or **Recent** tabs as well.

The **Libraries** tab is a new feature added in AutoCAD's 2021 version and if you are using an older version, you may find the **Other drawing** tab instead of the **Libraries** tab.

Modifying blocks

Blocks can be modified and when you modify a block, all cases of that block will change in the drawing. To explain this, I will use the door block that we created at the beginning of this chapter.

The door block we used contained a simple rectangle for the door and an arc representing the swing direction, shown in drawing **A** in the following figure. We want the block to change into the block shown in drawing **B**, with lines showing the wall distance and wall thickness, along with a dashed line along the length of the wall:

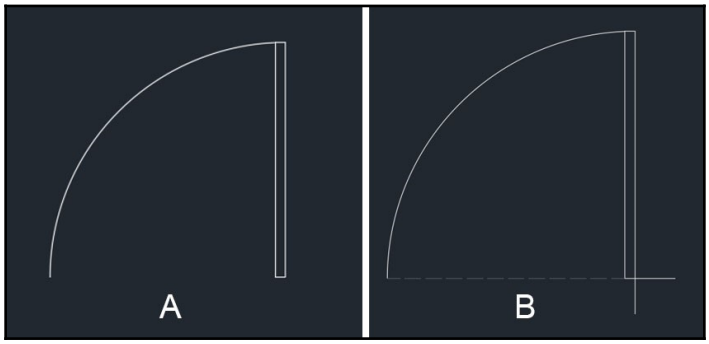


Figure 4.53: Block A, before modification, on the left and block B on the right, after making changes

Let's see how to modify the block in the Block editor environment:

1. Open the drawing that contains the "`Sample Door`" block and insert this block in the drawing area.
2. Double-click on this block and select its name from the **Edit Block Definition** window that pops up and click **OK**.
3. The block will open in the block editor environment, which has a gray background and a tab that says **Block editor**, as shown:

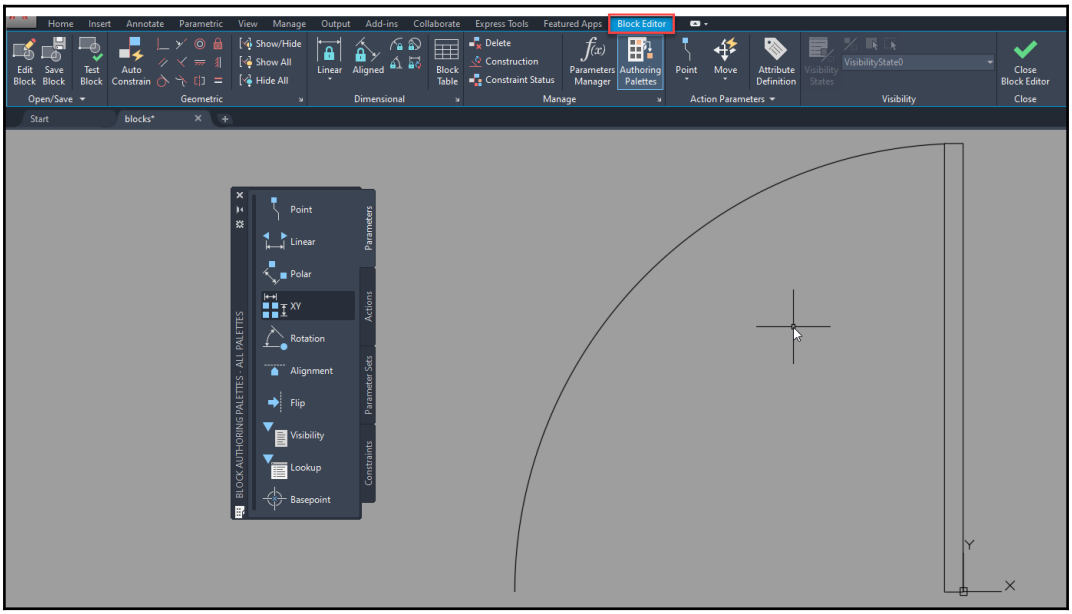


Figure 4.54: The door block open in the block editor environment

4. In this environment, the block will work like a normal drawing object and you can make changes to your block here. Switch to the **Home** tab to make modifications to this block, as per door block B.
5. When you are done making changes, click the **Close Block Editor** checkmark at the end of this panel and select "**Save the changes**" from the pop-up notification.
6. The block will be modified and all cases this block will also change in the drawing. If you insert the block again, it will be added with a modified form.

Modifying the block in the block editor is not the only way of making changes to your block; you can also modify the block right inside your drawing. To make changes inside your drawing, select the block then right-click and select **Edit Block in-place**. You can also select **Block Editor** if you want to modify the block inside the block editor environment, as shown here:

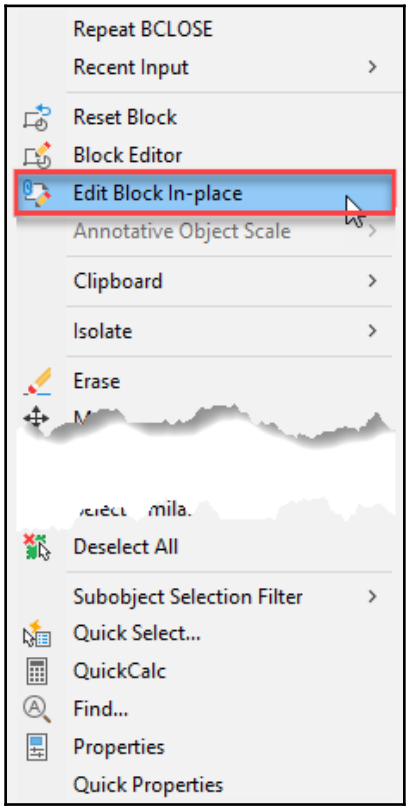


Figure 4.55: The Edit Block In-place option in the right-click context menu

When you select the **Edit Block In-place** option, AutoCAD opens a block selection window, where you can select the block you want to edit, and then click **OK**. The block opens right inside the same window and you can make changes to your blocks just like with a normal object. Once you are done making changes, click the **Save Changes** icon, which is now available at the end of the **Home** tab, as shown:

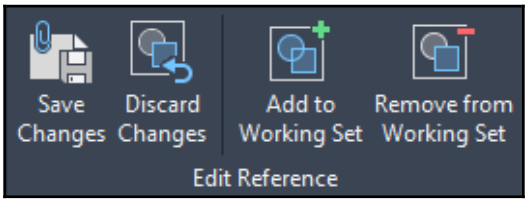


Figure 4.56: The Edit Reference panel and its options on the far-right side of the Home tab

The block will be saved with the modifications and if you add a new reference of the block, it will add the modified version of it.

Redefining blocks

If you want to replace a block with a new block or object, then you can use the Redefine Block option. In this example, I will redefine our existing sample door so that it looks like an entirely different kind of double door type block, as shown here:

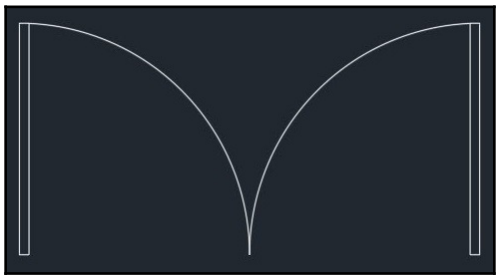


Figure 4.57: A double door block to be created by redefining the sample door block

Let's see how to redefine blocks:

1. For this, click on the **Create Block** option from the **Blocks** panel, or type `B` and press *Enter* to open the **Create Block** window.
2. Give this block the same name as the existing block in the drawing that you want to replace or redefine; in our example, the name will be "`Sample Door`".

3. Now, follow the next set of steps to specify the base point and number of objects and click **OK** when you are done.
4. After specifying all the options, my **Block Definition** window looked like this:

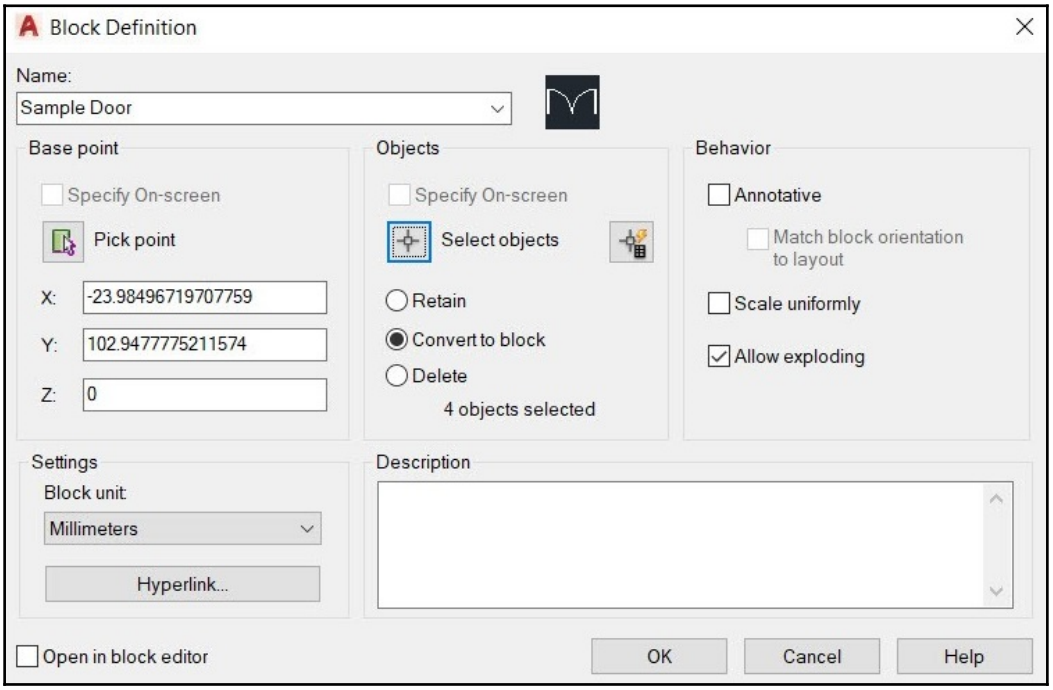


Figure 4.58: Options in the Block Definition window for redefining the sample door block

5. As soon as you hit **OK**, a block definition will show a pop-up message that says a block with the same name exists already and asks whether you want to redefine it. Select **Redefine block** and the block will be redefined:

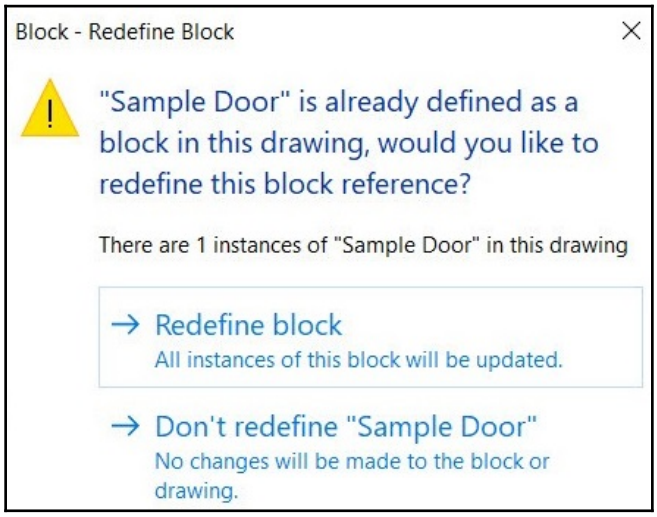


Figure 4.59: The Redefine Block pop-up message

Now, all the sample door blocks in the drawing will be replaced with the new door block type, which contains a double door, and the name of this new block will the remain same (`Sample Door`).

These are some of the ways blocks can be made and modified in AutoCAD. There is also another category of reusable content—called groups—which works as a block in some situations but is limited in features. It's good for situations where you need to make groups from multiple objects without all the features of the block. We will discuss groups and their properties next.

Working with groups

Blocks have lots of features, such as allowing you to resize blocks while inserting, make modifications in blocks that are transferred to all its references in the drawing, use blocks from other drawings, and so on. But we may not always need these features and for those situations, you can use groups, which in a way work like blocks but with limited functionality:

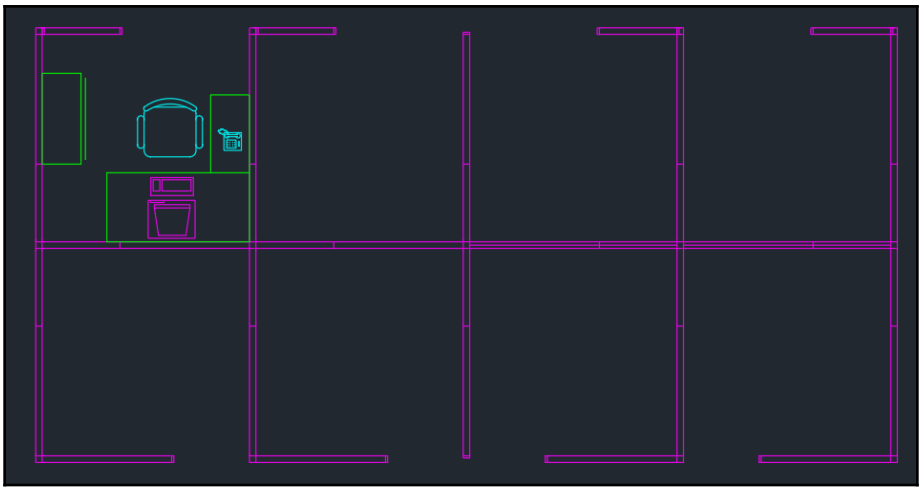


Figure 4.60: A sample drawing to be used for the groups feature

To understand groups, I will use the preceding drawing. Here, we have a set of cubicles, with the first cubicle containing office accessories such as a table, computer, phone, and so on. We will create our first group using these objects.

Creating groups

We will start the example here by creating a simple group comprising objects from our drawing:

1. To create a group, click on the **Group** icon in the **Groups** panel of the **Home** tab, as shown:



Figure 4.61: The Group option in the Groups panel of the Home tab

2. From the command line, select the `Name` subcommand, or type `N` and press *Enter* to select it. If you skip this option, you will end up with an unnamed group and AutoCAD will assign it a random name, which can be changed later:

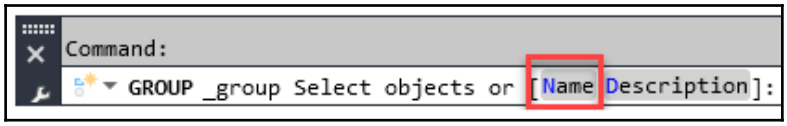


Figure 4.62: The Name subcommand in the command line

3. Type a name for the group but don't use spaces or special characters. In this example, I am using `office` as the name of the group. After typing the name, press the *Enter* key and the command line will again prompt you to select the objects for the group.
4. Select the chair, table, cupboard, computer, and telephone from the first cubicle in the drawing. Don't select the walls of the cubicle. Press *Enter* to create the group and exit the command.
5. A group called `office` is now created and you can use this group in your drawing.

If you select any object in the group, it will now select the whole group. If in your case the group is still not selectable, then make sure the **Group** selection option is active, as shown here:

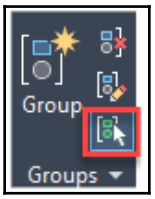


Figure 4.63: The Group selection option on the Groups panel

In the preceding example, we mostly selected blocks to create the group but it is not always necessary to do so and you can select simple objects to make groups as well.

Using groups

Now we have the `office` group, we can use it to add similar kinds of objects to other cubicles. Simply copy the group created in the first cubicle and paste it in other cubicles, as shown:

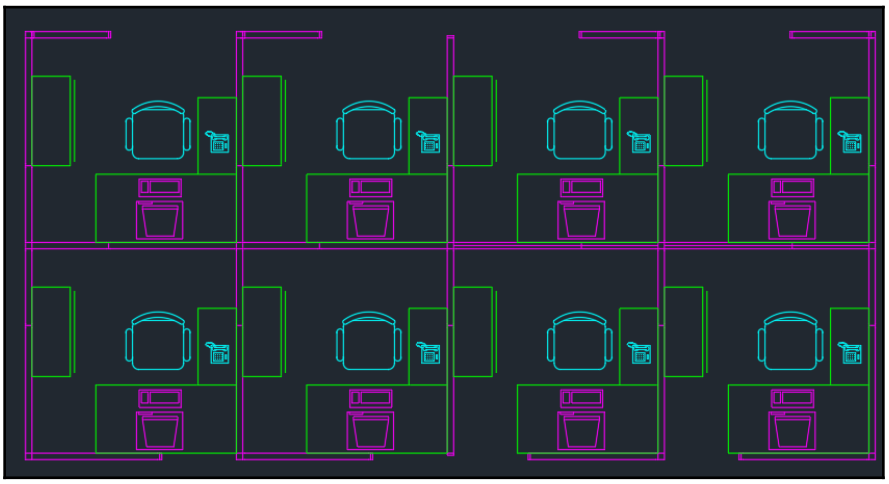


Figure 4.64: A single group copied to other cubicles in the drawing

If you want to add or remove objects from your group, then you can do so using the group edit tool. To explain this tool, I will remove the chair from our `office` group in the following example:

1. Click the group edit tool of the **Groups** panel, as shown:

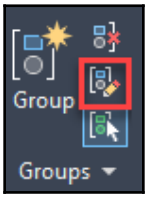


Figure 4.65: The group edit tool in the Groups panel

2. Now click on any object from the `office` group in the drawing area and the command line will now change to this:

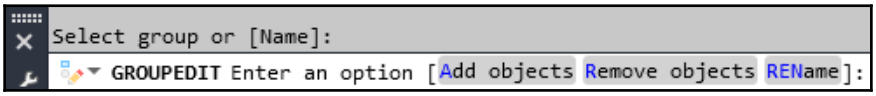


Figure 4.66: The GROUPEDIT options in the command line

3. As you can see, we have options for adding objects, removing objects, or renaming an existing group.
4. Click the `Remove objects` option from the command line or type `R` and press *Enter* to select it.
5. Now, click on the chair from the group you selected and press the *Enter* key.

You will notice that the group no longer has the chair in it. However, this change only happens to the selected group; the changes are not global and other cases of the `office` group remain unchanged—they will still have the chair as an object in the group.

If you want to explode a group so that all the objects of the group are again selectable separately, then you can use the ungroup option. To use it, click on the ungroup tool from the **Groups** panel, as in the following figure. Then, click on the group you want to explode, the group will break, and all the objects will now become separate once again:

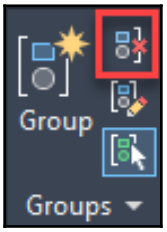


Figure 4.67: The ungroup option in the Groups panel

When you select a group, a bounding box shows up enclosing the objects of the group, as shown:



Figure 4.68: A group bounding box around objects of a group

If you don't want to see this bounding box, uncheck the **Group bounding Box** option in the **Groups** panel, as shown:

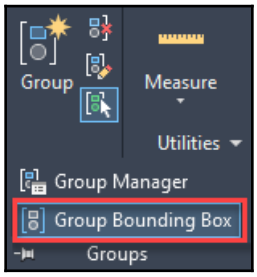


Figure 4.69: The Group Bounding Box toggle in the expanded Groups panel

So, as you have seen, groups are like a miniaturized version of blocks, with far fewer properties than blocks but with some completely different uses as well. They are mostly used as a tool to make temporary selection groups in a drawing that can be later ungrouped. In the next section, we will discuss another tool that will enhance the usability of blocks even further, called **attribute**.

Working with attributes

Consider the following example. Here, each cubicle has a separate cubicle number, shown by the yellow rectangular block that has a number in it. All the four cubicles' numbers are different but they were added using a single block, and the text that changes in all these block references is made with attributes:

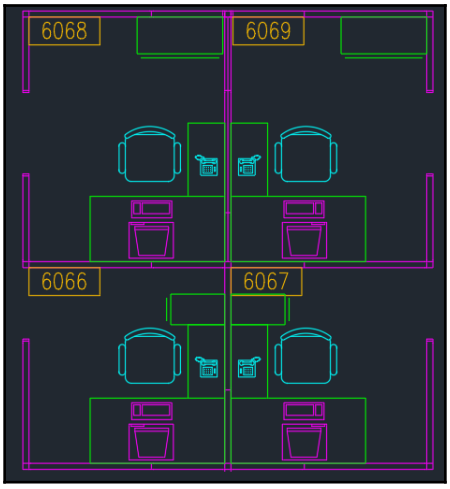


Figure 4.70: A sample drawing with blocks containing attributes

In this section, we will learn how to make these attributes, along with other kinds of attributes, and then we will learn how to use attributes in blocks.

Making attributes

In this example, I will make attributes for the cubicle number. We will make the rectangle first. For this example, I will make a rectangle with a length of `10` units and a width of `5` units. After we have a rectangle, we can start making the attribute:

1. Click the define attributes option in the expanded **Block** panel of the **Home** tab, shown in the following screenshot, or use its command alias, `ATTDEF`:

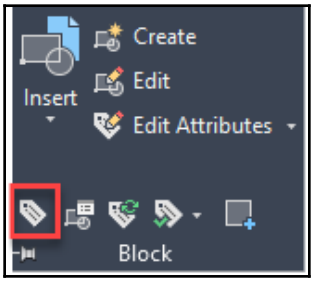


Figure 4.71: The define attributes option in the Block panel

2. The **Attribute Definition** window, shown in the following screenshot, will open with lots of options. From the **Mode** panel of this window, select **Lock position** and uncheck all the other options. We will talk about the other options in the **Mode** panel later in this chapter:

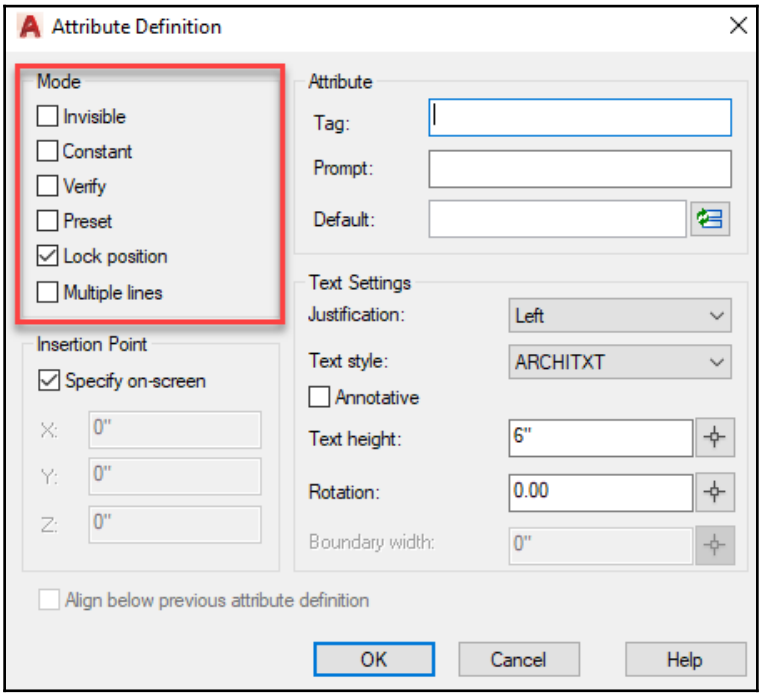


Figure 4.72: Attribute modes in the Attribute Definition window

3. In the **Attribute** panel on the top-right side, add the following values. In the **Tag** field, type `CNUM` (short for cubicle number). You can also type any other tag here— note that the space character is not allowed here.
4. In the **Prompt** field, type `Enter cubicle number` and in the **Default** section, you can type a default value or leave this field blank. In our example, I am leaving it blank.
5. In the **Text Settings** section, you can add the following values. For **Justification**, select **Middle center**. This will ensure your text always starts from the middle center of the rectangle. You can select any other justification value if you want to.

6. For **Text style**, select the style you have defined for the attribute; in this case, we have not defined any text style. In fact, we will learn about text style later in this chapter so I will leave this one as its default option, which is **STANDARD**.
7. Keep the **Annotative** checkbox unchecked and add the height for the text that you want. I am using a text height of `1` unit; the unit can be any unit that you have selected for the example.
8. Leave the rotation angle at `0` degrees and click **OK** to accept these settings. My final **Attribute Definition** window looks like this:

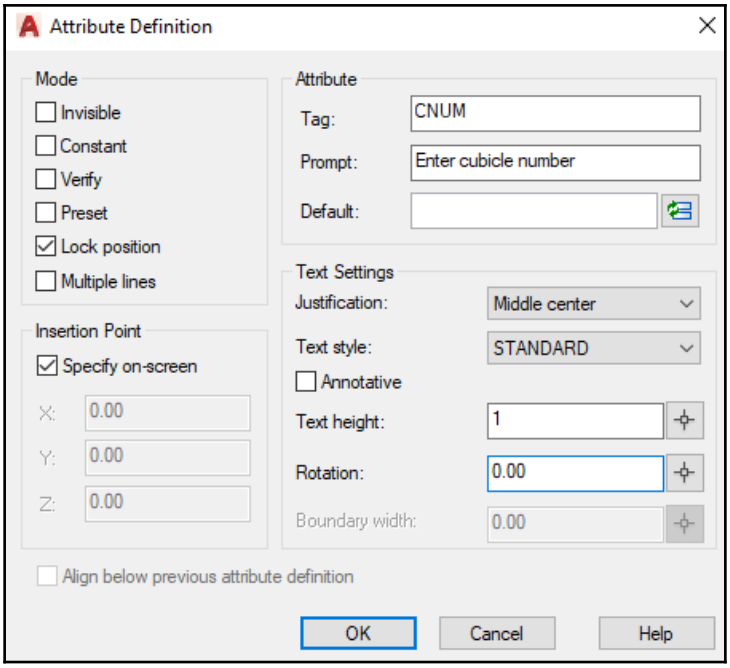


Figure 4.73: The Attribute Definition window after adding all the values in their respective fields

9. Now, the `CNUM` tag will show up on your cursor. Click on the geometric center of the rectangle and the final drawing will look like this:



Figure 4.74: A rectangle with the CNUM attribute at the geometric center

Now, we have a rectangle as well as the attribute text. It's time to convert these things into a block, and we will do just that in the following example:

1. Start the block command using the **Create** option in the **Block** panel or using its command alias, `B`.
2. In the block's window, add a name for this block. I am using `Cubicle Tag`.
3. Now, select the base point, the rectangle, and the attribute for the cubicle block. After making these settings, my **Block Definition** window looks like this:

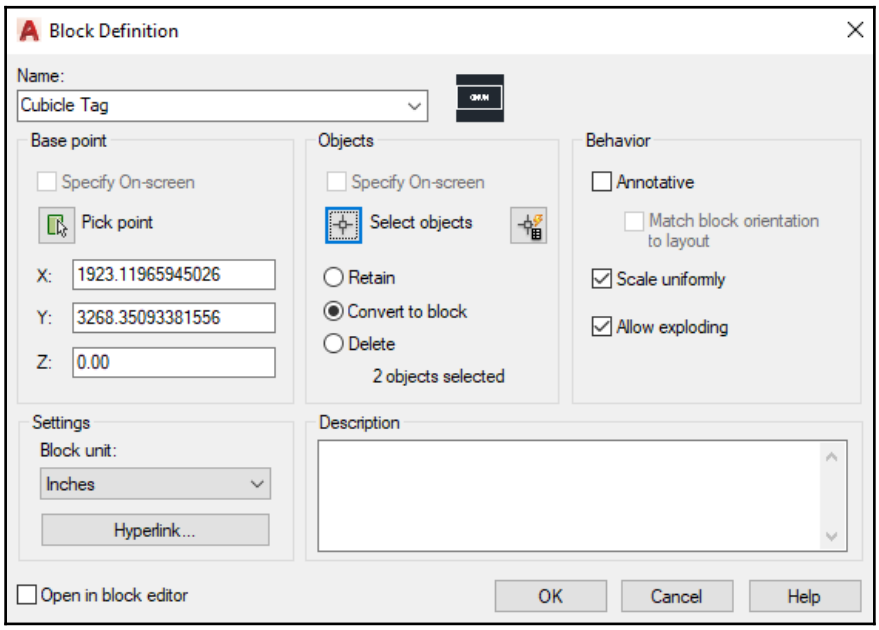


Figure 4.75: Options in the Block Definition window for converting the rectangle and attribute into a block

4. Click **OK** and the `Cubicle Tag` block will be created. As soon as you finish the block, the **Edit Attributes** window will open with the fields shown in the following screenshot:

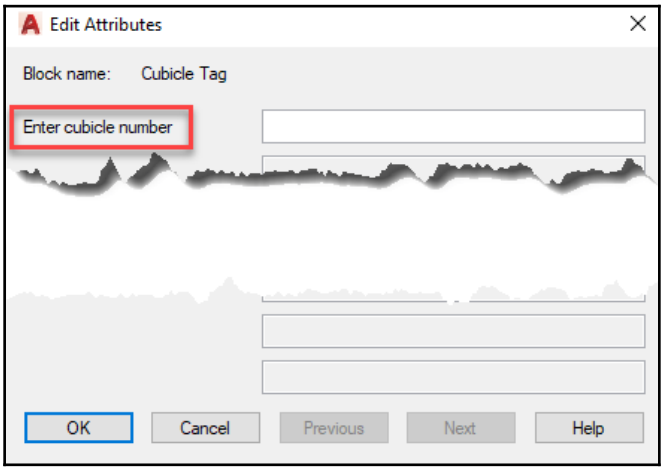


Figure 4.76: The Edit Attributes window with the Enter cubicle number field highlighted

5. In this window, the name of the block will show up at the top and you can see the prompt we used to create the attribute in the first field enclosed in the red box. Type any value in the first field and click **OK** to enter the block with that value.

We now have our first block, which contains an attribute that can be changed every time a new block reference is added. To add more block references, select the `Cubicle Tag` block from the **Insert** option in the **Block** panel. As soon as you insert a block in the drawing area, it will open the **Edit Attributes** window, where you can add the cubicle number and click **OK** to add a different cubicle number for every single block reference.

If you don't see the **Edit Attributes** window when inserting a block containing an attribute, you will see a similar prompt in the command line and you can directly type the attribute values in the command line:

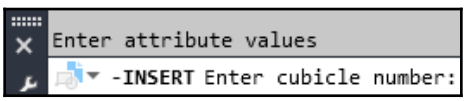


Figure 4.77: The edit attributes prompt in the command line



The display of the **Edit Attributes** window is controlled by an `ATTDIA` system variable. When the value of the `ATTDIA` system variable is set to `1`, then the **Edit Attributes** window will show up and if this value is `0`, then the prompt will appear in the command line.

If you want to modify the attribute value of the existing block, then double-click on the block and change the attribute from the **Value** field of the **Enhanced Attribute Editor** and click **OK**, as shown:

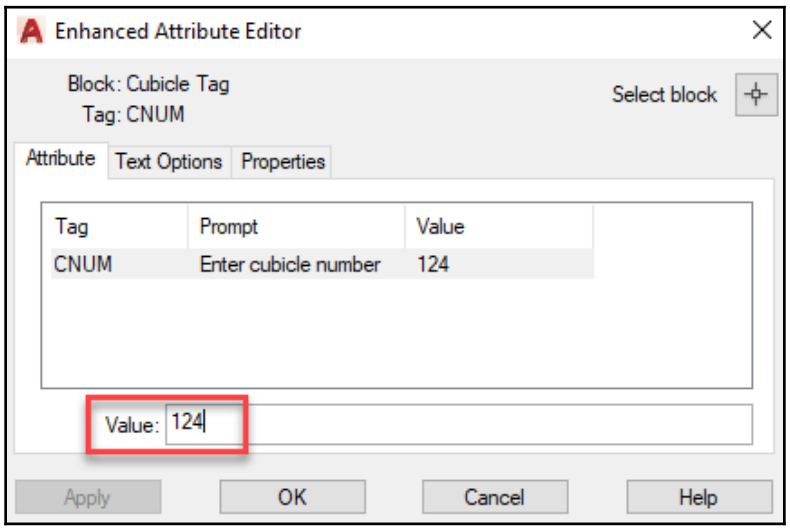


Figure 4.78: The Value field in the Enhanced Attribute Editor window

You can also change the formatting of an attribute using the **Text Options** and **Properties** tabs of the **Enhanced Attribute Editor** window. Some of the prominent properties you can change here are the color, justification, and text width.

Understanding attribute modes

There are some modes that are available to you when making attributes. We have selected only one of these modes, **Lock position**, and kept the rest of them unchecked. In this section, we will learn about the remaining modes and you can then decide whether to keep them checked or unchecked, depending on the type of block that you will use these attributes for:

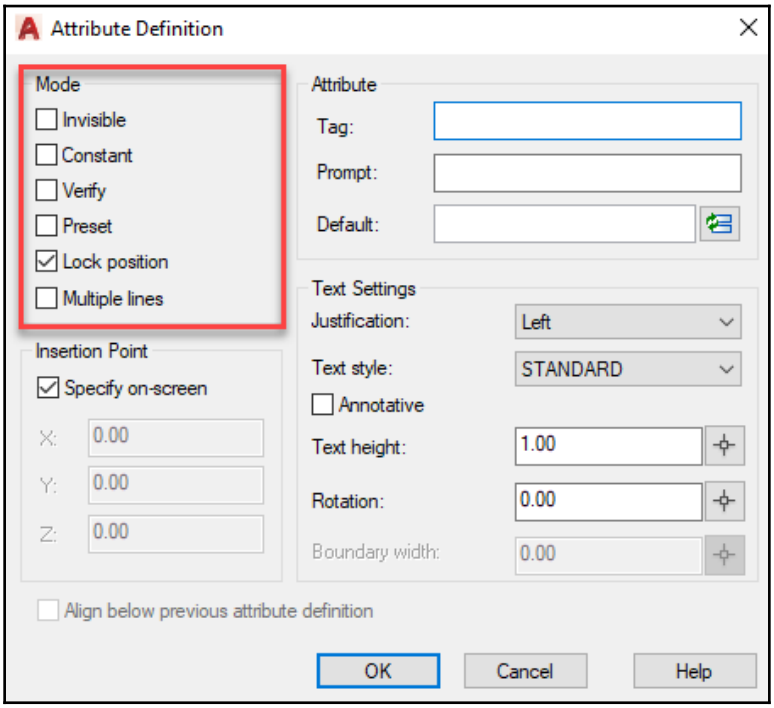


Figure 4.79: The attribute modes in the Attribute Definition window

Let's see each option one by one:

- **Invisible**: If you keep the **Invisible** mode checked, the attribute will no longer show in the drawing area but you can make it visible or invisible by using the attribute display options shown in the next section.
- **Constant**: As the name suggests, this mode will make the attribute value constant, which can't be changed once added to the block. As soon as you select the **Constant** mode, the **Prompt** field will gray out and now you can type a value in the **Default** field, which will be added automatically every time you add this block to the drawing:

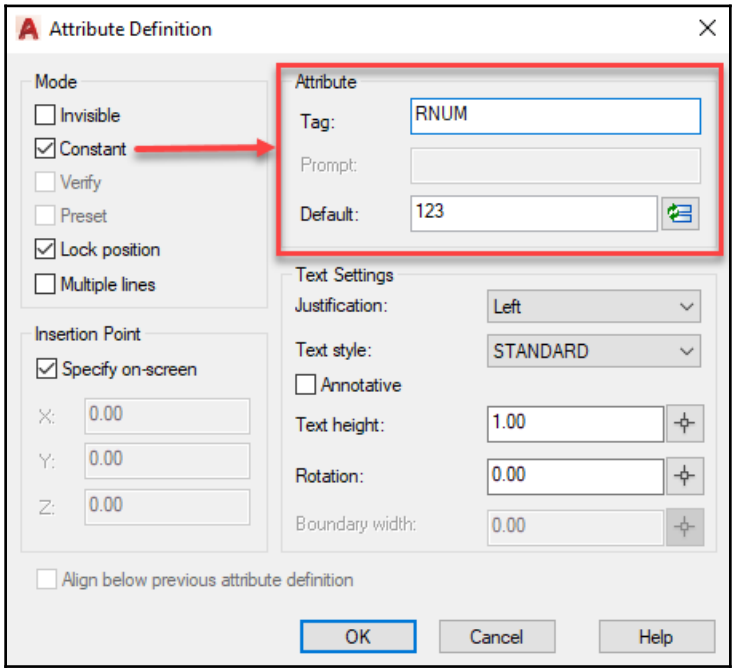


Figure 4.80: The Constant mode deactivates the Prompt field and adds a constant attribute value

- **Verify**: When this option is checked, AutoCAD will prompt you to verify the attribute that you have added at the time of adding the block. This option will work only when the `ATTDIA` system variable is set to `0`; that is, the **Edit Attributes** prompt is made to show only on the command line.

- **Preset**: When you check the **Preset** mode, the block will be added with the default value and it will not show any prompt. This will again happen only when the `ATTDIA` system variable is set to `0`, just like with the previous mode.

- **Lock position**: If you keep the **Lock position** mode checked, the attribute text will not move with respect to the block but if you keep it unchecked, you will be able to move the attribute text with respect to the block.

- **Multiple lines**: This mode allows you to add multiple lines for the attribute text. When this mode is checked, the boundary option in the **Attribute Definition** window will activate, where you can define the width of the text and if the text goes beyond this width value, it will be added to the next line:

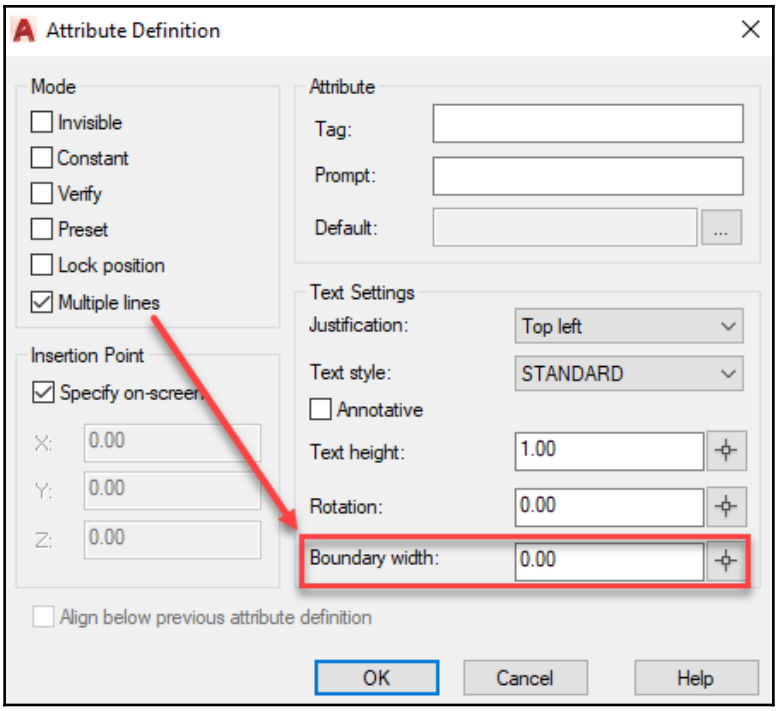


Figure 4.81: The Boundary width option activates when Multiple lines is selected in the Mode section

So, now that you know what these modes are, let's dig deeper into the first attribute mode—Invisible—as you are likely to use it often in a drawing.

Making invisible attributes

So far, we have worked with attributes that are visible in the drawing area, but you can make attributes invisible as well. There may be blocks where you want to keep some of its information hidden from the drawing, such as the name of the manufacturer of a chair block or the name of the person in the cubicle tag that we made in the preceding example. For these kinds of blocks, you can make invisible attributes that, by default, won't show in the drawing area, but you can still see them by using some extra options:

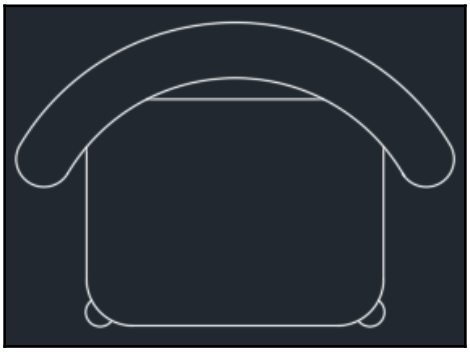


Figure 4.82: A chair drawing to be used to make an invisible attribute

In the following example, I will make an invisible attribute in the chair block using the drawing of the chair shown in the previous figure:

1. Click on the **Define Attribute** option of the expanded **Block** panel or use its command alias, `ATTDEF`.
2. Type `MANUF` for the tag and `Enter manufacturer name` in the **Prompt** field, then select the proper **Text Settings**, shown in the following screenshot.

3. Select **Lock position** and **Invisible** from the **Mode** panel. The **Invisible** mode will ensure that your attribute remains hidden when added to a block:

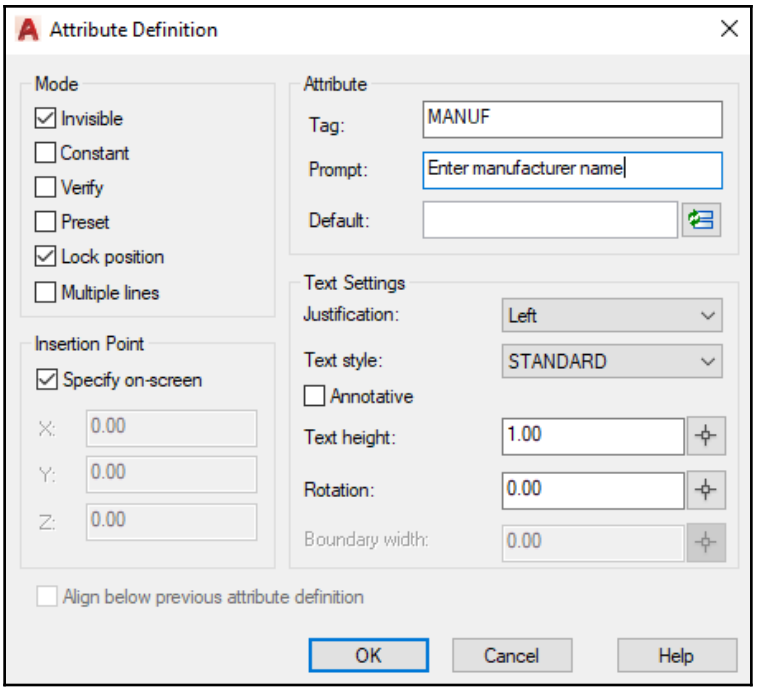


Figure 4.83: All the fields and modes selected to make an invisible attribute

4. Click **OK** and then click somewhere close to the chair to place this attribute, as shown:



Figure 4.84: The chair symbol with the attribute placed on the bottom-right side

5. Now, select the **Create Block** option from the **Block** panel or use its command alias, `B`.
6. Enter the name of the block. In this example, I am naming it `sample chair`. Select the chair and the `MANUF` attribute and specify a base point for the block in the **Block Definition** window and click **OK**.
7. The block will be created and you will now see the **Edit Attributes** window, where you can add the name of the manufacturer. Type `chair company` for the manufacturer's name and click **OK**.

You will notice that the block will be added to your drawing but the attribute will not show up in the drawing as it was made using the invisible mode. To make this attribute visible, you can change the attribute display settings.

From the expanded **Block** panel, select **Display All Attributes** to show the hidden attribute and it will show up as next to the chair. By default, this setting is set to **Retain Attribute Display**, which keeps any invisible attributes invisible and other attributes visible. You can also hide all the attributes—visible or invisible—using the **Hide All Attributes** option:

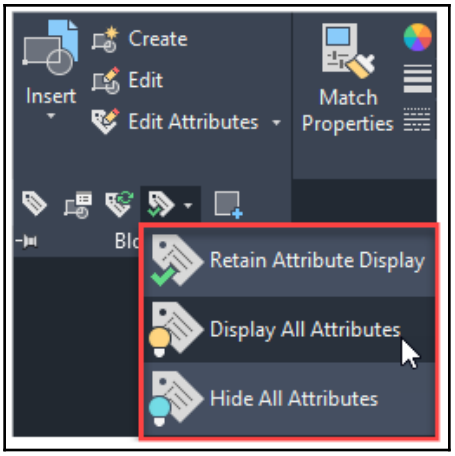


Figure 4.85: The attribute visibility options in the expanded Blocks panel

So, this was all about making attributes and blocks containing attributes. As you can see, the application of these tools is endless and the reusable nature of blocks makes this tool even more useful. So, before we begin the next chapter, let's summarize the content we have learned and the topics we will be learning in the next chapter.

Summary

This chapter began with the grid and snap status bar features, which allow you to visualize the drawing area and use the background grid as a reference for the drawing. We also learned about using the selection cycling toggle, which is great for making selections of overlapping objects. However, the main highlights of this chapter were the Array and block tools.

With Array, we learned how to create patterns along a rectangle, circle, and any other path. We learned how to create reusable content, called blocks, and also how to create blocks with attributes and a simple version of blocks, called groups.

In the next chapter, we will continue to learn the named objects and we will move one step further towards learning about drawing management tools, such as layers. We will also learn how to check the properties of objects and modify them for a drawing.

5
Managing Drawings with Layers and Properties

When you start adding up objects in your drawings, all the different types of objects can get really cumbersome to manage. Managing these objects and sorting them into different categories not only speeds up your drawing workflow but it can also make it easier for you to find the required objects when they are needed. So, this chapter is all about managing drawings using layers, and we will also learn how to assign different properties to our objects to sort them properly. In this chapter, we will also learn how to check the properties of objects, such as their area, length, and radius, which is essential to any drawing workflow. By the end of this chapter, you will be well equipped with the tools and commands required to effectively assign and manage the properties of objects.

The following are the topics we are going to discuss in this chapter:

- Managing object properties
- Using **Match Properties**
- Using enquiry commands
- Drawing management using layers
- Using the quick access tools in the **Layers** panel

Let's begin by understanding the grid settings and grid mode options of the status bar.

Managing object properties

In AutoCAD, you can change properties such as color, lineweight (to do with the thickness of a line), linetype, and transparency. These properties are required to properly display drawing types. As an example, obscured lines are generally shown by "hidden line" linetype and construction lines generally have transparency added to them so that they look less prominent than the actual drawing. In the sample drawing shown here, the hidden lines are shown by the dashed line type and the center line has transparency added to it:



Fig 5.1: Center and dashed lines with transparency

In this section, we will learn how to add these properties to different AutoCAD objects. The properties of an object can be changed from the **Properties** panel of the **Home** tab, as shown in the following figure:

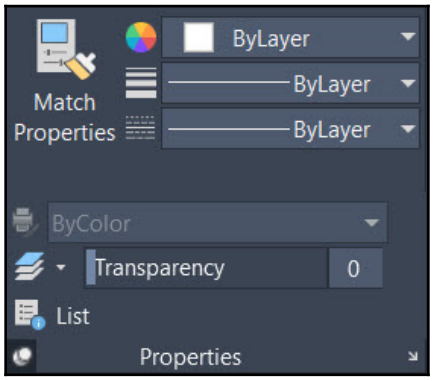


Fig 5.2: The Properties panel of the Home tab

To begin, I will use the drawing shown here. In this drawing, all the lines, the arc, and the circle have no transparency or linetype applied:

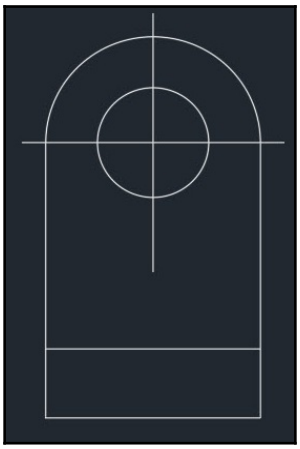


Fig 5.3: Sample drawing for applying properties

In the next section, we will learn how to apply linetype to construction lines and then we will add transparency to them.

Adding linetype

To add linetype, we need to first load a linetype in the drawing. The linetype menu is in the **Properties** panel, as shown in the following figure:

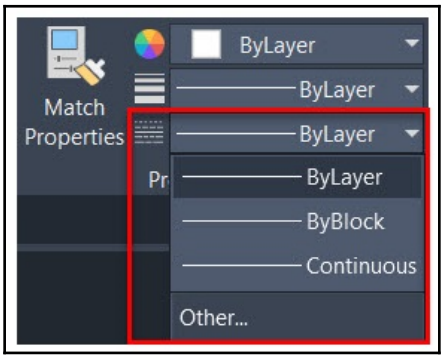


Fig 5.4: Linetypes available in the default linetype menu of the Properties panel

As you can see, by default, there are only three linetypes – **ByLayer**, **ByBlock**, and **Continuous** – and they all make a solid line. To load a linetype, click on the **Other...** option in the linetype menu and the **Linetype Manager** window will open up. Click on the **Load...** button of this **Linetype Manager** window and a new window with a list of all available linetypes in AutoCAD will open up:

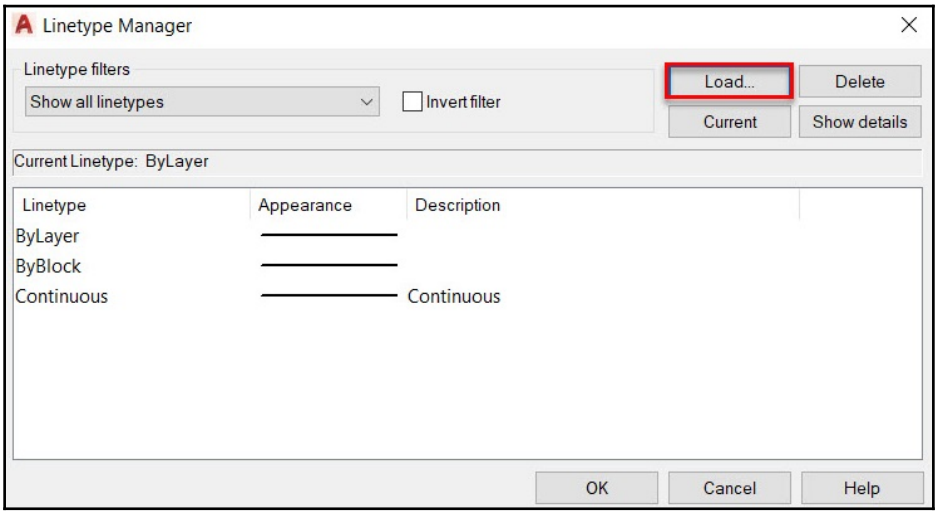


Fig 5.5: The Load... button in the Linetype Manager window

Select the linetype that you want to use in the drawing and click **OK** to load it in the **Linetype Manager** window. In this case, we want to use the **HIDDEN** and **CENTER** linetypes. So, select the **HIDDEN** linetype, then press and hold the *Ctrl* key and select **CENTER** and click **OK** to load it in the **Linetype Manager** window. Once the linetype is loaded in the **Linetype Manager** window, click **OK** to close it, and now both the linetypes will show up in the list of linetypes in the **Properties** panel.

To apply the linetype, follow these steps:

1. First, select a line from the drawing area. In this case, select the second horizontal line from the bottom.
2. Now click on the linetype menu of the **Properties** panel and select the **HIDDEN** linetype.
3. You will notice that the linetype will be applied to the selected line:

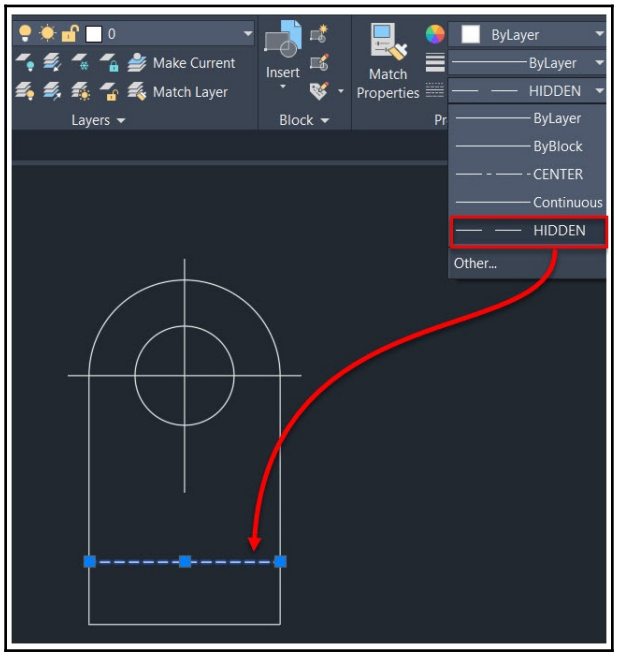


Fig 5.6: The HIDDEN linetype applied to the selected line from the drawing

In a similar way, we will apply the **CENTER** linetype to the two perpendicular lines at the center of the circle using the following steps:

1. Select both of the perpendicular lines that intersect at the center of the circle.
2. Now go to the linetype menu of the **Properties** panel and select the **CENTER** linetype from the linetype list.
3. The linetype will be applied to both perpendicular lines. Once the linetype is applied, you can press the *Esc* key to remove the line from the selection set:

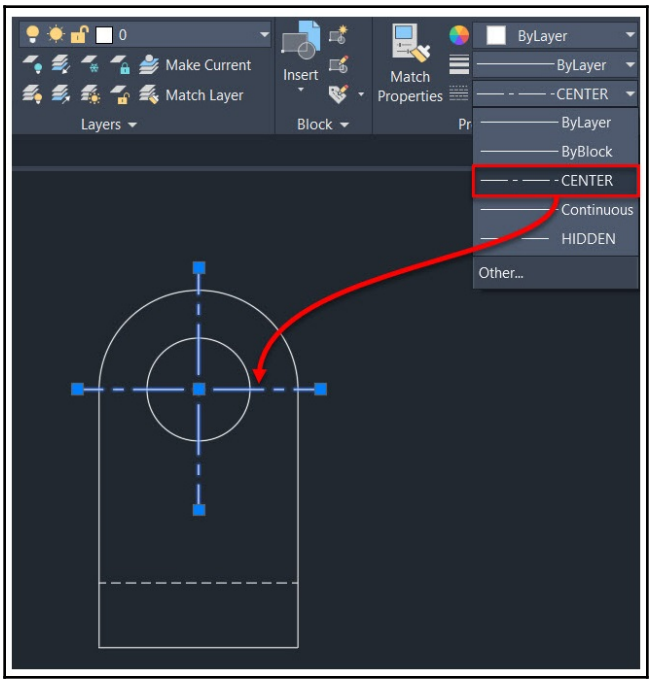


Fig 5.7: The CENTER linetype applied to two perpendicular lines from the drawing

In your case, the linetype may not look exactly like the one shown in the preceding figure, and if that's the case, then most likely it's an issue with the scale of the linetype, which you can change as mentioned in the next section.

What if the linetype does not look right in the drawing?

In many situations, you will find that even after applying the linetype, the line may not visibly change at all and may still remain a solid line; in some cases, the line's gaps or features will be too sparsely placed. In these situations, you need to change the scale of the linetype to any value that will ultimately make it look good.

To do this, you need to change the linetype scale from the **PROPERTIES** palette of the line. Select the line that is not displaying properly even after applying the linetype. Then, right-click and select **PROPERTIES** from the context menu or type `PR` and press *Enter*. Here, `PR` is the command alias of the **PROPERTIES** palette.

On the **PROPERTIES** palette, you will find an option called **Linetype scale**, as shown here:

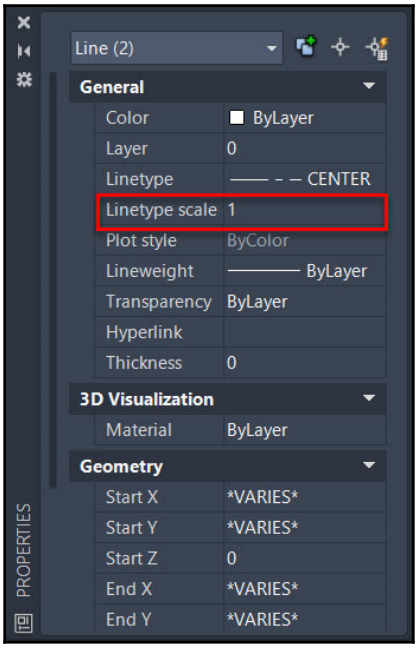


Fig 5.8: The Linetype scale option on the PROPERTIES palette

The default value for **Linetype scale** is `1`. Change this value to a smaller or higher number depending on the linetype you are getting. If your current linetype has features very closely spaced, then increase the linetype scale to a bigger number, and if they're very sparsely placed, then decrease the number. You may even need to use very small or very big numbers, such as `0.01` or `100`, in the **Linetype scale** field to make your lines look normal.

Although the name linetype may be a little confusing sometimes, as it seems as though it is a property specific to lines, that's not the case: you can apply linetype to other drawing objects, such as circles, polyline shapes, and ellipses.

Adding transparency

Now let's add transparency to the lines to which we added the **HIDDEN** and **CENTER** linetypes. The **Transparency** option is in the expanded **Properties** panel, as shown in the following figure:

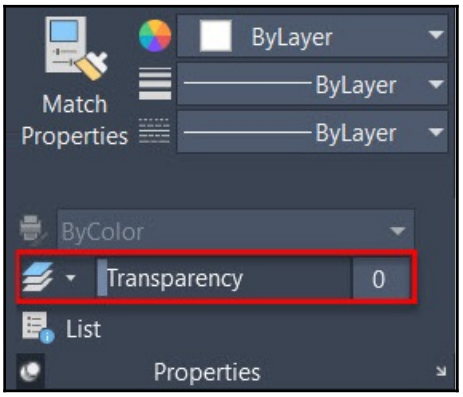


Fig 5.9: The Transparency option in the expanded Properties panel

The default value of **Transparency** is `0` and the maximum value is `90`. The higher the value, the more transparent your drawing will be; at a transparency value of `90`, an object will almost disappear. In the following figure, circle **A** has `0` transparency, circle **B** has `45` transparency, and circle **C** has `90` transparency:

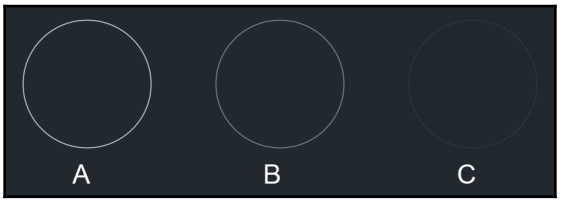


Fig 5.10 - Transparency values of 0, 45, and 90 applied to circles A, B, and C, respectively

To apply transparency, select objects from the drawing area; in this case, select the lines to which we applied different linetypes in the previous example.

Now go to the **Transparency** option and move the slider to the right to add more transparency; you can also type a value in the **Transparency** field, as shown in the following figure:

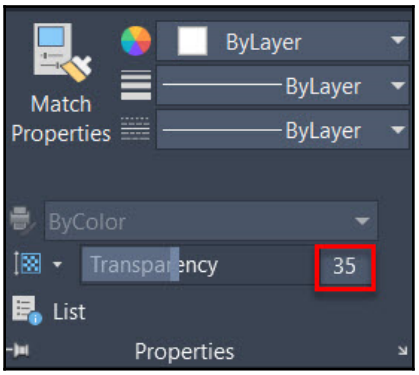


Fig 5.11: The Transparency slider in the Properties panel

Once you have selected the required transparency value from the field, simply press the *Esc* key and the transparency will be applied to the selected lines.

In some situations, even after applying the transparency, you may not see any effect on the drawing where transparency was applied. This happens when the **Transparency** status bar option is deactivated:

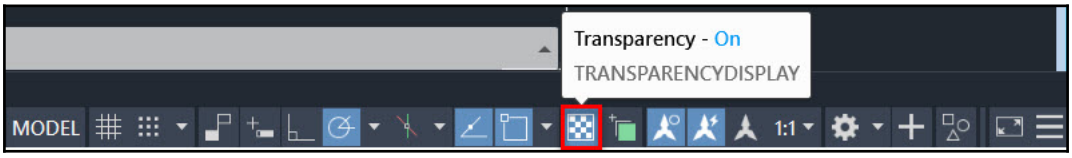


Fig 5.12 - Transparency display status bar toggle

To show the transparency in the drawing area, make sure the **Transparency** status bar option is active, as shown in the preceding figure. If you don't see the **Transparency** option on the status bar, then click on the icon of the three stacked horizontal lines at the end of the status bar and select the **Transparency** option from the list of status bar tools to make it visible on the status bar. Once it is on the status bar, you can then toggle its visibility.

After applying transparency of `60` to the **HIDDEN** line and the **CENTER** line, the drawing will look like the following figure:



Fig 5.13: Drawing after applying transparency of 60 to the HIDDEN and CENTER lines

As you can see from the preceding example, transparency is great for adding depth to a drawing and making construction geometries less prominent than the actual drawing. There are many other ways that you can use transparency in AutoCAD, but these are a couple of areas where they are mostly used.

Adding lineweight

Lineweight determines the thickness of a line and you can configure it using the lineweight menu of the **Properties** panel, as shown in the following figure:

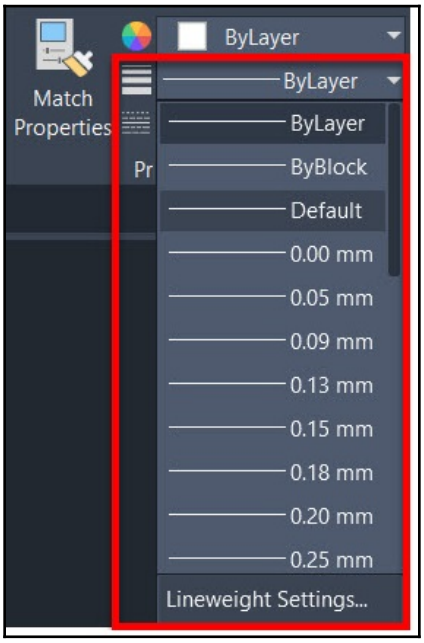


Fig 5.14: Lineweights in the lineweight menu of the Properties panel

Once again, to explain this, I will take the example drawing that we have used so far in this section. Here, I will make the outer boundary of the drawing thicker than the internal line and circle so that the end result looks like drawing **B**:

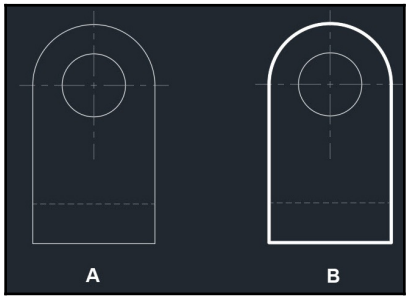


Fig 5.15: Drawing A with no lineweight and drawing B with lineweight applied to the outer polyline

The default value for lineweight is **ByLayer**, which applies a theoretical thickness of `0.00` units, which obviously is not possible in the real world but in AutoCAD lines can theoretically have a thickness value of zero. However, you can change this and add any thickness value from the lineweight list.

To explain this, I will add a lineweight of `0.5` units to the outer boundary of the preceding drawing using the following steps:

1. To add the lineweight, select the outer boundary from the drawing area.
2. Click the lineweight menu to expand it and from the list, select the lineweight that you want to apply. In this case, select a lineweight of `0.50` units. The units can be millimeters or inches, depending on the unit you have specified in the drawing template.
3. Once the lineweight is applied, press the *Esc* key to remove the objects from your selection and the boundary will now look bold with a thickness of `0.5` units. This thickness is relative thickness and if you zoom in or out, the thickness will always look the same.

If in this case too, after applying the lineweight, you don't see any change in the drawing, then activate the **Lineweight** display status bar option as shown in the following figure. If you don't even have the **Lineweight** display status bar option on the status bar, then click on the three-lines icon at the end of the status bar and select the **Lineweight** option:

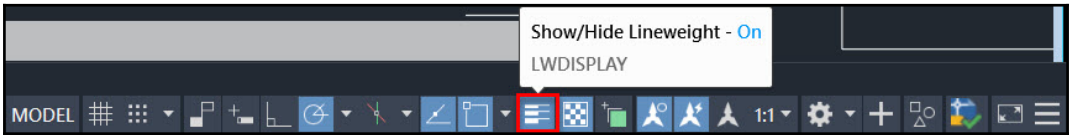


Fig 5.16: Lineweight display status bar toggle

Once the **Lineweight** option is visible on the status bar, make it active and you will be able to see the lineweight applied to objects in the drawing area.

Adding color

Your drawing does not have to be a completely black-and-white drawing; you can add colors to your drawing to make different objects look distinct. For example, in a floor plan drawing, you can make dimensions green, the walls blue, the doors and windows red, and interior objects cyan. The colors can be added to the objects directly or you can also add colors to the layers, which we will learn about later in this chapter.

To explain this, I will use this simple drawing of an office area:

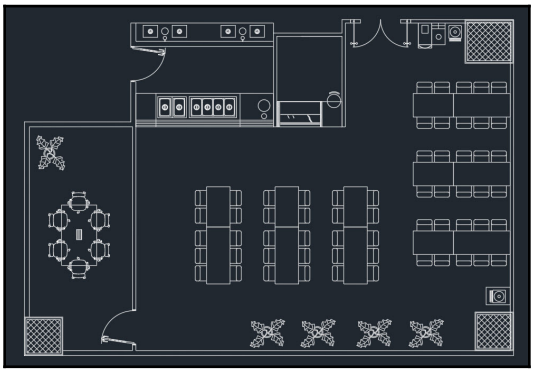


Fig 5.17: Sample drawing for adding colors

The color menu is at the top of the **Properties** panel of the **Home** tab, as shown in the following figure:

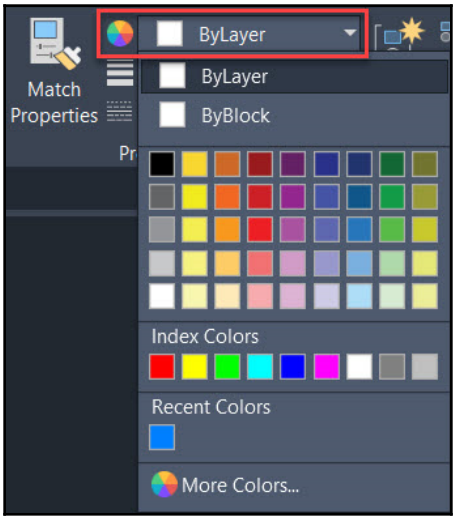


Fig 5.18: Color drop-down menu in the Properties panel

In the following example, I will start by adding colors to the plants:

1. Select all the plants that you see in the drawing area then expand the color menu.
2. Select the green color box from the **Index Colors** area as shown in the following figure.

3. Press the *Esc* key once you have applied the colors to the selected drawing:

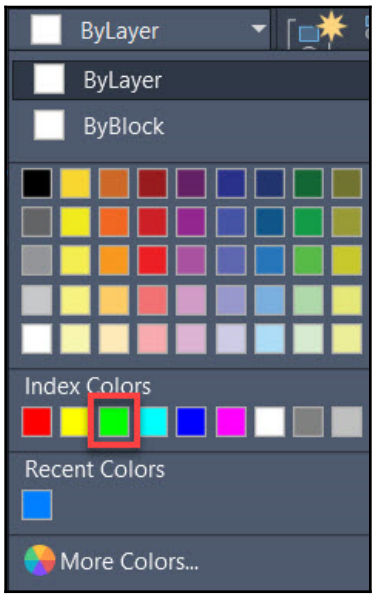


Fig 5.19: Green highlighted in Index Colors

The color of the objects will change as shown in the following figure. In a similar way, you can change the color of other objects in the drawing:

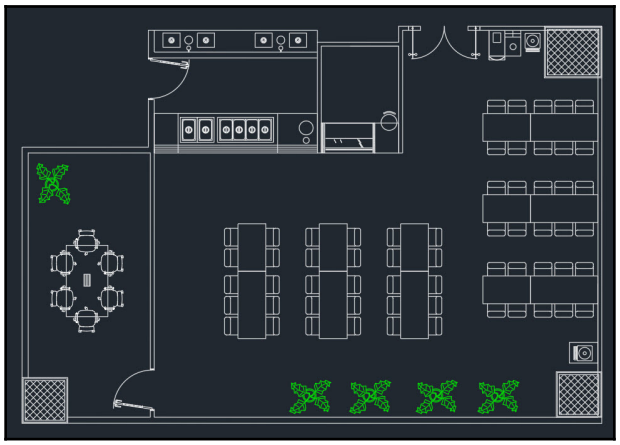


Fig 5.20: Green applied to the selected objects of the drawing

Although there are plenty of colors available in the colors menu, if you need more colors, then click on the **More Colors** option at the end of the colors menu and a **Select Color** palette will open. From this window, you can select colors using one of the three tabs– **Index Color**, **True Color**, and **Color Books**:

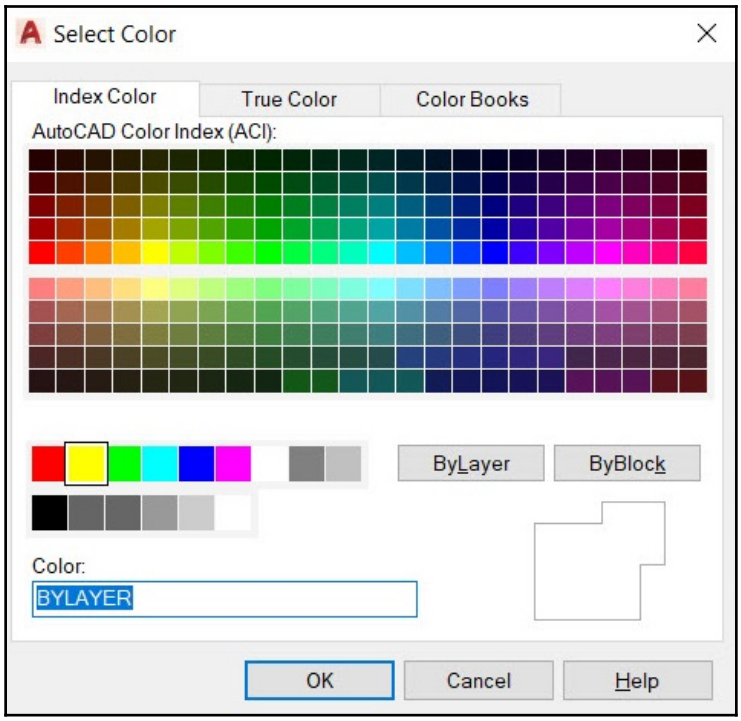


Fig 5.21: The Select Color palette, which opens upon clicking the More Colors... option

The **Index Color** tab has colors from the AutoCAD color index, offering 255 colors. The two panels at the top contain 240 colors from color numbers 10 to 249. The next panel contains 9 colors from 1 to 9 and the last panel contains mostly gray shades from color numbers 250 to 255. To choose the colors here, you can click on the box of the color or you can type in the color index number in the **Color** field. The color index number should range from 1 to 255.

In the **True Color** tab, you have the option of selecting color using **R** (red), **G** (green), and **B** (blue) values. For example, you can use `42,139,187` as the RGB value in the **True Color** tab and it will select a shade of blue.

Using the **Color Books** tab, you can choose from many standard Pantone color books. You can also switch to color books such as DIC Color Guide and RAL Design. Most often, you will find yourself selecting colors from the **Index Color** tab.

Changing properties using the PROPERTIES palette

As you might have noticed, the properties of an object can be changed by selecting the object and then modifying the properties from the **Properties** panel, but that's not the only option for changing properties. Properties can be changed or modified from the **PROPERTIES** palette as well. In the following example, we will learn how to apply properties to a selected object using the **PROPERTIES** palette:

1. To open the **PROPERTIES** palette, select an object from the drawing area. Now, right-click and select **Properties** from the context menu or type `PR` and press *Enter* to open the **PROPERTIES** palette. In this case, I selected a door from the drawing and selected the **Properties** option from the context menu as shown in the following figure:

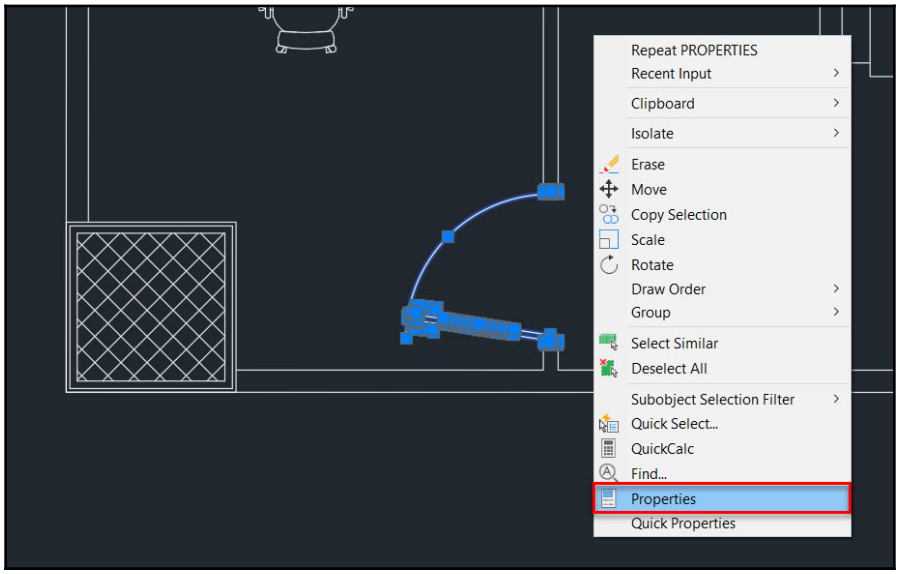


Fig 5.22: The Properties option in the context menu

2. The **PROPERTIES** palette, with properties such as **Color**, **Lineweight**, and **Linetype**, will show up as shown in the following figure:

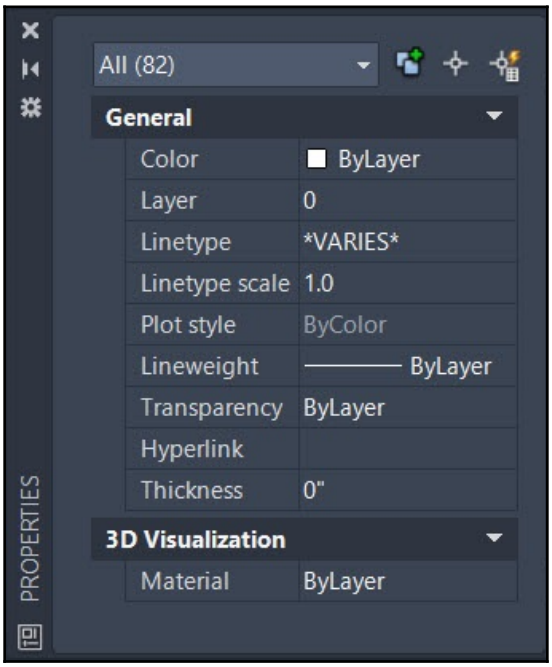


Fig 5.23: Color, Lineweight, Linetype, and other properties on the PROPERTIES palette

3. To modify or add any property on this palette, click on the field next to it and change the respective property. Here, we will add a color to the door using the **PROPERTIES** palette.
4. Click on the white box in the **Color** row under the **General** tab of the PROPERTIES palette and select a color from the dropdown.
5. Select red from the list or click on the **Select Color** option at the bottom of the list to select a color from the **Select Color** window.

6. Press the *Esc* key after selecting the color and the color will be applied to the selected objects. In this case, red will be applied to the door as shown in the following figure:

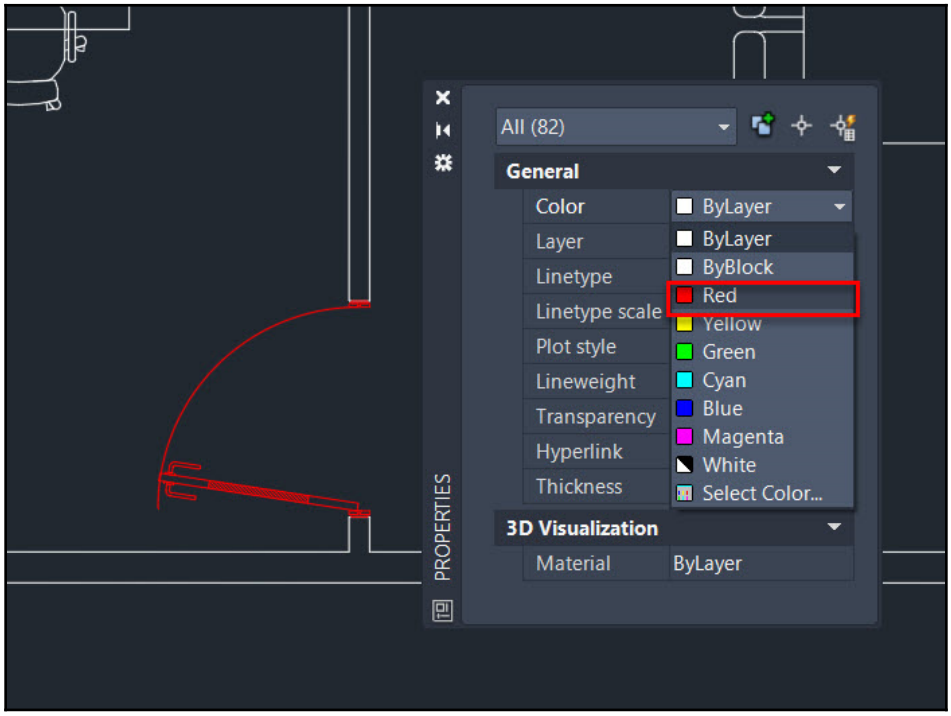


Fig 5.24: Red applied to the door using the PROPERTIES palette

In a similar way, you can change other properties too by selecting the respective field from the **PROPERTIES** palette; just make sure that you have the object selected to change the property.

Now you can see that adding properties can be a time-consuming thing to do if you want to add multiple properties for an object. But if you needed to transfer these properties to another set of objects, this could create issues. Fortunately, though, we have a tool that makes the transfer of properties a breeze. This tool is called **Match Properties** and it's discussed in the next section.

Using Match Properties

As you may have noticed, changing object properties is pretty simple and you can do it using the **PROPERTIES** palette. But, if you need to change the properties of multiple objects so that they all have the exact same set of properties, then doing it manually could cause errors and would certainly be a time-consuming thing to do.

So, in this case, you can use **Match Properties** to copy the properties of an object and transfer those properties to another object in the drawing. The **Match Properties** tool is found on the **Properties** panel and you can activate it using the icon shown in the following figure, or you can also use its command alias, `MA`:

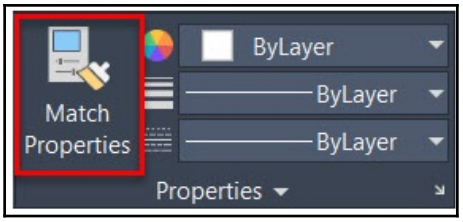


Fig 5.25: The Match Properties option on the Properties palette

Here, I will use a simple example of this circle, which is green, has the **HIDDEN** linetype, has a lineweight of `0.5`, and has a scale of `2` on the linetype. Now we can use **Match Properties** to transfer these properties to another set of objects– in this case to a rectangle and a circle, as shown in the following figure:

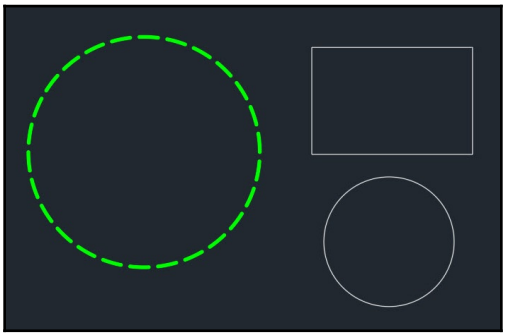


Fig 5.26: Applying the properties of one circle to other objects using the Match Properties tool

In the following example, we will transfer the properties of the larger circle to other geometries using the **Match Properties** tool:

1. Select **Match Properties** from the **Properties** panel or use its command alias, `MA`.
2. Click on the object whose properties you want to copy. In this case, click on the circle to which all the properties are applied.
3. Now a paintbrush icon will show up next to the cursor indicating that the properties are now copied and the cursor is ready to transfer these properties:



Fig 5.27: Paintbrush icon on the cursor when properties are copied

4. Click on the objects to which you want to transfer these properties. In this case, I will select another circle and a rectangle, and the properties will be transferred to those selected objects as shown in the following figure:



Fig 5.28: Properties transferred from a bigger circle to other objects in the drawing

Both objects, the small circle as well as the rectangle, will now have the same properties as the original circle.

In this case, the default properties are transferred from one object to another, but you can also control the properties that are transferred using the settings of the **Match Properties** tool, which is discussed next.

Modifying property types to match

To control the types of properties that are transferred from one object to another, you can modify the settings of **Match Properties** as shown in the following example:

1. To change these settings once again, select the **Match Properties** command and click on the object from where you want to copy the properties.
2. Now you will see the `Settings` option on the command line. Click on it or type `S` and press *Enter* to select the settings:

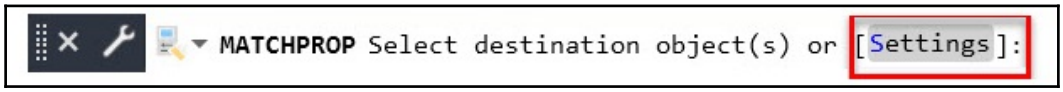


Fig 5.29: The Settings option of the Match Properties command

3. Now a list of properties will show in the **Property Settings** window. Check the properties from this list that you want to transfer and uncheck the properties that you don't and click **OK**:

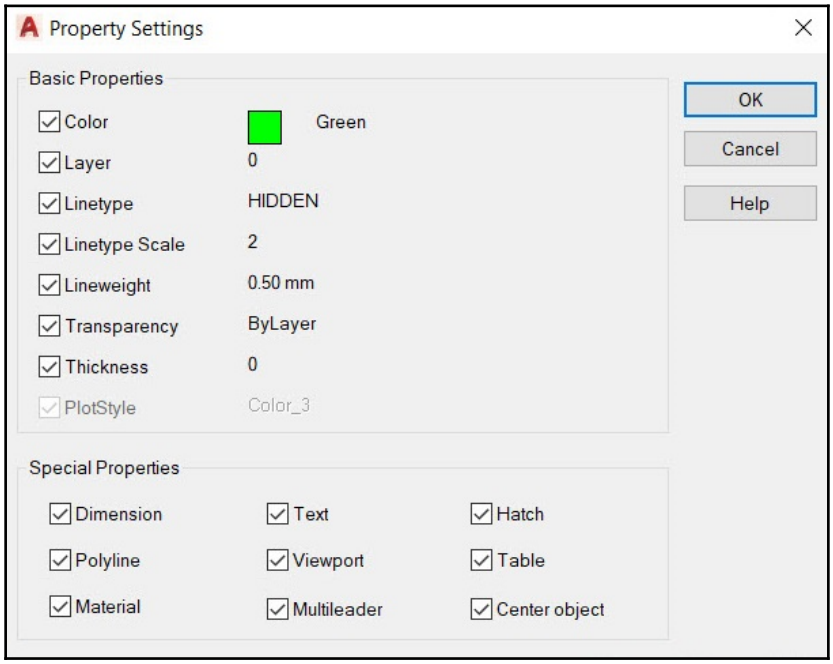


Fig 5.30: The Property Settings window of Match Properties

4. Now click on the objects to which you want to transfer these properties and only the properties you have selected in the **Property Settings** window will be transferred.

So, that was the **Match Properties** tool, which makes your drawing workflow quick and error-free as it can transfer lots of properties from one object to another with just a few clicks.

So, now that you know how to add properties to objects, we will move further to learn about checking the properties of objects using enquiry tools in the next section.

Using enquiry commands

Enquiry commands allow you to measure different geometrical properties in AutoCAD. The enquiry commands available in AutoCAD include **Distance**, **Radius**, **Angle**, **Area**, and **Volume**. The enquiry tools are on the **Utilities** panel of the **Home** tab, as shown in the following figure:

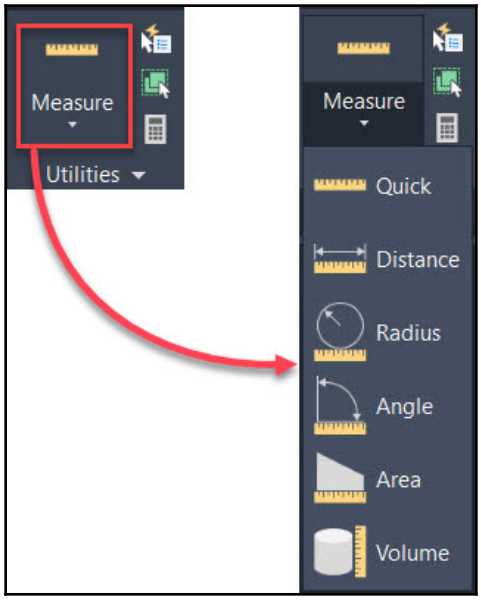


Fig 5.31: :Enquiry tools on the expanded Measure flyout of the Utilities panel

Click the arrow underneath **Measure** and a flyout with enquiry tools will show up. Let's see how we can use these enquiry tools to measure geometrical properties.

Quick

The first tool in this list, **Quick**, was added in the 2020 version of AutoCAD, and you can use it to measure dimensions quickly in a drawing. To use this tool, select it from the **Measure** flyout and move it in your drawing. As you move inside a room or closed area, it will show the length and width of the drawing area, as shown in the following figure:

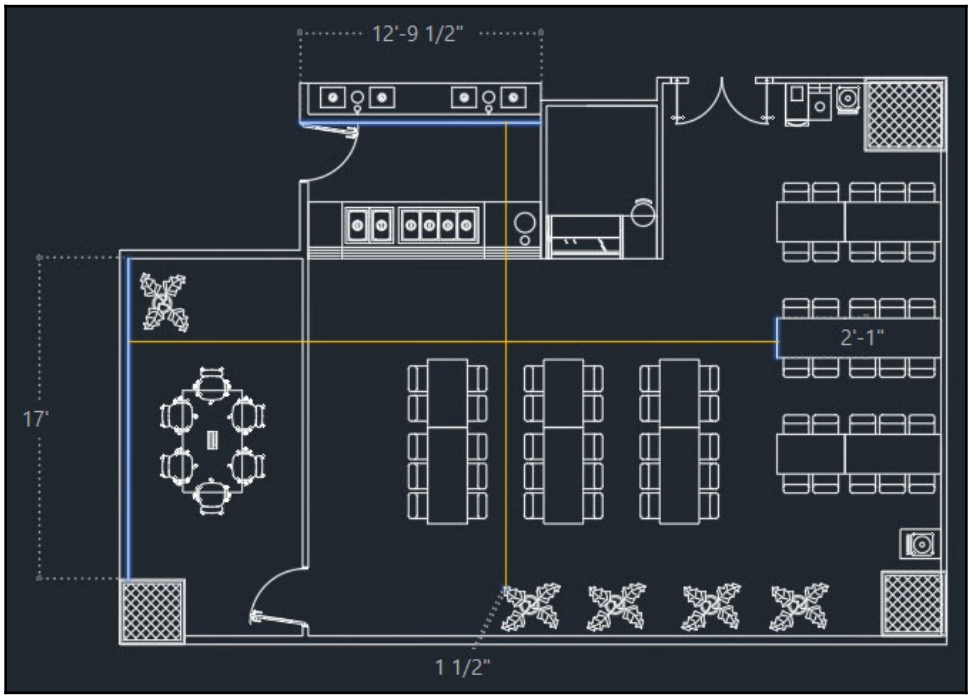


Fig 5.32: Distances between different walls visible when the Quick enquiry tool is used

These properties are not limited only to the length and width; when you move your cursor close to circular objects, it will show the radius of those objects and it will also show the angle of lines. In the 2021 version of AutoCAD, area calculation is also included with this **Quick** tool.

To find out the area of a closed space, just click and the area will show up on the tooltip. If you want to see the combined area of multiple regions, then click in a closed region, press and hold the *Shift* key, and click again in another closed area. The selected areas will be highlighted in green and you will see the combined area on the tooltip.

As mentioned earlier, the area feature of the **Quick** tool was added in the 2021 version of AutoCAD, so if you are working with the older version, you may not find this tool.

Distance

This enquiry tool can be used to measure the distance between two points. Select the **Distance** tool from the **Utilities** panel of the **Home** tab, as shown in the following figure:

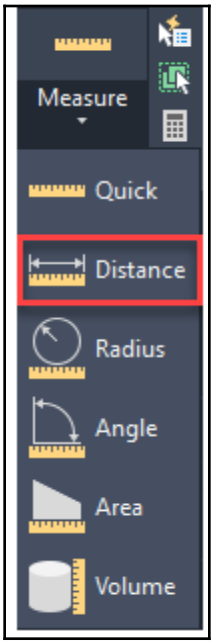


Fig 5.33: The Distance enquiry tool on the Measure flyout

Then, click at two different points in the drawing area and the shortest distance between those points will show up above the command line. To use this enquiry tool, you can also use its command alias, `DI`.

Radius

This enquiry command will measure the radius of circular objects such as circles, arcs, and fillets. To use this tool, select it from the list and then click on any circular object. The radius of the circular object along with the diameter will show up above the command line.

Angle

You can use this enquiry command to measure the angle between two intersecting lines. Select the **Angle** tool then click on two lines and you will see the angle between these lines above the command line.

Area

This is one the most useful enquiry commands and it is quite frequently used to measure the area in AutoCAD. You can start this enquiry command from the **Area** option of the **Measure** flyout or you can use its command, `AREA`. In the following drawing, we will find the area of the washroom with this enquiry tool using the following steps:

1. Start the **Area** tool from the **Measure** flyout or use its command, `AREA`.
2. Now the command line will prompt you to specify the first point. Click on one of the corners of the washroom and then keep clicking until you have selected all four walls, as shown in the following figure.

3. The selected area will show in green. To find the area, press *Enter* and the area will show up above the command line. The area will display in the current unit of the drawing template:

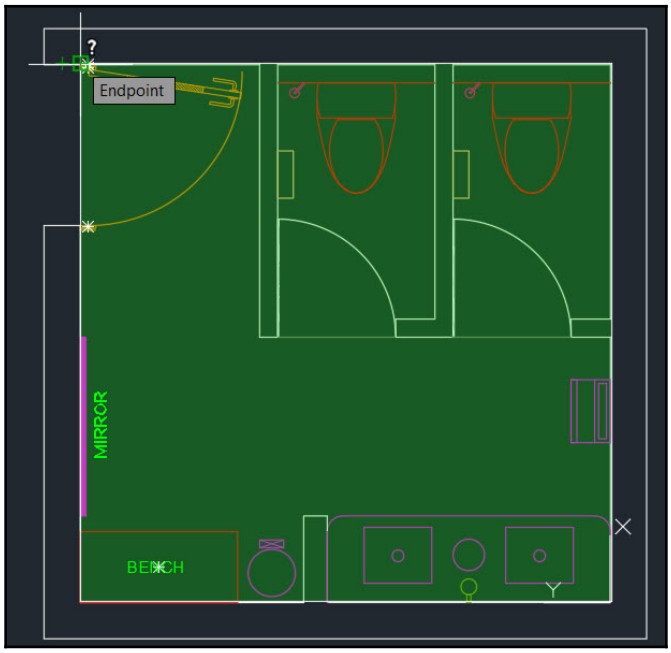


Fig 5.34: Selected area highlighted in the green box

As you may have noticed, this method was pretty straightforward because our example involved straight walls, but if you want to find the area of objects that are not made with straight lines, then you need to use a different method. In this case, we will find the area of the washing bay, which contains filleted edges, using the following method:

1. To find the area in question, select all the lines and arcs that make the boundary of the sink area and join them using the `JOIN` command to make a single polyline.
2. Start the `AREA` command and select the `Object` option from the command line, as shown in the following figure:

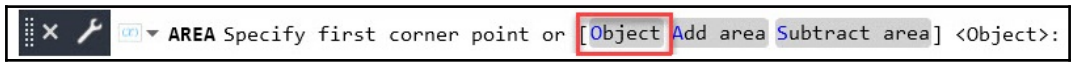


Fig 5.35: The Object option on the command line of the Area enquiry tool

3. Now click on the polyline you just joined in the previous step and the area of the polyline will show up above the command line:

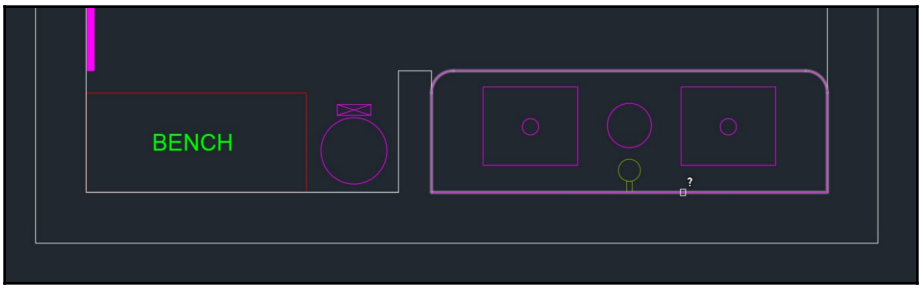


Fig 5.36: Polyline to be selected for calculating the area

In this case, the area shown above the command line is the complete area of the sink area, which includes the area of sinks and other objects inside the polyline. If, however, you want to remove some area from your calculation, then you can do that as well, as in the following example:

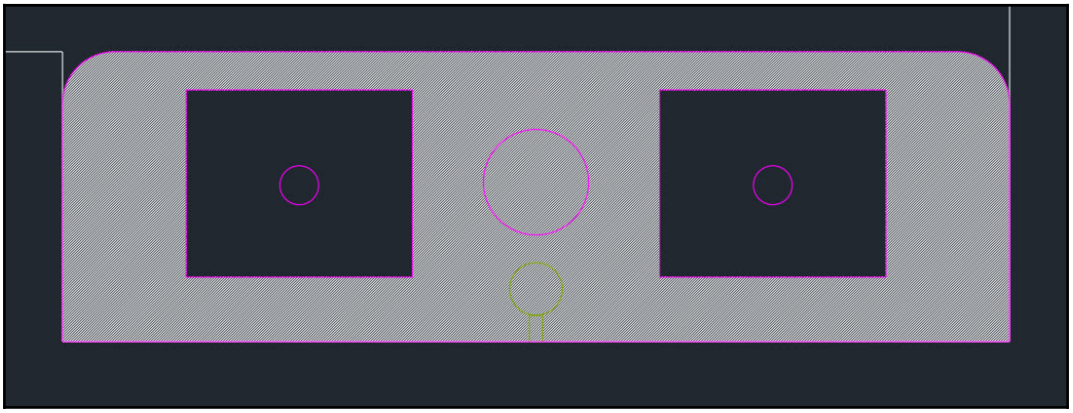


Fig 5.37: Area to be calculated shown by the hatch pattern

In the following example, we will once again calculate the sink area, but this time we will exclude the area occupied by sinks. So, essentially, we will find the area shown by the hatch pattern in the preceding figure:

1. To calculate this area, once again use the `AREA` command.
2. Select the `Add area` option from the command line and then click on the `Object` option in the command line.

3. Now click on the magenta-colored polyline marking the boundary of the sink area, then press *Enter* and you will notice that the entire sink area turns green, highlighting the selected area.
4. Now click on the `Subtract area` option from the command line and then the `Object` option from the command line.
5. Now click on the two rectangles inside the selected green region so that they turn brown, as shown in the following figure:

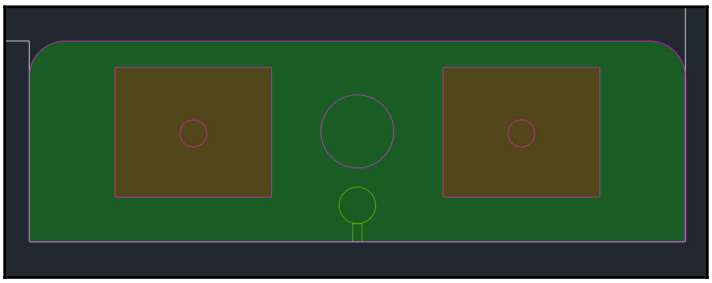


Fig 5.38: The removed area is shown in brown

6. Finally, press the *Enter* key and the area will show up above the command line.

In this case, the calculated area is the area shown in green and it does not include the area in brown. After having looked at the enquiry tools, it's time that we looked at the drawing management tool called layers. Layers is one of the most important features of AutoCAD drawings and we will discuss it in detail in the next section.

Drawing management using layers

Layers are really useful when you have added a great many objects to your drawing. Think of layers as sheets of tracing paper, which were pretty common in the hand-drafting days. We can draw different parts of a project on different sheets of tracing paper – you could have one sheet of tracing paper for the walls of a house, then another one for the internal furnishings of the rooms, and then another one for the dimensions. Then, we could overlap these sheets of tracing paper to get the complete house plan, and if we needed to show the drawings without any particular properties, such as dimensions, then we could simply remove the dimensions sheet from the project.

Layers in AutoCAD work in pretty much the same way, but in this case, it's obviously much more powerful and feature-packed than traditional tracing paper. In the drawing shown here, the different objects are placed on their respective layers, such as furniture on the furniture layer and doors on the door layer. Properties such as color are applied on the layers and all the objects on a particular layer take the property of the layer. For example, all the doors here are orange because orange is applied to the door layer:

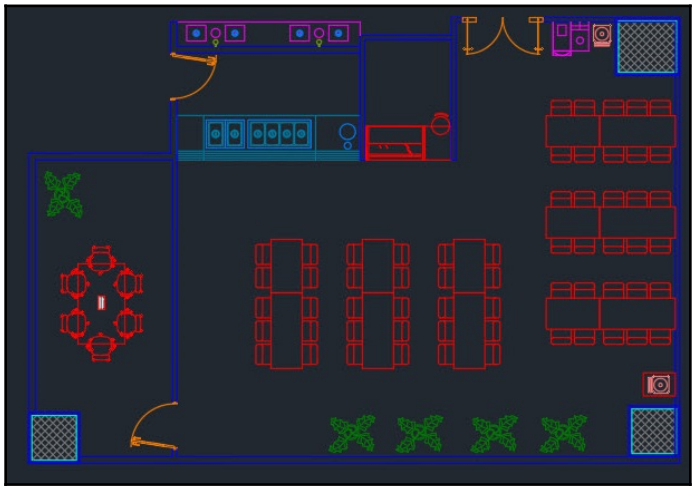


Fig 5.39: Sample drawing with multiple layers

In the following examples, we will learn how to make our own set of layers, apply properties to those layers, and assign objects to their respective layers.

Making layers

To make new layers in a drawing, open the **Layer Properties** palette, which is on the **Layers** panel of the **Home** tab, as shown in the following figure:

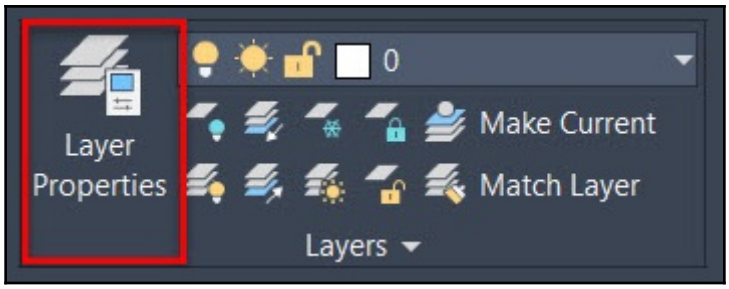


Fig 5.40: The Layer Properties option on the Layers panel

The **LAYER PROPERTIES MANAGER** palette will open up, which is just like any other palette of AutoCAD. You can resize this palette using the double-headed arrow, as shown in the following figure. You can close the palette using the close option and also hide it using the auto-hide icon. To move the palette around, grab the layer palette bar and then move it around:

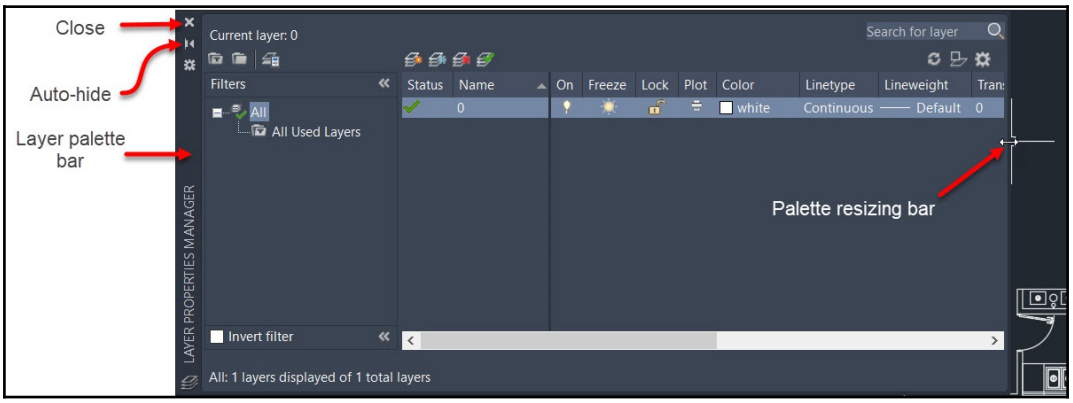


Fig 5.41: The LAYER PROPERTIES MANAGER palette

By default, the **LAYER PROPERTIES MANAGER** palette will contain only one layer, layer **0**, and all drawings are made on this layer. However, you can make as many layers as you want in your drawing. To make a new layer, click on the **New Layer** icon, as shown in the following figure and a new layer with `Layer 1` as the default name will be added in the **Layer Properties** palette. You can change the name of this layer to anything you want. To change the name of any existing layer, right-click on the layer name and select **Rename** from the context menu or click twice (slowly) on the layer name and the name editing field will show up. You can't change the name of the default layer, layer 0:

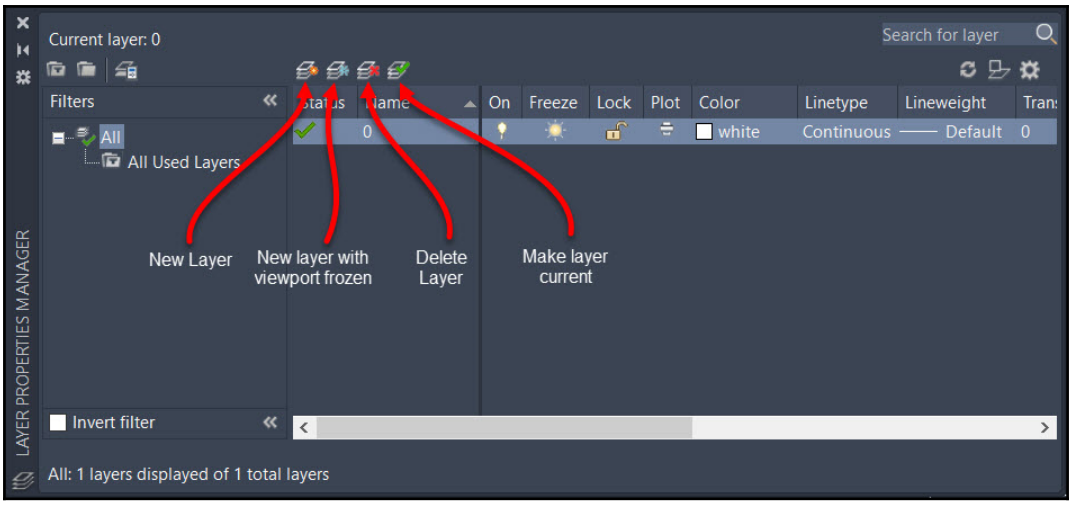


Fig 5.42: Layer options on the LAYER PROPERTIES MANAGER

In this example, we will add four layers in the **LAYER PROPERTIES MANAGER**, their names being `Walls`, `Furniture`, `Doors`, and `Plants`. The properties of the layer can be changed in the respective columns next to the layer name, such as **On**, **Freeze**, **Lock**, **Color**, **Linetype**, and so on.

To delete any layer, select it from the **LAYER PROPERTIES MANAGER** and then click on the **Delete Layer** icon as shown in the preceding figure. Not all layers can be deleted and when you try to delete any layer that can't be deleted, AutoCAD will show a warning message as shown in the following figure:

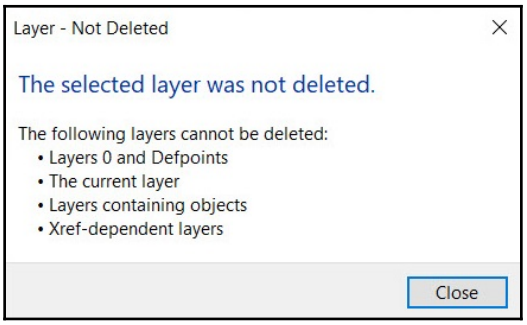


Fig 5.43: Layers that can't be deleted

As you can clearly see in the preceding figure, the list shows all the layers that can't be deleted. These layers include layer **0**, which is the default layer, and the **Defpoints** layer, which again is a default layer created in AutoCAD automatically when you add dimensions in a drawing. The current layer and the layer on which you have objects won't be deleted, and finally, layers used by external references (**Xref-dependent layers**) won't be deleted either. We will learn more about Xref in `Chapter 9`, *External References and Dynamic Blocks*.

Adding color to layers

We will change some properties of our layers, starting with the color. To change the color, click on the white box in the **Color** column next to the layer name. The **Select Color** window will show up, where you can select the colors for your layers:

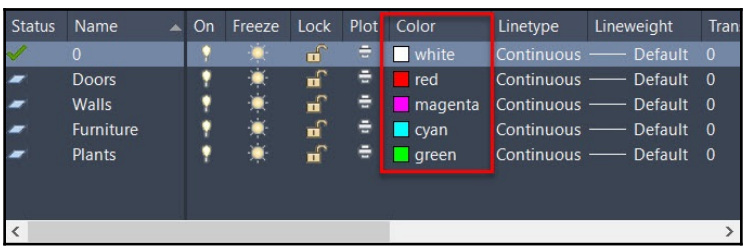


Fig 5.44: Color options for layers on the LAYER PROPERTIES MANAGER palette

In this case, I will select **red** for **Doors**, **magenta** for **Walls**, **cyan** for **Furniture**, and **green** for **Plants**. After applying the colors, the layers in the **LAYER PROPERTIES MANAGER** palette will look as shown in the preceding figure. Just like colors, you can add other properties to the layers as well, which we will learn about in the next section.

Adding linetype, lineweight, and transparency settings

After adding colors, let's configure linetype and lineweight for the layers. Here, we will add lineweight to the **Walls** layer. Click on the **Lineweight** column next to the **Walls** layer and a lineweight menu will show up. Select `0.50` units of lineweight from the **Lineweight** menu and click **OK**. Now the **Walls** layer has a linetype of `0.50` units.

In a similar way, click on the **Transparency** column for the **Furniture** layer and a **Layer Transparency** palette will show up. Select a transparency value from this list. The transparency values range between 0 and 90, where `90` is the maximum transparency you can apply. In this case, I have selected a transparency value of `80`. After applying these properties, the **LAYER PROPERTIES MANAGER** palette looked like the following figure in my case:

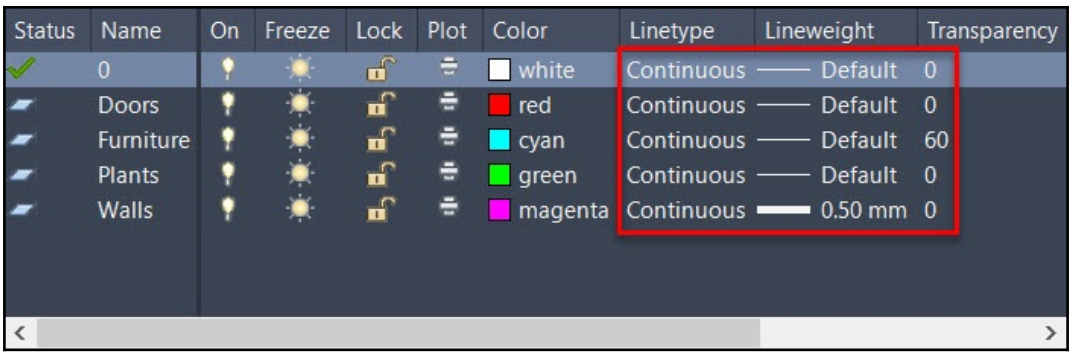


Fig 5.45: The Linetype, Lineweight, and Transparency options applied on different layers

Just like for lineweight and transparency, you can also add linetype settings to the layer using the **Linetype** column next to your selected layer. For linetype, click on the text that says **Continuous** in the **Linetype** column and then click on the **Load ...** button in the **Select Linetype** window. Select a linetype from the list of linetypes and click **OK**.

Assigning objects to layers

Now that we have prepared layers and assigned properties to them, it's time that we added objects to the layers. To do this, first close the **LAYER PROPERTIES MANAGER** palette. Now select all the doors in the drawing and click on the layer drop-down menu in the **Layers** panel and select the **Doors** layer from the list as shown in the following figure. Press the *Esc* key and now you will notice that all doors will turn red, indicating that they are now on the **Doors** layer and hence are taking properties from that layer:



Fig 5.46: The Doors layer with red applied to it

Again, select all the walls and then click on the layer drop-down menu from the **Layers** panel and select the **Walls** layer. The color of the walls will change to magenta, which is the color of the **Walls** layer. In a similar way, place **Furniture** and **Plants** from the drawing on their respective layers. After assigning all the objects to their respective layers, this is what my drawing looked like:

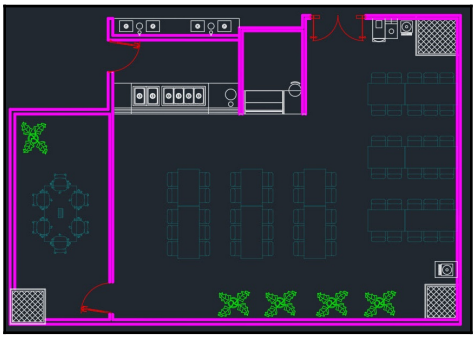


Fig 5.47: Objects put on their respective layers

As you can see, in this drawing, all the settings are applied to the respective objects, such as the walls, which now have `0.50` lineweight, and the furniture has a transparency value of `80`.

What if lineweight and transparency changes are not visible in the drawing?

Even after assigning the layers containing lineweight and transparency to objects, sometimes you will not see any of these settings taking effect in the drawing. This happens when the lineweight and transparency status bar options are not active:



Fig 5.48: Lineweight and transparency status bar toggles

To make these properties visible in the drawing, activate the lineweight and transparency status bar options from the status bar as shown in the preceding figure.

Making a drawing on an active layer

Now that you know how to assign layers to existing objects in the drawing area, let's talk about making objects directly on a layer. AutoCAD makes drawings on the current layer and the current layer is visible in the layer dropdown of the **Layers** panel, as shown in the following figure:

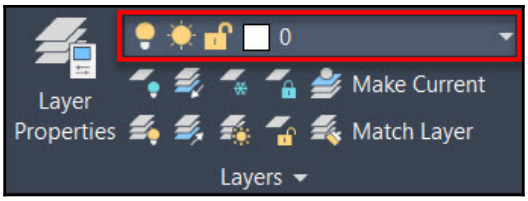


Fig 5.49: Current layer visible in the layer dropdown menu of the Layers panel

To make a layer current, select it from the layer drop-down list of the **Layers** panel. You can also make a layer current by double-clicking on the layer icon from the **LAYER PROPERTIES MANAGER** palette. You can also select a layer from **LAYER PROPERTIES MANAGER** palette and click on the Make Current icon at the top of **LAYER PROPERTIES MANAGER** palette to make that layer current, as shown in the following figure:

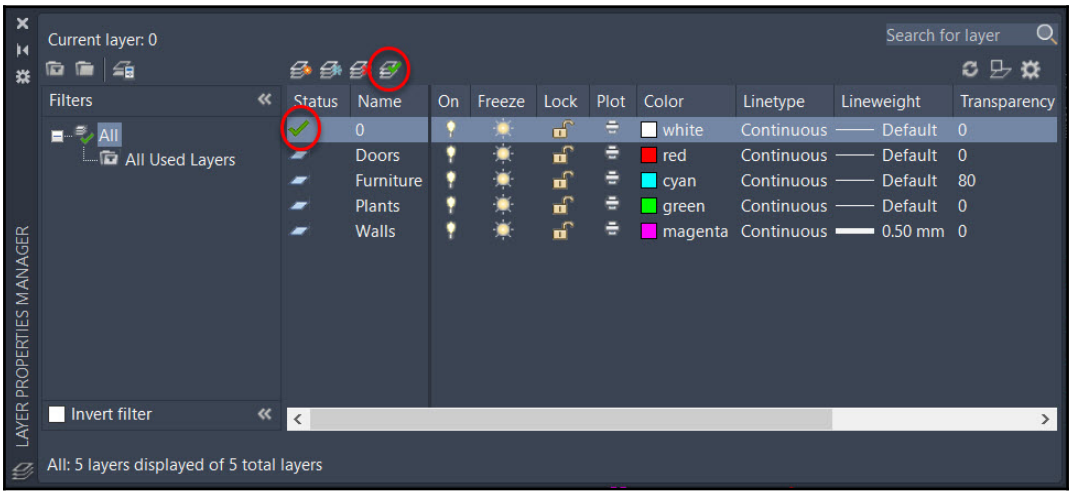


Fig 5.50: The Make Current option and the active layer on the Layer Properties palette

Once you have selected your current layer, start making a drawing and the drawing will be directly made on the current layer and will inherit properties from the current layer.

Now consider a scenario where you make a drawing on the **Doors** layer (which is red in color) or assign the **Doors** layer to any object, and then you select the object again and change its color to something else using the **Properties** panel. In this case, the color assigned using the **PROPERTIES** palette or the **Properties** panel will take precedence.

In the drawing here, I have assigned the sinks to the **Doors** layer and hence they turn red in color due to the color of the layer. Now select the sink on the left and then click on the color menu of the **Properties** panel of the **Home** tab and select yellow for it. You will notice that the color of the sink will change to yellow even though the object is still on the red layer, as shown in the following figure:

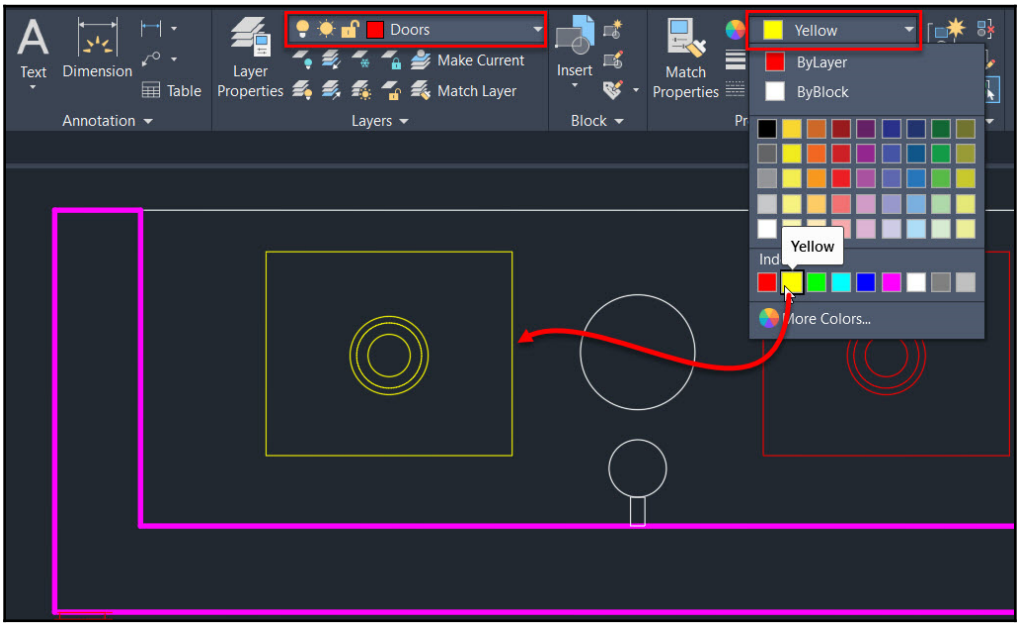


Fig 5.51: Yellow applied to an object using the Properties panel

The same applies for other properties too. In this case, if you once again want an object to take the properties of a layer, then select the object and, from the **Properties** panel, select **ByLayer** in the respective property field. In this case, changing the color from yellow to **ByLayer** will once again make the sink red in color, which is the color of the layer:

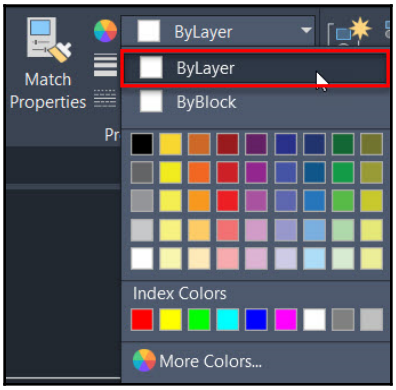


Fig 5.52: The ByLayer option on the Properties panel

Now that you know how to make a drawing on an active layer, let's talk about another set of frequently used layer settings: **On**, **Off**, and **Freeze**.

On, Off, and Freeze

Using the **On** and **Off** options, you can hide objects on selected layers. The **On**, **Off**, and **Freeze** options are in the layer dropdown menu of the **Layers** panel and also on the **LAYER PROPERTIES MANAGER** palette.

In the drawing that we are using in this section, we have the **Doors**, **Furniture**, `Walls`, and **Plants** layers. I will use this drawing to explain this topic. Click on the layer dropdown menu on the **Layers** panel and click on the light bulb icon next to the **Doors** layer. You will notice that the light bulb icon will turn gray and all doors from the drawing on that layer will disappear. The doors are still in the drawing and when you click on the light bulb icon next to the **Doors** layer, the doors will once again show up in the drawing area:

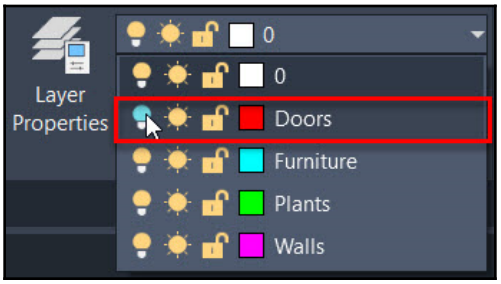


Fig 5.53: The Doors layer turned off using the light bulb icon

Alternatively, you can also turn the light bulb off from **LAYER PROPERTIES MANAGER** palette, as shown in the following figure:

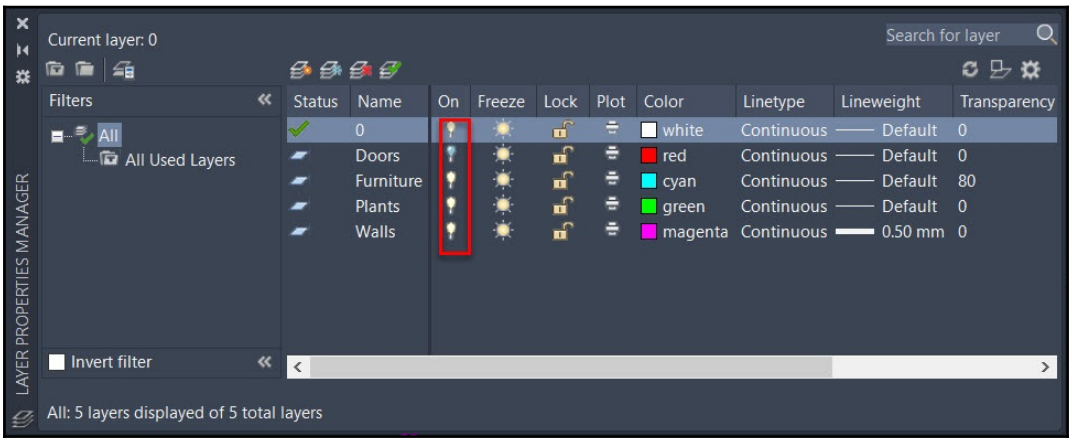


Fig 5.54: Light bulb icon for turning layers on or off on the LAYER PROPERTIES MANAGER palette

To bring back doors, click on the gray light bulb icon next to the **Doors** layer once again and the objects on the layer will show up. When you click the light bulb icon on the active layer, a warning message will show up (see the following figure). You can choose to keep the layer on or turn it off despite it being the current layer:

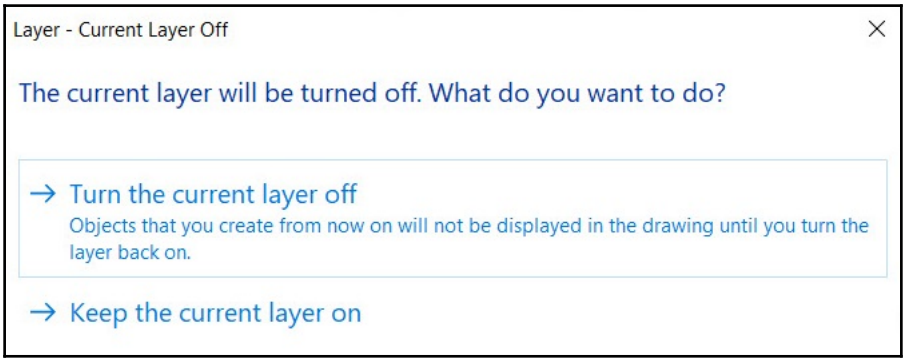


Fig 5.55: Turning off the current layer – warning message

The icon next to the light bulb icon that looks like the sun is for freezing a layer. Click on the freeze icon next to the **Furniture** layer and the icon will turn into a snowflake icon and all the objects on the **Furniture** layer will also disappear from the drawing area. Although visibly the **On**/**Off** and **Freeze** options seem the same, they work differently:



Fig 5.56: Furniture layer frozen in the layer drop-down menu

When you freeze a layer, AutoCAD removes it from memory as well and the layer won't be refreshed when you regenerate your drawing. Freezing a layer is especially helpful in drawings containing large datasets, where you can freeze layers that are currently not in use; this improves the performance of the drawing as AutoCAD no longer has to refresh data from layers that are frozen.

To thaw or unfreeze a layer, click on the snowflake icon in the freeze column and the objects on that layer will once again show up. The current layer will not freeze even when you try to freeze it and a pop-up message will show up when you try to freeze the current layer.

Layer lock

To make objects on any layer non-editable, you can lock the layer. The lock option on the layer is right next to the freeze option on the layer drop-down menu as well as on the **LAYER PROPERTIES MANAGER** palette.

To lock a layer, click on the padlock icon next to the layer that you want to lock and the layer will be locked and the transparency of objects on that layer will turn to 50%. This transparency is a visual cue that can be used to see the locked layers in a drawing. Also, when you hover your cursor over objects on a locked layer, a lock icon will show next to your cursor, as shown in the following figure:

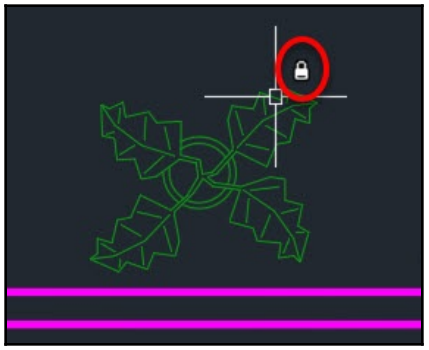


Fig 5.57: The padlock icon showing next to the cursor on locked layer objects

To change the transparency of the locked layer, click to expand the **Layers** panel and then move the slider for the **Locked layer fading** option to set the transparency value for the locked layer, as shown in the following figure:

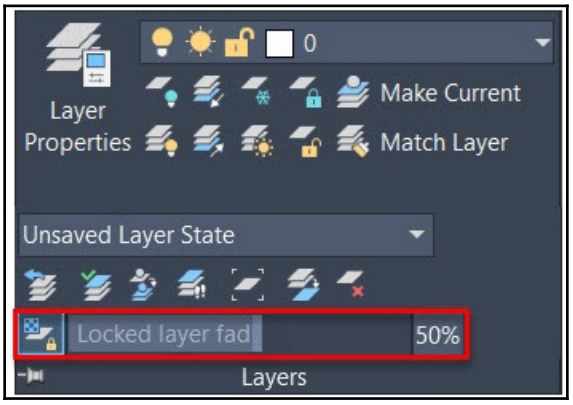


Fig 5.58: Locked layer fading percentage in the expanded Layers panel

The objects on a locked layer can't be modified or moved; however, you can take reference of the snap points of the objects on a locked layer.

To unlock the layer, click on the padlock icon next to the locked layer and the layer will be unlocked and the objects on the layer will once again become editable.

No-plot layers

If you don't want a layer to show up in your final prints, then there is an option for that as well on the **LAYER PROPERTIES MANAGER** palette. Click on the **Plot** icon next to the name of the layer. The **Plot** icon will also now show a rounded red cross and the layer will become a no-plot layer, meaning objects on the layer will no longer show when printed even though they will still show up in the drawing area:

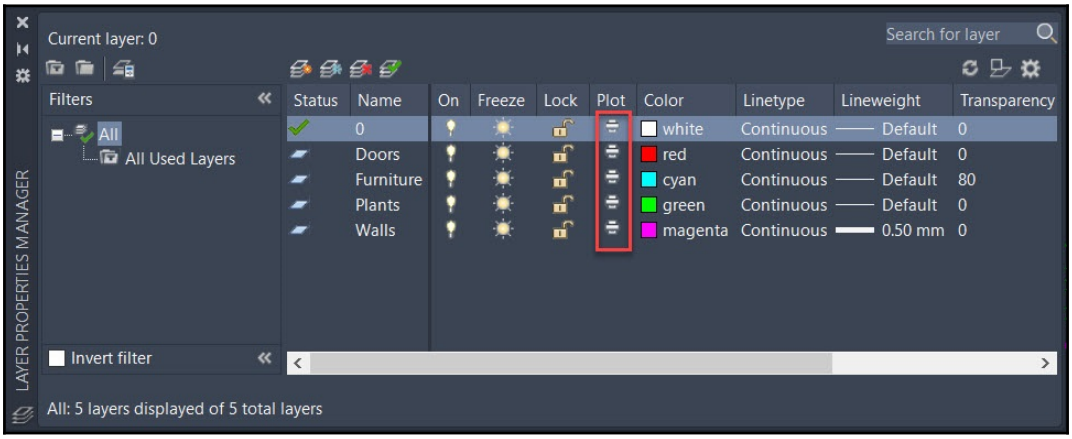


Fig 5.59: The Plot option on the Layer Properties manager palette

The **Defpoints** layer that we saw in earlier examples is set to "**no plot**" by default, and you can't change it back to a "**plot**" layer. Besides the standard the **LAYER PROPERTIES MANAGER** palette, you can also control the layer status using the **Layers** panel, which is a quicker way of managing layer status in drawings containing tens or hundreds of layers. We will discuss these quick access tools in the next section.

Using the quick access tools in the Layers panel

In our sample drawing, the list of layers is pretty small; in fact, there are just five layers including layer 0. However, most of the drawings that you will see in the workplace will probably contain tens or hundreds of layers, and managing a list that big is not easy. To manage such a big list, you can use the quick access tools on the **Layers** panel as shown in the following figure:

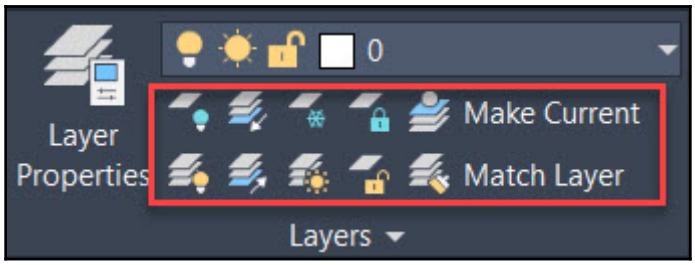


Fig 5.60: Quick access layer tools on the Layers panel

In the following examples, we will learn how to manage layer properties using the quick access tools of the **Layers** panel.

Layer off

The first icon in the first row, which looks like a layer with a blue light bulb icon, is for the layer off command. This tool will let you turn off a layer by selecting any object of that layer from the drawing area:

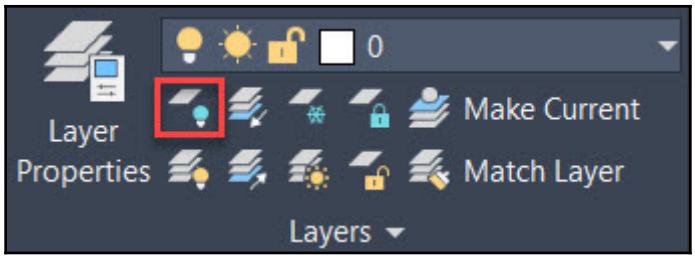


Fig 5.61: The layer off option on the Layers panel

To use this tool, select it from the **Layers** panel or use its command alias, `LAYOFF`, and then click on an object from the drawing area. The layer of your selected object will turn off and all the objects on that layer will disappear from the drawing too. You can select multiple objects from the drawing area too, and as soon as you click on an object, the layer of that object will turn off.

To turn all layers on, click on the icon that looks like three layers with a yellow light bulb right underneath the layer off icon in the second row of the **Layers** panel. As soon as you click on the icon, all the layers will turn on and all objects on the respective layers will also start to show up.

Layer isolate

The next icon, which looks like three layers with an arrowhead pointing downward, is for the layer isolate command. This command will let you hide all the layers of the drawing except your selected layers:

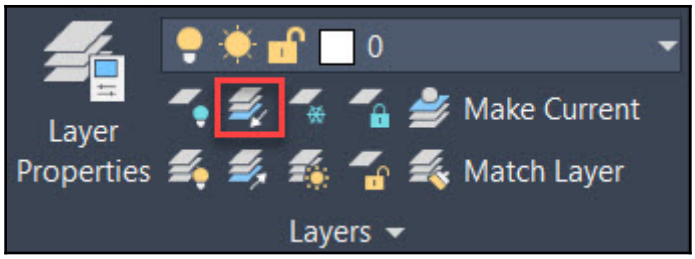


Fig 5.62: The layer isolate option on the Layers panel

To use this tool, click on its icon or use its command alias, `LAYISO`. Now click on the objects of the layer that you want to isolate and, once you have made your selection, press *Enter*. All the layers in the drawing will be hidden except your selected layers and the objects on the selected layers.

As an example, for this drawing, I will click on one of the walls and a door and press *Enter*. The final result in my case looks like the drawing shown here:

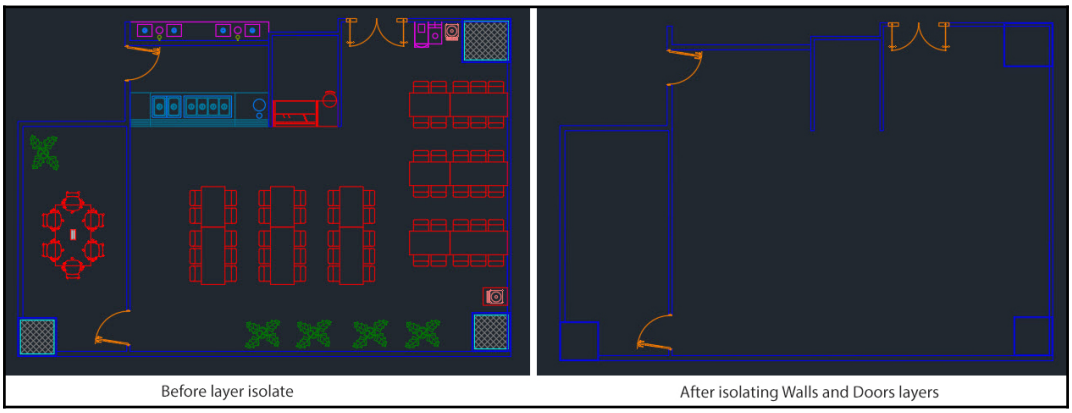


Fig 5.63: Layers isolated using the layer isolate option

To "de-isolate" the layers and bring all layers back into the drawing, just select the icon right underneath the first icon on the second row of the **Layers** panel. When you click this icon, all the layers will turn on and objects on those layers will show up again in the drawing.

Layer freeze

The next icon, which looks like a layer with a snowflake icon, is the layer freeze tool, and as the name suggests, you can freeze layers using this tool. Select the layer freeze icon from the **Layers** panel or use its command alias, `LAYFRZ`:

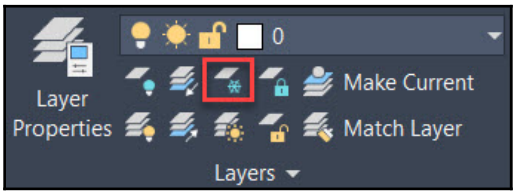


Fig 5.64: The layer freeze option on the Layers panel

Select an object from the drawing area and the layer will freeze and all the objects of that layer will disappear too. You can select multiple objects from different layers and all the layers will freeze. To thaw all the layers, click on the icon right underneath the layer freeze icon, which looks like three layers with a yellow sun.

Layer lock

This tool is next on the list and as the name suggests, this will lock the layers of selected objects. The icon of the layer lock tool looks like a layer with a blue padlock:

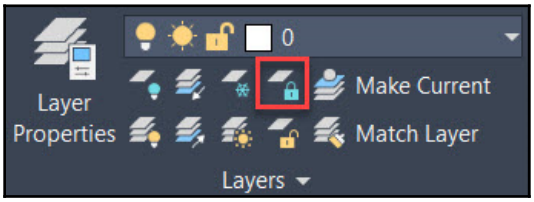


Fig 5.65: Layer lock option on the Layers panel

To lock a layer, select the tool from the **Layers** panel or use its command alias, `LAYLCK`. Once you have the command active, select an object from the layer that you want to lock and the layer will be locked. To unlock the locked layers, just click on the layer unlock icon, which is right underneath the layer lock icon, and click on any object of the locked layer and the layer will be unlocked.

The next option in the **Layers** panel, **Make Current**, will make your selected layer the current layer from the list of layers:

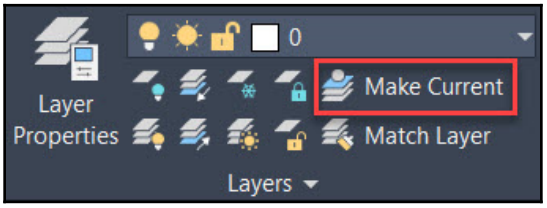


Fig 5.66: The Make Current option on the Layers panel

To use this tool, just select it from the **Layers** panel as shown in the preceding figure or use its command alias, `LAYMCUR`, and then click on an object of the layer that you want to make current.

Match Layer

Using the **Match Layer** tool, you can change the layer of selected objects to that of any other object from a drawing. To use this tool, select it from the **Layers** panel or use its command alias, `LAYMCH`. In this drawing, we will change the layer of the water dispenser unit, which is currently on layer **0**, to the **Furniture** layer. The water dispenser unit is shown in the yellow oval in the figure here:

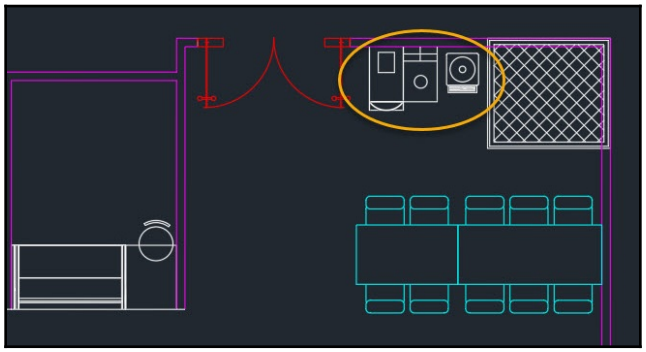


Fig 5.67: The water dispenser unit is shown in the yellow oval

After selecting the **Match Layer** tool, select the water dispenser unit from the drawing area and press *Enter*. Now click on any object of the layer on which you want to put the selected object. Click on the furniture right next to the water dispenser unit, which is **cyan** in color. The color of the water dispenser unit will change to that layer and it will be assigned to the **Furniture** layer's properties too, as shown in the following figure:

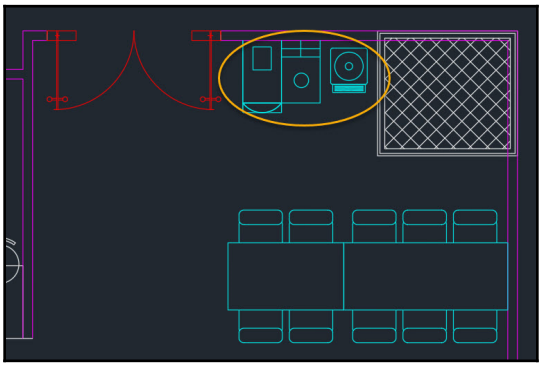


Fig 5.68: Water dispenser assigned to the Furniture layer

So, now that you know about using quick access layer tools to manage large lists of layers directly from the drawing area, let's move on to learn about another layer management tool, called the layer states manager, which helps in creating groups of layers with similar sets of properties.

The layer states manager

Imagine that you have lots of layers in a drawing and you need to show your drawing in various states, such as a state in which only walls and doors are visible, another state in which only walls and dimensions are visible, and so on. To get these states, you can hide the layers that you don't want and keep only required layers active. But doing this again and again would become repetitive and would take time every time you had to change the state of a layer. To make the states of these layers permanent, you can use the layer states manager, which is found in the expanded **Layers** panel as shown in the following figure:

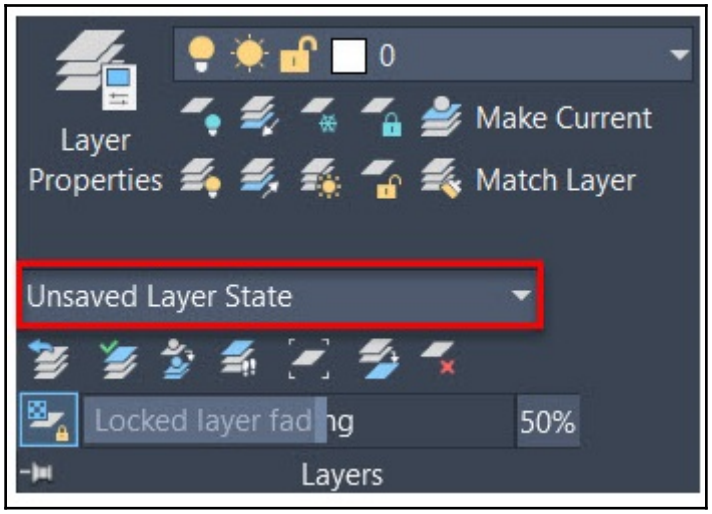


Fig 5.69: Layer states drop-down menu on the expanded Layers panel

In our example drawing containing five layers, we will create three layer states – a state where only walls and doors are visible, another one where only walls and furniture are visible, and another one where all the layers are visible:

1. To start with this tool, we will first switch all layers on so that all the objects are visible in the drawing area.
2. Now click on the **Unsaved Layer State** dropdown as shown in the preceding figure.
3. Now click on the **New layer state** option that shows up in the expanded panel. A new window will open up where you can add a name for this layer state. Let's call this layer state `All active` and then click on **OK** to save this layer state:

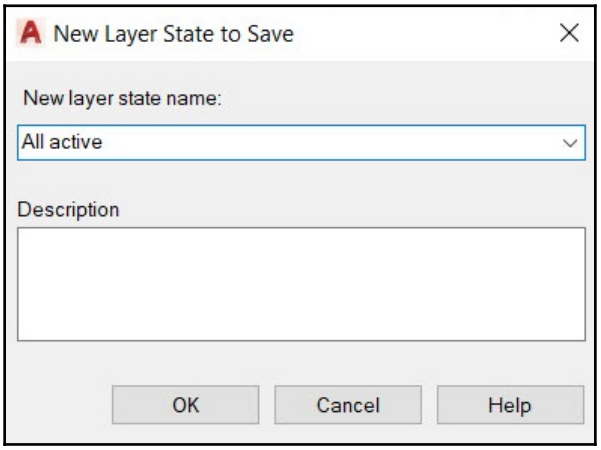


Fig 5.70: The All active layer state created with the layer states manager

4. Now turn off all the layers in the drawing except the **Walls** and **Doors** layers and then again go to the **New layer state** option.
5. Give your layer state a name in the next window and click **OK**. In this case, I will call this layer state `wall-door on`.
6. Once again, a new layer state will be created with your selected layers active.
7. Finally, turn off all layers except **Walls** and **Furniture** and then create a new layer state as explained in the preceding steps.

All the layer states that you make will be added to the layer states manager and you can then toggle between different layer states using the layer states manager panel, as shown in the following figure:

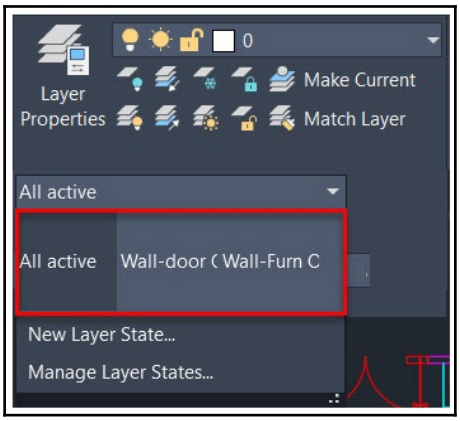


Fig 5.71: All the layer states on the layer states drop-down menu

By toggling the layer states, you can easily and quickly switch between different states of layers.

So, with our coverage of the layer states manager, we have finished this chapter, and now, before we move on to the next chapter, let's quickly summarize the topics we have learned about.

Summary

In this chapter, we learned about adding details to our drawings using properties such as linetype, lineweight, color, and transparency. We also learned how to apply these properties on a drawing level and on a layer level. We also learned about classifying drawing objects into layers and controlling their collective properties using layer tools. The tools related to managing lists of layers and layer states, and quick access tools on the **Layers** panel were also discussed.

In the next chapter, we will learn how to make symbols, often called blocks, in a drawing, and we will also learn how to use this reusable content. We will also learn how to add dimensions, text, and other annotations to our drawings, which will ultimately make our drawings more readable.

6
Working with Hatches, Text, and Dimensions

A drawing is meaningless without annotations such as text and dimensions. So far, we have learned how to create drawings and manage large datasets using different drawing management tools, but we have not yet looked at any dimensioning tools. In this chapter, we will learn how to add annotations such as dimensions, text, and multileaders to our drawing, which make our drawing relevant. Along with annotations, we will also learn how to make Hatch patterns and colored gradients in our drawings.

By the end of this chapter, you will be able to add dimensions, text, and multileaders to any drawings, and you will have also learned how to modify them as per the drawing standards and specific requirements.

The following are the topics discussed in this chapter:

- Filling an area with Hatches
- Adding color gradients
- Adding text to a drawing
- Working with dimensions
- Working with multileaders

So, let's begin this chapter by creating Hatches in our drawing.

Filling an area with Hatches

Using Hatches, you can add patterns in a closed area of a drawing. To understand the **Hatch** tool, I will use this sample drawing:

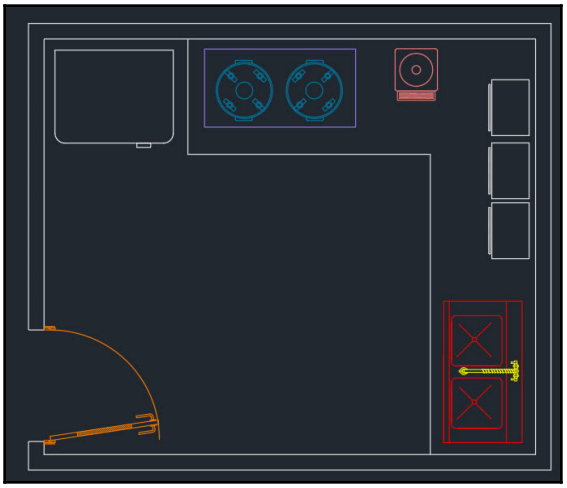


Fig 6.1: Sample drawing to be used for the Hatch tool

The **Hatch** tool is on the **Draw** panel of the **Home** tab, as shown in the following screenshot. In this case, I will apply Hatch patterns to the wall of the kitchen, the slab, and other places:

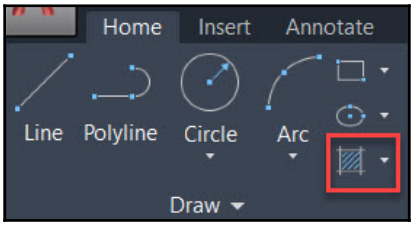


Fig 6.2: The Hatch tool on the Draw panel of the Home tab

To start the `Hatch` command, click on the **Hatch** tool from the **Draw** panel or use its command alias, `H`. Now, the command line will prompt you to specify a pick point and you will also see a temporary **Hatch Creation** tab, as shown in the following screenshot:

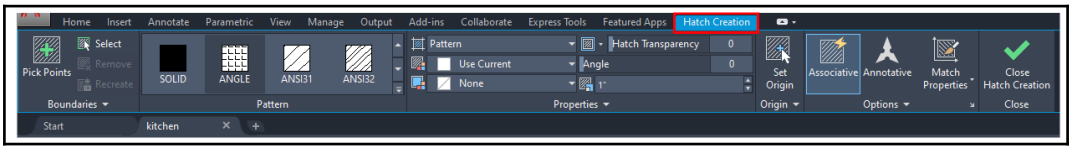


Fig 6.3: The Hatch Creation tab with Hatch tools

All the properties of the Hatch can be controlled with this **Hatch Creation** tab. The first panel on this tab is the **Boundaries** panel with two highlighted options, **Pick Points** and **Select**. The **Pick Point** option is selected by default and this option allows you to select a point in an area and then that enclosed area is selected. First, let's understand how pick point selection works and then we will look at the **Select** option in the next section.

Hatches with pick points

With the `Hatch` command active, click at a point inside the walls, as shown in the following screenshot:

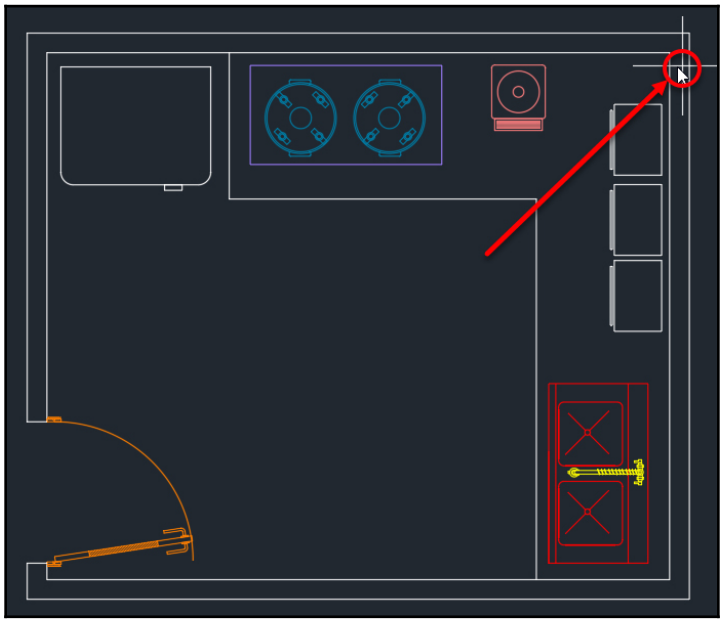


Fig 6.4: Point inside the walls selected to create the Hatch pattern

You will notice that the insides of the walls are selected, and a preview will show up with the default Hatch pattern. This type of selection method is called island detection, where the `Hatch` command detects boundaries around the point where you click. Once you have a Hatch area selected, you can select the Hatch pattern.

All the predefined Hatch patterns are on the next panel—**Pattern**. Click the down-pointing arrow on the bottom right of the **Pattern** panel and the panel will expand to show the list of Hatch patterns:

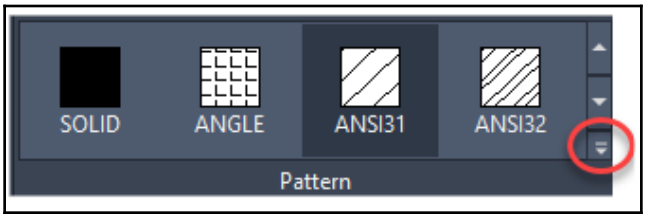


Fig 6.5: The down-pointing arrow that expands the Hatch pattern panel

Click on any Hatch pattern from this list. In this case, I am selecting the **ANSI31** hatch pattern from the pattern list. Now click at any point inside the walls and the **ANSI31** pattern will be applied in it. If the Hatch pattern does not look right within the walls, then change the scale of the Hatch pattern to increase or decrease the gap between the **ANSI31** Hatch pattern lines. The scale feature is on the **Properties** panel of the **Hatch Creation** tab, as shown in the following screenshot:

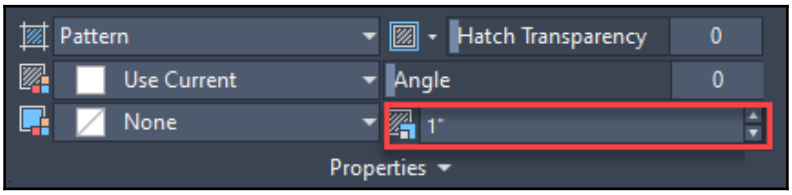


Fig 6.6: The Hatch scale field on the Properties panel

Increasing the scale will increase the gap between pattern lines and decreasing it will decrease the gap. In my drawing, I will keep the scale as **1** as it looks fine on the drawing:

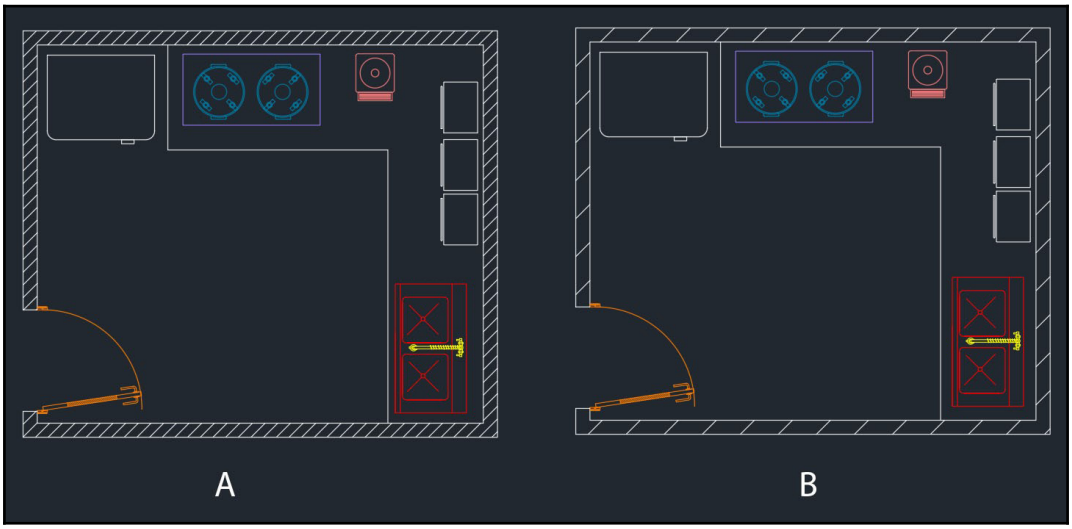


Fig 6.7: Hatch pattern ANSI31 applied with a scale of 1 in drawing A and with a scale of 3 in drawing B

Once you are done making this Hatch pattern, click the **Close Hatch Creation** green checkmark to apply the Hatch settings and close the **Hatch Creation** tab. My final Hatch pattern looked like drawing **A** when added with a scale of `1`; drawing **B** is the Hatch pattern with a scale of `3`.

So, now you are familiar with the way pick points work, let's look at how the **Select** option works in Hatch creation.

Hatches with the Select option

Start the `Hatch` command and, this time, click on the **Select** option from the **Boundaries** panel of the **Hatch Creation** tab. You will notice that, now, the cursor will turn into a pick box instead of a point selection cursor, and you can now select a boundary instead of an "island." In this case, click on the boundary of one of the cabinets next to the sink and it will be selected. You can also make multiple selections for the Hatch.

So, select all three cabinet boundaries so that they are included in the selection set. If you want to remove any selected boundary from your selection set, then you can click on the **Remove** icon on the **Boundaries** panel and select the boundary of the object that you want to remove.

Once you have objects selected, just go to the **Patterns** panel and select the pattern that you want to apply. In this case, I will select the first pattern, **Solid**, which will apply a solid color to the cabinets, as shown in the following screenshot:

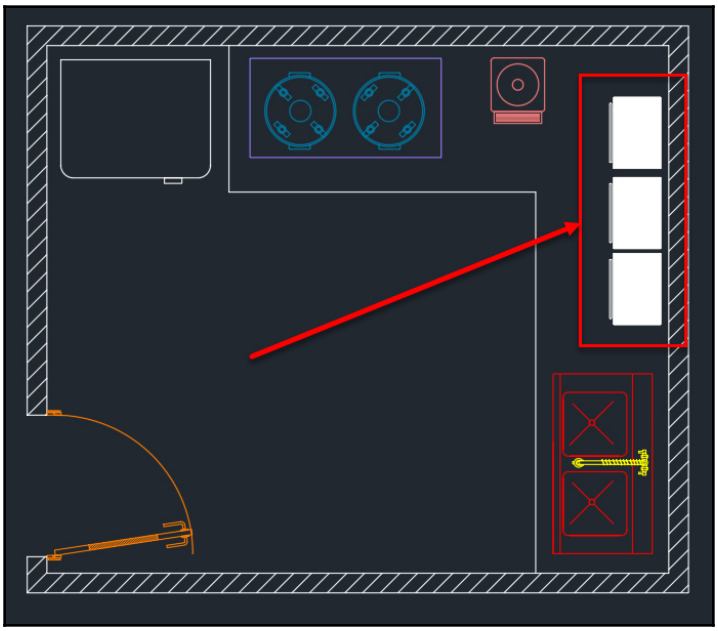


Fig 6.8: The Solid Hatch pattern added to the wall cabinets

So, now you are familiar with the basic method of adding Hatches, let's talk about different Hatch properties and options, and we will start with Hatch color in the next section.

Hatch color

You can apply different colors to Hatches using the Hatch color option. To begin with, start the `Hatch` command and select the **Pick Points** tool if it's not already selected, from the selection panel of the **Hatch Creation** tab. Now, click at a point inside the kitchen slab boundaries and select the **ANSI37** Hatch pattern.

Now, in the **Properties** panel, you will see two drop-down menus for color, as shown in the following screenshot:

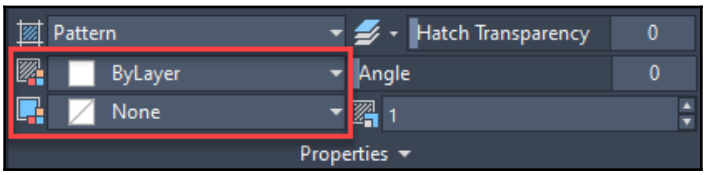


Fig 6.9: Hatch color and background color drop-down menus

The first menu, at the top, is for the color of the Hatch and the second menu, at the bottom, is for the color of the Hatch background. Select the color black for the Hatch color and the eighth color from the left for the background color, as shown in the following screenshot:

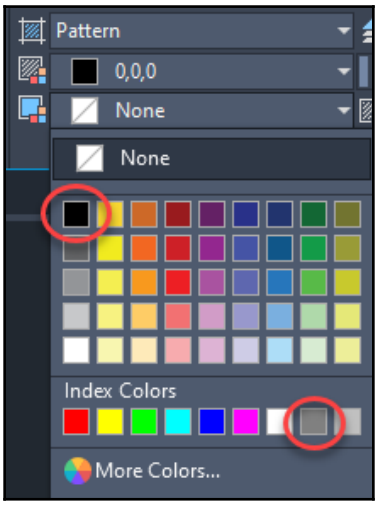


Fig 6.10: Selected colors highlighted on the color panel

As you can see, the list of colors is long, but if you want even more colors, then you can go to the **More Colors...** option of the color menu and select from the available colors. The result with the colors applied will look like the following screenshot:

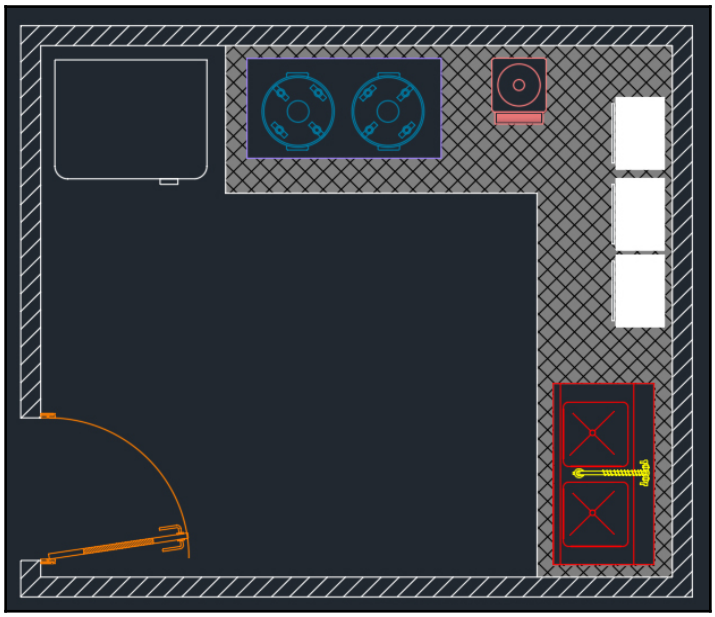


Fig 6.11: Hatches with colors applied to the kitchen slab

By using color, you can make Hatches that look different even when the same pattern is used for the Hatches. Another property that you can add to Hatches is transparency, which will help you add depth to a drawing—discussed in the next section.

Hatch transparency

Hatches have zero transparency by default, but you can make them look dull by adding transparency to them. Transparency is generally added to make objects of a similar color distinct, and it is also added when you want to print drawings so they consume less ink.

In the following example, I will add transparency to the Hatch pattern inside the gas stove of the kitchen drawing of the previous example:

1. Select the `Hatch` command from the **Draw** panel and click inside the gas stove to make a selection for the Hatch.
2. Now apply the **Solid** Hatch pattern and select the eighth color again, which we selected earlier for the slab. You will notice that the color of the slab and the gas stove is the same, and this makes the stove look like a part of the slab. To make it look different, I will add transparency to it.
3. The transparency option is on the properties panel of the **Hatch Creation** tab. The minimum value of transparency is `0`, which is also the default value, and the maximum value is `90`. In this case, I will add transparency of `60`. You can move the transparency slider to the right to add transparency or you can directly type in the transparency value:

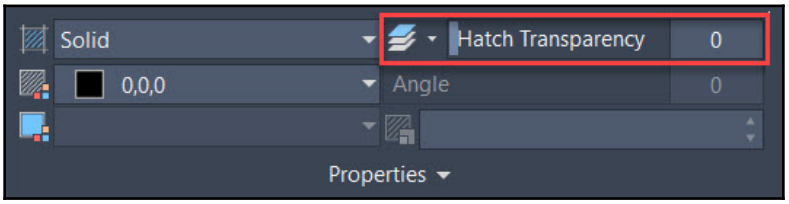


Fig 6.12: The Hatch Transparency option on the Properties panel

4. Once you are done adding the transparency, click on the **Close Hatch Creation** checkmark on the right of the **Hatch Creation** tab.

The final drawing in my case looked like the following image. The gas stove is highlighted in the red box:

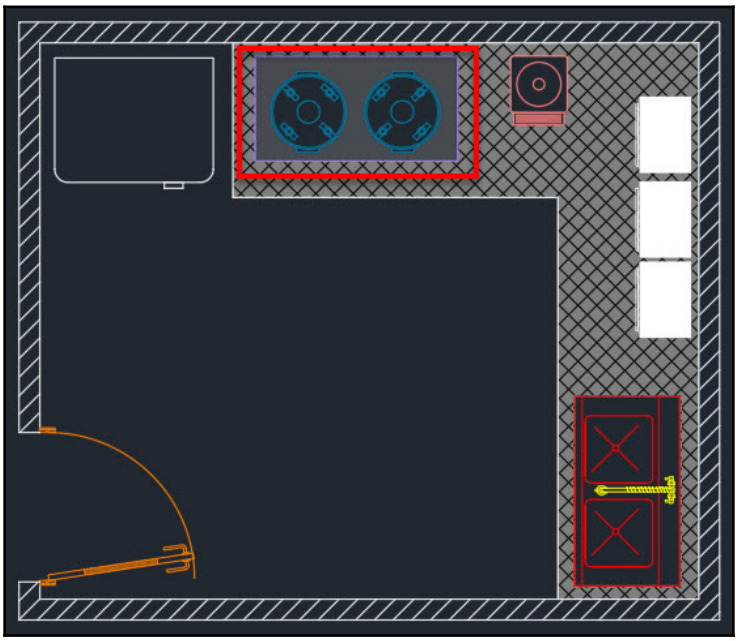


Fig 6.13: The gas stove after applying transparency to it

As you can see, we can add "depth" to our drawing using transparency and it can be used to distinguish different components of the same color too. So, after transparency, let's move to the Hatch **Angle** option, which is also useful if you are adding Hatches that contain linear patterns.

Hatch angle

The Hatch angle, as the name suggests, is the angle at which the Hatch is made in an area. Let's take the example of the **ANSI31** Hatch pattern. By default, the Hatch is aligned to an angle of 45 degrees with respect to the x axis, and if applied in a simple rectangle, it looks like this with the default setting:



Fig 6.14: The ANSI31 Hatch pattern with the default angle of 0 degrees

Now, if you want to rotate the Hatch pattern so that it becomes completely vertical, then you can add a Hatch angle of 45 degrees. The Hatch angle starts from the default angle of the Hatch pattern, so in this case, since the **ANSI31** pattern is already at an angle of 45 degrees, adding an **Angle** of **45** in the **Properties** panel, as shown in the following screenshot, will make the total angle 90 with respect to *x* axis:

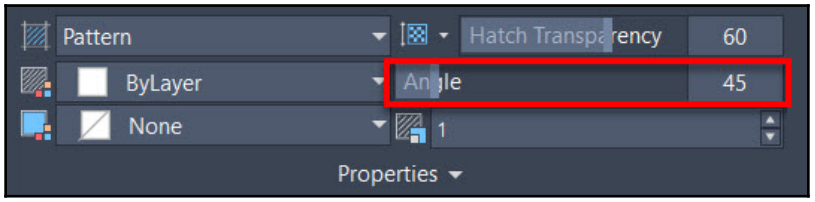


Fig 6.15: Angle changed to 45 degrees for the ANSI31 Hatch pattern

The final Hatch pattern will become vertical, as shown in the screenshot here:

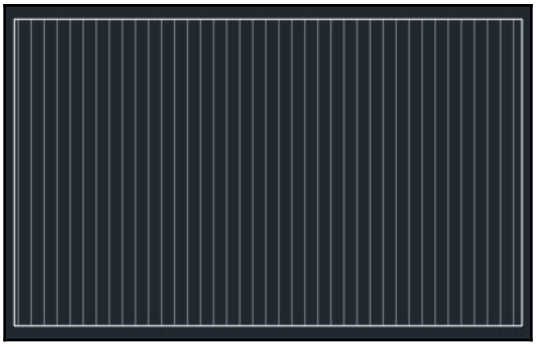


Fig 6.16: The ANSI31 Hatch pattern with a 45-degree Hatch angle

Similarly, you can add a Hatch angle of `-45` to the **ANSI31** Hatch pattern to make it horizontal. All the other Hatch patterns except the **Solid** Hatch pattern also behave in a similar way.

Set Origin

Every Hatch pattern has its origin point and when you apply a Hatch in an area inside your drawing, AutoCAD selects the origin point for it. But if you want to specify an origin and want to force AutoCAD to select your preferred point as the origin, then you can use the **Set Origin** option from the **Origin** panel of the **Hatch Creation** tab, as shown here:

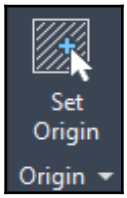


Fig 6.17: The Set Origin option on the Origin panel of the Hatch Creation tab

In the following screenshot, in **A**, the Hatch pattern has been applied normally without selecting the origin, whereas in **B**, the same pattern has been applied with the origin selected as the lower-left corner of the rectangle:

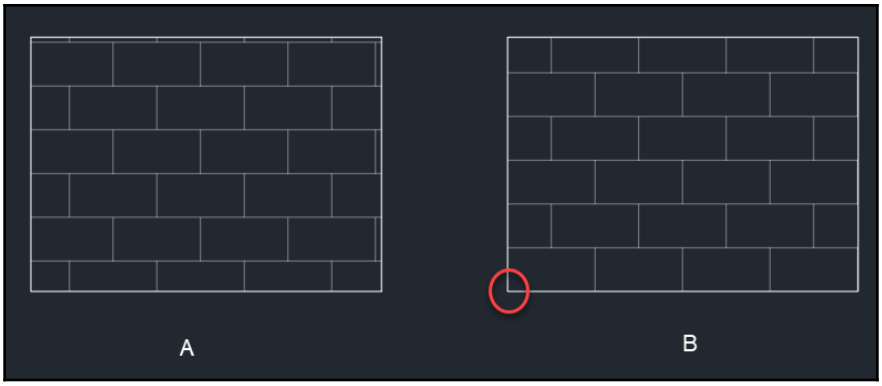


Fig 6.18: The Hatch pattern with a random origin point applied in rectangle A and the same Hatch pattern with the lower-left corner as the origin in rectangle B

As you will notice in **A**, the brick pattern that starts at the lower-left corner of the rectangle is smaller than a full block because the Hatch did not start from there, whereas in **B**, the lower-left corner brick marked with the red circle is a full block because the origin of the Hatch pattern is that point.

To set the origin, start the `Hatch` command, then select the area where you want to apply the Hatch. Then, click on the **Set Origin** tool from the **Origin** panel of the **Hatch Creation** tab, and click at the point that you want to specify as the origin. After specifying the origin, the Hatch will automatically adjust so that the pattern starts from the selected origin.

Associative Hatch

When adding a Hatch, you will see an option called **Associative** on the **Options** panel of the **Hatch Creation** tab. If you keep this option checked, it will create an **Associative** Hatch, and keeping it unchecked will let you create non-associative Hatches:

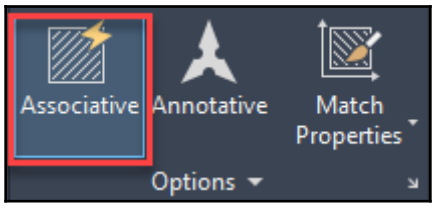


Fig 6.19: The Associative Hatch option on the Options panel

Associative Hatches are ones that are "associated" or linked with the Hatch boundaries, and when you modify the boundary of the Hatch, the Hatch automatically changes to fill the modified boundary, whereas non-associative Hatches have their own boundary and changing the boundary of the drawing does not affect the Hatch.

In the screenshot that follows, a Hatch has been applied in the rectangle and, later, one of the lines of the rectangle was changed to an arc—the resulting Hatch automatically adjusted as it is an associative Hatch:

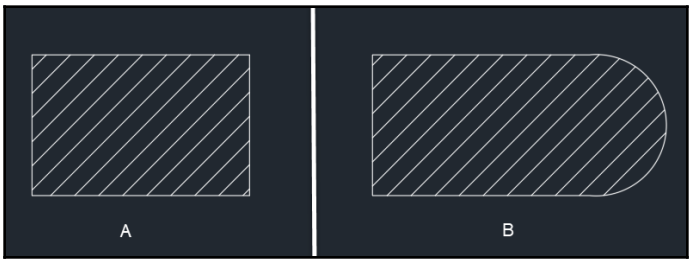


Fig 6.20: An associative Hatch adjusts automatically when the boundary of the Hatch changes

In the following screenshot, the same process has been repeated; the Hatch is applied to the rectangle and, later, one of the boundaries of the Hatch is converted to an arc, but it does not affect the non-associative Hatch:

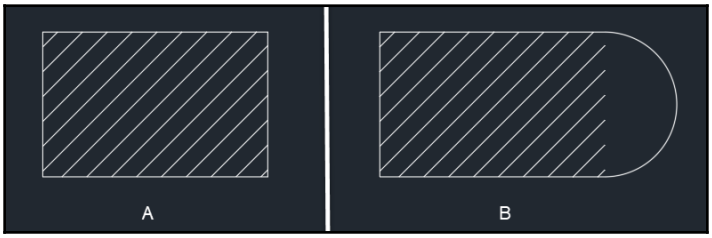


Fig 6.21: A non-associative Hatch pattern not adjusting to changes in the boundary shape

When you make Hatches associative and then want to change them back to a non-associative type, select the Hatch pattern again and then uncheck the **Associative** option from the **Hatch Editor** tab. Non-associative Hatches can't be changed to associative Hatches.

The next option is all about making Hatches in open boundaries with tiny gaps. We will use a tool called Gap Tolerance for this, which is discussed next.

Gap Tolerance

When making Hatches in a closed area, you might sometimes encounter this error message:

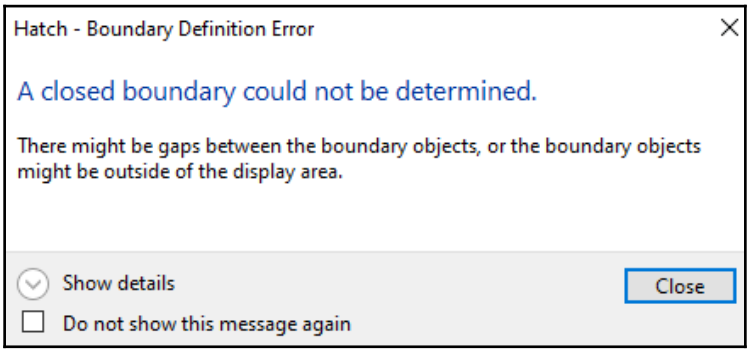


Fig 6.22: Error message when the Hatch tool is unable to find a closed boundary

As the message states, this error is related to a gap in the boundaries and to fix this error, you need to close any gaps that are keeping your drawing from creating a closed area. When you see this error while making Hatches, simply go to the **Options** panel and expand it. There, you will find the **Gap Tolerance** slider, as shown in the following screenshot:

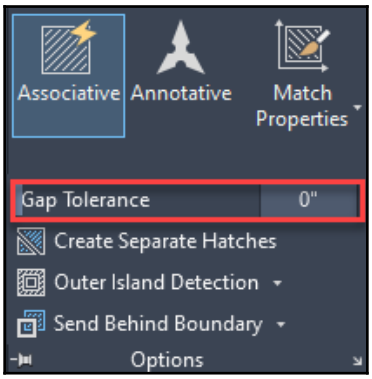


Fig 6.23: The Gap Tolerance slider on the Options panel

Type a value into the **Gap Tolerance** field that you feel is larger than the gap in the boundary where you are trying to apply the Hatch. You don't need to be precise but try to be close to the gap width to avoid unexpected results.

Just like Hatches, you can create colored areas with a solid color or a gradient of color, and we will discuss all that in the next section.

Creating color gradients

With the **Gradient** tool, you can add gradients of color to any selected area. The **Gradient** tool also works like the **Hatch** tool, but in this case, instead of a pattern, a color gradient is applied to the selected area.

In the example drawing of the kitchen that we used earlier with the `Hatch` command, we will add three-dimensional effects to the sink and the top of the refrigerator using the **Gradient** tool. You can start the **Gradient** tool from the expanded **Hatch** flyout on the **Draw** panel of the **Home** tab, as shown in the following screenshot. Alternatively, you can also use its command alias, `GD`:

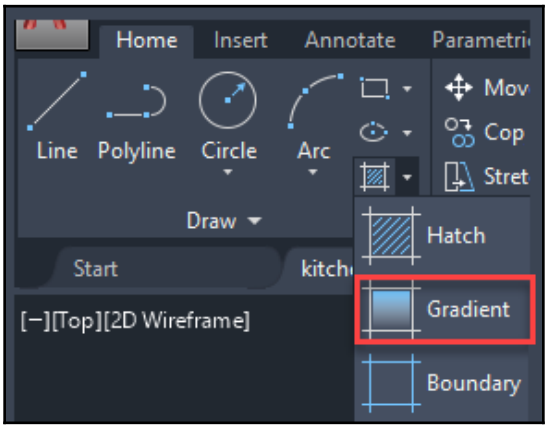


Fig 6.24: The Gradient tool in the Hatch flyout of the Draw panel

In the following example, we will learn how to apply gradients to the sinks of the kitchen drawing:

1. To apply the gradient, select the **Gradient** tool and then click at the point inside the area where you want to apply the gradient. The workflow for applying a gradient will be like that of applying a Hatch.
2. Once you click inside the selected area, a gradient will be applied with default colors. The first thing that you need to do here is to change the gradient type. You can change it from the **Pattern** panel of the **Hatch Creation** tab.

3. In this case, I am applying the gradient to the bowl shape of the sink and I am using one of the circular gradients, as shown in the following screenshot:

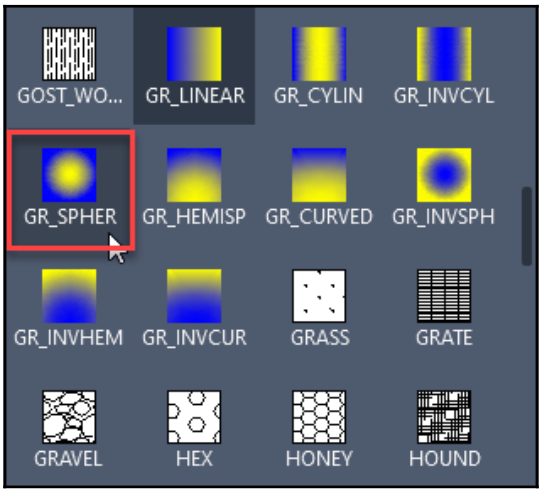


Fig 6.25: A circular gradient selected from the Patterns panel

4. Now we need to change the colors of the gradient, which can be done from the colors drop-down menu of the **Properties** panel on the **Hatch Creation** tab, as shown in the following screenshot. In this case, I am selecting a gray and white color combination for the sink:

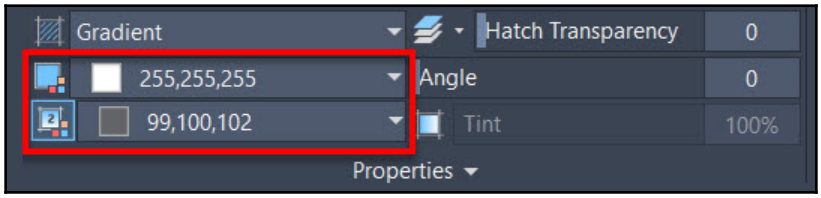


Fig 6.26: The colors white and gray selected for the gradient from the Properties panel

5. Once you are done selecting the colors, click on **Finish Hatch Creation** to exit the **Gradient** tool and the gradient will be applied as shown in the screenshot that follows.

In this case, I have applied the gradient to both sinks using the same gradient type:

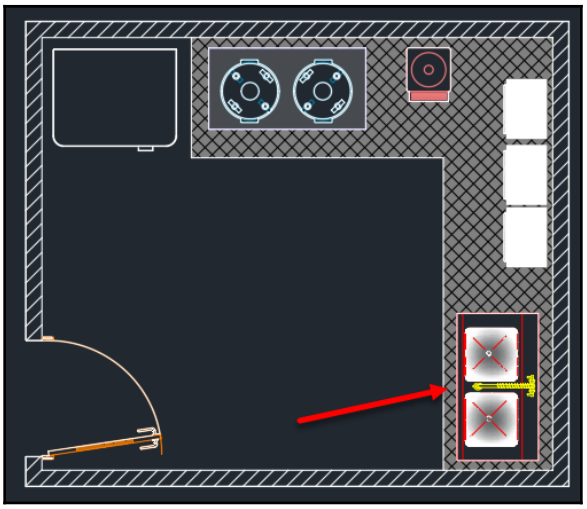


Fig 6.27: Gradient applied to the sinks in the drawing

Let's try another gradient on the top of the refrigerator:

1. Select the **Gradient** tool from the **Draw** panel or use its command alias, `GD`.
2. Now select the closed area inside the refrigerator and, from the `Pattern` panel, select any pattern that you like. In this case, I am again selecting a pattern that looks circular.
3. For the colors, I will select light blue and white. After applying the pattern, my drawing looked like the following screenshot:

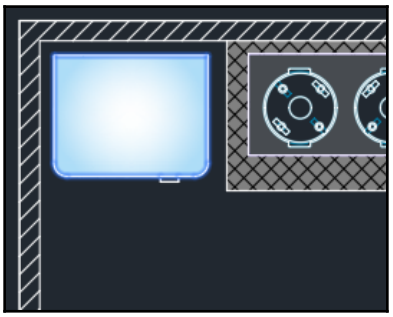


Fig 6.28: Gradient applied on the top of the refrigerator

As you can see, the white color of the gradient is exactly at the center of the selected area, but to give it a better three-dimensional look, I will make it off-center using the **Centered** option of the **Origin** panel:



Fig 6.29: The Centered option of the Origin panel of the Hatch Creation tab

I will simply uncheck this **Centered** option and the pattern will now look like this:

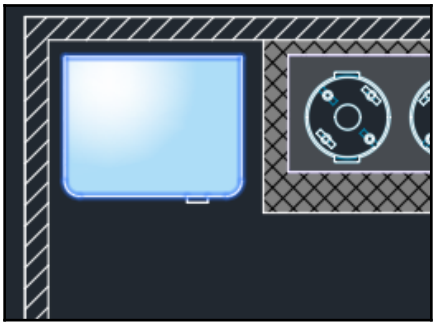


Fig 6.30: The gradient highlight changed to the top-left corner when the Centred option was unchecked

As you can see, in this case, the white gradient is more toward the top left of the selected area, giving it a much better appearance.

So, as you just saw, gradients can be applied to different areas of the 2D drawing to give them depth and to add colors to your drawing. However, if you want to apply only a single solid color, then you can select the **Solid** pattern from the **Pattern** panel and then apply it to the area where you want to add color.

The **Gradient** tool adds to the visual properties of the existing drawing. Just like a gradient, we can also use text in our drawing to add information, but unlike a gradient, information added with text can be precise. In AutoCAD, you can add single-line or multiline text—discussed in the next section.

Adding text to the drawing

Text is the most basic kind of annotation that you could add to a drawing. There are two text tools in AutoCAD: **Single Line**, which lets you add simple text entities without many formatting options, and **Multiline Text**, which has comprehensive formatting tools.

The **Text** tool is on the **Annotation** panel of the **Home** tab or you can also select the text tools from the **Annotate** tab, as shown in the following screenshot:

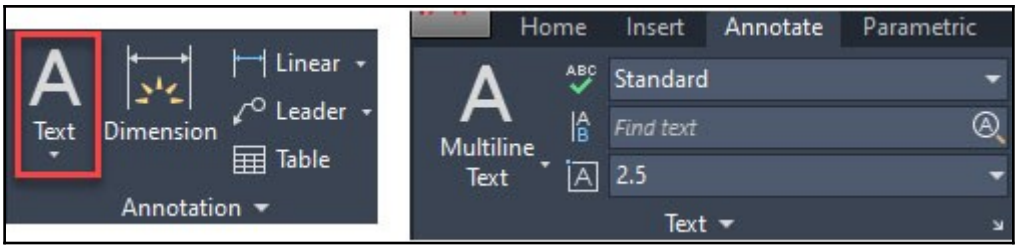


Fig 6.31: The Text tool on the Annotation panel of the Home tab on the left, and on the Text panel of the Annotate tab on the right

In all our examples, I will use the **Text** tool from the **Home** tab, though, if you are comfortable with the **Annotate** tab, you can use it to start the **Text** tool too. Before we add any type of text entity to our drawing, it is important that we learn about the properties of text, such as font, formatting, and scale. These settings are controlled in the **Text Style** window, which we will discuss next.

Creating a text style

The text style determines the properties of **Single Line** and **Multiline** text that is added to the drawing. By modifying the text style, you can control the properties of all the text entities that are using that style in the drawing.

In the following example, I will start by defining the text style, which basically determines the type of text settings that you will get in your drawing:

1. Expand the **Annotation** panel of the **Home** tab and click on the A the paintbrush icon, as shown in the following screenshot.

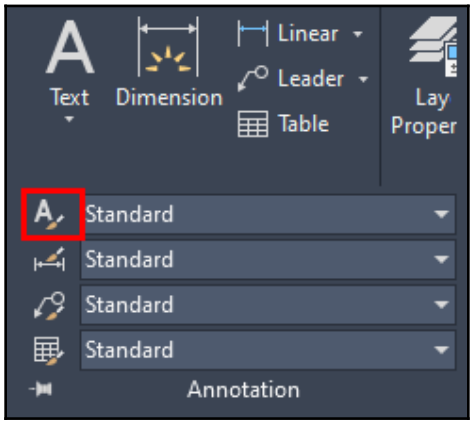


Fig 6.32: Text style options on expanded Annotation panel

2. This will open the **Text Style** window, as shown in the following screenshot. You can also use the command alias, `ST`, to open the **Text Style** window:
3. From the top right of the window, click the **New...** button, type the name of a text style, and click **OK**. In this case, I am adding the text style name as `Test`.
4. Now, a new text style, `Test`, will be added to the list of text styles and it will show up underneath the existing text styles, **Annotative** and **Standard**, on the left panel.
5. With the `Test` text style selected from the left panel, click the **Font Name** dropdown in the **Font** panel and select a font from this list. For our example, I am using the **Calibri** font from the list of fonts.

6. Select the font style, such as **Bold**, **Italic**, and so on, from the **Font Style** panel, as shown in the following screenshot. In our example, I will keep the **Regular** font style selected:

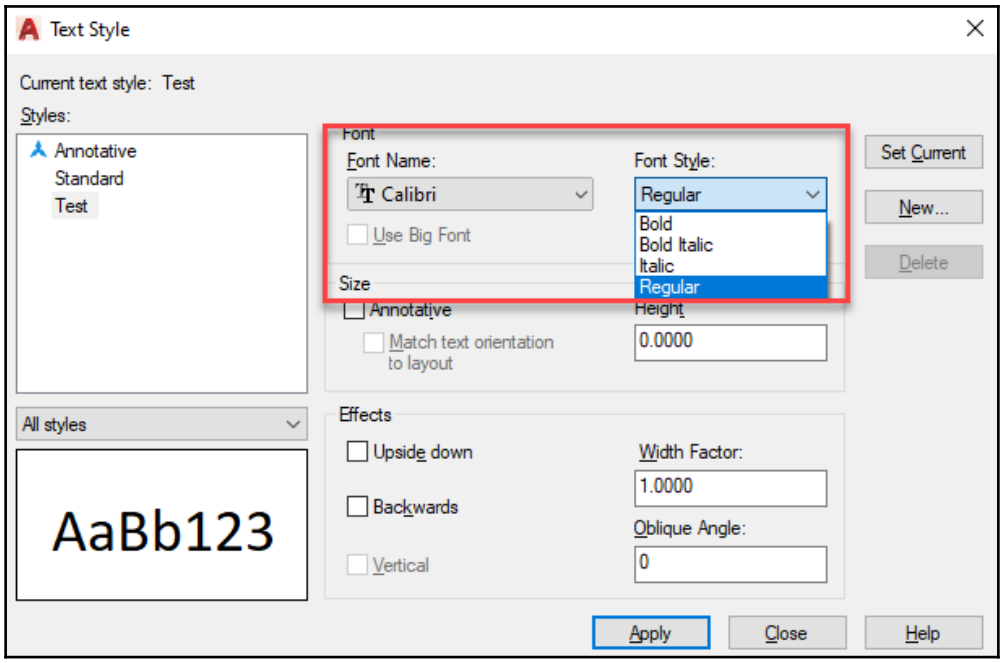


Fig 6.33: Font Name and Font Style options in the Text Style window

7. In the **Size** panel, uncheck the **Annotative** option if it is selected, and add the height of text you want to use in the drawing in the **Height** field. Leave the **Height** field blank if you want to specify the height value later in the drawing. For our example, I am using a height of `1.6`.
8. In the **Effects** panel, select different effects such as **Upside Down** and **Backwards** if you want these effects; otherwise, leave them unchecked. The **Vertical** effect won't show up for the Calibri font but it's available for shape fonts. I will leave all the effect boxes unchecked in this example.
9. You can also add a **Width Factor**, which will expand or shrink the complete text. A **Width Factor** between 0 and 1 will shrink the text and a **Width Factor** greater than 1 will increase the text width. The **Oblique Angle** option will make your text tilted at an oblique angle, which you will specify here. For our example, I will leave **Width Factor** as **1** and **Oblique Angle** as **0**.

After making these changes, click **Apply** and then click the **Cancel** button to close the **Text Style** window. After making all these changes, this is how my **Text Style** window looked:

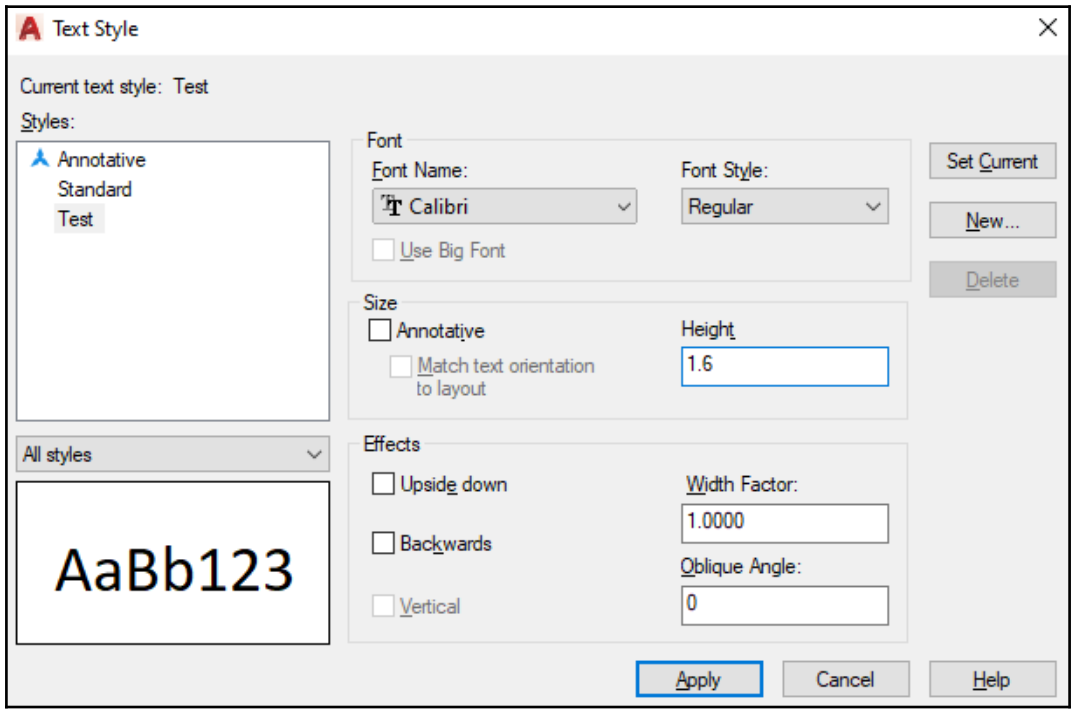


Fig 6.34: Text Style window with all the settings for the Test text style

The `Test` text style will now show in the list of text styles and you can select it from the **Text Style** panel, as shown in the following screenshot:

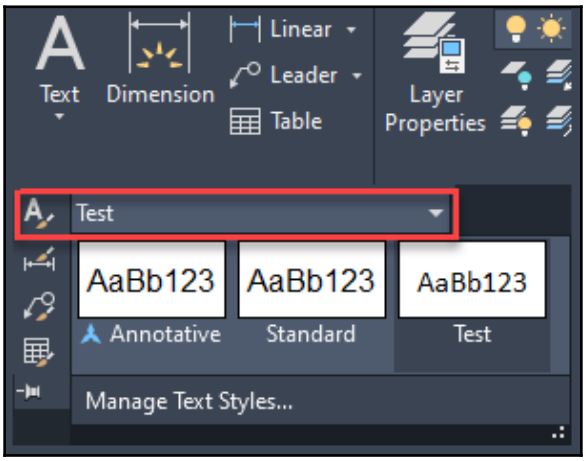


Fig 6.35: The Test text style in the list of text styles on the expanded Annotation panel

Once selected in the **Text Style** panel, the new text will be added to the drawing with the settings we have just added to this style. So, now our text style has been created, it's time to add some text to the drawing using our text style. We will discuss this in the next section.

Adding Multiline text

The **Multiline Text** tool allows you to add and modify multiple lines of text as a single entity, and this feature also allows you to change the formatting of the text. The **Single Line** tool does not allow changing formatting directly from the drawing area but you can modify some of the formatting settings for single line text from **Text Style** tab.

In the following example, we will learn how to add a simple piece of text using the **Multiline Text** option:

1. To start adding multiline text, click the **Multiline Text** option from the **Text** flyout of the **Annotation** panel of the **Home** tab or use its command alias, `T`:

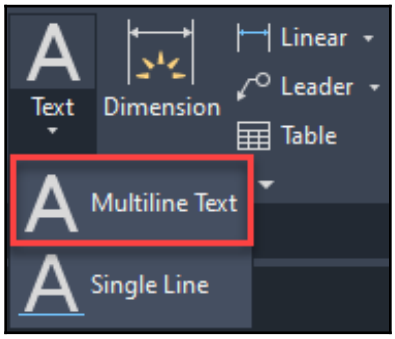


Fig 6.36: The Multiline Text option on the Annotation panel

2. Click anywhere in the drawing area, then let go of your cursor and click again to make a box. A blinking cursor will show up inside the text box and you will get the **Text Editor** tab, which can be used to modify the text formatting, as shown in the following screenshot:

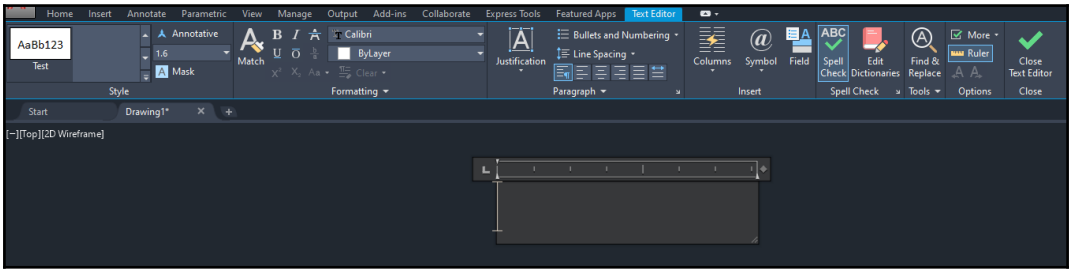


Fig 6.37: Text box with the Text Editor tab

3. Now you are ready to type in your text, type `The first line of sample text`, then press the *Enter* key and type `The second line of sample text`. Once you are done typing the text, click outside the text box to exit the `Text` command or click on the **Close Text Editor** checkmark to the right of the **Text Editor** tab.

4. If the text is very big or is added in multiple lines and you want to fit it on a single line, then click the diamond icon at the top of the text box, next to the ruler, and drag it to the right as shown in the following screenshot:

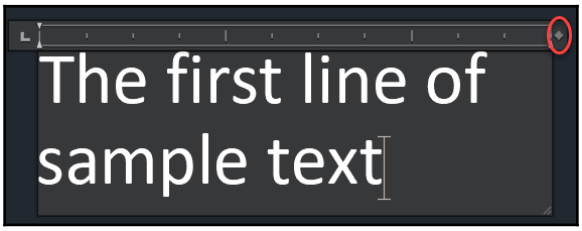


Fig 6.38: Ruler on the top of the text box

Your multiline text is now added to the drawing. If this text is very small or very large, then double-click your middle mouse wheel to fit it in the drawing area or zoom in and out to make it fit as per your requirements.

Changing text formatting

Text added in this way will take properties like the font and text formatting from the text style that we defined earlier. To change another text formatting, you need to once again go the **Text Editor** tab and change the formatting from there, as explained in the following example:

1. Double-click on the text that you added in the previous example and the **Text Editor** will once again show up and the text will become editable. You can now add to or remove the existing piece of text as well.
2. Select the first line of sample text from the textbox and then go to the **Style** panel and change the height of the text to `2`. By default, it will be **1.6**, which is the height of the text style we defined in earlier examples.
3. From the **Formatting** panel, click the **B** icon to make the text bold, **I** to make it italics, **A** to strike through the text, **U** to underline the text, and **O** to overline the text.
4. To stack the text, write a text with backslash like `a/b` then select it and click on the stack icon as shown in the following screenshot.
5. To add the text as superscript or subscript, select the text that you want to make superscript or subscript, such as `X2`, and then click on the superscript icon and it will become X^2.

6. To change the case of text, select the text and select the change case option, as shown in the following screenshot:

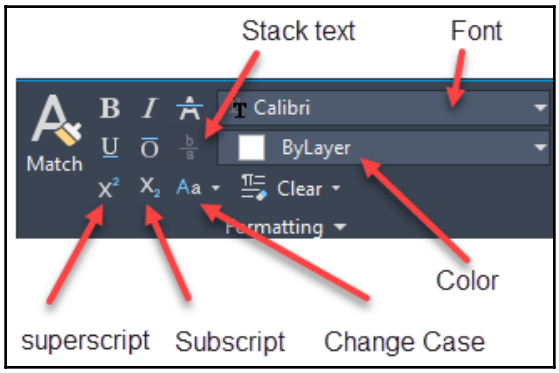


Fig 6.39: Change case and other options on the Formatting panel of the Text Editor tab

7. You can change the font from the **Font** drop-down menu, and right underneath it, we have the color option, which can be used to change the color of the selected text, as shown in the preceding screenshot.

From the next panel, **Paragraph**, you can add justification to the text, which will place it in different locations with respect to the textbox. You can also add bullets or numbering to selected lines of text from the respective option of the **Paragraph** panel.

Adding symbols

To add symbols in the text, select the **Symbol** option from the **Insert** panel and select a symbol from the list of symbols that shows up:

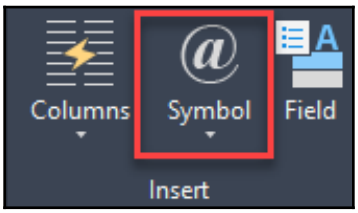


Fig 6.40: The Symbol option on the Insert panel of the Text Editor tab

If your symbol is not in this list, then click the **Other** option at the end of this list and the **Character Map** window will show up. Click on the symbol in the **Character Map** window and then click the **Select** button, then the **Copy** button. Now, close the **Character Map** window, right-click in the textbox, and select **Paste** from the right-hand context menu or press the *Ctrl + V* keyboard shortcut to paste the symbol:

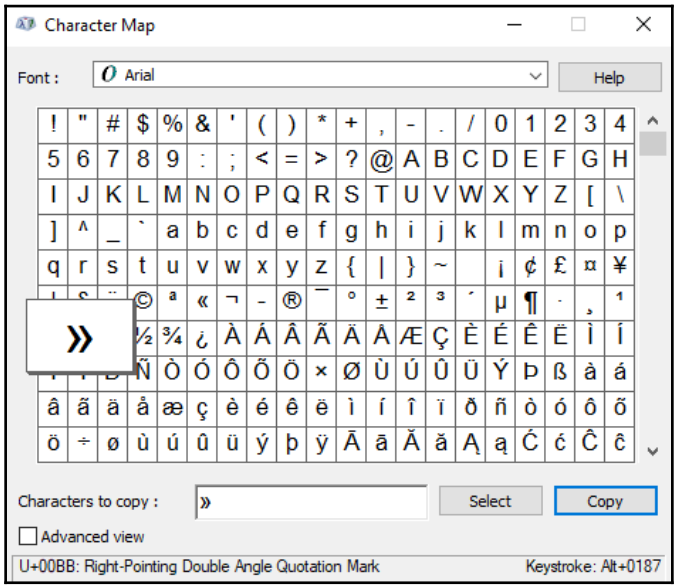


Fig 6.41: Character Map window with all the symbols in it

Character Map is a Windows operating system feature and you can also directly open Character Map from Windows and then copy a symbol on your clipboard and paste it into AutoCAD.

So, now you know about multiline text and how to add it to a drawing, in the next section, we will talk about adding single line text, which is a simple piece of text without direct formatting options.

Adding single line text

Single line text is a simple kind of text that does not offer many text formatting options, unlike multiline text. In the following example, we will learn how to add a simple piece of text using the **Single Line** text option:

1. To add single line text, select the **Single Line** option from the **Text** flyout of the **Annotation** panel, as shown in the following screenshot. You can also use its command alias, `DTEXT`:

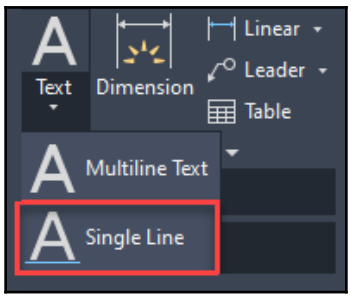


Fig 6.42: The Single Line text option on the Text flyout of the Annotation panel

2. Click at a point to specify the starting point of the text.
3. Now, the command line will prompt you to specify the rotation angle. Type `0` and press *Enter*. If the command line prompts you for the text height before the angle value, then type in a height for the text and then press *Enter* to go to the **Angle** option.
4. When you are done adding the angle, press *Enter* and a blinking cursor will now start showing up in the drawing. Now you are ready to type your text, type `first line of single line text`, then press *Enter* and type `second line of single line text`.
5. You will notice that, in this case, there is no **Formatting** tab available to change the text formatting. When you are done adding the text, press the *Enter* key twice to exit out of the single line textbox.
6. To modify the added text, double-click on the text and you will notice that, though you have made two lines of text with the command, each line is selectable separately and you can modify them separately too, hence the name **Single Line** text.

In this case, since we have already specified the height of the text in the **Text Style** option, the command line will not prompt for the text height while creating single line text. If you use another text style where text height is not specified, then the command line will prompt you for the height of the text and you can specify your own text height for the single line text:

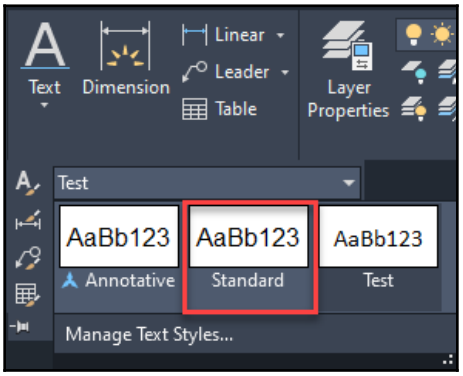


Fig 6.43: The Standard text style in the expanded Annotation panel

To check this, switch to the **Standard** text style from the **Text Style** menu, as shown in the preceding screenshot and again go to the **Single Line** text option from the **Text** flyout of the **Annotation** panel. Click on a point and now the command line will prompt you to specify the height of the text. Type in a text height value and press *Enter*. Now you can specify the rotation angle. Type `30` here; in this case, we will rotate the text to an angle and then press *Enter* and type in your text.

Once you are done typing, press the *Enter* key twice to exit out of the **Single Line** text command. With the preceding settings, my single line text looked like the following screenshot:



Fig 6.44: Single line text made with the preceding settings

As you may have noticed, adding text in AutoCAD is very similar to adding text in any other text editor, such as Microsoft Word. In AutoCAD, however, you have the option of adding single line text, which makes every line of the text separate, as well as multiline text, which makes adding paragraphs of text with formatting options a breeze.

To modify either of the text types, simply double-click on the existing text and the text will be highlighted and you can then modify it. Make modifications and then exit out of the textbox by pressing the *Enter* key twice in quick succession.

Having learned about text and text styles, it's time we look at another annotation tool, which is responsible for the most annotation you will ever add to a drawing, and that annotation is **Dimension**. The drawing will start making sense when you add dimensions to it and this is what the next section is all about, adding and modifying dimensions.

Working with dimensions

Dimensions are probably the most important kind of annotation you will add to your drawing. There are lots of dimension tools available in AutoCAD, which makes adding dimensions a breeze, and we will look at most of these tools. The **Dimension** tools are on the **Annotation** panel of the **Home** tab and on the **Dimension** panel of the **Annotate** tab, as shown in the following screenshot. For our example, we will mostly use dimension tools from the **Annotation** panel of the **Home** tab:

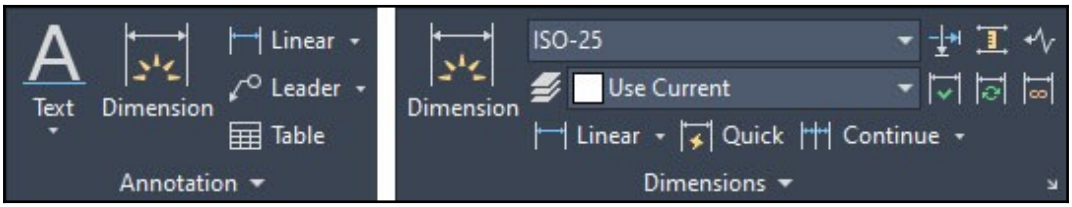


Fig 6.45: Dimension tools on the Annotation panel of the Home tab and the Dimensions panel of the Annotate tab

Just like text, the properties of dimensions are also defined by their style, which is called the dimension style. We will start by making our own dimension style in our drawing and then we will use this dimension style to add dimensions to our drawing.

Making a dimension style

To open the Dimension Style Manager, click the dimension with the paintbrush icon on the expanded **Annotation** panel, as shown in the following screenshot, or use its command alias, `D`:

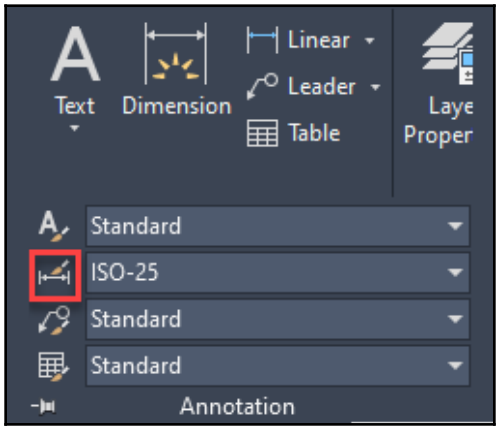


Fig 6.46: Dimension Style Manager on the expanded Annotation panel

The **Dimension Style Manager** window will open with a list of standard dimension styles in the left panel, as shown in the following screenshot. The dimension styles in your case might be different if you have another template, but that won't matter here as we will be making our own dimension style:

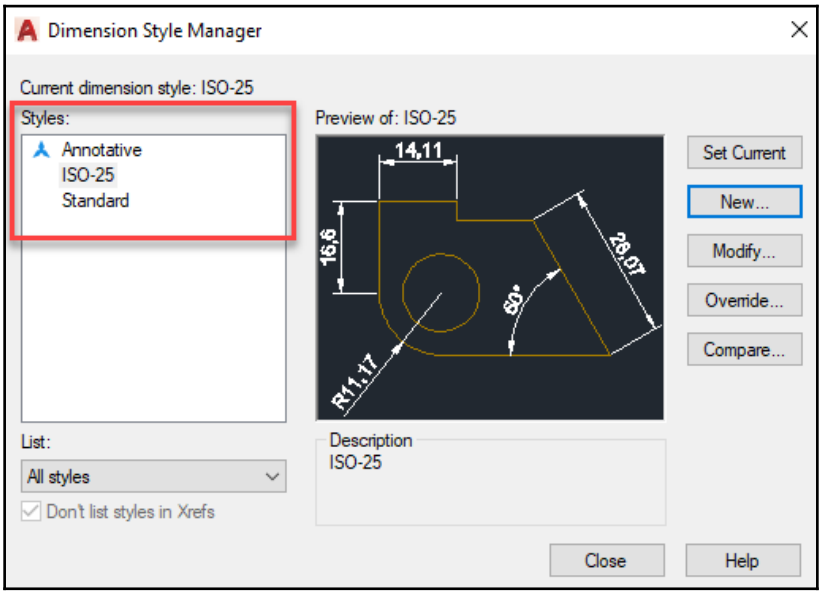


Fig 6.47: Dimension styles in the Dimension Style Manager

Click the **New...** button and give your dimension style a name. I will call it `Sample Dim` for our example. From the **Start With** menu, select the **Standard** dimension, make sure the **Annotative** checkbox is not selected, and click the **Continue** button. This will make a copy of the **Standard** dimension style and name it `Sample Dim`, which we will modify:

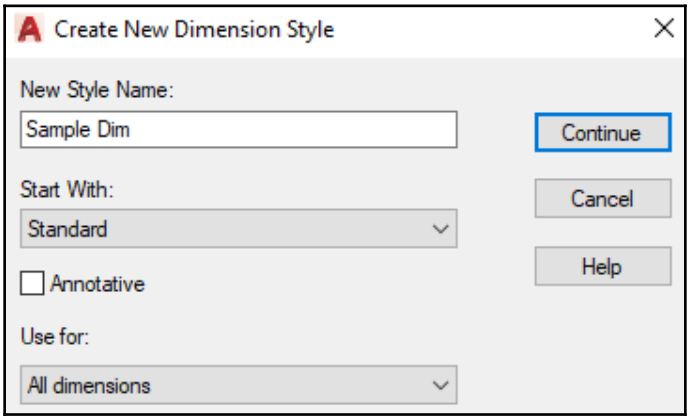


Fig 6.48: The Sample Dim dimension style

A new **Dimension Style** window will open with lots of tabs and panels. We will look at these options and how they affect our dimension style. But before we go further, let's familiarize ourselves with dimension terminology using the following screenshot:

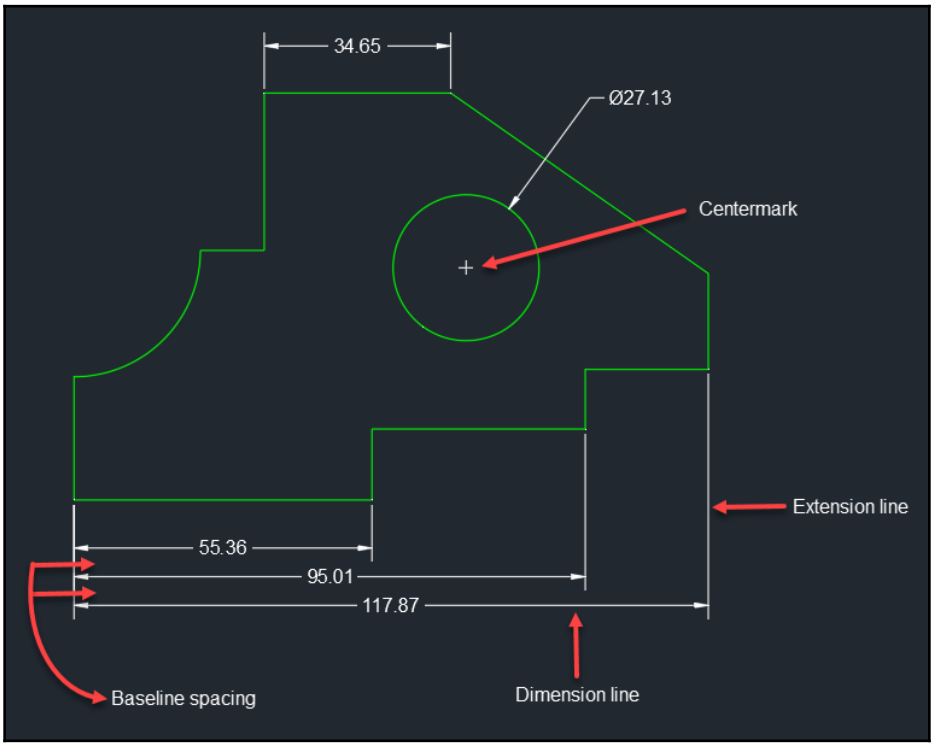


Fig 6.49: Different parts of the dimensions labelled in the drawing

Now that you are familiar with the terminology used for dimensions, let's start with the **Lines** tab of the **Modify Dimension Style: Sample Dim** dimension style. All the changes you make to the dimension style will show up in the **Preview** panel of the **Dimension Style** window, which shows a sample drawing and its dimensions in the top right of the window.

Using the Lines tab

The **Dimension lines** panel has settings for changing the **Color**, **Linetype**, and **Lineweight** of dimension lines only. You can also change the **Baseline spacing** of the dimension, which is the gap between the dimension lines of the baseline dimension using its respective field, and hide the left or right sides of dimension lines using the **Suppress** checkboxes:

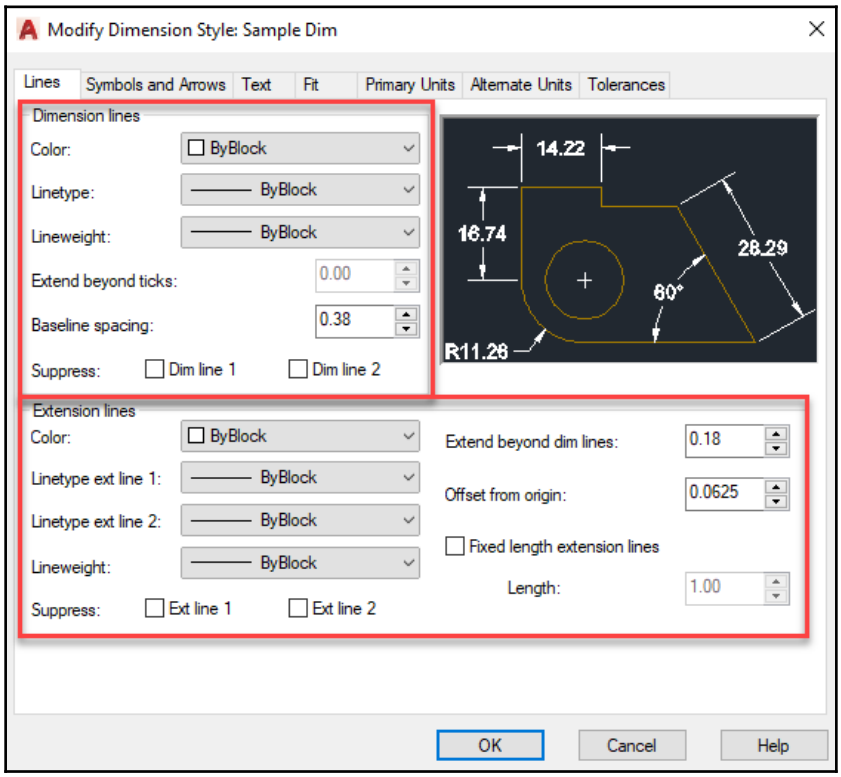


Fig 6.50: Dimension and extension line settings on the Lines tab

Similarly, the settings in the **Extension lines** panel will affect only the extension lines and you can use its settings to change the **Color**, the linetype of extension lines 1 and 2, and the **Lineweight**, and hide the extension lines using the **Suppress** features. **Extend beyond dim lines** is the tiny line segment length shown in the following screenshot, and you can choose to give it a different length or make it zero. **Offset from origin** is the gap between the extension line and the actual geometry as shown in the following screenshot, which you can control from the **Offset from origin** field:

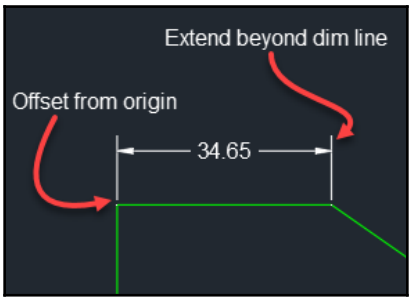


Fig 6.51: The Offset from origin and Extend beyond dim line features of a dimension

If you want a fixed length for your extension lines, then check the box that says **Fixed length extension lines** and then type in the length of the extension line. By default, the length of the extension line will change with respect to the position of the dimension. So, if you move your dimension far from the drawing the extension line will be longer and if you keep it close to the drawing it will be shorter.

Using the Symbols and Arrows tab

As the name suggests, this tab will allow you to change the settings related to the symbols and arrows of the dimension. From the **Arrowheads** panel, select the type of arrow you want in your dimension style using the **First** drop-down menu. When you change the **First** arrow, the **Second** one will change automatically to match the **First** one but if you want a different **Second** arrow, then you can change the **Second** arrow from the **Second** dropdown. The **Leader** drop-down menu will allow you to change the type of arrow for the leader line, which we discuss later in this chapter. You can change the size of the Arrow from the next field, **Arrow size**:

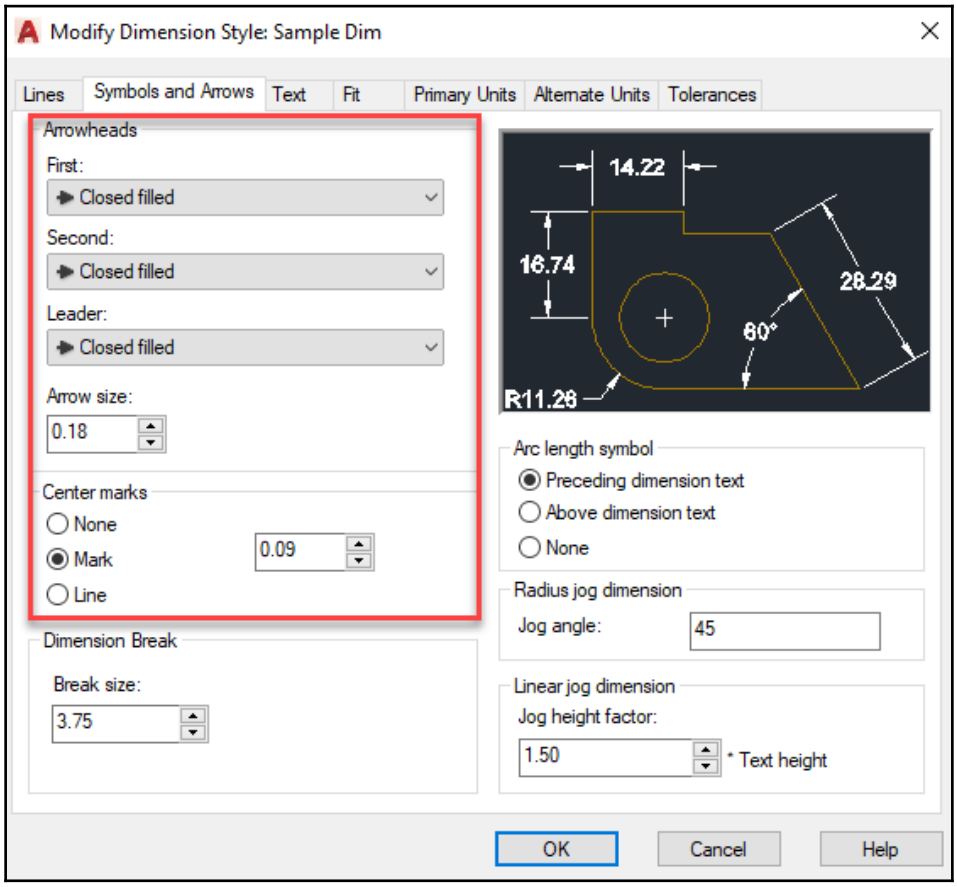


Fig 6.52: Arrow and center mark settings highlighted on the Symbols and Arrows tab

The next panel is **Center marks**. You can select **None** if you don't want a center mark at the center of a circle or an arc in the drawing. Select **Mark** if you want a tiny center mark and the **Line** option if you want to add an extra line along with the center mark. The field next to the **Mark** option can be used to specify the length of the center mark lines.

Using the Text tab

Using this tab, you can modify the settings of the text in your dimension. The first panel, **Text appearance**, will help you change the appearance of text:

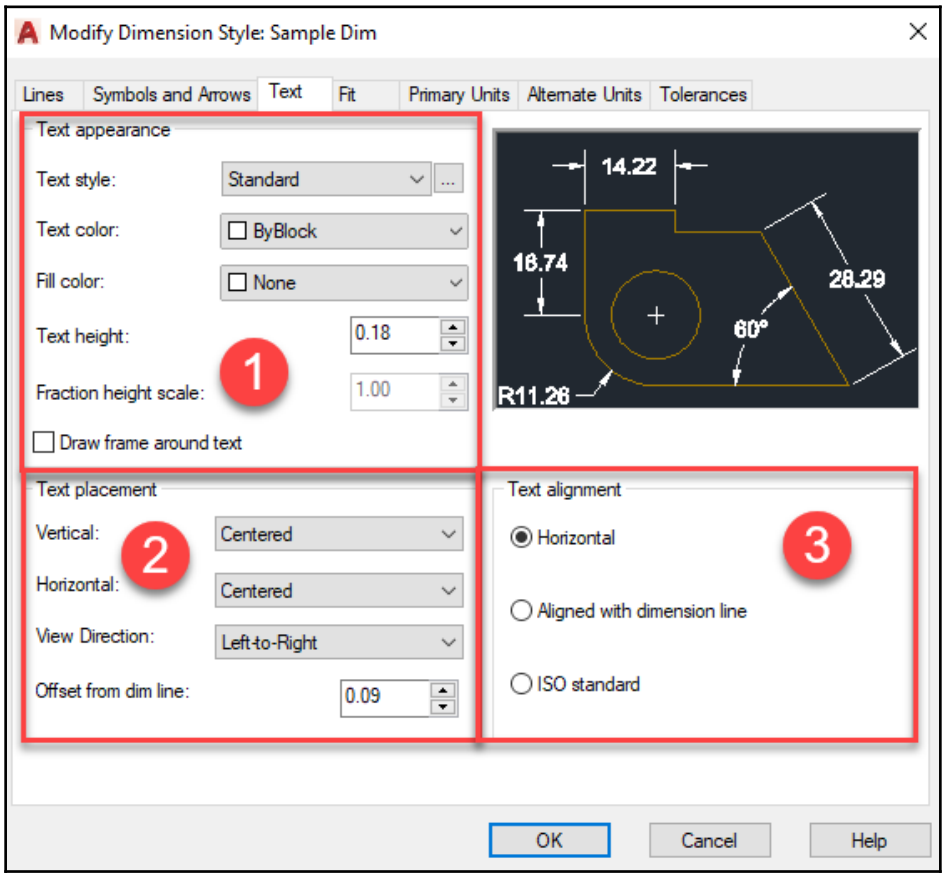


Fig 6.53: The Text appearance, Text placement, and Text alignment panels on the Text tab

You can change the text style using the **Text style** dropdown. If you want to modify the text settings, then click on the ellipses next to the **Text style** drop-down menu, as shown in the following screenshot, and the **Text style** window will open where you can modify text settings further:

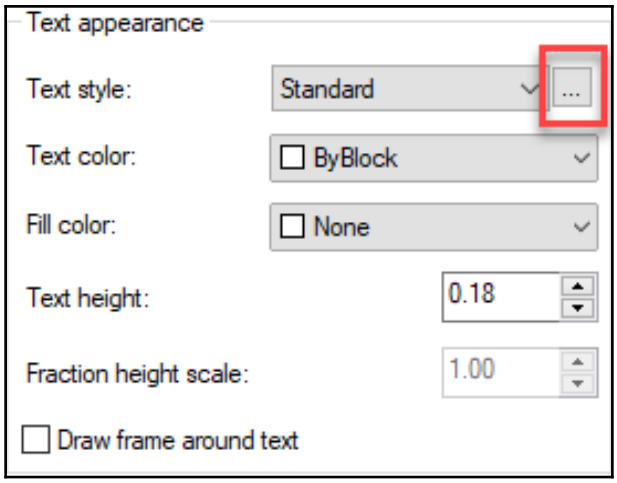


Fig 6.54: Text style ellipses highlighted in the red box of the Text appearance panel

You can change the color of text using the **Text color** option and the background color of the text can be changed using the **Fill color** option. The height of the text is the next field, called **Text height**.

When you have changed the arrow size on the **Symbols and Arrows** tab, then it is recommended to change the text size as well to the same value, so that both remain consistent. Click on **Draw frame around text** to enclose the dimension text in a box.

The second panel, called **Text placement**, will change the placement of your text on the dimension or extension lines. Change the options in this panel to move the dimension text around. The third panel, **Text alignment**, has three options: **Horizontal**, which makes the dimension text horizontal with respect to drawing; then the second option, **Aligned with dimension line**, aligns the dimension text with the alignment of the dimension line; and finally, **ISO standard**, which will align dimension text as per ISO standards.

Using the Fit tab

The **Fit** tab has various settings that will help you further refine the properties of a dimension style. Most of these settings are self-explanatory and a good amount of description is already available on the respective panel of the **Fit** tab:

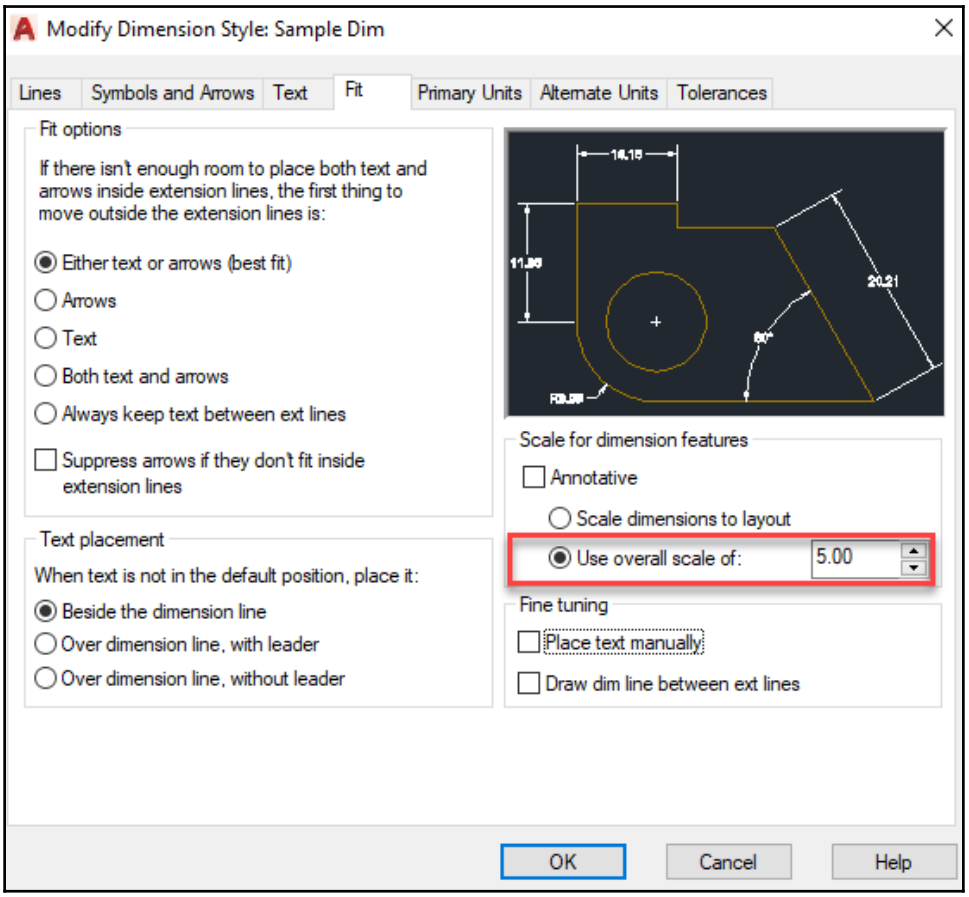


Fig 6.55: The Use overall scale of option on the Fit tab

The option we will look at on the **Fit** tab is the **Use overall scale of** field. If your dimension and arrow size is very tiny or very big, then you can change the overall scale of the dimension style from this option. As an example, if your drawing text and dimension looks like image **A** in the following screenshot, then change the value on the **Fit** tab from `1` to `5` and you will notice that all the parameters, such as dimension text size, arrow size, and various gaps, will increase by five times. The resulting dimension will look like **B** in the following screenshot:

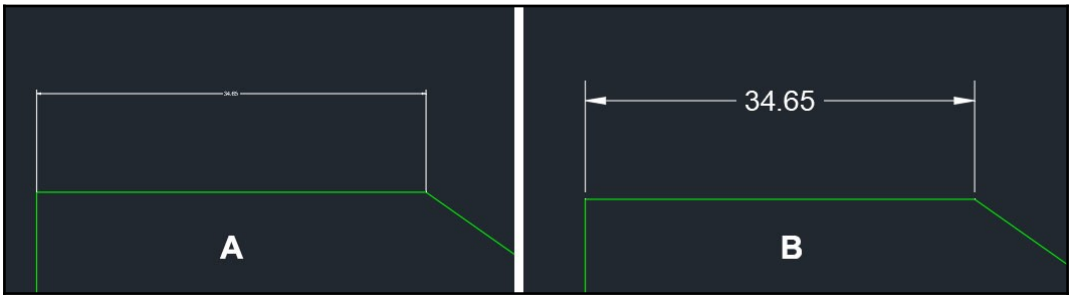


Fig 6.56: Dimension size before and after changing the scale from the Fit tab

This method of increasing the size of the dimension is generally helpful when you want the size of the dimension to scale with respect to the drawing. From the **Fit** Tab, this is the only option that we will look at now, but feel free to explore other options on the **Fit** tab. The next tab that we will look at is the **Primary Units** tab. The options on this tab will help you add dimension unit formatting and precision related settings.

Using the Primary Units tab

Using the **Primary Units** tab, you can change settings such as the units and the precision of a dimension. There are two **Primary Units** panels—one for **Linear dimensions** and another for **Angular dimensions**:

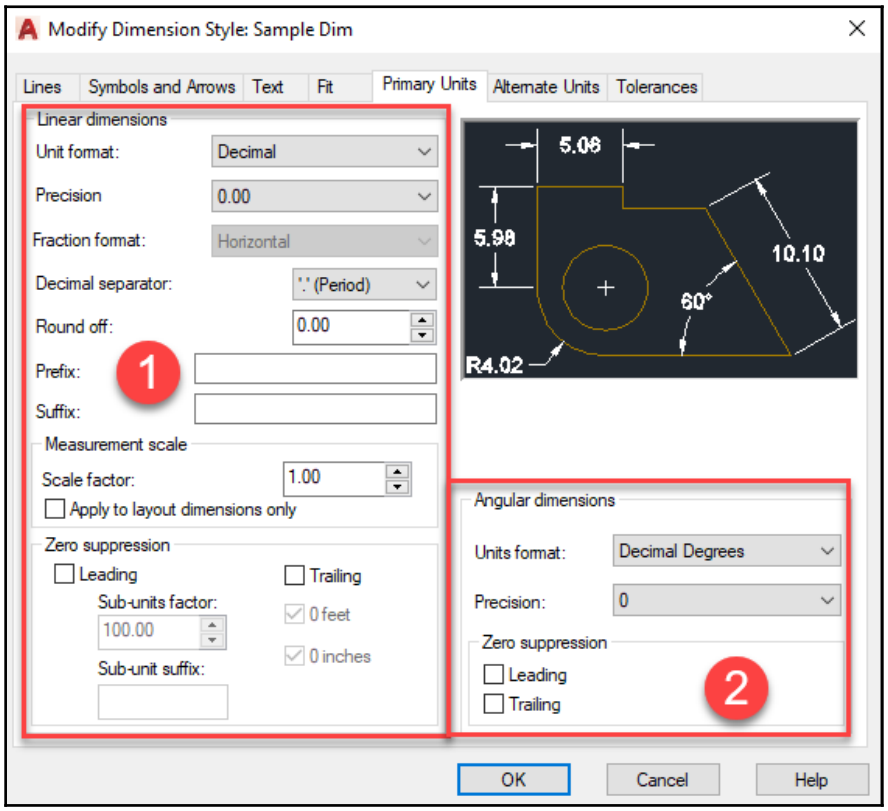


Fig 6.57: The Linear dimensions and Angular dimensions panels of the Primary Units tab

To change the unit display format of linear dimensions, click the **Unit format** dropdown and select the unit you want to display from the **Linear Format** panel. Here are some examples of dimensions displayed with different unit formats:

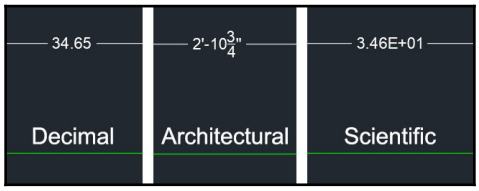


Fig 6.58: Dimensions displayed in the Decimal, Architectural, and Scientific formats respectively

The **Precision** dropdown can be used to change the display precision of the unit and this basically controls the number of decimal points visible in the dimension. The **Decimal separator** field changes the decimal separator to a period, comma, or space.

Similarly, in the **Angular dimensions** panel, we have settings for angular dimensions and the first drop-down menu here is for the **Unit format** for the angle. You can choose between different angle formats such as **Decimal Degrees**, **Degree minute second**, and **Radians**. The next drop-down menu is **Precision** and this again controls the number of decimal places in the angular dimension.

We will discuss the last two tabs, **Alternate Units** and **Tolerances**, after finishing the basic dimensioning tools later in this chapter. In the next section, we will learn about adding dimensions to our drawing using different tools, such as **Linear**, **Aligned**, **Baseline**, and **Ordinate**.

Adding dimensions

So, now you are familiar with the basic dimensioning tools, it's time to start adding dimensions to our drawing. For this, I will be using the drawing shown here, where vertices are labelled with letters:

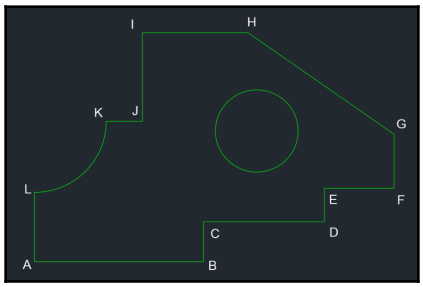


Fig 6.59: Sample drawing to be used for adding dimensions

In the following example, we will learn how to add our first dimension to this sample drawing and we will start with the **Linear** dimension:

1. Click the arrow right next to the **Linear** dimension tool on the **Annotation** panel on the **Home** tab, as shown in the following screenshot:

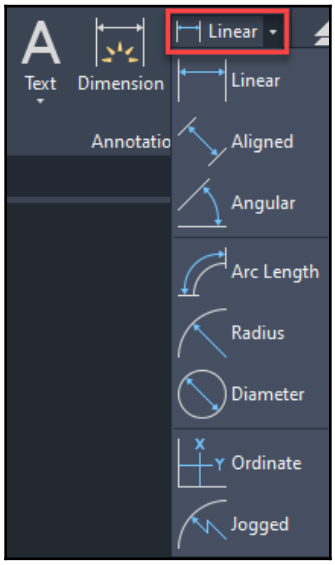


Fig 6.60: The Linear tool and other dimension tools on the Dimension flyout of the Annotation panel

2. A menu with a list of dimension tools will show up, click **Linear** from this list of dimension tools.
3. Now click on point **A** and then on point **B** and then move your cursor away from line **AB** and click again to place the dimension.

You've added your very first dimension in the drawing and, as you might have noticed, AutoCAD will automatically find out the length of the line and will put that length on your dimension line:

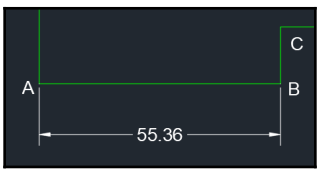


Fig 6.61: Linear dimension added on line AB

You can add this dimension in a slightly different way as well, using the following example:

1. Select the **Linear** dimension tool once again and then, instead of clicking on any point, directly press *Enter* key.
2. Now click line **AB** and then click again some distance from the line to place your dimension.
3. You will notice that, in this case, AutoCAD automatically selected the endpoints of the line **AB** and will place the dimension accordingly.

The dimensions added here are associative, which means if you change the geometry, the dimension will update itself. So, in this example, if you increase the length of line **AB**, the dimension will automatically update to reflect the updated length of the line.

Using the Linear dimension tool

The **Linear** dimension tool will add horizontal or vertical linear dimensions only. So, the very first dimension that we added here was a horizontal one from point **A** to **B**. Now, select the **Linear** dimension tool again and then click on point **H** and then on point **G** and move your cursor away from line **HG** toward the right and you will notice that now we are adding a vertical dimension, as in the following screenshot:

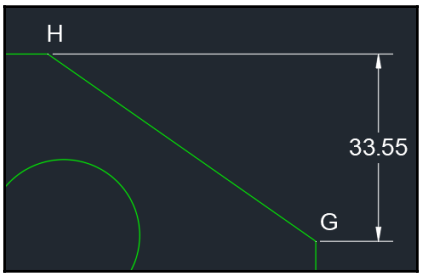


Fig 6.62: Vertical linear dimension between points G and H

Line **HG** is inclined to an angle, still the linear tool will only create the dimension that is either horizontal or vertical and the type of dimension, that is, horizontal or vertical, that you will get in this case will depend on the point where you put it. So, instead of moving toward the right, if you move your cursor in a downward or upward direction after selecting points **G** and **H**, a horizontal dimension will be created.

Using the Aligned dimension tool

In the previous example, we were able to add either a horizontal or vertical dimension on line **HG** using the **Linear** tool. If you want to add a dimension that is aligned to the line, then select the **Aligned** option, which is the next option in the **Dimension** menu. Select the **Aligned** tool, then click on point **H** and then on point **G** and move your cursor away from line **HG**:

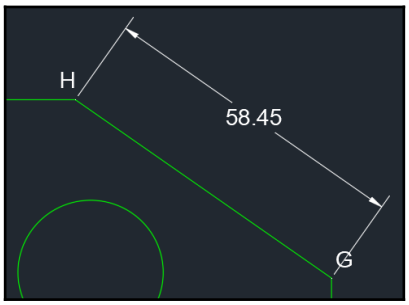


Fig 6.63: Aligned dimension added between points G and H

You will now get a dimension that is parallel to the aligned line **HG**, as shown in the preceding screenshot. The **Aligned** dimension tool can be used for horizontal as well as vertical dimensions, but we still need the **Linear** dimension tool as there may be cases where you would want to apply a linear dimension to even an inclined line and the **Linear** dimension tool will work for those situations.

Using the Angular dimension tool

As the name suggests, this dimension tool will help you add angle dimensions. In the following example, we will add an angular dimension on vertex **G** of our sample drawing:

1. Select the **Angular** tool from the **Dimension** drop-down menu and then click on line **HG** and then on line **GF** and now you will see the angular dimension on your cursor.
2. Move your cursor away from vertex **G** toward the left and you will see an angle value toward that side. If you move toward the right, you will see the angle value along that side of the lines.
3. Click on the side of the vertex where you want to put the dimension. In my case, I added the angular dimension toward the left and it looked like the following screenshot:

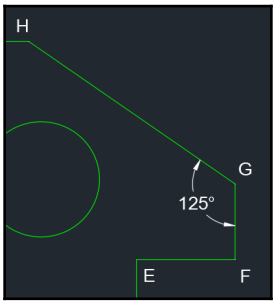


Fig 6.64: Angular dimension added on vertex G

As shown in the preceding example, you can use the **Angular** dimension tool to add angles to intersecting lines. The **Angular** dimension tool can be used for lines that are not actually intersecting but that, if extended, would intersect. The **Angular** dimension tool can't be used for parallel lines.

Using the Arc Length dimension tool

This dimension tool will show the length of the arc along the circumference. Don't confuse the arc length with the radius or diameter of the arc; there are other dimension tools for radius and diameter. To use this tool, click on the **Arc Length** option and then click on arc **KL** in the drawing and click at another point away from arc **KL** to place the dimension. You will now get an arc length dimension that will have an arc length symbol preceding it, as shown in the following screenshot:

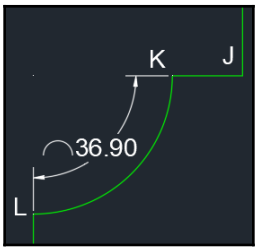


Fig 6.65: Arc length dimension added on arc KL

The preceding arc length symbol will appear before the arc length dimension automatically but you can turn this symbol off in the **Dimension Style Manager**.

Using the Radius dimension tool

This dimension tool will add the radius dimension on a circle or arc. In the following example, we will add the radius dimension on arc **KL** of the sample drawing:

1. Select the **Radius** tool from the dimension drop-down menu and then click on arc **KL**, then move your cursor away from arc **KL** and click again to place the dimension.
2. The radius dimension will be added now, and the dimension will have the **R** prefix, indicating the radius value.
3. You can add a radius dimension to a circle as well, using a similar workflow:



Fig 6.66: A radius dimension added on arc KL

After adding the radius dimension on the arc, my dimension looked like the preceding screenshot.

Using the Diameter dimension tool

Just like the **Radius** tool, you can use the **Diameter** dimension tool to add the diameter to a circle or arc. In the following example, we will add the diameter dimension to the circle of our sample drawing:

1. Select the **Diameter** tool and then click on the circle.
2. The diameter value will now show up on your cursor with the **Ø** symbol as the prefix.
3. Click at any point away from the circle and the diameter dimension will be added:

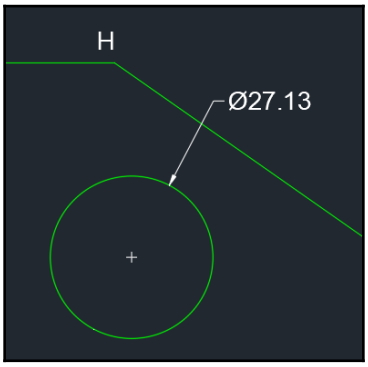


Fig 6.67: Diameter dimension added on the circle

The **Ø** symbol indicates the diameter value just like **R** indicated the radius value. My diameter dimension on the circle looked like the preceding screenshot.

Using the Ordinate dimension tool

This is another great way of adding linear dimensions to your drawing and is most frequently seen in drawings that contain lots of features, such as a sheet metal part containing many grooves and punches. To use this dimensioning tool, we will first move point **A** of our example drawing to the origin. When we are done adding the ordinate dimensioning, you will understand the significance of moving point **A** to the origin:

1. To move the drawing so that point **A** goes to the origin, select the **Move** tool from the **Modify** panel, then select the entire drawing and press the *Enter* key.
2. Now click on point **A** to select it as a base point and then type `0,0` and press *Enter* again.
3. The drawing will now move so that point **A** overlaps the origin.

Now we are ready to add the ordinate dimension to our drawing and in the following example, we will learn how to do just that:

1. Click on the **Ordinate** tool from the **Dimension** drop-down menu and click on point **A**.
2. Move your cursor below line **A** and click again to place the dimension. You will notice that the dimension value now displays `0`, which is basically the **X** coordinate value of point **A**.

3. Select the **Ordinate** tool again and click on point **B** and again move the cursor below line **AB** and click again. This time, you will get the value of the cursor too, which is basically the **X** coordinate value of point **B**.
4. Similarly, add the ordinate dimension values to points **D** and **F**. The final dimensioning should look like the following screenshot:

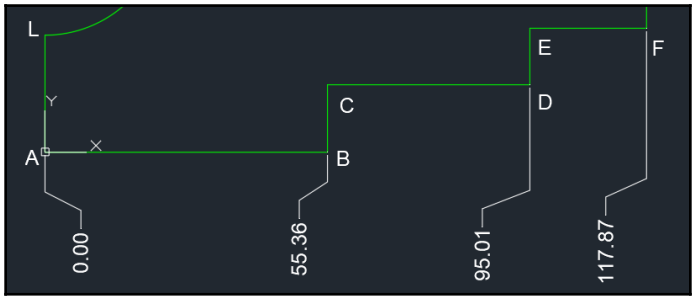


Fig 6.68: Ordinate dimension added along the X axis with point A at the origin

So, in this case, the dimensions are showing the **X** coordinate value of the selected point and since we have moved the first point, **A**, to the origin, the respective distances are measured with respect to point **A**. So, at point **B**, we have the distance between point **A** and **B**, and at point **D**, we have the distance between point **A** and **D**. Similarly, at point **F**, we have the distance between point **A** and **F**. So, essentially, using ordinate dimensioning we are adding dimensions with respect to the origin. This method of adding dimensions is cleaner than simple linear dimensions as you don't need the start and end points for every dimension.

We can also add a vertical ordinate dimension, and in that case, the distances will also be measured in the vertical direction with respect to the origin, or in this case, point **A**. The ordinate value will be the **Y** coordinate value of the selected point in the vertical direction:

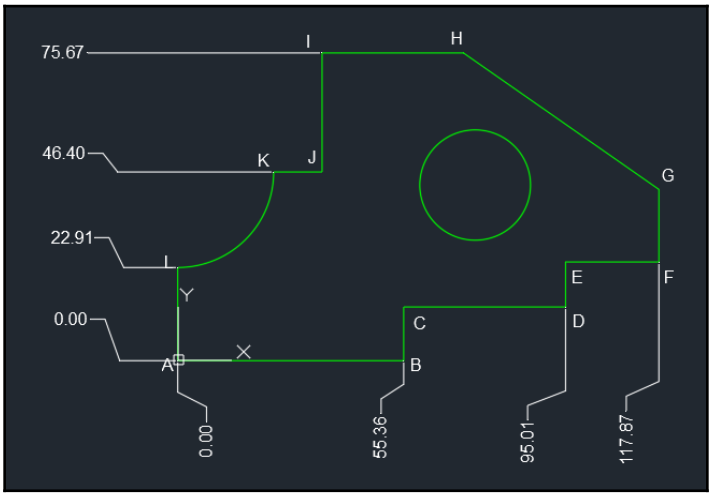


Fig 6.69: Ordinate dimensions added along the horizontal and vertical direction with point A at the origin

In my case, the vertical and horizontal ordinate dimensions looked like the preceding screenshot. Here, all the distances of points **L**, **K**, and **I** are measured with respect to point **A**, or the origin.

Using the Jogged dimension tool

This dimension tool is used to add a radius dimension to a circle or arc, but it is meant for a situation where the center of the arc or circle is located off the drawing sheet or far from the circumference of the circle or arc. This dimension tool allows you to specify the location of the center for those kinds of circles or arcs using the center location override and a zigzag line is added to the radius line, which indicates that the actual center is located toward the direction of the zigzag line.

In the following example, we will add a jogged radius to an arc:

1. Select **Jogged** from the **Dimension** drop-down menu and click on the arc or circle on which you want to add the jogged radius.
2. When the command line prompts you to specify the center location override, click at a point toward the center of the circle or arc, and then click at a point to place the dimension, and then again to place the jogged line.

3. The final dimension that you will get will look like the one shown in the following screenshot:

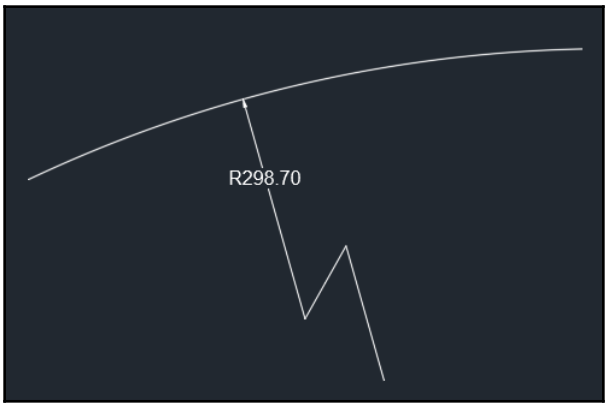


Fig 6.70: The jogged dimension added to an arc

This was an overview of the dimension tools we have in AutoCAD. You can use these tools to add almost any kind of dimension to your drawings. There are also some specialized kind of dimension tools, such as baseline and ordinate, which we discuss later in this chapter. In the next section, we will discuss the **Dimension** tool, which can make almost all of the dimension types we just discussed.

Understanding the Dimension tool

Right in the middle of the **Annotation** panel of the **Home** tab, you will find the **Dimension** tool, which is really a great way of adding dimensions, and using this single **Dimension** tool you can add almost all of the types of dimensions that we have added to the drawing so far. To explain this tool, we will once again use the drawing we have used in previous examples:

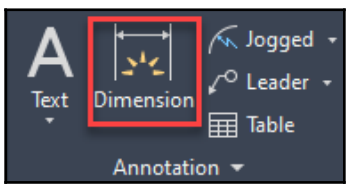


Fig 6.71: The Dimension tool on the Annotation panel of the Home tab

In the following example, we will add linear, aligned, angular, and other dimensions using the **Dimension** tool:

1. Select the **Dimension** tool from the **Annotation** panel of the **Home** tab or use its command, `DIM`.
2. Click on point **A**, then point **B**, and click away from line **AB** to make your linear dimension. Similarly, click on point **H** and then **G** and click away from line **GH** to drop the dimension and you will have an inclined dimension:

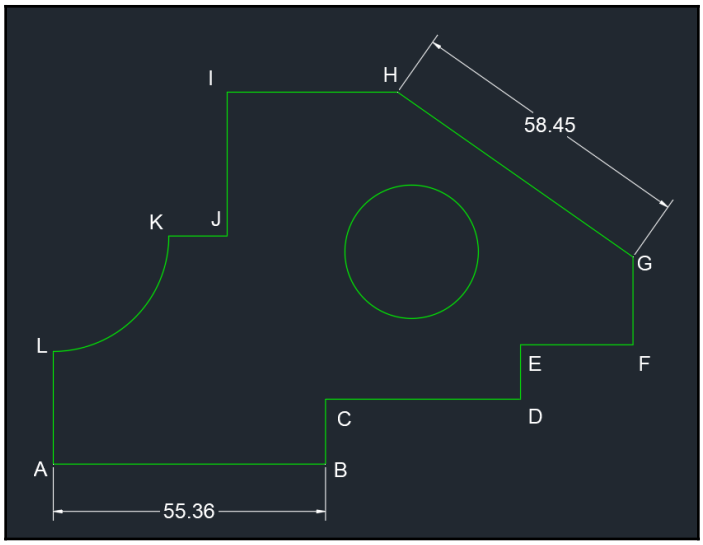


Fig 6.72: Linear and inclined dimensions added with the Dimension tool

3. For the inclined line, if you move your cursor toward the right, you will get a vertical dimension, and moving it toward the top will make a horizontal linear dimension.
4. Select the **Dimension** tool again and click on the arc **KL**, then click at a point away from the arc and you will have a radius dimension. Similarly, select the **Dimension** tool and click on the circle and you will get a diameter dimension.

5. To make the arc length dimension using the **Dimension** tool, select the **Dimension** tool then hover your cursor over the arc in your drawing and you will see an option on the command line for making the arc length dimension. Type `L` and press *Enter* and now, when you click on the arc, you will get an arc length dimension:

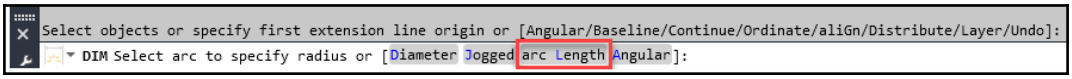


Fig 6.73: The arc Length subcommand on the command line of the DIM command

6. To make an ordinate dimension, select the **Dimension** tool, then select **Ordinate** from the command line, and now when you click at a point, you will have an ordinate dimension for that point:

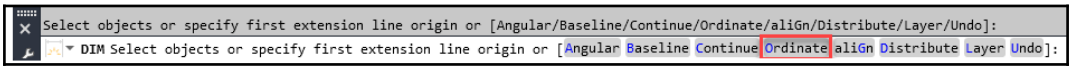


Fig 6.74: The Ordinate subcommand on the command line of the DIM command

There are also other options on the command line that you can use to add even more kinds of dimensions using only the **Dimension** tool.

Adding multiple dimensions to a drawing can take a lot of space and make dimensions look untidy as well. In the next section, we will learn about the **Continue** dimension tool, which allows us to add many linear dimensions to our drawing with a single tool and it also keeps drawing clean.

The Continue dimension tool

This method of dimensioning is also called chain dimensioning as dimensions are added like chain links, one after the other. The **Continue** dimension tool is on the **Dimension** panel of the **Annotate** tab, as shown in the following screenshot:

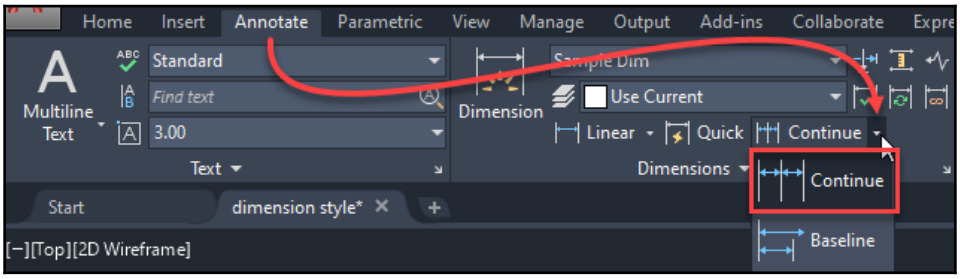


Fig 6.75: The Continue dimension type on the Dimension panel of the Annotate tab

Before you use the **Continue** dimension tool, you need to make a reference dimension, which is explained in the following steps:

1. Click on the **Linear** dimension tool from the **Annotation** panel of the **Home** tab or the **Dimensions** panel of the **Annotate** tab, or use its command alias, `DLI`.
2. Now click on point **A**, then on point **B** to make the first horizontal dimension.
3. Now select the **Continue** dimension tool and you will notice that the dimension will automatically start from point **B**, and all you now need to do is specify the next point for the dimension.
4. Click on point **D**, then on point **F** and you will end up with a continue dimension up to point F.

When you are done adding the continue dimension, press the *Esc* key to exit the command and the dimensions will look like the following figure.

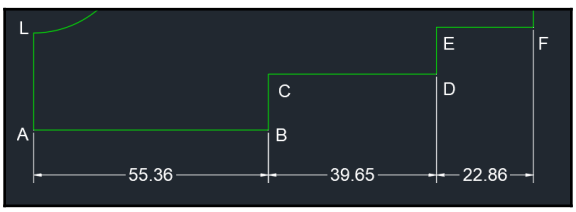


Fig 6.76: Continue dimensions added to the drawing

As you may have noticed, this method of dimensioning removes the hassle of selecting start and end points in situations where you want to make multiple dimensions in the chain form. The next dimension type, **Baseline**, is also similar, but instead of a chain, this dimension type adds a dimension using a reference point called a baseline. We will discuss it in the next section.

The Baseline dimension tool

Just like **Continue**, this is another method of adding dimensions using a reference dimension. So, here too, start with the linear dimension and click on point **A**, then point **B** to make the first linear dimension. Now select **Baseline** from the **Dimension** panel of the **Annotate** tab. The **Baseline** tool is right underneath the **Continue** tool.

Now you will notice that the next dimension starts from point **A** or the first point of the reference dimension. In this case, point **A** is the baseline for this dimension. Now click on point **D**, then on point **F** and you will get dimensions that start from baseline **A**, as shown in the following screenshot:

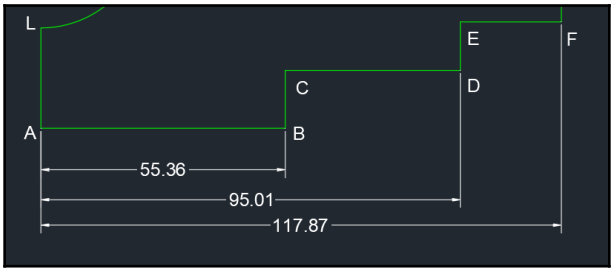


Fig 6.77: A baseline dimension added to the drawing

To exit the command, press the *Esc* key. The gap between the baseline dimensions is controlled by the **Baseline spacing** option, which is on the **Lines** tab of the **Modify Dimension Style** window. So, now we know the different types of dimension tools, it's time we learn how to modify the dimensions and their placement options in the drawing. We will discuss that in the next section.

Modifying dimensions

So far, we have seen different methods of adding dimensions to our drawing. In this section, we will learn how to modify these existing dimensions using their multi-function grips.

Let's add a linear dimension on line **AB**. Now select the dimension and you will notice multi-function grips (blue square boxes), as shown in the following screenshot:

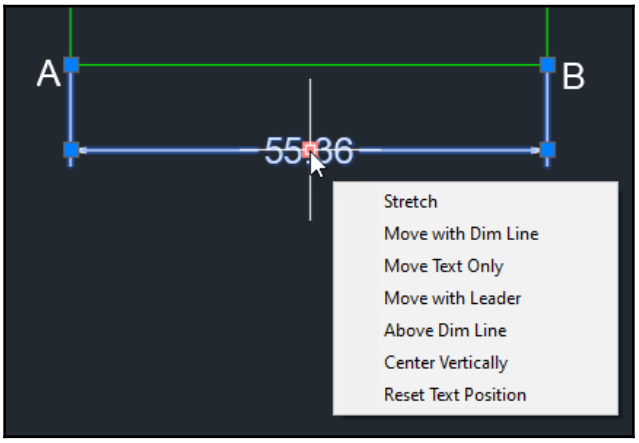


Fig 6.78: Dimension edit options on the multi-function grip menu

Hover your cursor over the grip above the dimension text and you will see a menu with a bunch of options, as shown in the preceding screenshot. You can use these menu options to further modify the dimension properties. Let's have a look at these menu options one by one:

- **Stretch**: Select this option to move the dimension text as well as the dimension line with respect to line **AB**.
- **Move with Dim Line**: This option is also very similar to the **Stretch** option and it allows you to move the dimension text along the dimension line, and this also allows you to move the dimension line.
- **Move Text Only**: This option lets you move only the dimension text along the dimension line, keeping the location of your dimension line intact.
- **Move with Leader**: This option lets you move your dimension text only, but in this case, a leader line will be made, which attaches the text to the dimension line, as shown in the following screenshot:



Fig 6.79: Dimension text with a leader line attached to it

- **Above Dim Line**: This option places your dimension text above the dimension line.

- **Center Vertically**: This option places the dimension text at the center of the dimension line. The current location of the dimension text is **Center Vertically** in the preceding screenshot.
- **Reset Text Position**: This option resets all the customizations you have made so far and resets the position of the dimension text as per the dimension style. If you hover your cursor over the end point grips of the dimension, you will get another set of options, as shown in the following screenshot. Let's discuss these options too:

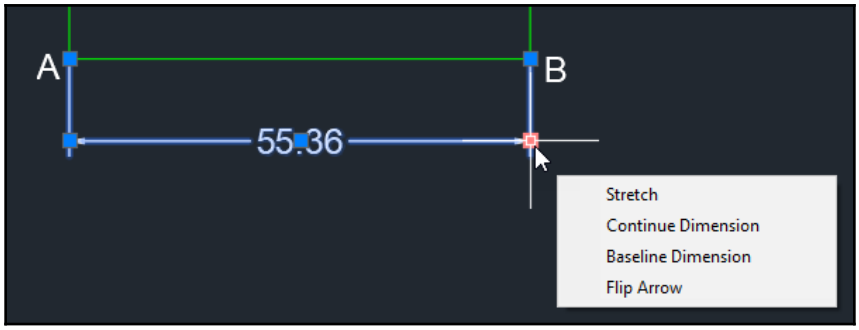


Fig 6.80: Dimension edit tools on the multi-function grip menu

- **Stretch**: Just like the previous **Stretch** option, this option lets you move the dimension line with respect to line **AB**.
- **Continue Dimension**: Select this option to start making the continue dimension from the selected grip location. If you select the grip at point **B**, then the next option, the continue dimension, will start from point **B**.
- **Baseline Dimension**: This will allow you to make the baseline dimension from the select grip as the baseline. So, in this case, our selected grip is at point **B**. The baseline will become point **A** and the rest of the dimensions will follow baseline **A**. As for the dimension on line **AB**, the baseline or the first point was point **A**.
- **Flip Arrow**: This option allows you to flip the direction of the dimension arrow. This option is especially helpful in places where you have no room for the dimension text and you want to create some room for the dimension text by flipping the arrow direction.

Similarly, if you select other dimension grips, such as the grip for the angular dimension or the radius dimension, you will get other sets of option that will be relevant to that dimension, and you can further modify dimensions using those options.

So, these were the options you can use to modify dimensions in your drawing and, depending on the type of drawing and space available for the dimension, you may need to use these tools. In the next section, we will learn about adding dimensions using multiple units, and we can do this using the **Alternate Units** feature of **Dimension Style Manager**, which is discussed next.

Using alternate units in dimensions

Generally, you will see dimensions in one unit but often it is required to add dimensions in multiple units. For those situations, you can use the **Alternate Units** feature of **Dimension Style Manager** to add dimensions in more than one unit.

In our sample drawing, we will use **millimeters** (**mm**) and **inches** (**in**) as the two units. In the following example, I will use inches as the primary unit and millimeters as the alternate unit.

1. Open the drawing and type `UN` and press *Enter*.
2. Now set the insertion unit to **Inches** and the unit type to **Decimal**, then click **OK.**

So now we have set our unit format to inches, it's time we modify the dimension style as well. We will do it in the following example:

1. Select the **Dimension Style Manager** option from the expanded **Annotation** panel of the **Home** tab as shown in the following image or use its command alias, `D:1`.

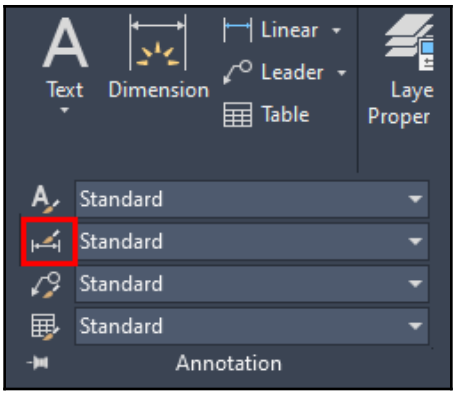


Fig 6.81: Dimension style manager option on Annotation panel

2. Select the `Sample Dim` dimension style that we created in the earlier examples and click the **Modify** button on the **Dimension Style Manager** window.
3. Now go to the **Alternate Units** tab on **Modify Dimension Style** window and click the first checkbox at the top, which says **Display alternate units**. The options on the **Alternate Units** tab will become active and you can now modify them.
4. Select **Decimal** as the **Unit format** and **Precision** of two decimal places.
5. In **Multiplier for alt units**, type `25.4`. This value of `25.4` will be multiplied by the primary unit to get the alternate unit. As you know, 1 inch equals 25.4 mm, hence this multiplier will convert the primary unit into the alternate one.
6. Click **OK** when you are done making changes and close the **Modify Dimension Style** and **Dimension Style Manager** window:

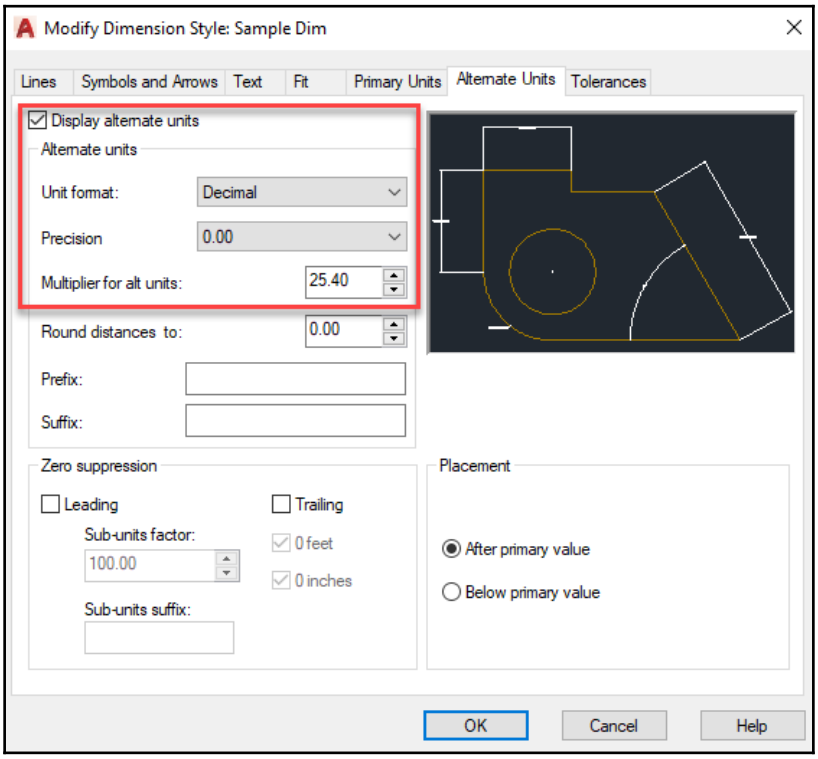


Fig 6.82: Alternate unit options on the Alternate Units tab

7. Add a linear dimension on line **AB** and, this time, the dimension will show in inches as well as in millimeters. The first dimension is in inches and the alternate dimension will show in square brackets in millimeters, as shown in the following screenshot:

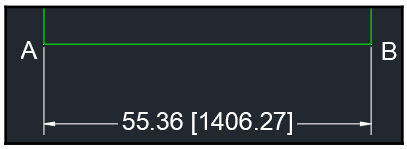


Fig 6.83: Dimension with inches as the primary unit and millimeters as the alternate unit

8. In a similar way, you can use the **Alternate Units** tab to create any alternate unit dimension using different multiplier values. To get rid of the alternate unit, simply modify your **Dimension Style Manager** and uncheck the **Display alternate units** checkbox on the **Alternate Units** tab of the **Dimension Style Manager** window.

So, now we know about adding multiple dimension unit types, it's time to go back to the **Dimension Style Manager** to learn about adding tolerances to our dimensions. We will learn about adding tolerances, which are an essential part of mechanical drawings, in the next section.

Adding tolerances

Drawings made in any CAD drafting software are precise, but in the real world, the parts are not precise and there is always some tolerance or deviation from the theoretically exact dimensions, which can be displayed in the drawing using the **Tolerances** feature of **Dimension Style Manager**.

Let's add tolerance to our example drawing, and we will start with the symmetrical tolerance type:

1. Open the drawing and open the **Dimension Style Manager** window using the **Expanded Annotation** panel of the **Home** tab or using its command alias, `D`.
2. Now select the **Sample Dim** dimension style and click on the **Modify** button.

3. Now select the **Tolerances** tab, and from the **Method** drop-down menu of the **Tolerance format** panel, select **Symmetrical** and **Precision** of two decimal places.
4. In the **Upper value** field, type `0.5` and click **OK** to close the **Dimension Style Manager** with these changes:

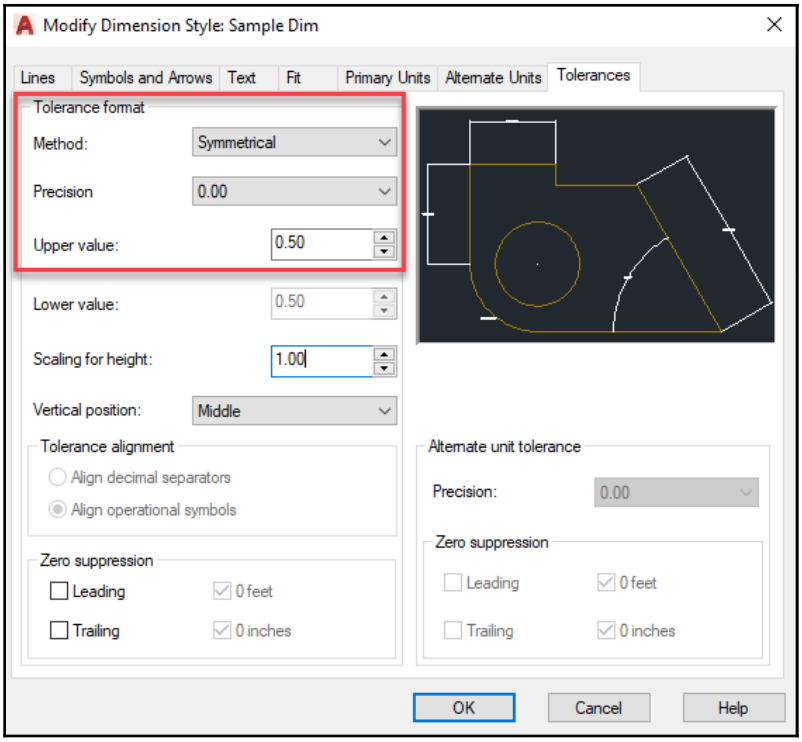


Fig 6.84: Tolerance options on the Tolerances tab

5. Now apply a linear dimension on line **AB** and the dimension should look like the following screenshot:

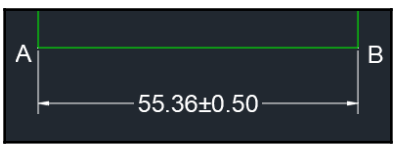


Fig 6.85: Dimension with symmetrical tolerance of 0.50 applied to it

As we used symmetrical tolerance, we got tolerance of `±0.50` for the upper and lower limit. If, however, you want separate upper and lower limits, then change **Tolerance format** to **Deviation** and specify separate upper and lower limits as shown in the following screenshot:

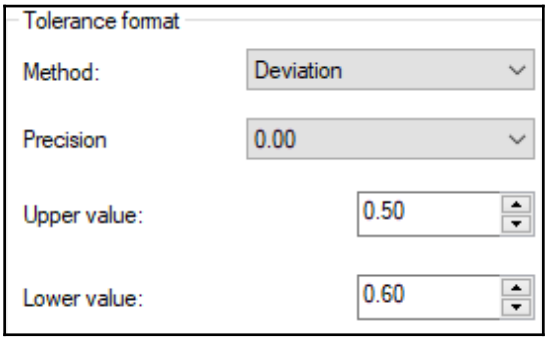


Fig 6.86: Deviation tolerance with different upper and lower limit values

The resulting tolerance will look like the following screenshot:

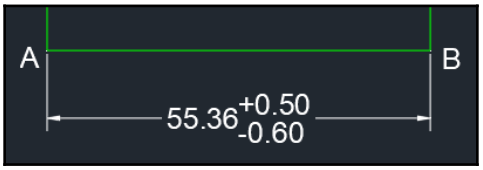


Fig 6.87: Dimension with deviation tolerance applied to it

You can also try other tolerance formats from the **Method** drop-down menu of the **Tolerances** tab in Dimension Style Manager. To remove tolerance from your dimension, switch back to **None** in the **Method** drop-down menu.

The dimensions are added automatically and whenever you add a dimension to your drawing, AutoCAD finds the right value and adds it for you. You can certainly modify this dimension by double-clicking on the dimension value, but in that case, the dimension will lose its associativity and modifying geometry will no longer change the dimension automatically.

If you, however, want to label an object with your own custom text instead, then multileaders are is the best tool for that and we will explore them in the next section.

Multileaders

Multileaders are added to a drawing when custom information is needed; instead of relying on dimensions, you need to put in your own values. As an example, here, we have a subassembly where different parts are labelled using multileaders:

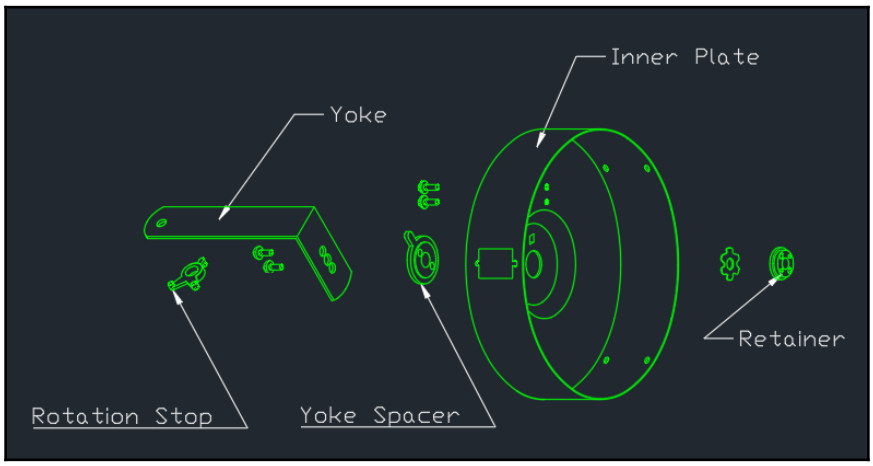


Fig 6.88: Parts of an assembly labeled with multileaders

In this section, we will learn to create a multileader style and then use that multileader style to label different parts of the subassembly shown in the preceding example.

Creating a multileader style

Just like a dimension style, you can create a multileader style in your drawings as well, and then assign a different set of properties to it. In the following example, we will learn how to create a custom multileader style:

1. Click the multileader style icon from the expanded **Annotation** panel of the **Home** tab, as shown in the following screenshot, or use its command alias, `MLEADERSTYLE`:

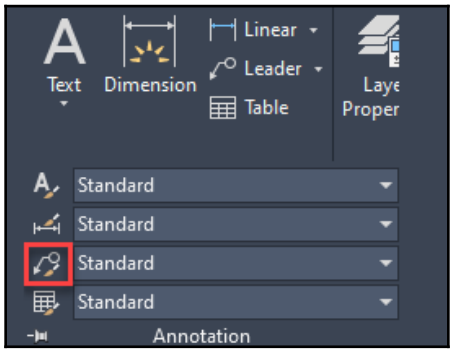


Fig 6.89: Multileader style option in the expanded Annotation panel

2. Click the **New** button in the **Multileader Style Manager** window and give this new multileader style a name. For this example, let's call it **Sample leader**. Click the **Continue** button after adding this name.
3. The **Modify Multileader Style** window, shown in the following screenshot, will open with three tabs—namely, **Leader Format**, **Leader Structure**, and **Content**:

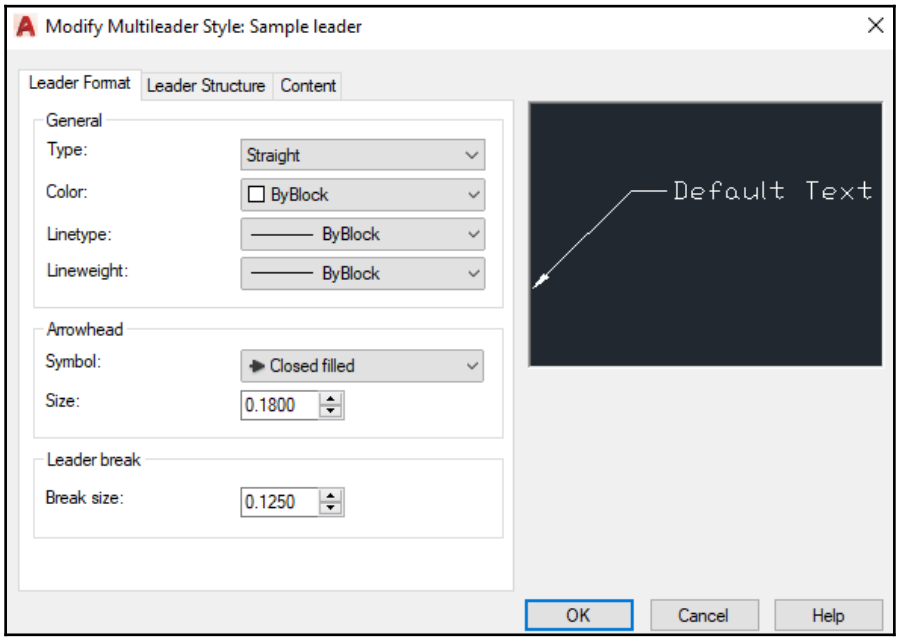


Fig 6.90: The Modify Multileader Style window with options to modify the style of the multileader

4. Click **OK** to close the **Modify Multileader Style** window and you will have a new multileader style called **Sample leader** in the **Multileader Style Manager**. Close the **Multileader Style Manager** window by clicking the **Close** button.

Once you have the multileader style in your drawing, you can open it again and modify the properties. Open the **Multileader Style Manager**, then select the **Sample leader** style and click on the **Modify** button. Now the **Modify Multileader Style** window, as shown in the preceding screenshot, will open. We will discuss the options in this **Modify Multileader Style** window in the next section.

Using the Leader Format tab

You can change the **Type** of leader line to **Straight**, which makes the leader line use straight lines; **Spline**, which uses spline for the leader line; and none, which will remove the leader line completely from the **Type** drop-down menu of the **General** panel. Other options on the **General** panel are the **Color** of the leader line, its **Linetype**, and **Lineweight**:

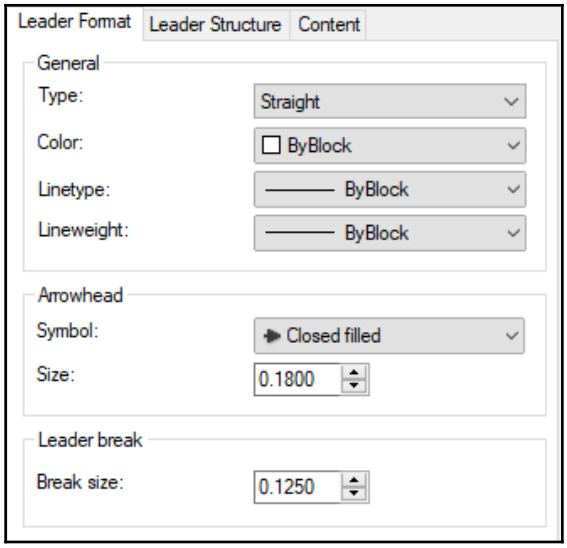


Fig 6.91: Options on the Leader Format tab

The next panel has options for changing the arrowhead type of the leader line and then the size of the arrowhead.

Using the Leader Structure tab

On the **Leader Structure** tab, we again have lots of settings, but the settings we will discuss here are **Maximum leader points** and **Scale**. **Maximum leader points** will determine the number of points required to make the leader line. The **Scale** option will increase the overall scale of the leader line, and by doing so, we will essentially make everything in the leader line, such as the arrow size and the text, bigger or smaller. So, if your leader arrow and text look too small or too big, then change the value of **Scale** to fix it:

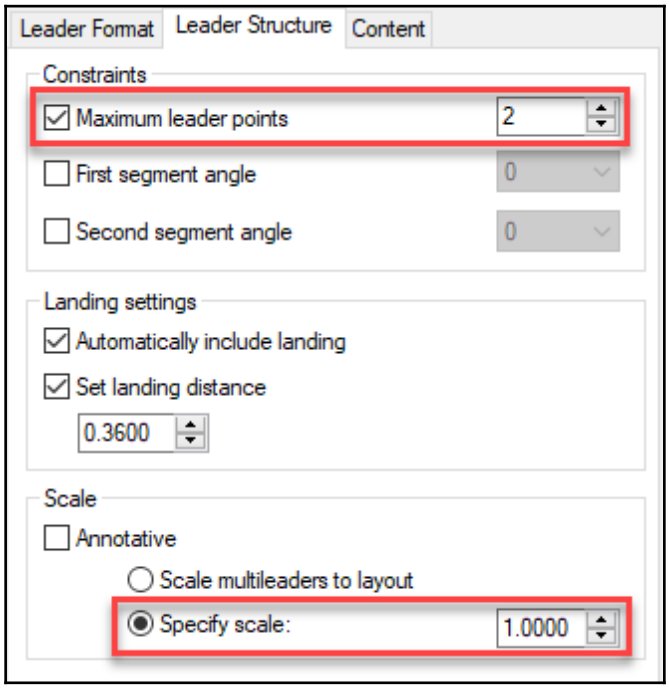


Fig 6.92: Maximum leader points and multileader scale fields on the Leader Structure tab

The last tab on the **Modify Multileader Style** window is **Content**, and from this tab, you can change text-related settings.

Using the Content tab

This tab has options related to the content of the multileader, and that content can be text or a block. From the **Multileader type** drop-down menu, select **Mtext** and you will see text-related options in the **Content** panel, as shown in the following screenshot:

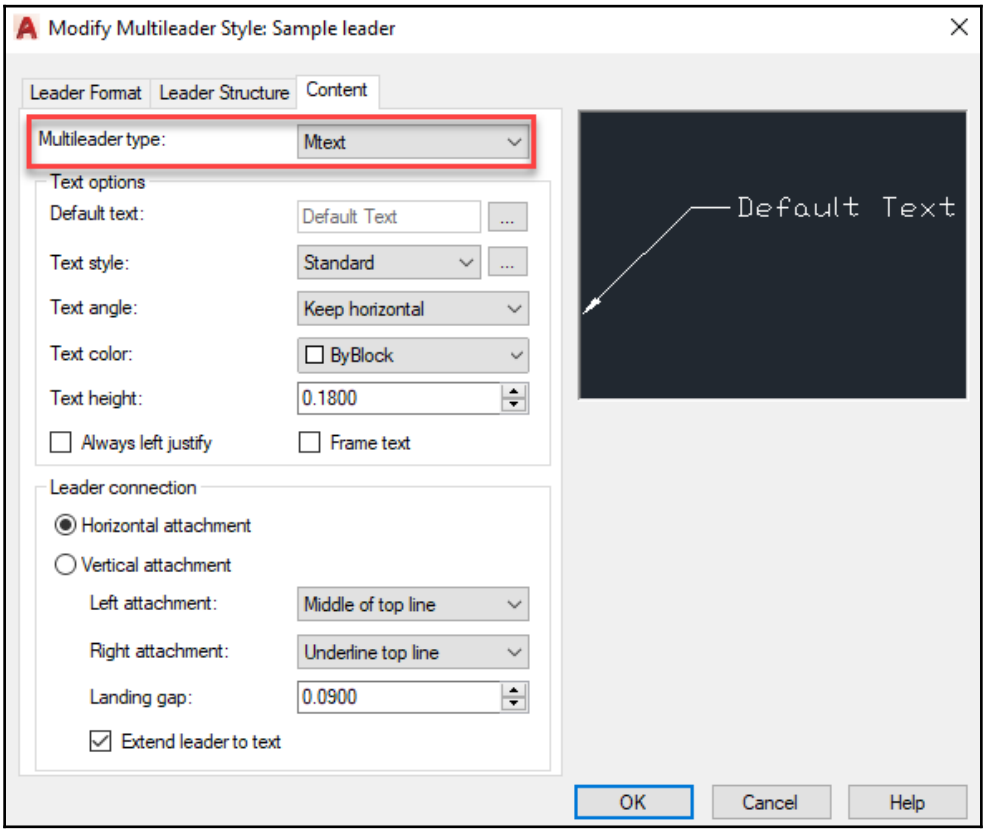


Fig 6.93: The Multileader type option on the Content tab

You can change the **Text style**, **Text angle**, **Text color**, and **Text height** using the fields shown here:

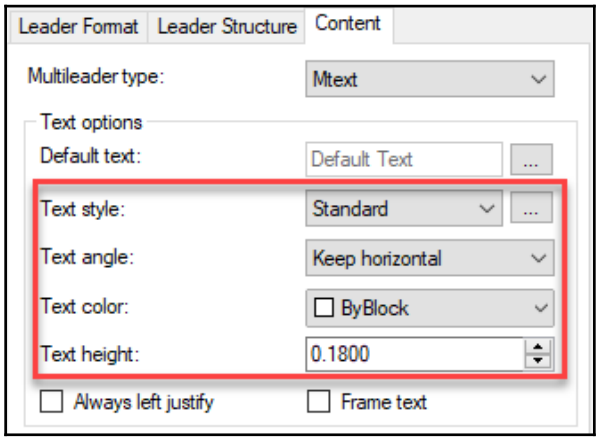


Fig 6.94: Text settings on the Content tab

If you select **Block** from the **Multileader type** drop-down menu, the options on the **Content** tab will change completely to show you options related to multileaders containing blocks. For our example, we will select a multileader with **Mtext** in the **Multileader type** drop-down menu.

So, now you know about adding and modifying a multileader style, which changes the way a multileader looks, it's time to add the actual multileader to our drawing. We will do this in the next section.

Adding a multileader

We will add a couple of multileaders to our sample subassembly to label different part names. In the following example, we will use the multileader style we created in the previous example to create multileaders:

1. Select the **Leader** tool from the **Annotation** panel of the **Home** tab or use its command alias, `MLD`:

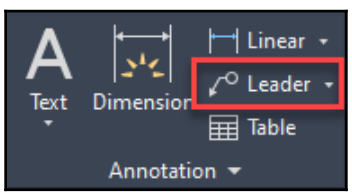


Fig 6.95: The Leader tool on the Annotation panel of the Home tab

2. Click inside the L-shaped part on the left and then click again to place the next point of the multileader.
3. Now you can add your text at the end of the multileader. Type `Yoke` and then click outside the textbox to finish adding the multileader.
4. The multileader will be added as shown in the following screenshot. Here, the arrow is added at the first point and the text at the second:

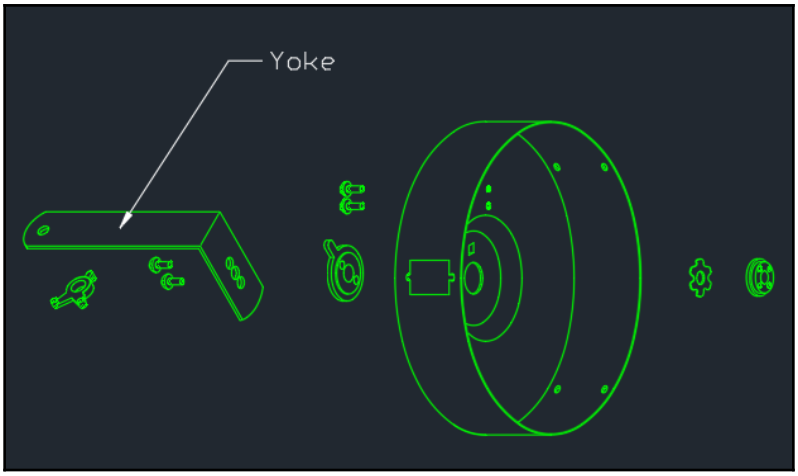


Fig 6.96: Multileader with an arrowhead on the part and text at the other end

In a similar way, you can add other multileaders to the drawing. To modify a multileader, select it from the drawing area, then click on the grips to modify the location of the text or arrow of the multileader.

With this topic, we have finished this chapter. Before we move on to the next chapter, let's summarize the topics we have learned about in this chapter.

Summary

In this chapter, we learned to fill the enclosed area with hatches and gradients and we also learned to add annotations including text, dimensions, and multileader. We not only learned to create these objects but we also learned to modify the properties of these objects as per our drawing.

In the next chapter, we will learn about making tables, and isometric drawings, and we will also learn to use parameters and constraints to make a drawing.

7
Tables, Isometric, and Parametric Drawings

In this chapter, we will explore some advanced tools, including isometric and parametric drawings. Isometric drawing is a method of making 3D drawings inside a 2D workspace, and parametric drawing is a technique that enables you to make drawings with the help of parameters. In parametric drawings, you can make different parts of a geometry interconnected; this kind of drawing is used for making simple dynamic blocks as well.

We will learn how to add tables to our drawings using the table feature of AutoCAD, and we will also learn how to insert Microsoft Excel tables into our drawings. These tables can be used for objects such as a bill of materials. Tables can be added as linked entities, using data links, or as standalone AutoCAD tables. We will learn about both methods in this chapter.

By the end of this chapter, you will be fairly confident with parametric drawing techniques and will also be able to make isometric drawings and use tables in AutoCAD.

The following are the topics that will be covered in this chapter:

- Working with AutoCAD tables
- Understanding fields
- Making isometric drawings
- Working with parametric drawings

So, let's begin this chapter by creating tables right inside AutoCAD using the table tool.

Working with AutoCAD tables

Just like text, dimension and multileader tables are also used for adding annotations to a drawing. Tables can be used to create a bill of materials or title block, or even just to add a simple set of information. The application of the table tool in AutoCAD is limited only by your imagination.

In AutoCAD, you can make a table directly or you can also import an external table made with Microsoft Excel (or other programs) that is saved in Excel format.

In this section, we will start by learning how to add a simple AutoCAD table, which will be a simple cost estimation table in this case. After using the simple table tool, we will move on to learn about importing external tables and exporting AutoCAD tables as well. We will also learn how to make table styles within AutoCAD, which will basically define the properties of our table.

So, let begin by creating a simple table in AutoCAD.

Making a table in AutoCAD

The **Table** tool is on the **Annotation** panel of the **Home** tab, as shown in the following screenshot. In the following example, we will start by making a table and then we will add information to it:

1. You can start the **Table** tool from the **Annotation** panel or use its command alias, `TABLE`:

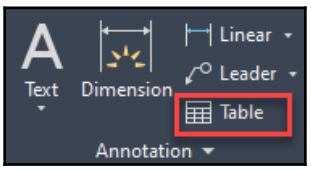


Figure 7.1: The Table tool in the Annotation panel

2. The **Insert Table** window with all the table insertion options will open. Click the **Specify insertion point** radio button from the panel toward the right side of the **Insert Table** window, as shown here:

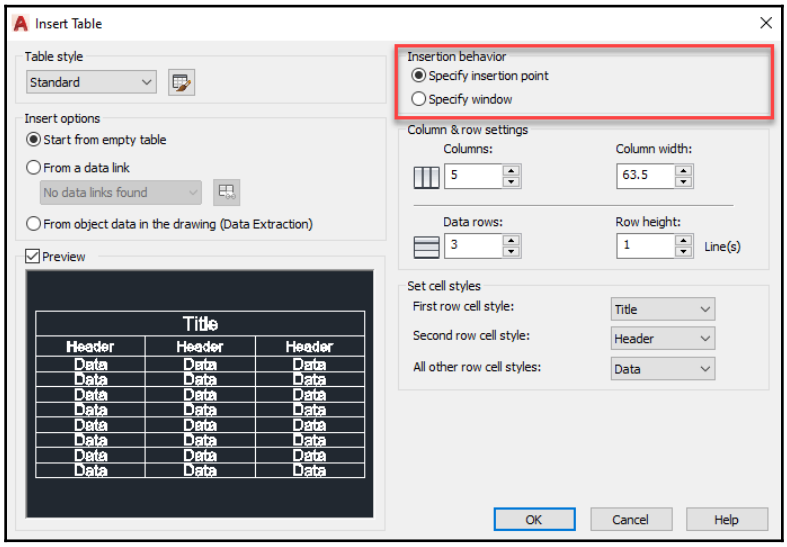


Figure 7.2: The Specify insertion point option in the Insertion behavior section of the Insert Table window

3. In the next panel, you can specify the number of columns, column width, number of rows, and row height. Type `5` in the **Columns** field and `50` in the **Column width** field.
4. Type `4` in the **Data rows** field and, in the **Row height** field, type `1`, as shown in *Figure 7.3*:

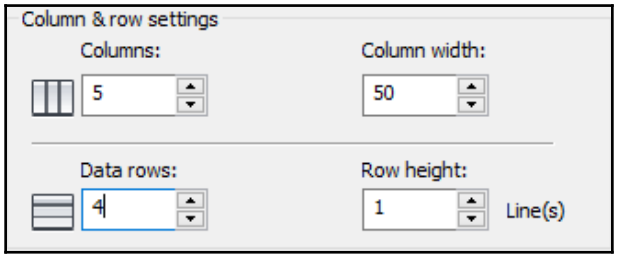


Figure 7.3: The Data rows and Row height fields

5. Leave the options in the **Set cell styles** panel unchanged, set **First row cell style** to **Title**, **Second row cell style** to **Header**, and **All other row cell styles** to **Data**. **Title**, **Header**, and **Data** are just the types of formatting that will be applied to the first, second, and remaining rows of the table.

6. Click **OK** and you will have a table. Click on a point in the drawing area and the table will be added as shown in *Figure 7.4*. If the `TABLE` command is active even after adding the drawing, press *Esc* to exit the `TABLE` command:

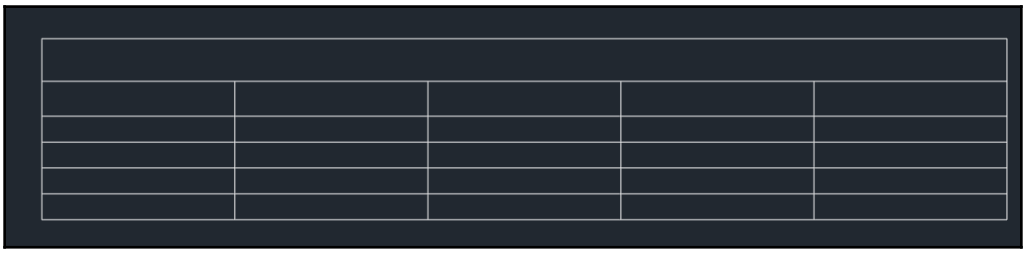


Figure 7.4: A table made with four data rows, one header row, one title row, and five columns

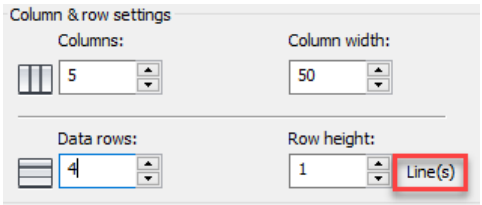
As you will notice, although we created a table with four rows, we ended up with a table containing six rows. The two extra rows are the **Title** and **Header** rows, which are added automatically as we selected them in the **Set cell styles** panel of the **Insert Table** window. The number of columns will remain exactly as defined in the **Insert Table** window.

The specific case of row height in AutoCAD tables

If you check the height of the row in the table that we just created, it will be different to the value that we specified in the **Insert Table** window. In our example, we specified the row height as 1 unit but, upon checking, I found it to be 6.667 units; yours could be different. So, why this difference?



If you look at the **Row height** field, you will notice that it has been labeled in a slightly different way:





The **Row height** field has **Line(s)** next to it, as highlighted in the preceding screenshot. So, instead of adding absolute row height, here we are adding the height in terms of lines, where "one line height" equals a number obtained by taking into consideration the text height and the margins of the text. So, the row height is controlled by the number of lines instead of the absolute height.

If you change the workflow of inserting a table slightly and instead of selecting **Specify insertion point**, you select the **Specify window** option from the **Insert Table** window, then you will have access to only one option each (for columns and rows) from the **Column & row settings** section, as shown in the following figure:

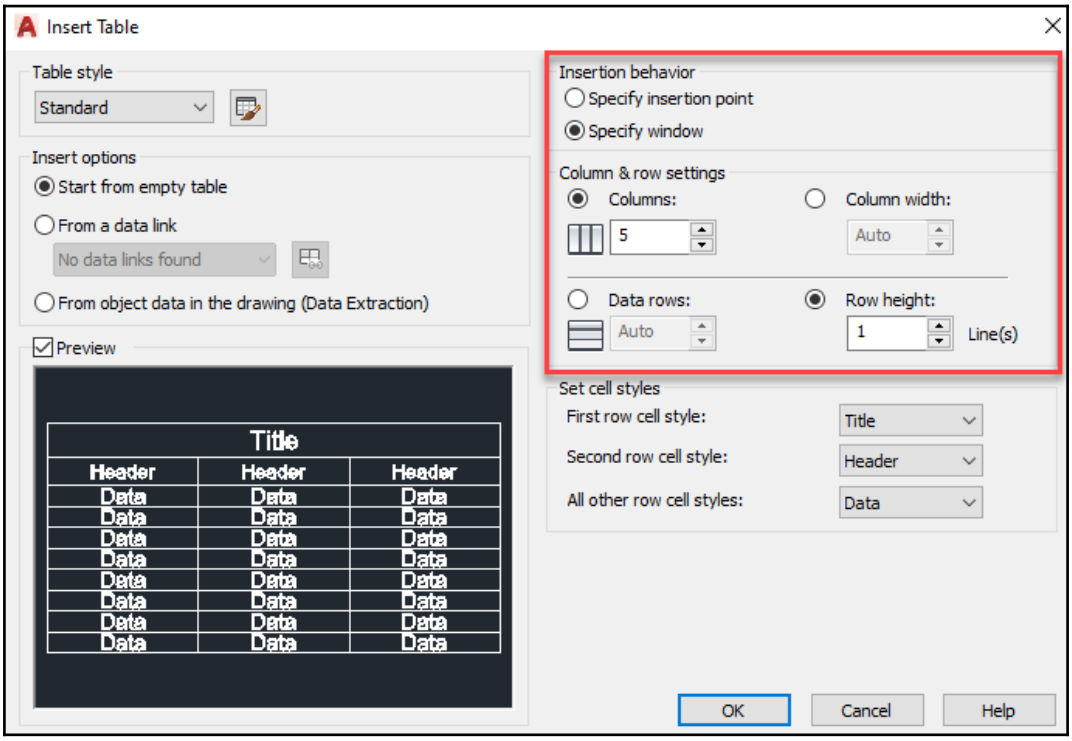


Figure 7.5: Only one field each in the Column & row settings section is active when the Specify window option is selected

Here you can choose to configure only one option each (for columns and rows) from the **Column & row settings** section; the other options will be set automatically when you add the table in the drawing area:

1. In this example, select the **Columns** and **Data rows** radio buttons, add `5` columns and `3` rows, then click **OK**.
2. Click on a point in the drawing area and move your cursor. You will notice that now you can increase the row height and column width by moving your cursor, and hence these options were disabled.
3. Click again to make the table with the **Specify window** option active. If, after making the table, the command is still active, press the *Esc* key to exit the command.

A table made using this method is much more dynamic and offers us the flexibility of changing the size as per the available drawing area.

Modifying tables

So, now that you know how tables are added to a drawing, let's move on to modifying a table and adding text to it. Select the table you made in the previous example and you will now notice the blue multi-function grips. You can click and drag these grips to change the size of different features of your table, as shown in *Figure 7.6*:

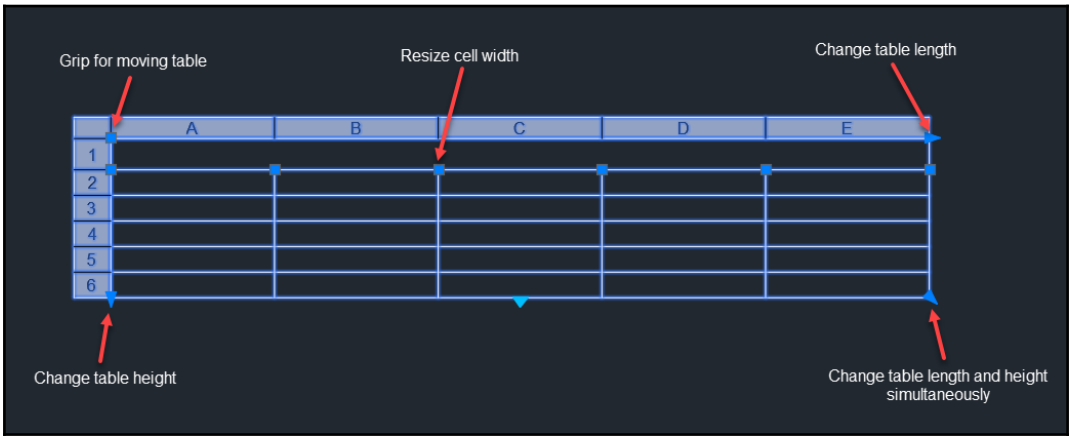


Figure 7.6: Grips for modifying and resizing the table

Try changing the size of the table you have created using these grips. The table size can be changed even for tables that were made with specific row and column height width options.

Press the *Esc* key to get rid of all selections or commands and then click once inside any of the table cells. You will notice that the cell is highlighted, and you will also get a **Table Cell** tab, which has options related to changing table cell settings, as shown in *Figure 7.7*:

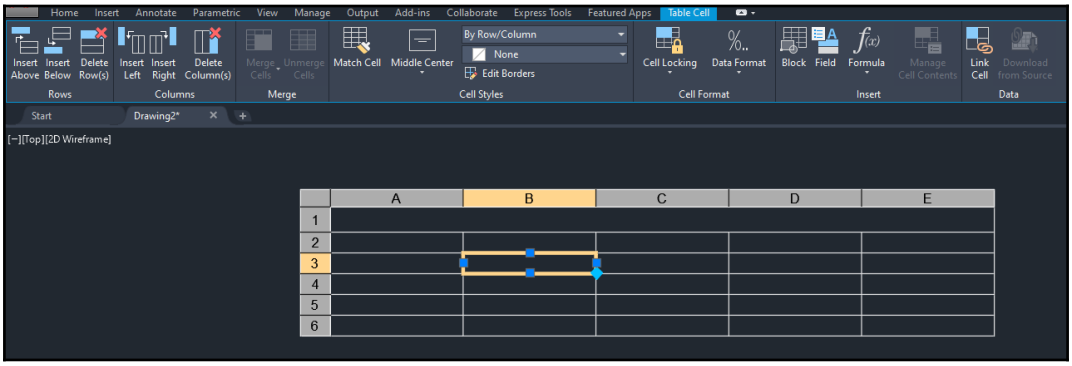


Figure 7.7: The Table Cell tab becomes active when the cell of a table is selected

When you double-click on a table cell, you will get the **Text Editor** tab, and this allows you to add text inside a table cell, as shown in *Figure 7.8*:

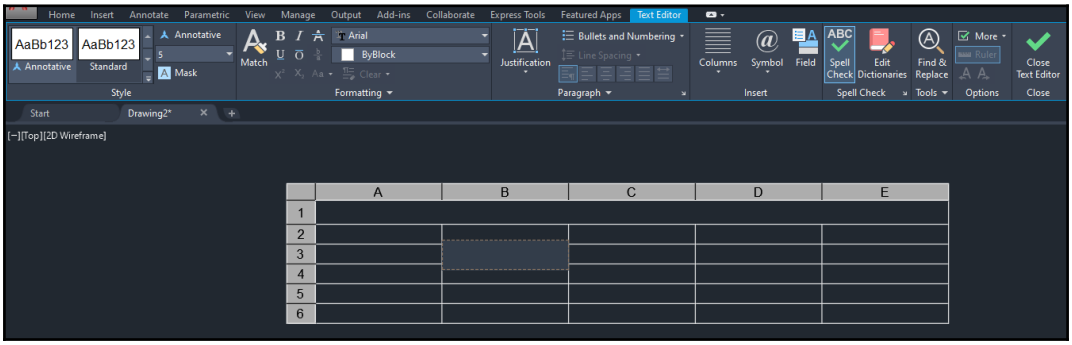


Figure 7.8: The Text Editor tab, visible when you double-click inside a table cell

You will also notice that this text is multiline text with formatting options available on the **Text Editor** tab. The default text style will be used here, but you can still modify text properties using the **Text Editor** tab.

Making a sample table

To show you how tables work in AutoCAD, I will make a sample table that contains the cost of some components of an assembly. We will not only modify the table, but also change the formatting of the text inside the table and the settings of the table. Let's start by making a simple table:

1. Erase all the tables you have in your drawing and then click on **Table** in the **Annotation** panel or use its command alias, `TABLE`.
2. Select the **Specify window** option from the **Insertion behavior** section of the **Insert Table** window.
3. In the **Column & row settings** section, set the values to `4` columns and `5` rows. The **Column width** and **Row height** options will be disabled in the **Insert Table** window as we can specify these values on the screen.
4. Select **Title**, **Header**, and **Data** for the options in the **Set cell styles** panel, respectively. After configuring these settings, my **Insert Table** window looked like this:

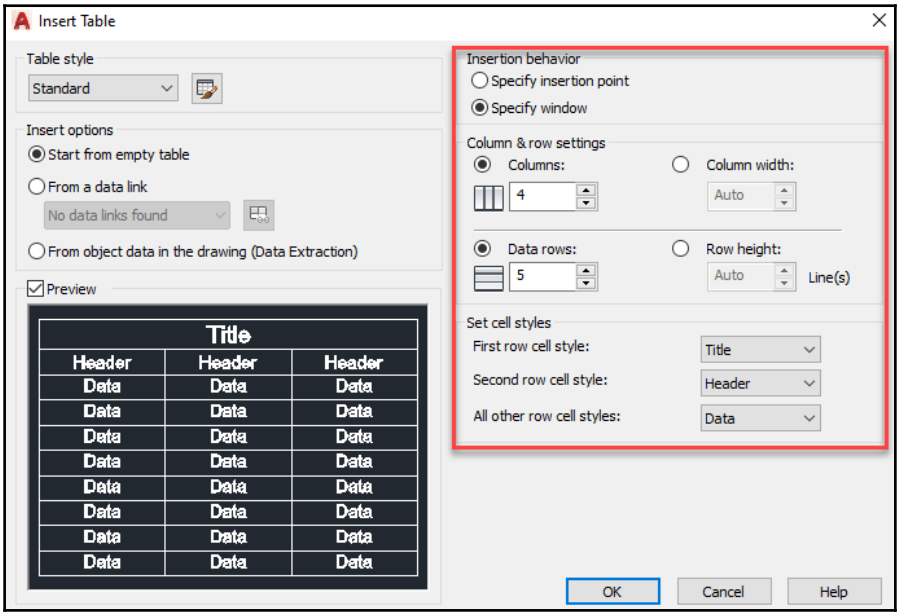


Figure 7.9: Table settings in the Insert Table window

5. Once you are done configuring these settings, click **OK** to exit the **Insert Table** window.

6. Now, click on a point in the drawing area, and then click on another point toward the bottom right of the first point so that you end up with a table that looks like this:

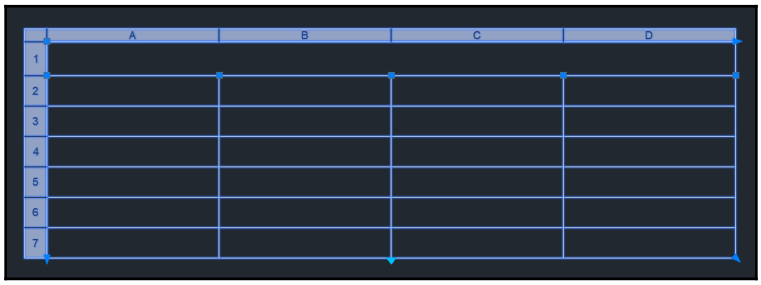


Figure 7.10: Table made with the Specify window option

As you can see, this table has four columns, five data rows, one header row, and another title row, so in total we have seven rows. Also, when you select the table, the cells at the top and the left are labeled with letters and numbers, respectively. So, we can use these letters and numbers to designate a cell. As an example, the cell situated in the bottom right of the preceding *Figure 7.10* can be called D7 as it's in column D and row 7. In the next examples, we will use a similar cell-naming convention.

Adding data to a sample table

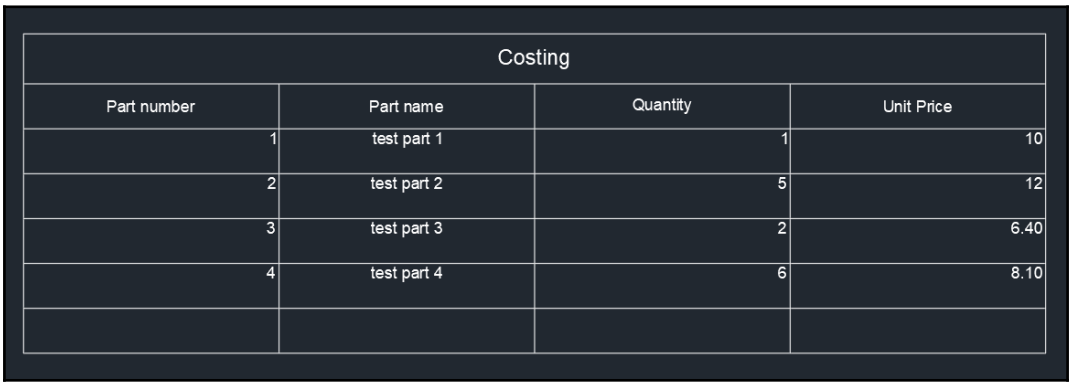
So, now that our table is prepared, let's start by adding some data to it. In the following example, we will make a sample costing table:

1. Double-click the title cell at the top of the table and type `Costing` in it, and then click outside the table to exit the `TEXT` command. The text should now be added with the default text style, as shown in *Figure 7.11*:



Figure 7.11: Title text added with the default text style

2. If your text is very tiny or very large, then you need to modify the text style so that the size of the text fits the table cell. You can select any font in the text style, but for this example, I am using a text style with Arial font. We discussed making text styles in the previous chapter.
3. Type `Part number` in cell A2, `Part name` in cell B2, `Quantity` in cell C2, and `Unit Price` in cell D2.
4. Now, fill the remaining cells as shown in *Figure 7.12*:



Costing			
Part number	Part name	Quantity	Unit Price
1	test part 1	1	10
2	test part 2	5	12
3	test part 3	2	6.40
4	test part 4	6	8.10

Figure 7.12: The header and data cells filled with text in the default text style

In this table, we have lots of things to fix, such as the justification of text, the merging of some cells, and the addition of a new column for the total price. Let's start with the text justification settings.

Adding text justification

Click and drag from cell A3 to cell A6 so that all the cells between those cells are highlighted, click on the justification option of the **Table Cell** tab, and then select **Middle Center**, as shown in *Figure 7.13*:

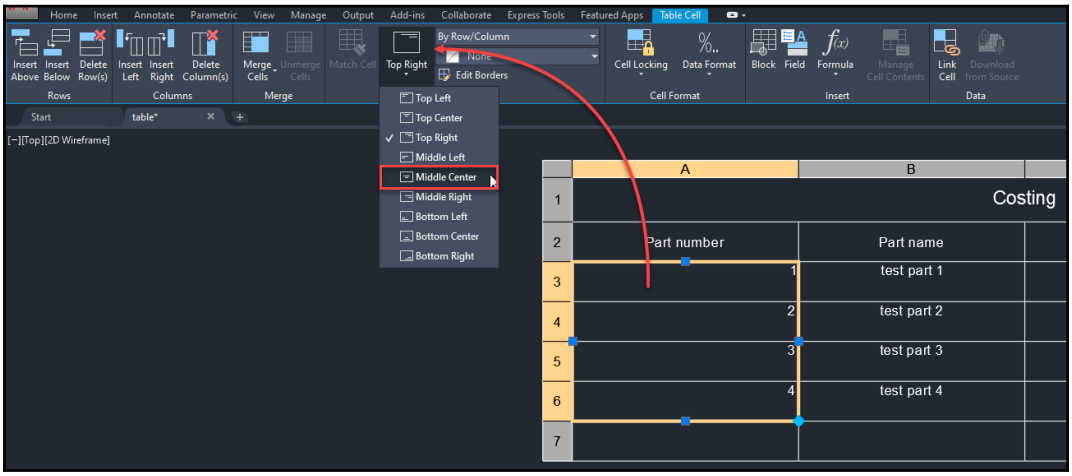


Figure 7.13: Text justification option for selected cells

The text will align in the middle center of the cells. You can configure similar text justification settings for other text entities in the table. In my example, I justified all text to the middle center of the table cells.

Adding new columns

To add a column, select any cell beside which you want to add the row or column; for this example, select cell D7 and then select the **Insert Right** option from the **Columns** panel of the **Table Cell** tab, as shown in *Figure 7.14*, and a new column will be added to the right side of cell D7:

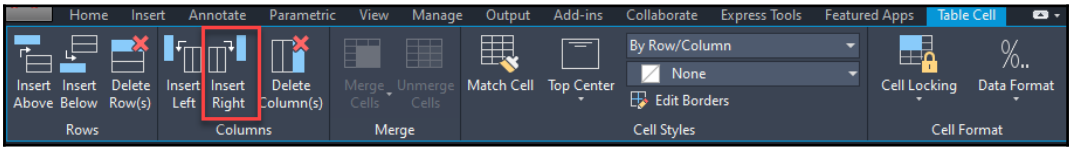


Figure 7.14 - Inserting a column to the right of the selected cell

Similarly, you can add a row also from the **Rows** panel. Deleting a row or column is also similar in terms of procedure: just select the cell in that row or column and click the **Delete Row** or **Delete Column** option from the **Rows** or **Columns** panel.

Merging cells

To merge cells into a single cell, select the cells first. In this example, I will merge cells A7, B7, C7, and D7 into a single cell. Select all the cells from A7 to D7 and then click **Merge All** in the **Merge** panel of the **Table Cell** tab, as shown in *Figure 7.15*:

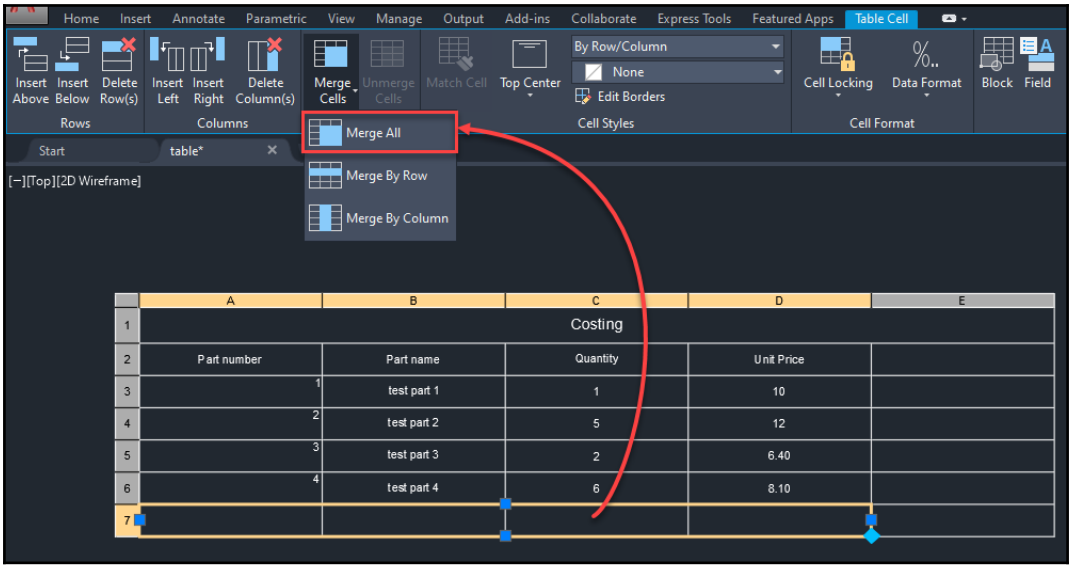


Figure 7.15: The Merge All option in the Table Cell tab

Now, you will end up with a single cell in place of four selected cells.

So, now we have created some new cells by merging cells and by adding a new column. Click on cell E2 and type `Total price` in it. Also, type `Total cost` in the merged cell ABCD7. After making these changes, the final table should look like this:

Costing				
Part number	Part name	Quantity	Unit Price	Total price
1	test part 1	1	10	
2	test part 2	5	12	
3	test part 3	2	6.40	
4	test part 4	6	8.10	
Total Cost				

Figure 7.16: Final table with data added in the cells

Now, we will add data to the empty cells using formulas, which is a standard feature in Microsoft Excel and is also available in AutoCAD with similar feature sets.

Adding data using formulas

In the **Total price** column of the table, we want to add the total price of the parts, which is the product of the part quantity and the unit price.

In the following example, we will use a formula that will allow us to multiply the value in column D by the value in column C, and the resulting value will be added to column E:

1. Double-click on cell E3, which is right underneath the **Total price** cell, and then type `=C3*D3`.
2. This string of characters is a formula, which begins with the = sign. `C3*D3` indicates that we are multiplying the value in cell C3 by the value in cell D3.
3. Once you are done typing this in cell E3, click outside the table and you will end up with 10 as the value inside cell E3. Here, the value inside cell C3 is 1, and when it is multiplied by the value inside cell D3, which is 10, the result is 10 x 1, or 10.
4. Now, we can copy the same formula for other cells in column E. For that, select cell E3 and then click the diamond grip at the bottom right of the cell and drag it up to the cell E6, as shown in *Figure 7.17*.
5. You will notice that the formula will be copied to all the selected cells of the table and it will be updated as per the values in the respective **Unit Price** and **Quantity** columns, as shown here:

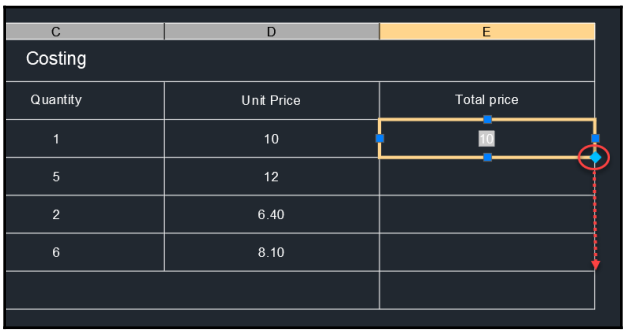
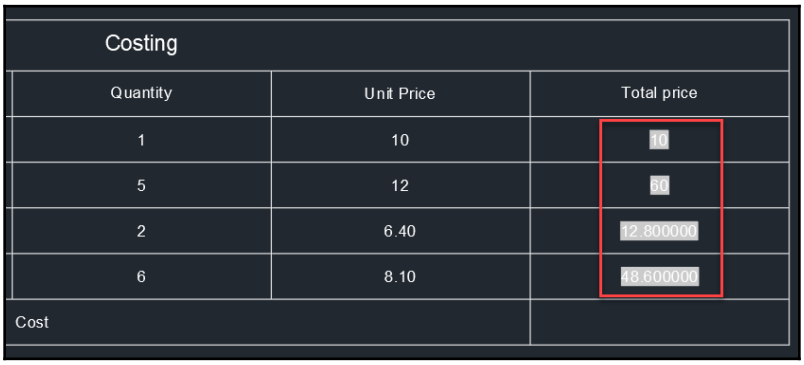


Figure 7.17: Selecting the cell and copying its formula to other cells

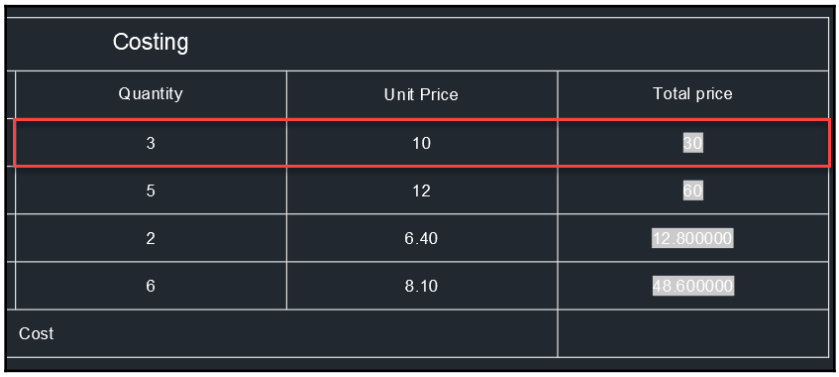
All the values added in the **Total price** column will have a gray background, which indicates that this data is different from normal text. This is called a field in AutoCAD, which is dynamic text linked to some other features in the drawing; this field can change according to the data that is linked to it:



Costing		
Quantity	Unit Price	Total price
1	10	10
5	12	60
2	6.40	12.800000
6	8.10	48.600000
Cost		

Figure 7.18: Formula added to the selected cells

To see the field in action, double-click on cell C3 and change the quantity from 1 to 3, and you will notice that the value in cell E3 will update to 30 instead of 10, as 3 x 10 is 30. We will learn more about fields in the next section:



Costing		
Quantity	Unit Price	Total price
3	10	30
5	12	60
2	6.40	12.800000
6	8.10	48.600000
Cost		

Figure 7.19: When changing the value in the numbered cell, its related field updates in the highlighted row

The gray background of the field is visible only in the drawing area; when you plot your drawing, it will plot like normal text without any gray background. So, basically, we have a gray background in the drawing to distinguish simple text from fields.

Adding a formula to the Total cost cell

Just like we added a formula in the **Total price** column, we can add another formula that calculates the total cost of components by adding up all the values inside column E. Here are the steps required to do this:

1. Select cell E7 and then select **Sum** from the **Formula** drop-down menu of the **Insert** panel of the **Table Cell** tab, as shown in *Figure 7.21*.
2. Now, click on cell E3 and then click on cell E6. This will include all these cells in our calculation and the formula will be added automatically.
3. Click outside the table to finish adding the formula and, once again, you will end up with a formula that shows the sum of all the values in the **Total price** column:

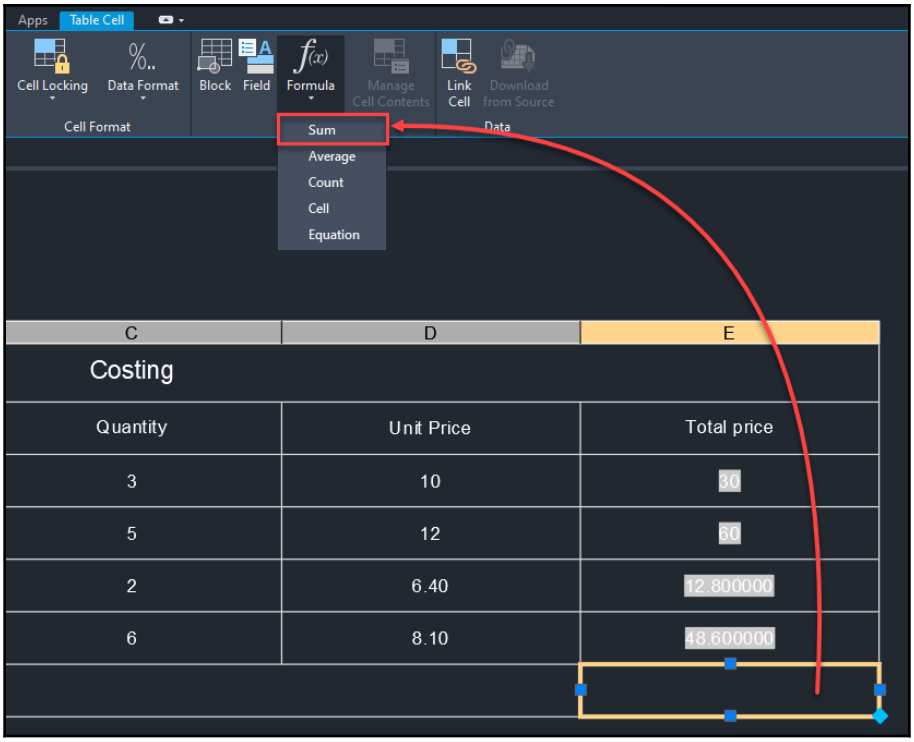


Figure 7.20: Using the Sum formula in the selected cell

My final table looked like *Figure 7.21*:

Costing				
Part number	Part name	Quantity	Unit Price	Total price
1	test part 1	3	10	30
2	test part 2	5	12	60
3	test part 3	2	6.40	12.800000
4	test part 4	6	8.10	48.600000
Total Cost				151.400000

Figure 7. 21: Final table with all the data and formulas in it

In this table, changing any parameter, such as the quantity of any part or its unit price, would update the entire table to always reflect the correct data; this is all possible because of formulas.

So, now our table has been made in AutoCAD and it can be added anywhere in the drawing, but if you want to save this table in a Microsoft Excel-compatible format, then you need to export it using a simple method shown in the next section.

Exporting tables in Excel format

The table we made in AutoCAD can be exported in an external file format such as CSV, which you can open with Microsoft Excel. The following steps will show you the method of doing just that:

1. To export the table, select it from the drawing area and then right-click and select **Export...** from the context menu, as shown in *Figure 7.22*:

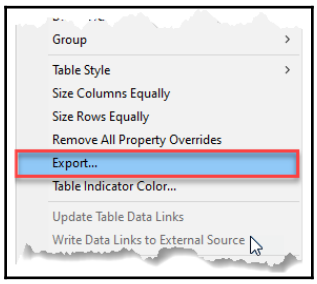


Figure 7. 22: The Export... option in the right-click menu of the table

2. Specify a location in the next **Export Data** window, give your file a name, and then click the **Save** button.
3. You will notice that the table is now saved at your specified location in CSV format. Double-click on the CSV file and the table will open in Excel in a simple text-only format with no formatting.
4. The table we have made in this example, when exported to CSV format, looks like *Figure 7.23* in Excel:

	A	B	C	D	E
1	Costing				
2	Part numb	Part name	Quantity	Unit Price	Total price
3	1	test part 1	3	10	30
4	2	test part 2	5	12	60
5	3	test part 3	2	6.4	12.8
6	4	test part 4	6	8.1	48.6
7	Total Cost				151.4

Figure 7. 23: Table exported from AutoCAD to Excel in CSV format

So, as you can see, we can export AutoCAD tables in an Excel-compatible format, with the only disadvantage being the loss of the formatting settings. Just like exporting a table from AutoCAD to Excel is possible, the reverse is also possible, which is explained in the next section.

Importing an Excel table into AutoCAD

Just like you can export a table from AutoCAD to Excel, you can do the reverse too with similar ease. In this example, I have merged some cells in the table and I exported it in XLSX format. Now, we will try inserting this table into AutoCAD:

1. Start a blank drawing in AutoCAD and click on **Table** from the **Annotation** panel, or use its command alias, `TABLE`.
2. Now, in the left section of the **Insert Table** window, click the **From a data link** radio button of the **Insert options** panel, as shown in *Figure 7.24*:

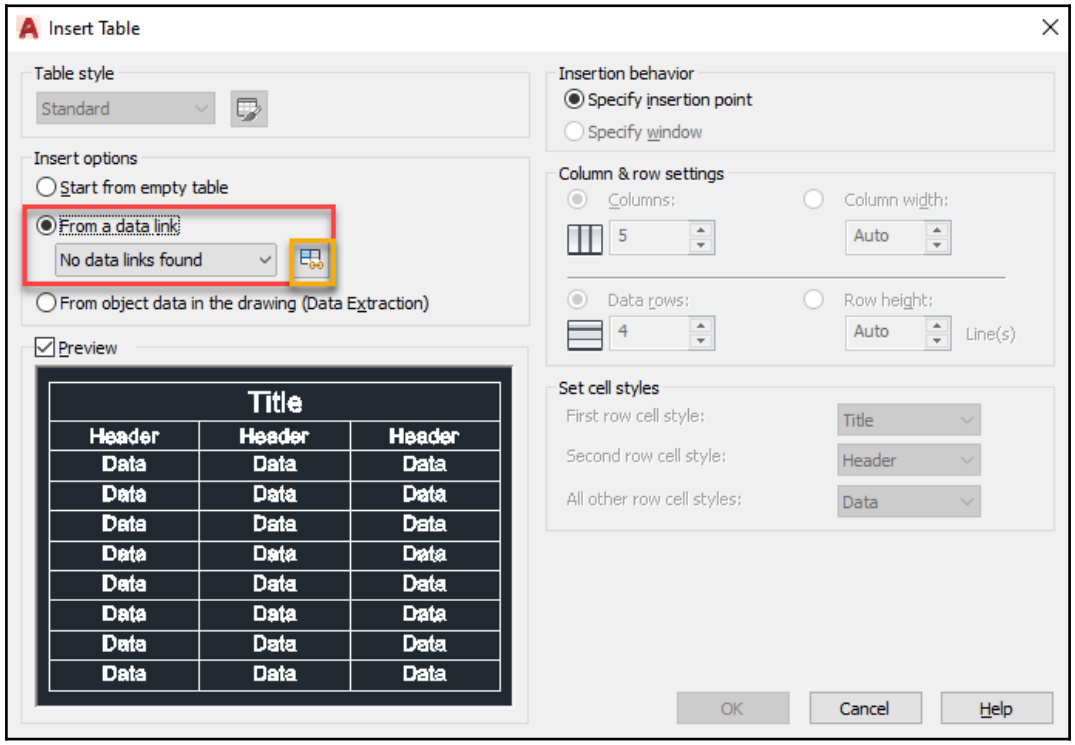


Figure 7. 24: Options for inserting an external table in the Insert Table window

3. Now, click the data link icon highlighted inside the yellow box in *Figure 7.24*.
4. The **Select a Data Link** window will open. Click **Create a new Excel Data Link** in the **Links** panel and give this link a name, and then click **OK**. For this example, I am naming it `sample table`.
5. Now, a **New Excel Data Link: sample table** window will open. Click the **...** button next to **Browse for a file ...**, as shown in *Figure 7.25*:

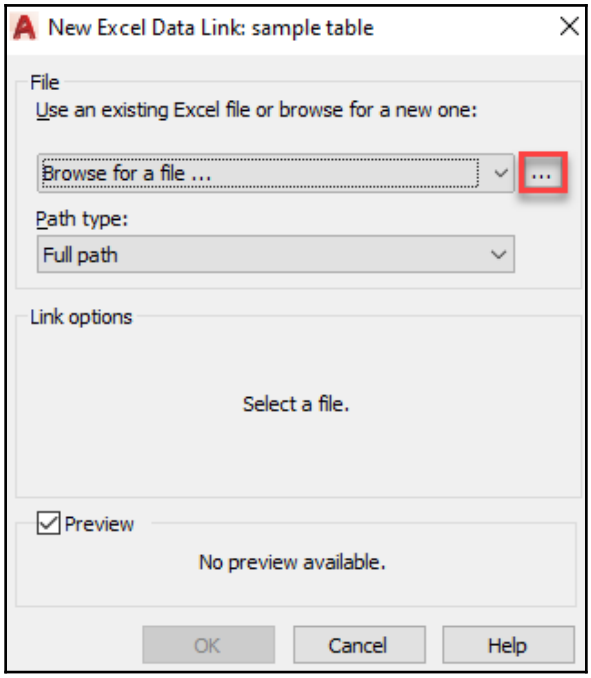


Figure 7. 25: The browsing option highlighted in the red box

6. Now, locate the Excel file and click the **Open** button on the File Explorer window.
7. Click **OK** to close all open windows and then click on a point in the drawing area to insert the table.

You will notice that the table will be added to the drawing and the cell margins will also be retained with it, as shown in *Figure 7.26*:

Costing				
Part number	Part name	Quantity	Unit Price	Total price
1	test part 1	3	10	30
2	test part 2	5	12	60
3	test part 3	2	6.4	12.8
4	test part 4	6	8.1	48.6
Total Cost				151.4

Figure 7. 26: Table inserted from Excel to AutoCAD

This table is a linked table, and if you change data inside the Excel file and save it, the table in your AutoCAD will show a pop-up bubble in the bottom-right area indicating that the table has been updated. You can click the link in the pop-up bubble to update the table too.

If, for some reason, you don't see the pop-up bubble even after changing the Excel table, then you can right-click the data link icon on the status bar and select **Update All Data Links...** to update the table, as shown in *Figure 7.27*:

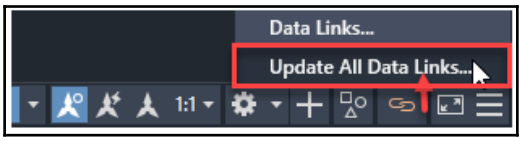


Figure 7. 27: The Update All Data Links... option in the status bar

We can insert the Excel table and other external content as a data link directly too, and we will learn about data links in detail in the next section.

So, we have almost finished our work the table tool. Now is the time to finally look at table styles and the table style manager.

Working with the table style manager

The table that we added in the preceding examples took properties such as the text style of table cells, the thickness of table lines, the background color of table cells, and the color of the table text from the table style. So far, we have worked with the default table style, but in the following example, we will create our very own table style and make a table with that custom table style:

1. To make the table style, expand the **Annotation** panel and click the table style icon indicated in *Figure 7.28*:

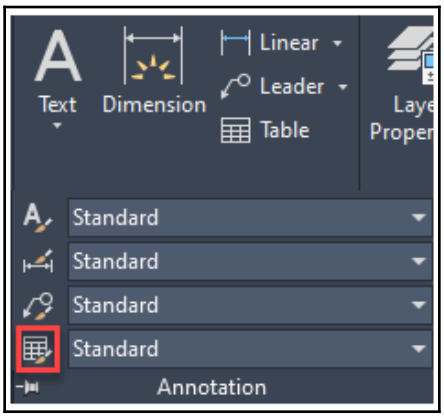


Figure 7. 28: The table style option in the expanded Annotation panel

2. Click the **New** button in the **Table Style** window and give your new table style a name (I am naming it `Test style`) and then click **Continue**.

3. Now, the **New Table Style: Test style** window will open, and this is the place where we can change the table style. Go to the **Cell styles** panel and select **Title** from the drop-down menu. This will ensure that the properties underneath the **Cell styles** panel will only affect the title cell:

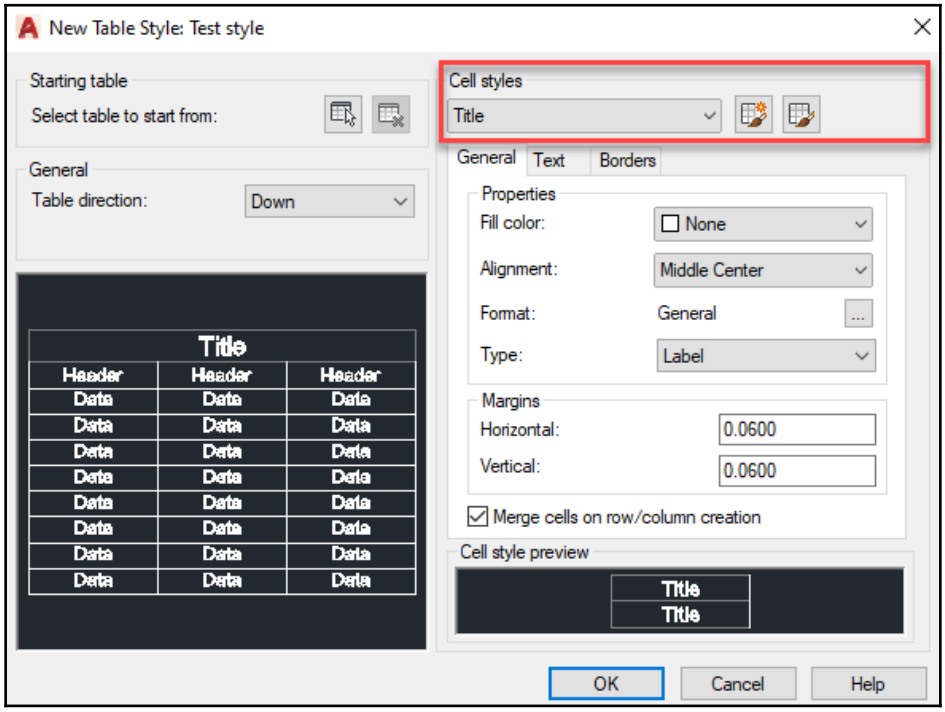


Figure 7. 29: The Cell styles panel in the New Table Style: Test style window

4. Now we have three tabs: **General**, **Text**, and **Borders**. Their names suggest the properties that they are concerned with. Select **Red** in the **Fill color** dropdown and leave the other settings as they are by default.
5. Select a text style from the **Properties** panel of the **Text** tab or set a custom height for the title text. In this example, I will only change the text color to green.
6. From the **Borders** tab, select a lineweight and a linetype or color for your table borders. In this case, I am changing the lineweight to 0.5 and the color to blue. To apply these lineweights, click the second box on the **Properties** panel, as highlighted with the red box in screenshot **C** of *Figure 7.30*:

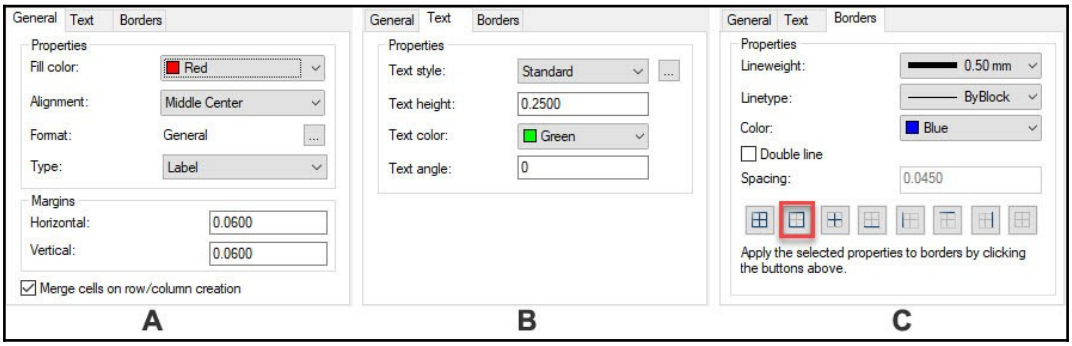


Figure 7. 30: The General, Text, and Borders settings for the table

7. Once you are done making these changes, click **OK** to close the table style manager and make a table with this table style.

You will notice that the table cell will inherit the properties that we applied. Similarly, you can modify the table styles again to add more properties to other cell types, such as header and data cells. In our example, the final table looked like *Figure 7.31*, with these settings applied to only the title cell:

Costing				
Part number	Part name	Quantity	Unit Price	Total price
1	test part 1	3	10	30
2	test part 2	5	12	60
3	test part 3	2	6.40	12.800000
4	test part 4	6	8.10	48.600000
Total Cost				151.400000

Figure 7. 31: Cell style applied to the title cell of the table

So, finally, we are done making tables and modifying their properties too. It's time now to move on to the next topic, which is data links. We have already seen an example of data links in action, but we will explore them further in the next section.

Working with data links

Data links are connected pieces of data in AutoCAD. As an example, if you insert a table in AutoCAD from an Excel file as a data link and then modify the Excel file, the table inside AutoCAD will update automatically.

In this section, we will learn how to import tables and Word files in our drawings using data links. In the following example, we will learn how to import a table using a data link:

1. Open a table in Excel. I am opening the Excel table that we used in the previous examples.
2. Now, select all the cells that contain text and press *Ctrl* + *C* to copy all the content on the clipboard. The selected cells will now have a "marching ant" kind of frame around them, as shown in *Figure 7.32*:

	A	B	C	D	E
1	Costing				
2	Part number	Part name	Quantity	Unit Price	Total price
3	1	test part 1	3	10	30
4	2	test part 2	5	12	60
5	3	test part 3	2	6.4	12.8
6	4	test part 4	7	8.1	48.6
7	Total Cost				151.4
8					

Figure 7. 32: "Marching ant" selection box in the Excel table

3. Now, open a blank drawing in AutoCAD and expand the paste icon from the **Clipboard** panel and select **Paste Special** from the drop-down menu, as shown in *Figure 7.33*:

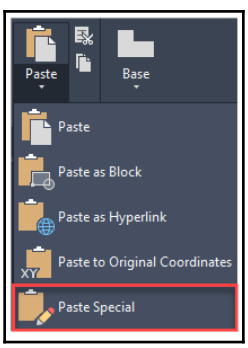


Figure 7. 33: The Paste Special option in the Paste drop-down menu

4. In the **Paste Special** window that pops up, select the **Paste Link** radio button from the left panel. Now, the right panel will show two options: **Microsoft Excel Worksheet** and **AutoCAD Entities**. Select **AutoCAD Entities** from the panel and click **OK**.
5. Now, click on a point in the drawing area to add your table.

You will notice that a table that looks exactly like the AutoCAD table will show up, but this table is a data link that is linked to the original Excel file. If the size of the table is very small or large, then zoom in or out or use the `SCALE` command to change the size of the table as per your requirements.

In the preceding example, if, instead of **AutoCAD Entities**, you selected **Microsoft Excel Worksheet** from the **Paste Special** drop-down menu, then you would have a linked table that looked more like an Excel sheet.

If you modify the original Excel sheet and save your Excel file, and then go to the AutoCAD drawing that contains the linked table, a pop-up bubble will appear mentioning a change in the data link. You can click the **Update data link** text in the pop-up bubble to update the data link. Alternatively, you can also right-click on the data link icon in the status bar and then select **Update All Data Links...** from it, as shown in *Figure 7.34*:

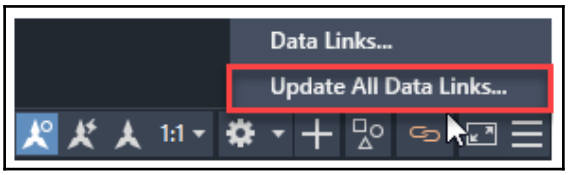


Figure 7. 34: The Update All Data Links... option in the status bar

To add a Microsoft Word file as a data link, you can follow the same workflow. In this case, I will import a simple piece of text from Microsoft Word to AutoCAD:

1. Open a Word file that contains text, press *Ctrl + A* to select all the text in the Word file, and then press *Ctrl + C* to copy the text to the clipboard.
2. Now, go to the **Paste Special** window from the expanded **Paste** panel and select the **Paste Link** radio button, as shown in *Figure 7.35*:

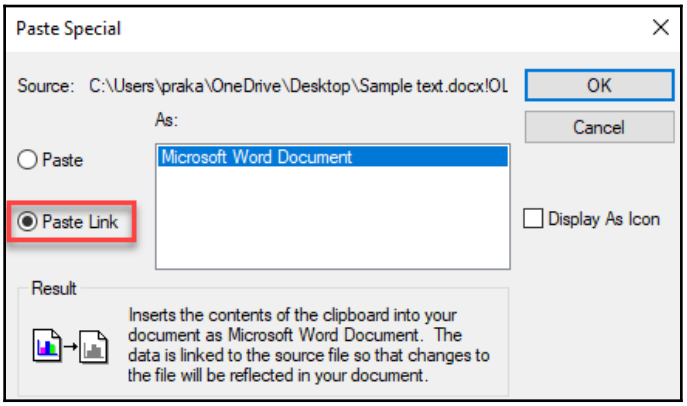


Figure 7. 35: The Paste Link radio button in the Paste Special window

3. Select **Microsoft Word Document** from the panel in the middle and click **OK**.
4. Click on a point in the drawing area to paste the data link.

The linked text is now added to your drawing, and if you now modify the text in the Word file and save the Word file, the text inside AutoCAD will change automatically to reflect the changes. If, for some reason, your linked text is not changing, then once again go to the data link icon of the status bar, right-click on it, and select **Update All Data Links...** to update the data link.

As you may have noticed, data links are great if you primarily work with Excel tables and want to avoid the hassle of modifying or formatting tables in AutoCAD. You can make tables in Excel and import them to AutoCAD with just a couple of clicks.

Similar to data links, there is also a feature inside AutoCAD that lets you link a property to text: fields. We used formulas in our earlier table example. Fields are like formulas, but there is a lot more to this tool than just formulas. In the next section, we will explore this tool in detail.

Understanding fields

Field is a linked piece of text that shows the property of the object it is linked to. Properties of Field text are determined by the text style, and for that reason, we will modify the text style in our drawing before we start learning about fields.

In the following example, I will make a text style called `Field Style` and then I will use this text style for all the fields:

1. Select the text style icon from the expanded **Annotation** panel of the **Home** tab or use its command alias, `ST`.
2. Click the **New** button on the **Text Style** window and give the text style a name. For this example, name it `Field Style`, and then click **OK**.
3. Now, change the font to `simplex.shx` and the font height to `2.5000`.
4. Click **Apply** and then **Close** to apply these changes, and then close the **Text Style** window. My final text style settings for **Field Style** appeared as shown in *Figure 7.36*:

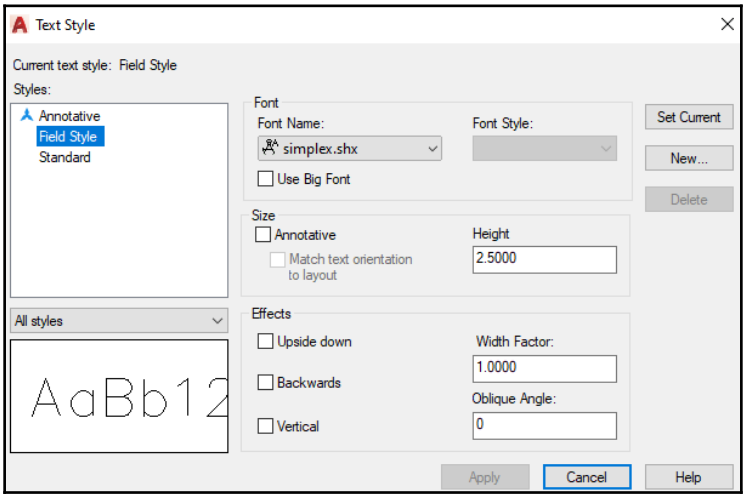


Figure 7. 36: The Field Style text style with all the settings

Now that we have our text style, we are ready to add our fields to our drawing, and the fields will now use the current text style, **Field Style**, which we created just now.

In the next example, we will create a field that represents the area of a closed polyline shape:

1. Make a closed polyline shape with lines and arcs as shown in *Figure 7.37*. In my drawing, the length of the horizontal line at the bottom of this polyline shape is `50` units, and the dimensions of other lines and arcs are randomly selected:



Figure 7. 37: Closed polyline shape

2. Type `FIELD` in the command line and press the *Enter* key; the **Field** window will open.
3. In the **Field category** dropdown, select the **Objects** option and the **Field names** panel will update with a different set of options.
4. Select **Object** from the **Field names** panel and then click the box near the **Object type** option that looks like an AutoCAD pointer.
5. Now, the **Field** window will be hidden and you will see a selection box. Click the polyline and the **Field** window will show up again.
6. Now, the **Property** panel underneath the **Object type** option will populate with a list of properties associated with our selected polyline. Select **Area** from the list of properties and then select **Decimal** from the **Format** panel.

7. Click **OK** when you are done making these changes in the **Field** window. The workflow of creating fields is also shown in *Figure 7.38*:

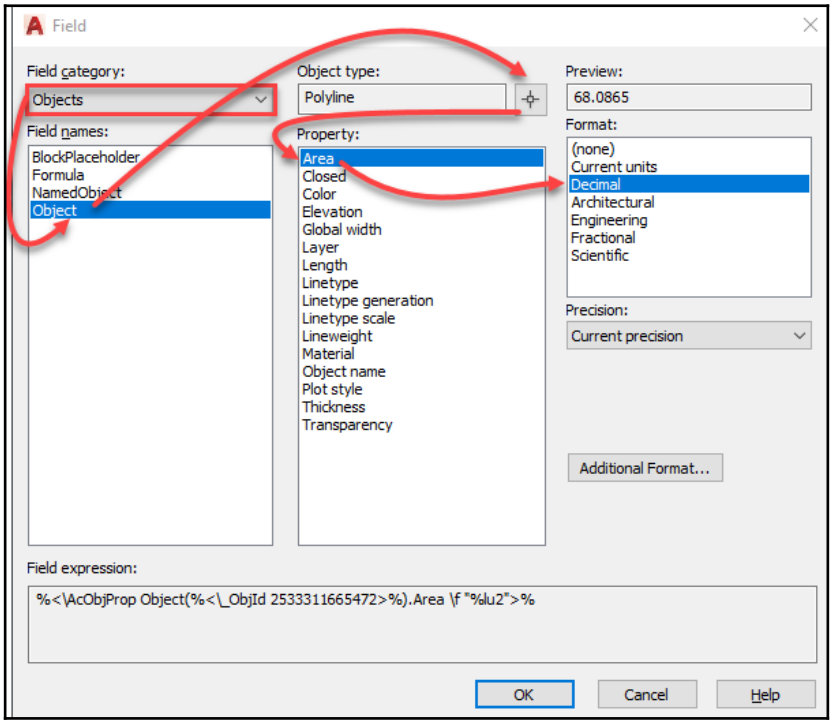


Figure 7. 38: Workflow for adding the Area field for the drawing

8. Now the area of the polyline will show up on your cursor. Click inside the polyline to place the field and you now have an area field that is linked to the polyline.
9. Change the area of the polyline by moving the lines or arcs using grips and then type `RE` and press *Enter* to regenerate the drawing; you will notice that the area shown in the field will also update.

The hallmark of a field is a gray background, and you will see this for the area field too. This gray background of the field is only visible in the drawing area and will not show up when you plot your drawing. The fields of a drawing will be plotted as simple text without any gray background. In my example, the field with the area looked like *Figure 7.39*:

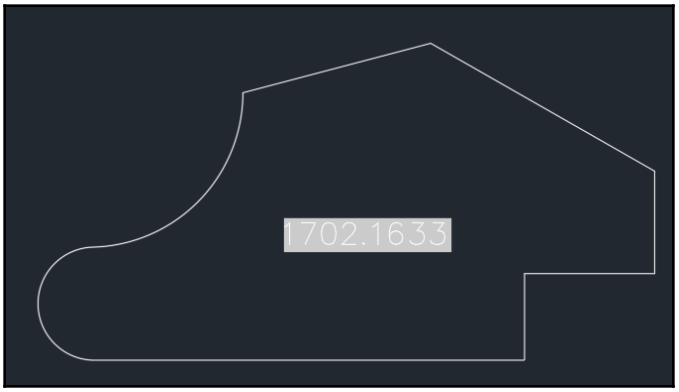


Figure 7. 39: The area field with the value of the area inside the polyline

So, this was just one example where fields can be used. There are many other object types for which you can add fields; for instance, you could add a field for viewport scale, which we discussed in our chapter on printing and plotting, or a field for data and time.

Now, we will move on to another topic that is completely different from fields: custom linetypes and hatches. In the next section, we will learn to make custom linetypes using linetype code and custom hatches using express tools.

Making isometric drawings

Isometric drawings are 3D drawings that are made with vertical lines of 90 degrees and horizontal lines at an angle of 30 degrees. An example of an isometric drawing is shown in *Figure 7.40*:

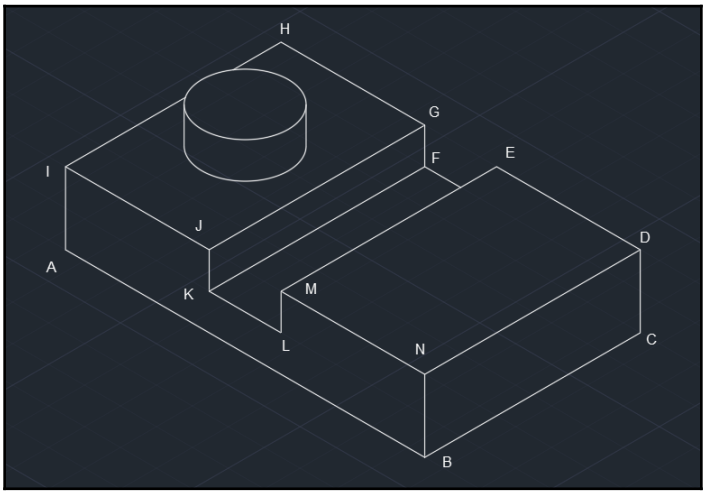


Figure 7. 40: Sample isometric drawing

As you can see, this drawing looks as though it is 3D even though it has been made on the normal 2D plane in an AutoCAD workspace. We will make this drawing using the isometric feature in the following examples, but before we do, we need to prepare.

Preparing a workspace for isometric drawing

As isometric drawing differs from a normal drawing. It requires its own set of customizations before we can do it. Click the grid mode and isodraft toggles in the status bar, as shown in *Figure 7.41*:



Figure 7. 41 - The grid mode and isodraft options in the status bar

If you don't see these options in the status bar, then click the icon that looks like three horizontal lines, one on top of the other, at the far right of the status bar and select the **Grid** and **Isometric Drafting** options, as shown in *Figure 7.42*:

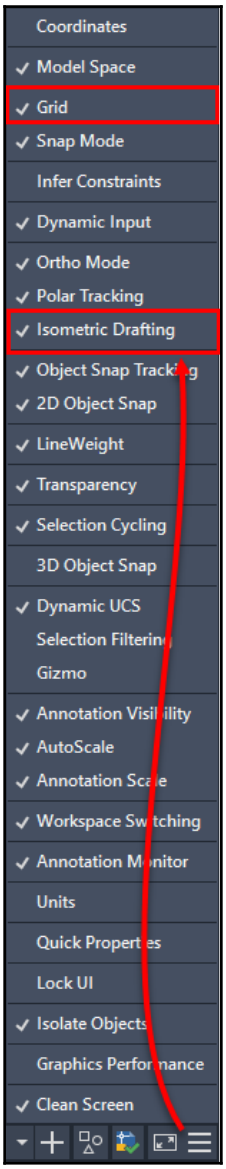


Figure 7. 42: The Grid and Isometric Drafting options in the customization menu

When checked, the options will show in the status bar, and then you can click the options in the status bar to activate them.

Click the arrow right next to the polar tracking status bar option and make sure **30, 60, 90, 120…** is selected from the list of angles, as shown in *Figure 7.43*:

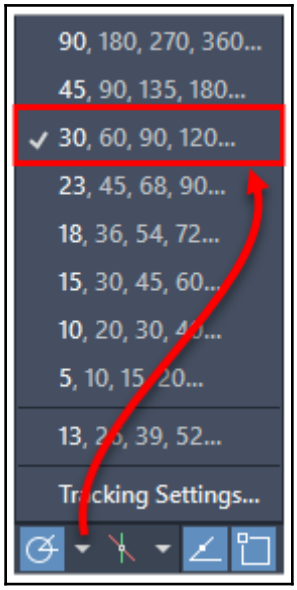


Figure 7. 43: Angle values in the polar tracking status bar option

This will ensure that our cursor aligns to the 30-degree angle when making an isometric drawing. Finally, we need to make changes in the object snap toggle of the status bar. Click the arrow right next to the object snap icon and activate the object snaps as shown in *Figure 7.44*:

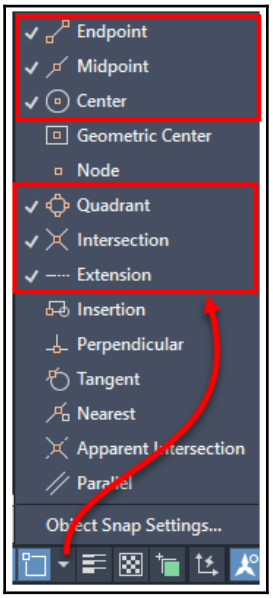


Figure 7. 44: Object snaps to be activated using the status bar

Now, we are all set to start making our isometric drawing; we will use *Figure 7.40* as a reference.

Making the drawing

As you can see, I have labeled vertices of the isometric drawing with letters, and we will use these letters as a reference in our drawing. Also, with the isodraft option active in the status bar, you will notice that the cursor will now align to an isometric plane, as shown in *Figure 7.45*:

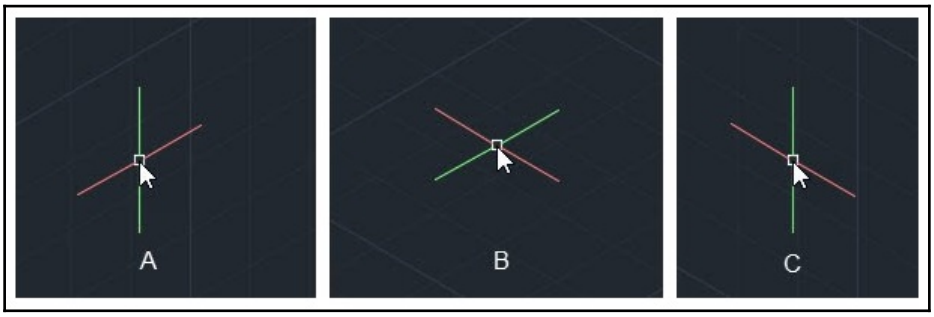


Figure 7. 45: Cursor aligning to different isometric planes when the isodraft option is active in the status bar

You can change the alignment of the cursor using *F5*. Every time you press the *F5* key, not only will the cursor change, but also the plane on which the isometric drawing will be made will change. *Figure 7.45* shows the different types of cursor alignments that you will get when pressing *F5*. Keep the grid active from the status bar to see this effect clearly with respect to the background.

With all the settings in place, let's now start making the isometric drawing:

1. To start making the drawing, select the Line command from the **Draw** panel of **Home** tab or use its command alias, `L`.
2. Press *F5* so that the cursor reaches position **B**, as shown in *Figure 7.45*, and then click on a point to start the Line command.
3. Move your cursor vertically upward, and when the green tracking line shows, type `1` and press the *Enter* key. You will then have the first line of the drawing, which is line **AI** in *Figure 7.40*.
4. Now, move your cursor to the right so that the tracking vector shows an angle of 30 degrees, as shown in *Figure 7.46*, and then type `3` and press the *Enter* key:

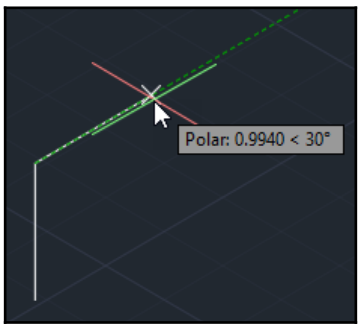


Figure 7. 46: Line made at an isometric angle of 30 degrees

5. Now move the cursor to the right so that the tracking angle now shows 330 degrees. Then, type `2` for the distance and press *Enter*, and you will have a line with a length of 2 units. The cursor should now be at point **G** of the initial drawing.
6. Move your cursor vertically down, and then type `0.5` and press the *Enter* key. This will add another line, **GF**, to the drawing.
7. Again, move your cursor to the right at an angle of 330 degrees and then type `1` for the distance and press *Enter* again.
8. Now, move your cursor vertically up and type `0.5` and press *Enter*, and you will now reach point **E** of the initial drawing. The drawing so far should look like *Figure 7.47*:

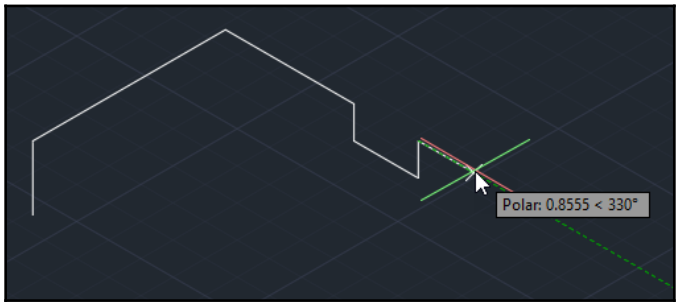


Figure 7. 47: Isometric angle of 330 degrees on the line

9. Again, move your cursor to the right so that it shows an angle of 330 on the tracking vector, as shown in *Figure 7.47*, and then type `2` and press the *Enter* key.
10. Move the cursor vertically down, type `1`, and press *Enter*, and you will end up at point **C** of the drawing, making line **DC**.
11. Move the cursor to the left now so that the angle for the tracking vector is 210 degrees, type `3`, and press *Enter*, and you will reach point **B** of the drawing:

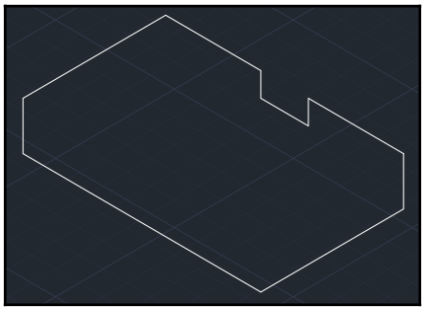


Figure 7. 48: Closed loop formed after adding the lines

11. Click point **A**, which is the starting point of this drawing, to close this shape, and the final drawing that you will get should look like *Figure 7.48*. Press *Enter* again to exit the `Line command` as well.
12. Start the `Line` command again, click on point **I** of the drawing, and move your cursor to the right so that the tracking angle shows 330 degrees. Type `2` for the distance and press *Enter*, and you will end up at point **J** of line **IJ**.
13. Now, move the cursor vertically down, type `0.5` and press *Enter* again, then move the cursor to the right so that the tracking vector again shows 330 degrees. Then, type `1` and press *Enter* again, and finally move the cursor vertically up, type `0.5`, and press *Enter* again. You will reach point **M** of the drawing. The drawing so far should look like *Figure 7.10*:

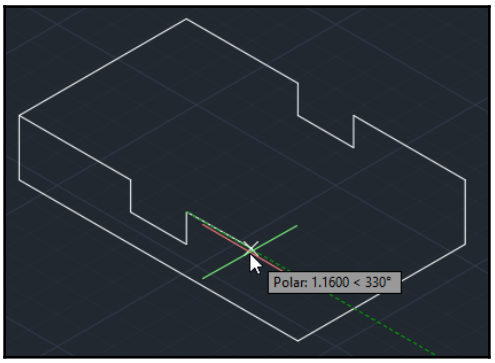


Figure 7. 49: Line with an isometric angle of 330 degrees

14. With the `Line` command still active, move your cursor to the right so that the angle for the tracking vector shows 330 degrees, then type `2` and press *Enter*. Now, you will reach point **N** of the drawing. From here, click point **D** to join the line to the existing vertex and press *Enter* to exit the Line command.

Now that we have made the basic sketches for the isometric drawing, it's time to add the remaining lines and other shapes to the drawing:

1. Start the `Line` command again and click on point **B**. Then, click point **N**, and we now have a line connecting **B** and **N**. Press *Enter* again to exit the `Line command`.
2. Start the `Line` command again and click on point **J** and then click on point **G**, so that a line is made connecting points **J** and **G**. press *Enter* to exit the command.
3. Similarly, make a line connecting points **K** and **F** and then another line connecting points **M** and **E**. The final drawing after making these lines should look like *Figure 7.50*:

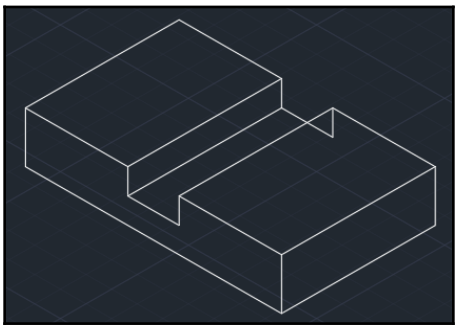


Figure 7. 50: Isometric drawing after adding all the outlines

4. Now, select the `Line` command again and click on the midpoint of line **IH**. Then, move it to the right at an angle of 330 degrees and then join it with the midpoint of line **JG**. Once you have the line, press *Enter* to exit the `Line` command. This line is a construction geometry that we will delete after using its reference point.

To make the cylindrical part on the plane **IJGH**, we will use the isocircle tool, which is like the circle tool but for isometric drawing. We can't use the circle tool directly in an isometric drawing, as the circle tends to look more like an ellipse in the isometric view.

So, now we will add details to the drawing:

1. Start the **Axis, End** ellipse from the expanded ellipse drop-down menu of the **Draw** panel on the **Home** tab, as shown in *Figure 7.51*:

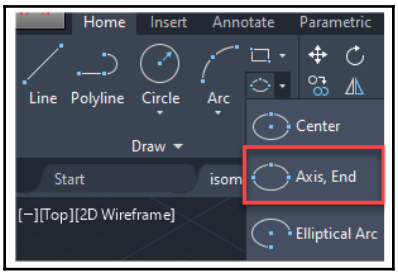


Figure 7. 51: The Axis, End ellipse option in the expanded ellipse dropdown of the Draw panel

2. Now, the command line will show `Arc`, `Center`, and `Isocircle` in the subcommands, as shown in *Figure 7.52*. Select the `Isocircle` subcommand by clicking on it or type *I* and press *Enter* and the `Isocircle` command will activate:

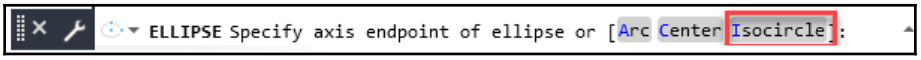


Figure 7. 52: The Isocircle option on the ellipse command line

3. Click on the midpoint of the construction line that we made in the previous example, the line that touches the midpoint of lines **HI** and **JG**. In the `Specify radius of isocircle` prompt, type `0.6` and press *Enter*.
4. Select the isocircle we made in the previous step and start the copy command from the **Modify** panel of the **Home** tab, and then click on the center of this isocircle as the base point. Move your cursor vertically upward, type `0.5`, and then press *Enter*.
5. Delete the construction line we used to make the isocircle.

6. Select the `Line` command and join the quadrants of the two isocircles as highlighted in *Figure 7.53*:

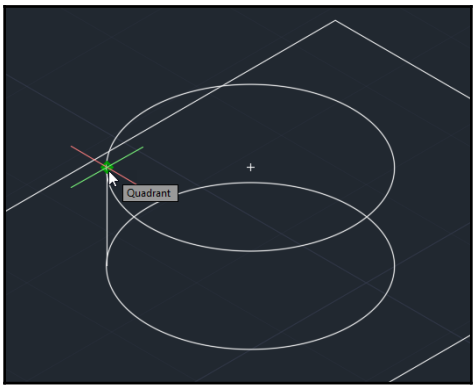


Figure 7. 53: Joining the quadrants of the isocircle using a line

7. Now, our isometric drawing is almost done: all we need is a bit of cleanup, and for this, we can use the trim command. Select the trim tool from the **Modify** panel of the **Home** tab and then press the *Enter* key to select all the objects in the drawing as the trimming boundaries.
8. Click the lines and other segments highlighted in *Figure 7.54* and those segments will be trimmed from the drawing area. You will have your final isometric drawing:

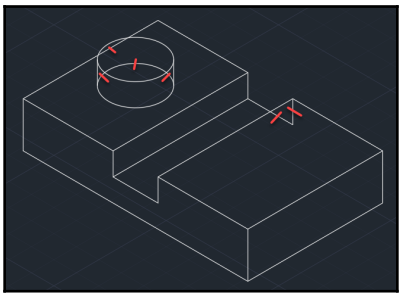


Figure 7. 54: Lines and other curves to be trimmed highlighted with red lines

Once you are done making the isometric drawing, you can click the **ISODRAFT** status bar toggle to turn off isometric mode and then you will once again have a normal **XY** axis system.

So, this was one of the ways whereby you can make isometric drawings in AutoCAD. We have 3D tools as well that can make this kind of 3D drawing, and we will learn about those tools later in the 3D-related chapters.

After having looked at isometric drawing, let's look at another great feature in AutoCAD that is very different from the isometric tool but is equally interesting. This tool is the parametric feature. Using this tool, you can make drawings using parameters or equations, which we will learn about in the next section.

Working with parametric drawings

Parametric drawing is the technique of making drawings using a set of parameters that help maintain the relationships between different components of a drawing, part, or assembly. AutoCAD is not a parametric software; software such as Fusion 360 and Autodesk Inventor are intrinsically parametric. But despite AutoCAD not being parametric software, it has tools that help us make 2D drawings with parameters and constraints.

In the following section, we will learn how to make a simple drawing using parameters and constraints. We will also use the formulas to add relations between different parts of the drawing.

Using the Infer Constraints status bar toggle

On the status bar, activate the Infer Constraints toggle as shown in the following screenshot. If you don't have this toggle on the status bar, then click the icon of three stacked horizontal lines at the end of the status bar and activate the Infer Constraints option from the list that is displayed. Once it is activated in the list, the option will show on the status bar and you can then toggle it on or off:



Figure 7. 55: The Infer Constraints status bar tool

In the following example, we will learn how to make constrained geometries using the Infer Constraints tool:

1. Click the Infer Constraints status bar mode to activate it so that it turns blue in color. Now we are ready to make drawings using this feature.
2. Select the `Line` command from the **Draw** panel of the **Home** tab or use its command alias, `L`.
3. Click on a point in the drawing area to start the `Line` command, then move the cursor to the right so that it is completely horizontal, and then click again at any point in the horizontal direction. The horizontal line will be made, and a new horizontal constraint will also be added, as shown in *Figure 7.56*:

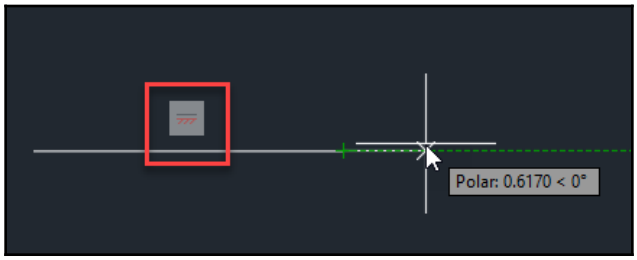


Figure 7. 56: Horizontal constraint added along the horizontal line

4. Move your cursor up in the vertical direction and then, at some distance in the vertical direction, click again. Another set of constraints will be added, as shown here:

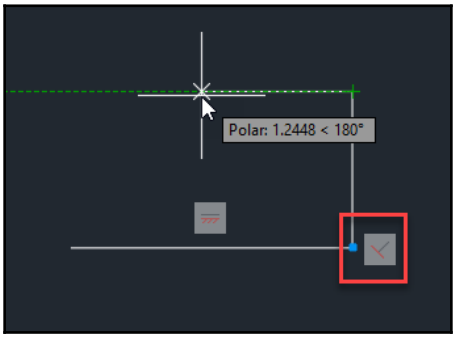


Figure 7. 57: Perpendicular and coincident constraints added along the vertical and horizontal lines

5. Similarly, complete the rectangle shape by making another horizontal line and then another vertical line.

You will notice that every time you add a new horizontal or vertical line, constraints are automatically added to the drawing because we have activated the Infer Constraints mode from the status bar. To understand these constraints, select any vertex of the rectangle we made in the previous example and try to move it in the drawing area.

You will observe that no matter how you move the vertex, the horizontal line will always remain horizontal and the vertical line will always remain vertical. Moreover, we made this rectangle using lines, but other vertices will not break apart when you move any line from its place, as shown in *Figure 7.58*.

All of this is possible because of constraints that are automatically added to the drawing while making them. In this drawing, horizontal, perpendicular, and coincident constraints are added to the drawing:

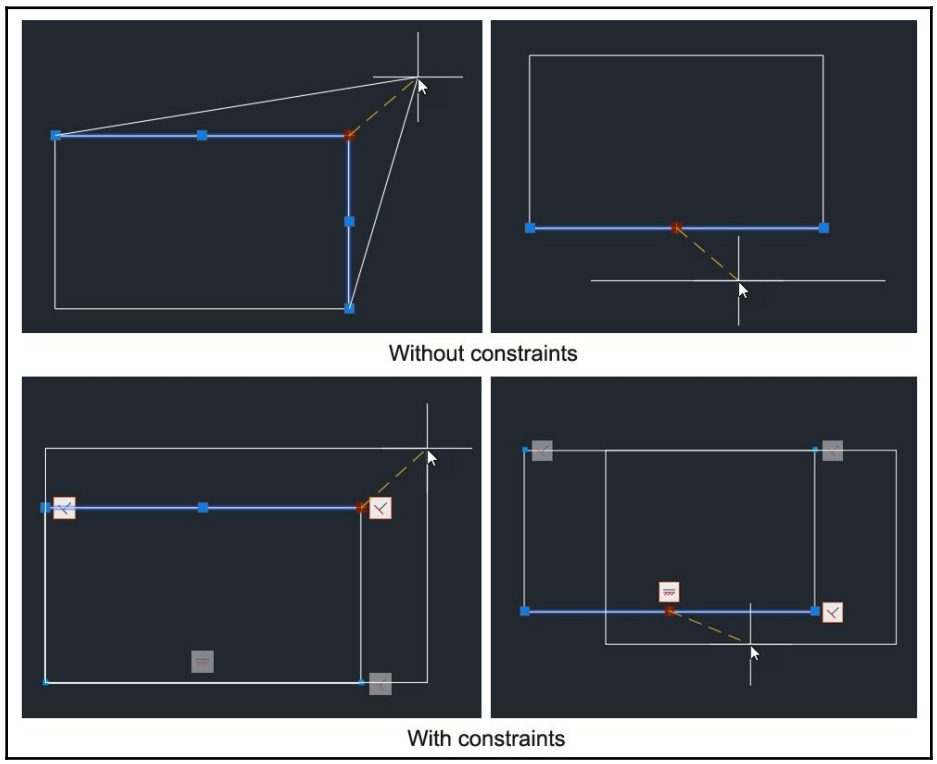


Figure 7. 58: Rectangle with and without constraints

Figure 7.56 highlights the horizontal constraint added to the horizontal line, and this constraint will keep this line horizontal. *Figure 7.57* shows the perpendicular and coincident constraints. The perpendicular constraint is shown by its icon and the coincident constraint is shown by a blue dot at the vertex of the drawing. The perpendicular constraint will keep the vertical line perpendicular to the horizontal one. The coincident constraint will keep the vertical and horizontal lines connected, or coincident.

Similarly, if you keep the Infer Constraints mode active and continue to make your drawing, other constraints will be automatically added to the drawing. You can also apply these constraints manually, which we will learn about in the following section.

Applying geometric constraints

To apply constraints manually, deactivate the Infer Constraints status bar toggle. This will ensure that the drawing is made without automatic constraints, and then we will be able to apply selected constraints to our drawing.

For this example, I will make the drawing shown in *Figure 7.59* and apply constraints to it:

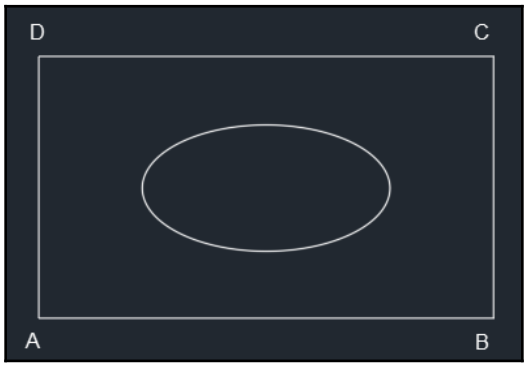


Figure 7. 59: Sample drawing for geometric constraints

Select the **Parametric** tab and you will find all the tools related to the parametric drawing on it, as shown in *Figure 7.60*:

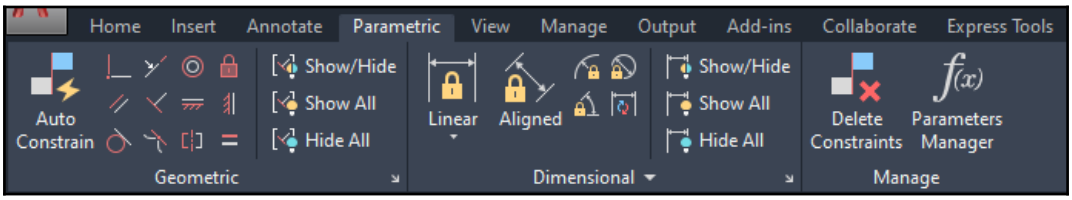


Figure 7.60: Parametric constraints options on the Parametric tab

This tab has three panels, namely, **Geometric**, **Dimensional**, and **Manage**. The **Geometric** panel has all the geometric constraints, the **Dimensional** panel has all the dimensional constraint tools, and the **Manage** panel has tools for managing the dimensional constraints and formulas.

So, here I will apply the constraints to the drawing shown in *Figure 7.59* and I will apply geometric as well as dimensional constraints:

1. Click the horizontal constraint icon in the **Geometric** panel of the **Parametric** tab, then click anywhere on line **AB**, and the horizontal constraint will be applied to the line. To see the name of the constraint, hover your cursor over the icon and the name of the constraint will show up on the tooltip, as shown in *Figure 7.61*:

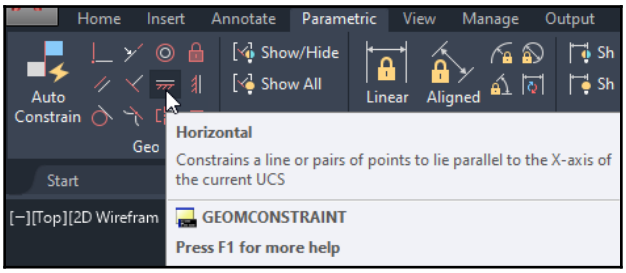


Figure 7. 61: Name of the geometric constraint on the tooltip

2. Select the vertical constraint from the **Geometric** panel and click on line **BC** to make it vertical.
3. Select the parallel constraint from the **Geometric** panel and then click on line **AB** and then on line **CD**, and both lines will now be parallel. Similarly, apply a parallel constraint between lines **BC** and **AD**.

4. Now we have constrained our rectangle geometrically, and if you now try to make changes to the shape of the rectangle by moving the lines or vertices, the geometry will remain a rectangle because of the constraints.
5. You can also apply all of these constraints automatically using the **Auto Constrain** tool of the **Geometric** panel. Click the **Auto Constrain** button on the **Geometric** panel of the **Parametric** tab, and then select the entire drawing shown in *Figure 7.59* and press *Enter*. The relevant geometric constraints will be applied to the drawing to keep its current shape.

So, now that our geometric constraints are in place, let's start adding dimensional constraints to the drawing.

Adding dimensional constraints

Now, we will start adding dimensional constraints to the rectangle, and later we will add dimensional constraints to the ellipse as well with the help of formulas.

So, let's begin by adding dimensional constraints to the drawing:

1. Select the linear constraint from the **Dimensional** panel of the **Parametric** tab.
2. Now click close to the midpoint of line **AD** and a red marker will highlight the midpoint, as shown in *Figure 7.62*. Now, click close to the midpoint of line **BC** and a dimensional constraint will show up on the cursor. Click above line **DC** and then click anywhere in the blank drawing area to place this dimensional constraint:

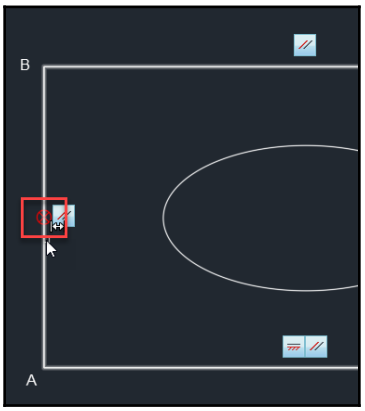


Figure 7. 62: The midpoint of line AB highlighted when the cursor hovers close to the midpoint

3. Select the linear constraint icon again from the **Dimensional** panel and then click close to the midpoint of line **AB**, then click close to the midpoint of line **CD**, and then click to the left of line **AD**. Click again in the blank area of the drawing to place this vertical dimensional constraint. The final drawing after adding the horizontal and vertical dimensional constraints will look like *Figure 7.63*. In this case, the dimensional constraints will constrain the length and width of the rectangle and you will no longer be able to change the length and width of the rectangle by moving lines or vertices:

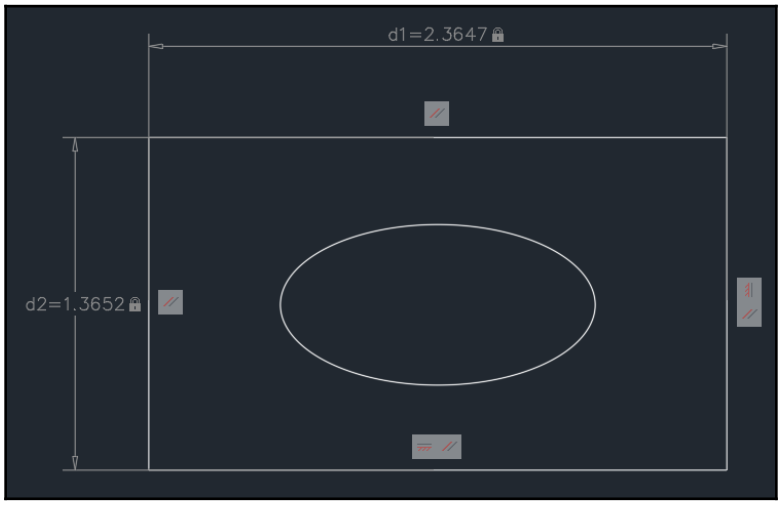


Figure 7. 63: Drawing after adding horizontal and vertical constraints

So, now our rectangle is almost fully constrained, but if you select the rectangle using any grip, you will still be able to move it, although, this time, the shape and the size of the rectangle will not change. Also, when you move the rectangle, the ellipse inside the rectangle will not move with it because we have not added any constraints to link the ellipse with the rectangle. So, in the next exercise, we will add the remaining constraints to our drawing, and we will also build some relationships between different geometries of the drawing using formulas.

Adding relationships between different geometries

In the following example, we will make the rectangle fully constrained and then we will add other geometric relationships to the drawing:

1. Click the fix constraint icon, which looks like a padlock, from the **Geometric** panel of the **Parametric** tab and click close to point **A** of line **AB**.
2. Point **A** will be highlighted with a red marker and the fix constraint will be applied to point **A**, making it fixed. With respect to point **A**, other lines will also become fixed as we have already applied other geometric constraints to them.
3. The rectangle is now fully constrained, and if you try adding more constraints to the rectangle, an error message will pop up:

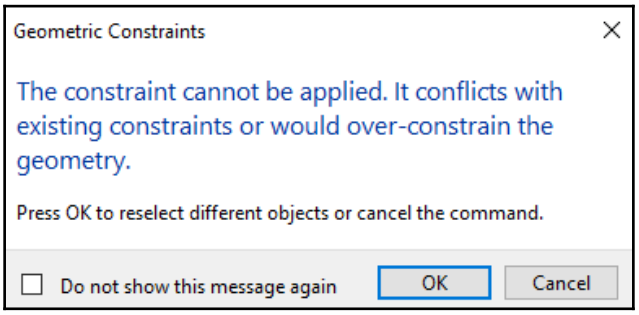


Figure 7. 64: The message that pops up when you try constraining a fully constrained drawing

Now, we will try to change some parameters in the drawing and see how this affects our drawing:

1. Double-click the **d1** dimensional constraint and the dimension text will be highlighted.
2. Change the length of the **d1** constraint from its original value to something else. In our case, the original length is 2.3647 and we will change it to `4`. Make sure you only change the number and not `d1= text`. So, the new value should be `d1=4`.

3. Click outside the dimensional constraint anywhere in the drawing area and you will notice that not only does the dimensional constraint value change, but also the length of the rectangle will now change to 4 units. This is all possible because of dimensional constraints.
4. Similarly, double-click the **d2** dimensional constraint and change its value to `d2=2`, and then click outside the constraint to apply the changes.

In this example, you saw that dimensional constraints control geometry and that you can change geometry by changing the values of dimensional constraints. Our constraints are applied only to the rectangle and, hence, the ellipse remains unaffected by all these changes.

Adding formulas to constraints

In order to make the ellipse constrained with respect to the rectangle, you need to add more dimensional constraints to it. We will use dimensional constraints and formulas to align the ellipse with the center of the rectangle:

1. Select the linear constraint from the **Dimensional** panel of the **Parametric** tab.
2. Click on the ellipse, then click close to the midpoint of line **AB**, and then click close to line **BC** to place the dimensional constraint. Click again in the empty drawing area to apply the changes and exit out of the linear dimensional constraint. This will add the **d3** dimensional constraint.
3. Click the linear dimensional constraint again and click the ellipse, then click close to the midpoint of line **AD**, and then move the cursor close to line **AB** to place the dimensional constraint. Finally, click on the empty area to apply the changes and exit the linear dimensional constraint. This will add the **d4** dimensional constraint.
4. Double-click the **d3** constraint and change its value to `1`, which is half the length of line **AD**, or the width of the rectangle.
5. Double-click the **d4** constraint and change its value to `2`, which is half the length of line **AB**, or the length of the rectangle.

6. The ellipse will now sit exactly at the center of the rectangle, as shown in *Figure 7.65*:

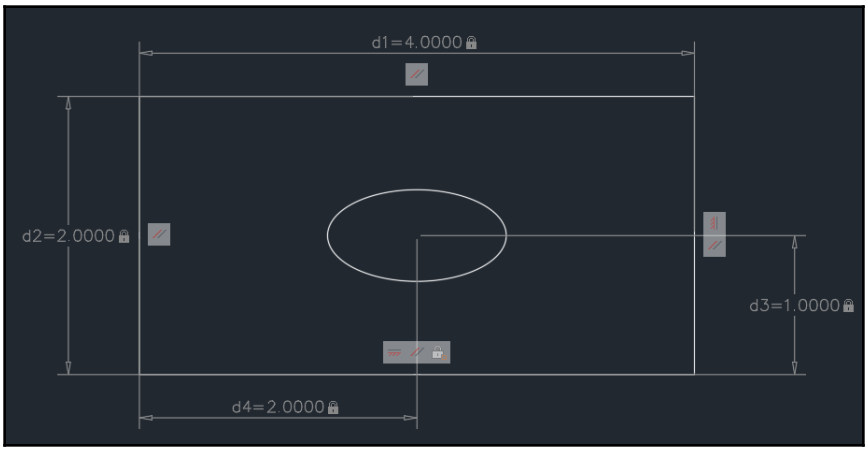


Figure 7. 65: Ellipse at the center of the rectangle

In this drawing, let's change the value of the **d1** dimensional constraint once again. Double-click on the **d1** constraint and type `d1=3`, and then click on the blank drawing area to apply the constraint. You will notice that although the length of the rectangle will now change to `3` units, the ellipse will remain where it was. Once again, the location of the ellipse will no longer be at the center of the rectangle.

If you want to fix this issue so that no matter what value you use for the length or width of the rectangle, the ellipse remains at the center, then you need to add some formulas. The following are the steps for adding formulas to dimensional constraints:

1. Double-click the **d3** dimensional constraint and change its value to `d3=d2/2`, and then click anywhere in the blank drawing area to apply the changes and exit the dimensional constraint command: we have our first formula.
2. In this case, the value of the **d3** dimension is now linked to the **d2** dimension, and it will now remain always half of **d3**, as defined in the formula, and hence the center of the ellipse will remain exactly at the center of line **AD**.
3. For the other direction, we need to repeat the steps for the **d4** dimensional constraint. In this case, use `d4=d1/2` as the value, and now we will have a drawing where the ellipse will always remain at the center of the rectangle.

4. You can verify the results by changing the value of **d1** or **d2** to any value. The final drawing, with all the constraints and formulas, will look like *Figure 7.66*:

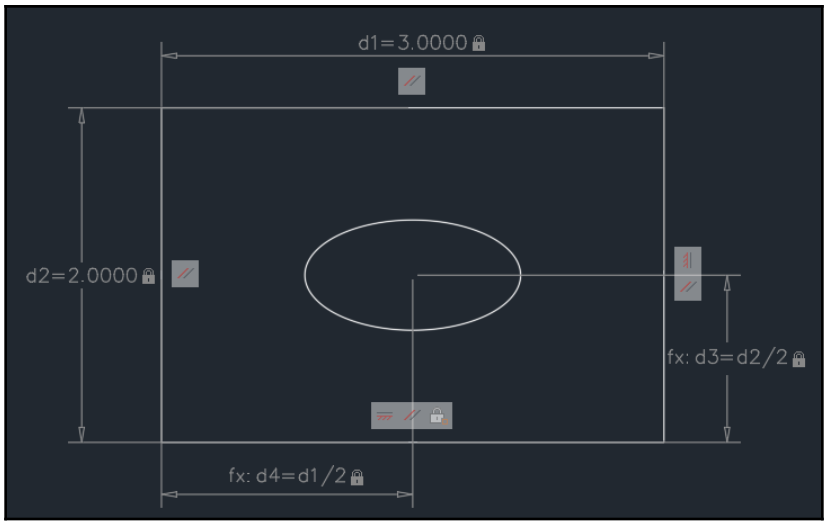


Figure 7. 66: Final drawing with all constraints and formulas

Although we added a very simple formula here, you can make even more complex formulas involving other parameters, such as multiplication, factors, squares, and square roots.

This was the parametric tool that you can use anywhere you want to link different parts of the geometries with parameters. We will use some of these parametric tools with dynamic blocks as well. With this topic we have also concluded this chapter and before we move on to the next chapter, let's summarize the topics we have discussed so far.

Summary

This chapter was about the advanced tools and features of AutoCAD. In this chapter, we learned how to make AutoCAD tables and also use Excel tables right inside AutoCAD. We learned how to add tables as data links and use formulas in tables. We also learned how to add fields in our drawing, which are dynamic pieces of text connected to other drawing properties.

We learned how to make isometric drawings, which are drawings that look like 3D drawings with *X*-, *Y*-, and *Z*-axis values. Finally, we learned about parametric tools, which are especially helpful when it comes to dynamic blocks. Using the parametric feature, we also learned how to make drawings with constraints and formulas.

In the next chapter, we will continue to learn about more advanced tools; we will mostly focus on customization-related features of AutoCAD, such as customizing linetypes and the user interface.

8
Customization Tools

This chapter is all about customizing the AutoCAD user interface and custom objects. In this chapter, we will learn how to make our own linetypes and Hatches patterns, and we will also learn how to transfer these custom objects from one drawing to another. We will then learn how to customize the AutoCAD user interface to make it look more personalized, and later we will talk about using some internal and external resources for blocks, such as Design Center and tool palettes. We will conclude the chapter by learning about complex polylines and splines, and we will also learn how to modify existing polylines and splines.

These are the topics that we are going to cover in this chapter:

- Making custom linetypes
- Making custom Hatches
- Customizing the user interface
- Using Design Center
- Using tool palettes
- Working with complex polylines and splines

So, let's get started with creating custom linetypes in AutoCAD.

Making custom linetypes

There are plenty of linetypes in AutoCAD, such as Hidden, Dashed, Border, and Center, but sometimes we may just need a linetype that is custom made for our drawing, and AutoCAD has tools for just that. You can not only make custom linetypes, but also custom Hatches using these tools.

In this section, we will learn how to make linetypes using the Make Linetype Express tool and also using linetype code. Similarly, we will also learn how to make Hatches using SuperHatches Express tool. So, let's get started making a custom linetype using the Make Linetype tool.

Making a simple linetype using the Make Linetype tool

Linetypes can be made with text, symbols, and complex shapes, but we will begin by making a linetype with simple text and symbols:



Figure 8.1: Sample linetype with text and symbols

In the following example, we will make a linetype containing the text `water` wrapped inside two double arrows pointing towards the text and surrounded by lines, as shown in *Figure 8.1*:

1. To begin making this linetype, select the **Mtext** tool from the **Annotation** panel of **Home** tab or use its command alias, `MT`.
2. Now, make a text box and type `water`.
3. To add the symbol, click the **Symbol** icon on the **Insert** panel of the **Text Editor** tab and select **other** option from the list of symbols.
4. The **Character Map** window will open. Select the double arrow symbol from the character map, then click on the **Select** button and then the **Copy** button, and close the **Character Map** window:

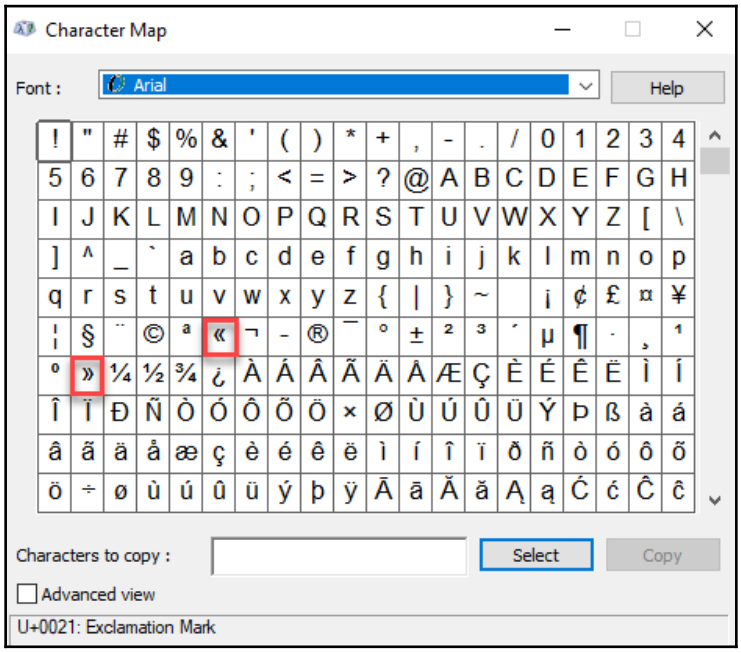


Figure 8.2: Symbols in the Character Map window

5. In the textbox, right-click next to water and select **Paste** from the context menu. Repeat the process to paste the arrow pointing in the other direction too.
6. When you are done adding the text and the symbols, exit the **Mtext** tool and add two lines, one to the left and another one to the right of the text.
7. Linetypes can only be made with single-line text, and we have used multiline text here. So, we need to convert the multiline text into single line text and we can do it by using the Explode command.
8. Select the **Explode** option from the **Modify** panel of the **Home** tab or use its command alias, X.
9. Select the text and the arrow symbols and press *Enter*, and the text along with symbol will be exploded and you will end up with single line text.
10. Also, make sure you have made the lines with the line command and not with the polyline command, or you won't be able to make the linetype.

So, now that we have made the first pattern of the linetype, we will convert this pattern into a linetype. Follow these steps to convert this pattern into a linetype:

1. Go to the **Express tools** tab and, from the expanded **tools** panel, select the **Make Linetype** option. Alternatively, you can also use its command alias, `MKLYPE`:

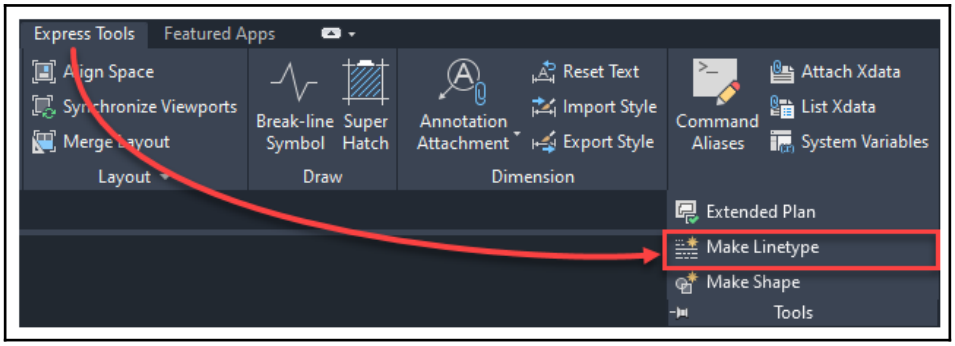


Figure 8.3: The Make Linetype tool on the expanded tools panel of the Express tools tab

2. Now, the **Select Linetype File** window will open. Here, you need to save the linetype file, which will hold the code required to make this linetype. Specify a location and a name and then hit the **Save** button and the linetype file will be saved with the LIN extension. In our example, I have named my linetype file `Sample line`, and I saved it to the desktop:

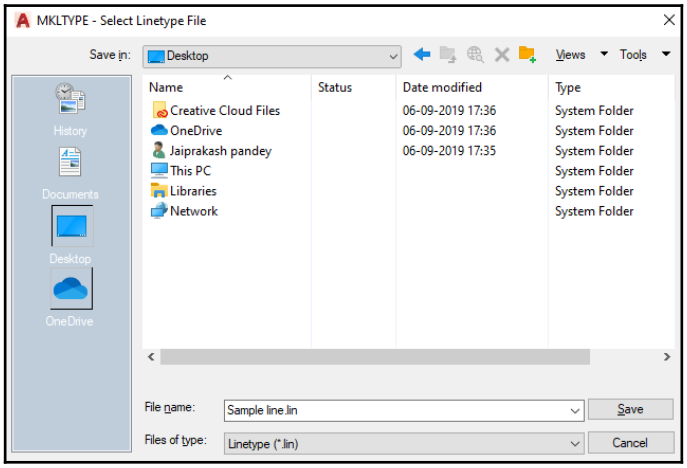


Figure 8.4: The Sample line.lin file saved on the desktop using the Select Linetype File window

3. Now, the command line will prompt you to specify the name of this linetype. Type `waterline` in the name and press *Enter*.
4. The next prompt is for the description. Type `linetype for water pipeline` and press *Enter* again.
5. Now, the command line will prompt you to specify the starting point of the linetype definition. Click on point **A**, and for the ending point click on point **B**, as shown in *Figure 8.5*:



Figure 8.5: Starting and ending point of the linetype labeled with point A and B

6. The next prompt is **Select objects**. Here, make a selection box to include all the objects (the text, the symbols, and the two lines) and press *Enter*.
7. You will see a message that says **WATERLINE created and loaded**. This indicates that we have successfully created a new custom line and that it is also loaded in our drawing and ready to use.

To test this linetype, make a line that is larger than the text, symbol, and line segments we used to make the linetype. Select this line, then go to the **Properties** panel of the **Home** tab, expand the linetype drop-down, and select the **WATERLINE** from the linetype list, as shown in *Figure 8.6*:

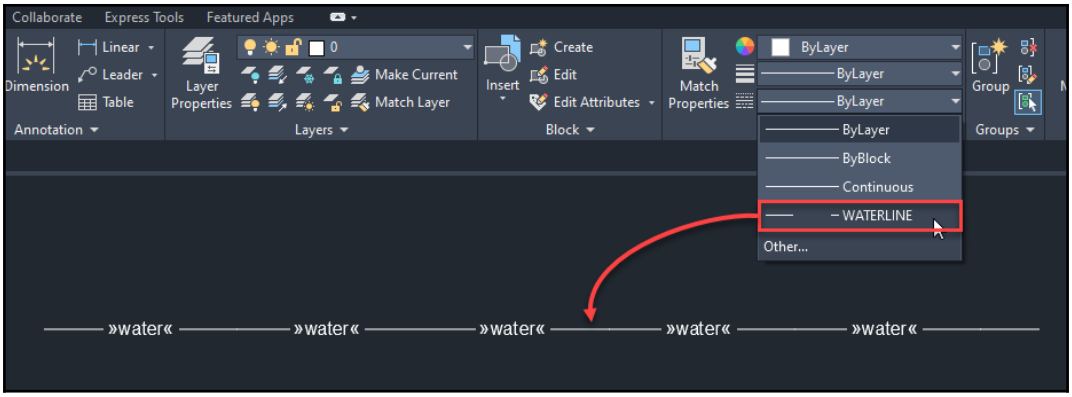


Figure 8.6: Waterline added to the list of linetypes in the properties panel

The custom linetype will be applied to this line. You can apply this linetype to a circle or other geometries as well, and you can even change the scale of this linetype from the properties palette to make the linetype bigger or smaller.

This was an easy and straightforward method of making a linetype in AutoCAD, and we used this method to make a simple linetype. In the next section, we will make another custom linetype using complex shapes.

Making complex linetypes using Express tools

The Make Linetype tool is restrictive: it only allows us to make linetypes using single-line text and lines and it certainly does not allow us to make linetypes using geometries such as a polyline or an arc. But despite these limitations, you can still use Make Linetype to make linetypes containing complex shapes. And in this section, we will learn how to do that. We will make a linetype using complex shapes, as shown in *Figure 8.7*:



Figure 8.7: Linetype segment made with a line and arcs

In this case, we will use two different tools to make a linetype containing a line and two arcs. As you know, we can't use arcs in our linetype, so we will instead use a shapefile, which can be used for linetypes. But to do that, we need to first convert the drawing shown in *Figure 8.7* into a shape that can be then used in linetypes.

Here is the workflow of converting our drawing into a shapefile:

1. Go to the **Express tools** tab and select **Make Shape** from the expanded **tools** panel, as shown in *Figure 8.8*:

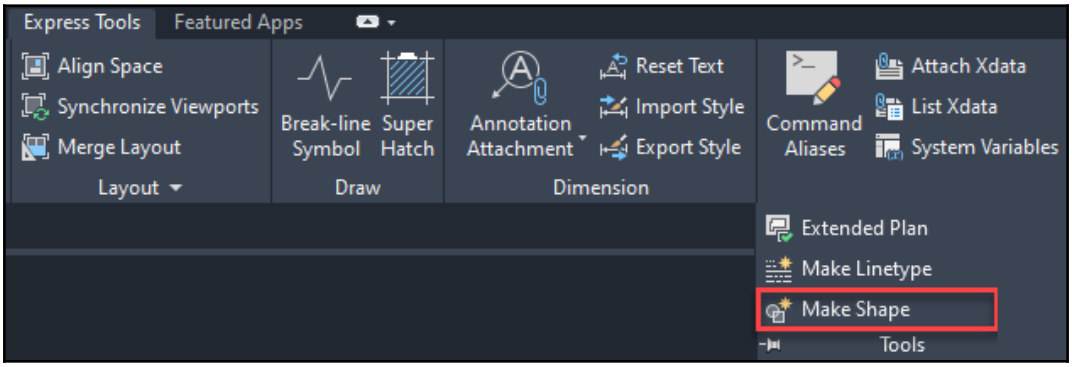


Figure 8.8: The Make Shape option on the expanded tools panel of the Express tools tab

2. The **Select Shape File** window will open. Specify a location and a name for the shapefile. In this example, I am using the desktop as the location and naming this file `test shape`. Click **Save** when you are done.
3. Now, the command line will prompt you to specify the name of the shape. For this example, I have used `SHP`. So, type the name `SHP` and press *Enter*.
4. Now, press *Enter* again to accept the default resolution of `128` from the command line.
5. For the next prompt, you need to specify the base point, which in this case is the leftmost point of the drawing, as highlighted by the red circle in *Figure 8.9*:



Figure 8.9 - Basepoint of the shape highlighted in the red circle

6. The next prompt is **Select Objects**. In this case, make a window to select all the objects, which in this case is the line and two arcs, and then press *Enter* key.

7. The `shp` shapefile has been created, and now it's ready to use. We will insert this shapefile in our drawing and then we will use this shapefile instead of our original drawing. To insert the shapefile, type `SHAPE` and press *Enter*.
8. When the command line prompts for the name of shapefile, type `SHP` and press *Enter* again.
9. You will notice that a shapefile will now show up on your cursor. Click on the base point of the existing drawing, which is highlighted with the red circle in *Figure 8.9*.
10. Press *Enter* to accept the default height from the next prompt, and press *Enter* again to accept the default rotation angle as well.

The shape will be added overlapping the original drawing, and now we can use this shapefile to make our linetype. Follow these steps to convert this shapefile into a linetype:

1. Select the **Make Linetype** tool from the expanded **tools** panel of the **Express Tool** tab or use its command alias, `MKLTYPE`:

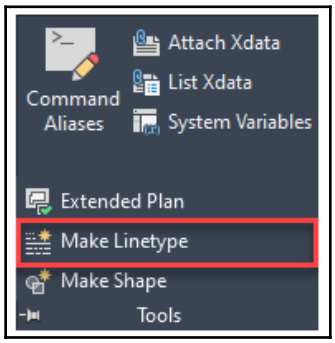


Figure 8.10: The Make Linetype tool on the expanded tools panel

2. Once again, the **Select Linetype File** window will open. Specify the desktop as the location and `new linetype` as the name of the linetype file, and then click the **Save** button.
3. Now, the command line will prompt for the name of the linetype. Type `shapeline` and press *Enter*.
4. In the next prompt, which is for the description, type `linetype made with shapefile` and press *Enter* again.

5. For the starting point, click on point **A**, and then for the end point, click on point **B** as shown in *Figure 8.11*:



Figure 8.11: The start and end point of the linetype labeled with points A and B

6. The next prompt is for selecting the objects. Click on the shapefile from the drawing area and don't select the arc or line underneath the shapefile.
7. Press *Enter*, and you now have a new linetype called `shapeline` made with a shapefile.

To use this linetype in a drawing, we will make a line that is longer than the shapefile, then select the line and click on the linetype drop-down in the **Properties** panel of the **Home** tab, select the **SHAPELINE** linetype, and the linetype will be applied to the selected line:

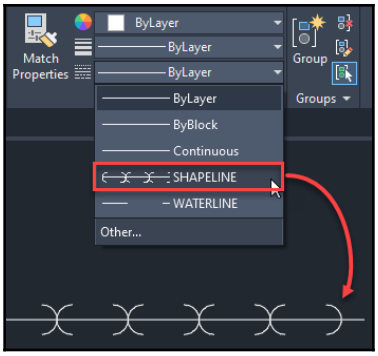


Figure 8.12 - Shapeline added to the list of linetypes in the Properties panel

You can also apply this linetype to a circle or other drawings, and you can even change the scale of this linetype to make it look bigger or smaller using the linetype scale option on the Properties palette.

So far, we have made linetypes using **Express tools**, but you can also make linetypes using a type of code exclusive to AutoCAD, which we will call linetype code here. In the next section, we will learn about linetype code, and we will also make a linetype using linetype code.

Making linetypes using code

To understand how linetype code works, we will look at the code created for the Waterline linetype, which we created in an earlier example. Open the `Sample line.lin` file from the desktop using Notepad or any other plain text editor. To open it, select the `Sample line.lin` file, then right-click and select **Open with**, then **Notepad**, from the series of options. The file will open, and you will see code that looks like this:

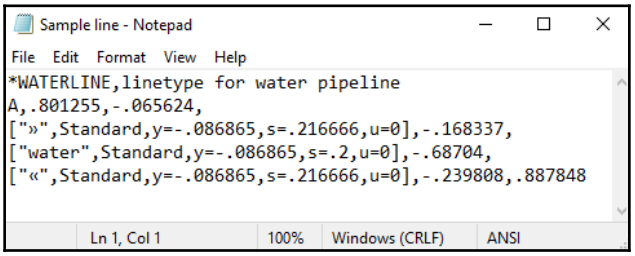


Figure 8.13: Linetype code in the Sample line.lin file

Before we try to understand this linetype code, let's simplify the existing code with the following steps:

1. Copy the entire text and paste it underneath the existing code so that we end up with the same copy of code twice in the file, as shown in *Figure 8.14 A*.
2. Now, delete the highlighted text in *Figure 8.14 A* so that the resulting code looks *Figure 8.14 B*:

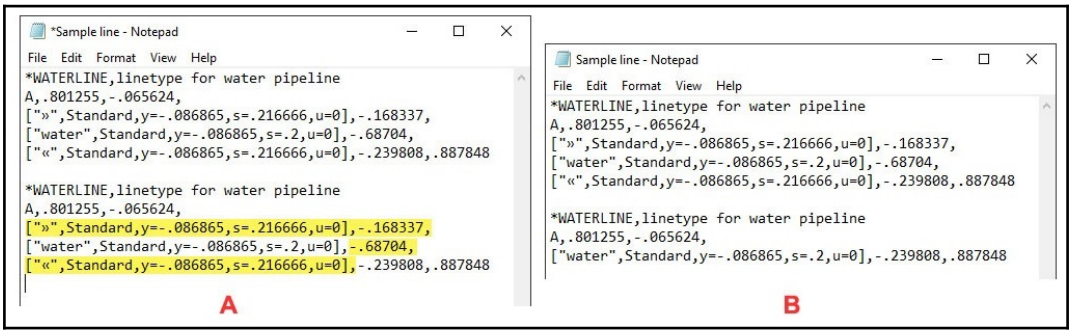


Figure 8.14: Linetype code before and after modifications

3. In this simplified code, we will modify some more values to make it even more simple. I will swap 0.801255 with 0.80, -0.065624 with -0.07, 0.086865 with 0.09, -0.239808 with -0.24, and 0.887848 with 0.89.

So, the final simplified code in just two lines will look like *Figure 8.15*. I have replaced the multiple decimal places with just two, rounding off the remaining decimal places:

```
*NEWWATERLINE,linetype for water pipeline
A,.80,-.07,["water",Standard,y=-.09,s=.2,u=0],-.24,.89
```

Figure 8.15: Simplified linetype code of the new waterline linetype

Now, that we have a simplified code, let's try and understand what each component of this code means.

In the first line of the code, `WATERLINE` is the name of linetype, and after the name we have a description of the linetype.

In the second line, we have the letter `A`, which indicates the point where the linetype begins.

After A, the value `.80` is the length of line and the next value, `-.07`, is the gap width. So, the length of the line has a positive value and the gaps have negative values.

The next part of this code is text inside square brackets. Here, water is the text that will show up on the linetype and Standard is the text style. The `y` value of `-.09` is the offset value of the text along the *y* axis, and `S=.2` is the scale of the text. U=0 is the upright value.

The next set of values are outside the square brackets, and this is once again a gap of .24 (indicated by its negative value) and then the length of line is `.89`.

So, finally, if you read the code you will get the length of the line, `.80`, then a gap of `.07`, then the text water, then a gap of `.24`, and finally, a line of length `.89`. To see this linetype code in action I will change the name of this linetype, and to do this, change `WATERLINE` to `NEWWATERLINE` in the first line of the linetype code and save the Notepad file again as `sample.lin` on the desktop. Make sure you save the Notepad file with the LIN extension, or this linetype code will not work.

Follow these steps to use this new linetype code in our drawing:

1. To use this linetype code, open a blank drawing, go to the linetype drop-down menu of the **Properties** palette on the Home tab, and click on the **Other...** option:

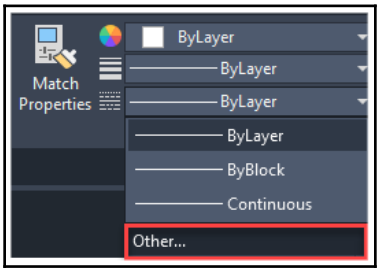


Figure 8.16: The Other... option in the linetype drop-down menu of the properties panel

2. Now, click the **Load** button, and then click the **File** button in the **Load or Reload Linetypes** window:

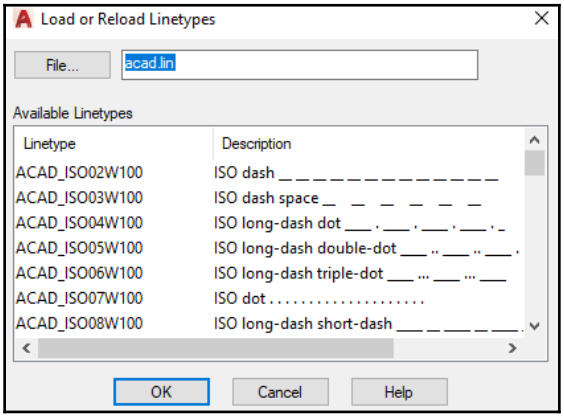


Figure 8.17: The File button in the top left of the Load or Reload Linetypes window

3. Locate the `sample.lin` file from desktop and click on **Open** button. You will notice two linetypes, `NEWWATERLINE` and `WATERLINE`, load up.
4. Select the linetype that you want to load, or add bot**h the linetypes by pressi**ng and holding the *Ctrl* key and then click **OK**. Click **OK** again, and you will have both linetypes loaded in the linetype menu for use.
5. In this case, we made a copy of the existing linetype code and renamed the new code `NEWWATERLINE`, so we have a new linetype here. The properties of this linetype will be different as we have modified it.
6. Apply both the linetypes to two different lines, and you will have linetypes that look like this:

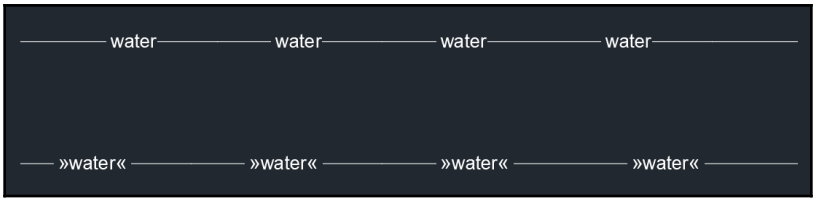


Figure 8.18: Newwaterline and Waterline linetypes applied on two lines

The linetype at the top is the linetype that we modified, and the second linetype is the one we created using the `MKLTYPE` Express Tool. The first linetype, which we modified, has only text and a line. We removed the lines from the linetype code that contained information related to symbols, and hence we have a simplified linetype here.

So, this is how we can make custom linetypes using **Express tools** and linetype code. To export custom linetypes, you need to just send the linetype file, which has a LIN extension, and then import this linetype file in the drawing where you want to use the custom linetype. Just note that LIN files will not work with linetypes with shapefiles because linetypes containing shapes need the shapefile as well as the linetype file.

Now, that we are familiar with making custom linetypes, let's move onto custom Hatches, which are like making linetypes.

Making custom Hatches patterns

Just like custom linetypes, you can use "hatch code" as well as an Express Tool called "Super Hatch" to make custom Hatches in AutoCAD. In this section, we will make a custom Hatches using Super Hatches.

Before we begin making the Hatches pattern, we need to make the first component of the pattern and convert it into a block. This block can then be used to create the required Hatches. For this example, I will create a pattern that looks like this:



Figure 8.19: Sample shape for making Hatches pattern

In the following example, we will convert the drawing shown in preceding diagram into a block which will be later used in the custom Hatch pattern:

1. To convert this drawing into a block, select the **Create** option from the **Block** panel of the **Home** tab or use its command alias, `B`.
2. Give this block a name. I will name it `Hatches` pattern. Then specify the base point at the center of the drawing, which is the intersection point of the two perpendicular lines.
3. Now, click on the **Select Objects** box and select all the objects in the drawing. Make sure the **Convert to block** radio button is checked.
4. Click **OK** when you are done, and you will have a block made from the drawing shown in *Figure 8.19*.

Now, that we have our block in the drawing, it's time to make the Hatches pattern using this block. But before you make the Hatches pattern, make a rectangle or any other closed drawing that is larger than the block. This will be the closed boundary where the Hatches pattern will be applied.

So, in the following example, we will learn to make our custom Hatch pattern using block and we will also learn to apply it in a closed area.

1. Select the **Super Hatch** tool from the **Draw** panel of the **Express tools** tab, as shown in *Figure 8.20*, or use its command alias, `SUPERHatches`:

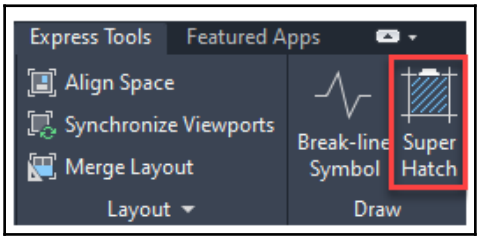


Figure 8.20: Super Hatches tool on the Draw panel of Express tools tab

2. The **SuperHatch** palette will open with many options on it. Select the **Block** button from the palette, click the **Block** button in the top left of the **SuperHatches – Insert** window, select the **Hatches Pattern** block from the list of blocks, and click **OK**. Click **OK** again to close the **SuperHatch** palette:

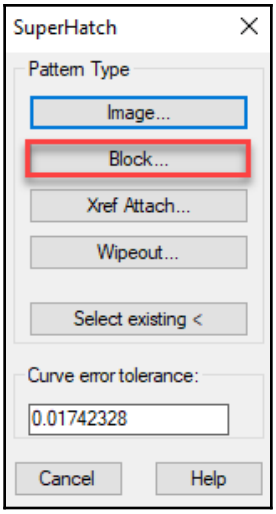


Figure 8.21: The Block button on the SuperHatch palette

3. Now, click inside the Hatches area or the closed drawing where you want to add the Hatches pattern.

4. Press *Enter* in the next prompt to accept the default scale of `1` along the *x* axis, and then press *Enter* again to accept the default scale of `1` along the *y* axis too.
5. Press *Enter* again to accept the default rotation angle of `0` degrees.
6. Now, the command line will prompt "**Is the placement of this BLOCK acceptable**", press *Enter* to select the default option, which is `Yes`.
7. Now, the command line will prompt you to specify the block extents, and the block will also show a purple box around it, as shown in *Figure 8.22*. Click the lower-left corner of this purple box and then click on the upper-right corner of this box and press *Enter*:

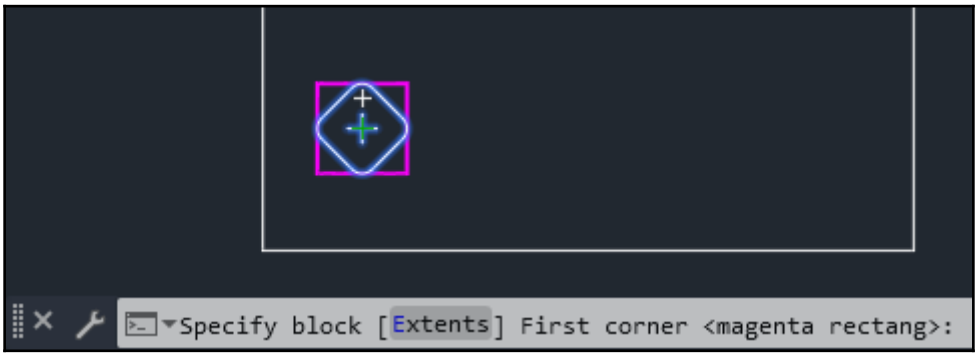


Figure 8.22: Hatches pattern bounding box shown in purple

8. Now, click anywhere inside the area where you want to add the Hatches pattern and press *Enter*.
9. The Hatches pattern will be added in the selected area, as shown in *Figure 8.23*:

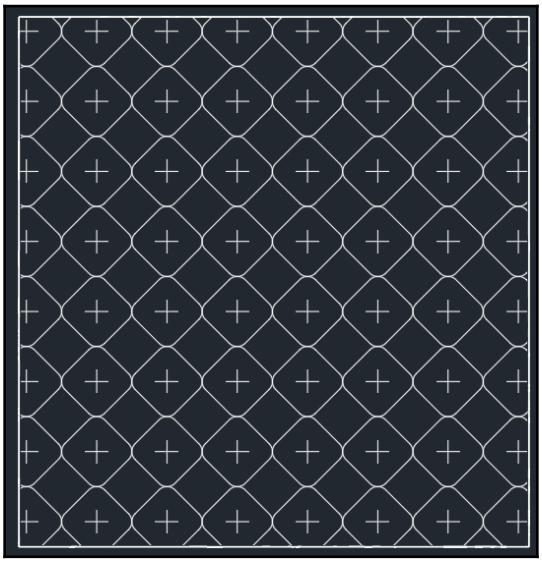


Figure 8.23: Hatches pattern applied in the selected area using the SuperHatches tool

In a similar way, you can add this custom Hatches pattern to any other closed area in your drawing. The custom Hatches pattern made here using SuperHatches has its very own limitation as well. This Hatches pattern is limited to the current drawing and you can't transfer this Hatches pattern to any other drawing. This may be a bit limiting in situations where you want to use the same pattern on different drawings. But nonetheless, you now know about a tool that makes custom Hatches a breeze.

After making custom linetypes and Hatches, it's time we move onto customizing the user interface and making it much more personalized. The next section is dedicated to the customization of the user interface and saving those customizations.

Customizing the user interface

The default workspace of AutoCAD, which is also called the Drafting and annotation workspace, has a clean and intuitive layout, with all the tools and commands easily available on their respective tabs and panels. But sometimes this user interface is just not enough, and you may need some custom commands in an easily accessible place, such as the Home tab. If that is the case, then you can customize the workspace as you like.

In this section we will do just that. We will learn how to make our own custom panel with our own custom commands, we will add that panel to the Home tab, and finally we will save these settings and other settings too as a custom workspace.

So, let's get started with adding a custom panel to the **Home** tab.

Making a custom panel

If you look at the **Home** tab, you will find panels such as **Draw**, **Modify**, and **Annotation**. These panels contain the default AutoCAD commands, but now we will add our own custom panel to this list of panels, and we will also add our own list of commands to this panel. Follow these steps:

1. To make the customization go to the **Manage** tab and select the **User Interface** tool from the **Customization** panel, as shown in *Figure 8.24*, or use its command alias, `CUI`:

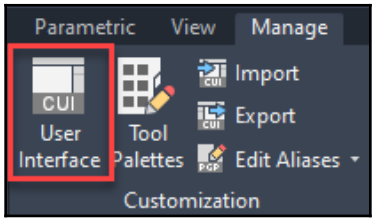


Figure 8.24: User Interface option on the Customization panel of the Manage tab

2. In the **Customize User Interface** window, select the **Customize** tab from the top of the window and then expand the **Ribbon** tree, as shown in *Figure 8.25*:

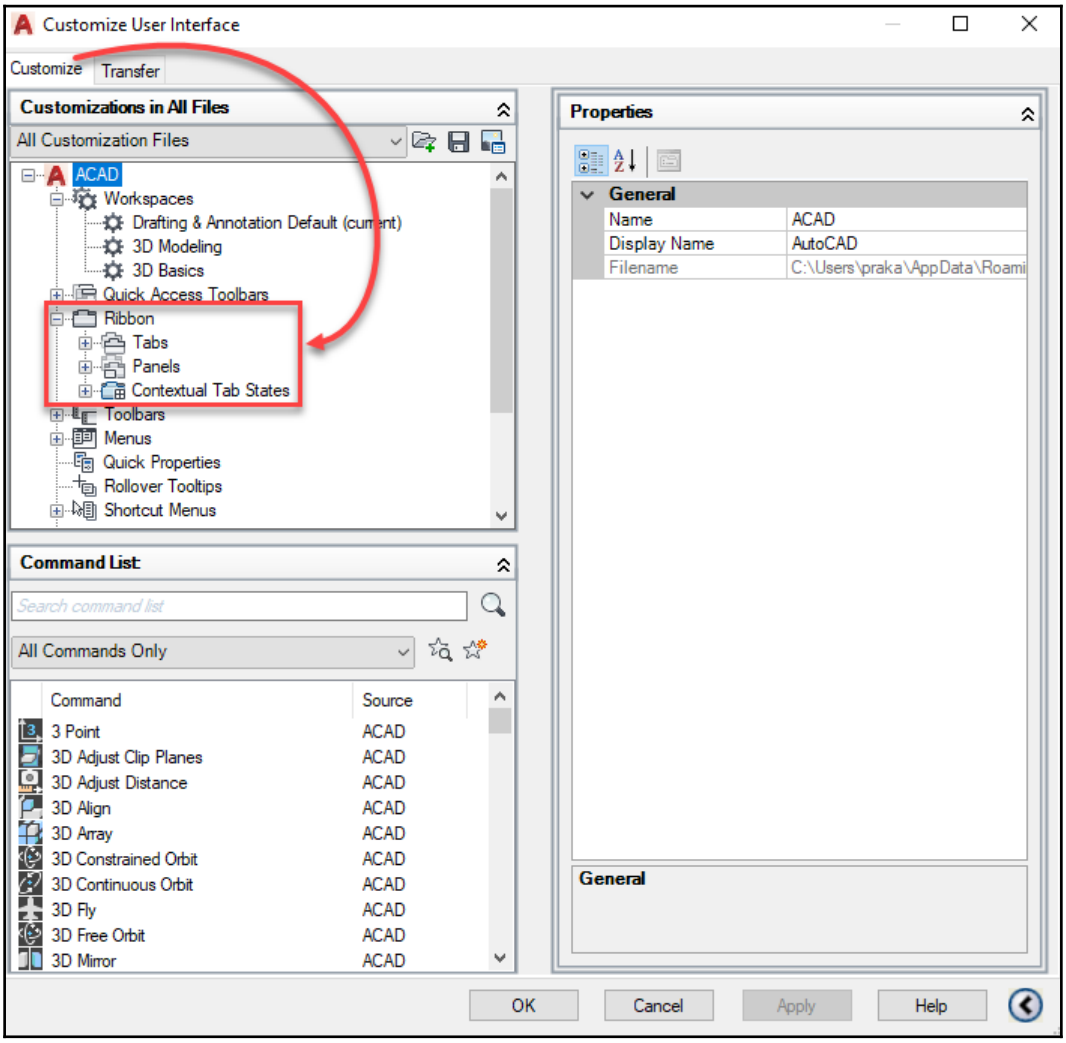


Figure 8.25: Ribbon tree expanded in the Customize tab

3. Right-click on the **Panels** option from the **Ribbon** tree and select **New Panel** from the menu. The panel will be added to the bottom of the list of panels in the panel tree; give it a name. For this example, name it `Sample Panel`.
4. Now, that we have a new panel, it's time to add the commands to this panel. Go to the **Command List** panel and search for the command you want to add in the search bar, as shown in the yellow box of *Figure 8.25.*

5. In this case, I will start by searching for the `Purge` command in the command list search box. There may be more than one command in the list of commands. Select the one you need, then right-click on it and select **Copy** from the context menu, as shown in *Figure 8.26*:

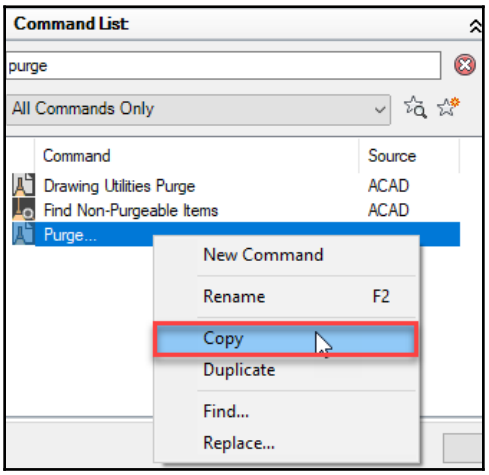


Figure 8.26: Copy the command using the right-click menu

6. Now, click on **Row 1** under `Sample Panel`, and from the right-click menu select **Paste**, and the command will be added to the newly created `Sample Panel`, as shown in *Figure 8.27*:

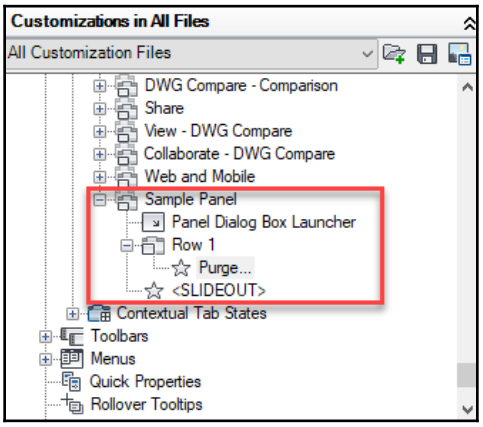


Figure 8.27: The Purge command added to Sample Panel

7. Repeat the process to add other commands to **Row 1** of `Sample Panel`. In this example, I have added the `Circle` command and the `Units` command to the first row.
8. Right-click on `Sample Panel` and then select **New Row** from the menu. A new row will be added to `Sample Panel`. You can add more commands to this new row, and you can create as many rows as you want.
9. To add the commands in the expanded panel, copy the command and paste it in the `<SLIDEOUT>` option of `Sample Panel`.
10. When you are done adding commands to the panel, right-click on `Sample Panel` and select **Copy** from the context menu. Now, collapse the **Panels** tree by clicking the - sign next to the panels, and then expand the **Tabs** tree under **Ribbon** in the customization panel, as shown in the red box of *Figure 8.25*.
11. Select the **Home – 2D** tab, right-click on it, and select **Paste** from the context menu. You will notice that `Sample Panel` will be added to the bottom of the list of panels on the **Home—2D** tab, as shown in *Figure 8.28*:

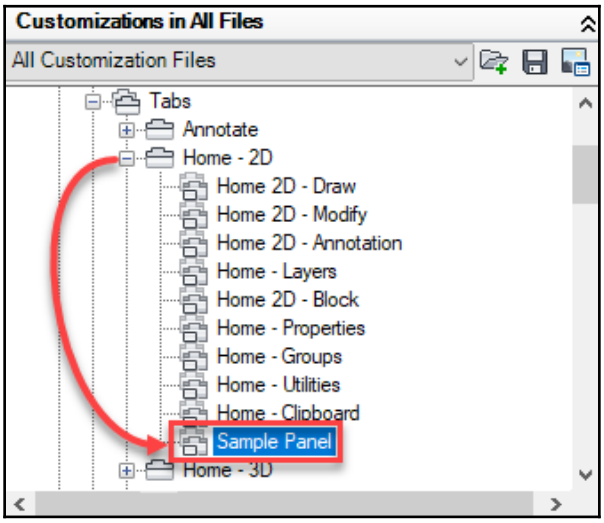


Figure 8.28: Sample Panel added to the Home tab

12. Click **OK** to close the **Customize User Interface** window. If you look at the **Home** tab now, you will find your own `Sample Panel` with your selected set of commands on it, as shown in *Figure 8.29*. You can now select the commands from this panel just like any other command:

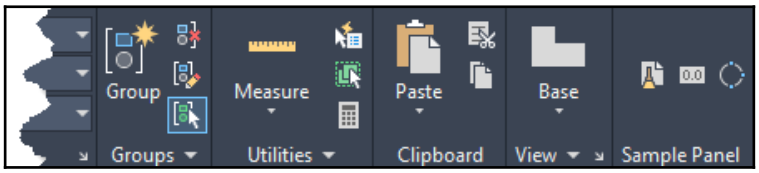


Figure 8.29: Sample Panel at the end of the Home tab with selected commands

To delete the panel, go to the **Customize User Interface** window, expand the **Panels** tree under the **Ribbon** tree, and select `Sample Panel` from the bottom of the panel list. Then right-click on it and delete it. Click **OK** to close the **Customize User Interface** window, and you now have the default **Home tab** without `Sample Panel`.

This is one of the customizations that you can implement in AutoCAD. There are other customizations as well. So, let's now discuss customizing the user interface with existing panels and toolbars, and then we will learn how to save these customizations as a workspace in the next section.

Making user interface changes

You can change the way AutoCAD looks by moving the panels and tabs and by activating the toolbars. Let's start by modifying the ribbon area, which has all the panels and tabs.

Click and drag on the blank part of the **Draw** panel of the **Home** tab and move your cursor in the drawing area. You will notice that the **Draw** panel will detach from the **Home** tab and will now show up in the drawing area. Similarly, you can drag out other panels from different tabs and place them anywhere in the drawing area.

To send the panels back to their original location on the ribbon area, click the return panel icon, as shown in *Figure 8.30*:



Figure 8.30: Return panel icon highlighted in the red circle

You can also open the command toolbars, which were a standard feature in classic AutoCAD. To open the toolbars, click the down arrow on the quick access bar and select the **Show Menu Bar** option, as shown in *Figure 8.31*:

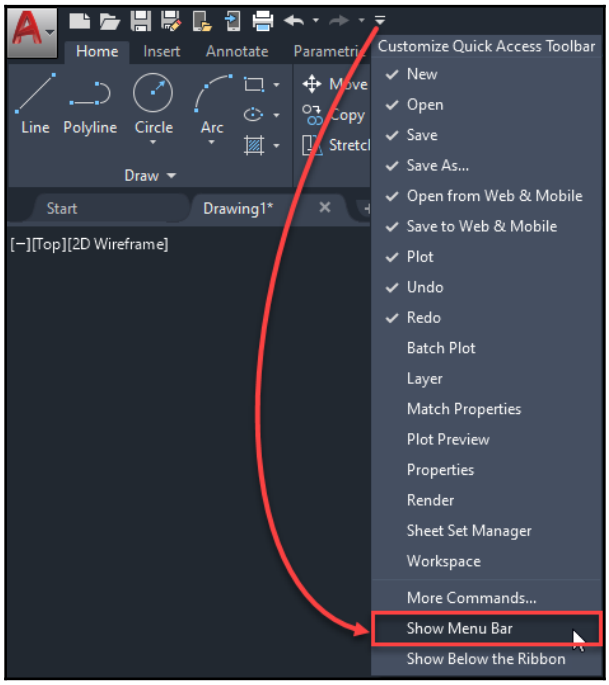


Figure 8.31: Show Menu Bar option

Now, a menu bar on top of the ribbon area will appear. Click on **tools**, then **Toolbar**, and then **AutoCAD** from the menu, and there you will see a list of toolbars that you can activate:

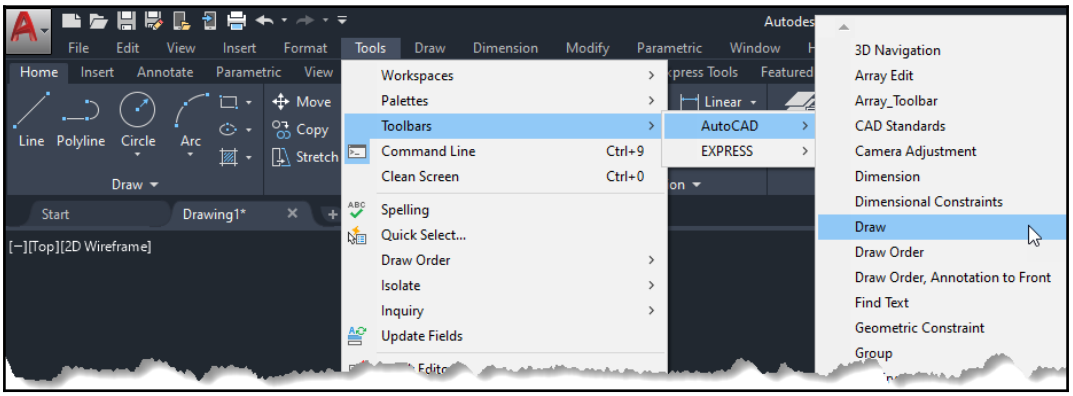


Figure 8.32: Toolbars available in the tools drop-down menu of the menu bar

Select **Draw** from the toolbar list and then select **Modify** and **Layer**, and the three toolbars will be added to the drawing area. These toolbars are floating toolbars, which means you can grab the side of these toolbars and move them wherever you want in the drawing area. You can also dock these toolbars to the side or to the top of the drawing area.

Now, that we have made lots of customizations in AutoCAD, you can save it as a workspace if you want. To save these customizations as a workspace, click the gear icon on the status bar and select the **Save Current As...** option from the menu, as shown in *Figure 8.33*:

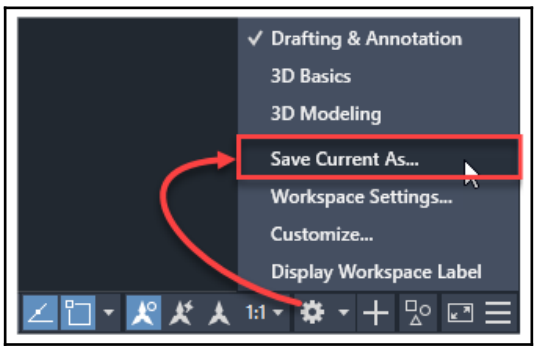


Figure 8.33: The Save Current As... option for saving the current settings as a workspace

Give this workspace a name. I will name it `Sample workspace` and then click the **Save** button. We now have a new workspace with a customized user interface.

To switch to the default workspace, click on the gear icon again, select Drafting & Annotation from the menu, and the default workspace will return. To switch back to the workspace with the customizations, click on the gear icon again and select `Sample workspace` from the list; you will once again get the customized workspace.

If you want to delete the custom workspace, switch to the Drafting & Annotation workspace, type `CUI` and press *Enter*, and the **Customize** user interface window will open. This window will show your custom workspace in the list, as shown in *Figure 8.34*:

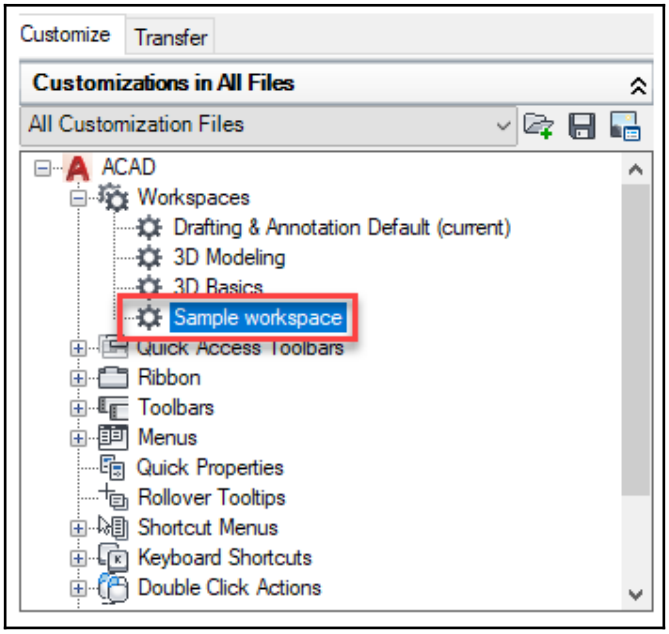


Figure 8.34: Sample workspace added to the list of workspaces

Select the workspace, then right-click and select delete. Once the custom workspace is deleted from the list of workspaces, click **OK** to close the **Customize User Interface** window, and your custom workspace will disappear.

So, this was AutoCAD interface customization. Now, let's move onto an entirely different are of AutoCAD. We will talk about working with blocks, which are available in the AutoCAD library of blocks. The tools with which you can access library blocks are called Design Center and the tool palette, and in the next section, we will discuss these tools.

Using Design Center

Design Center is like the inbuilt library of AutoCAD blocks, which can be used to access frequently used blocks for your project. Design Center has blocks from different disciplines, such as mechanical, architecture, civil, and electrical. In this section, we will learn to use Design Center to access the blocks, and we will also learn how to use it to access blocks from our own drawings.

In the following example, we will explore different parts of the Design Center tool:

1. To open Design Center, select the Design Center tool from the **Palettes** panel of the **View** tab, as shown in *Figure 8.35*, or use its command alias, `ADC`. You can also use the *CTRL + 2* keyboard shortcut to open Design Center:

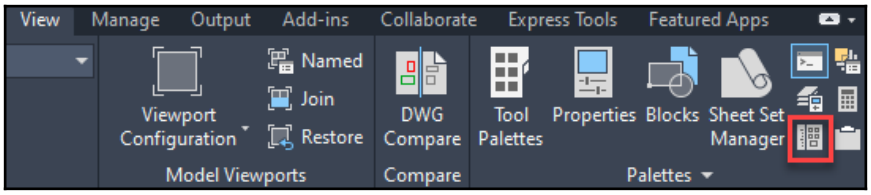


Figure 8.35: Design Center tool on the Palettes panel of the View tab

2. The Design Center palette will open with the last viewed settings. Click the **Home** icon on the heads-up toolbar of Design Center, and you will get the default view of Design Center, which looks like this:

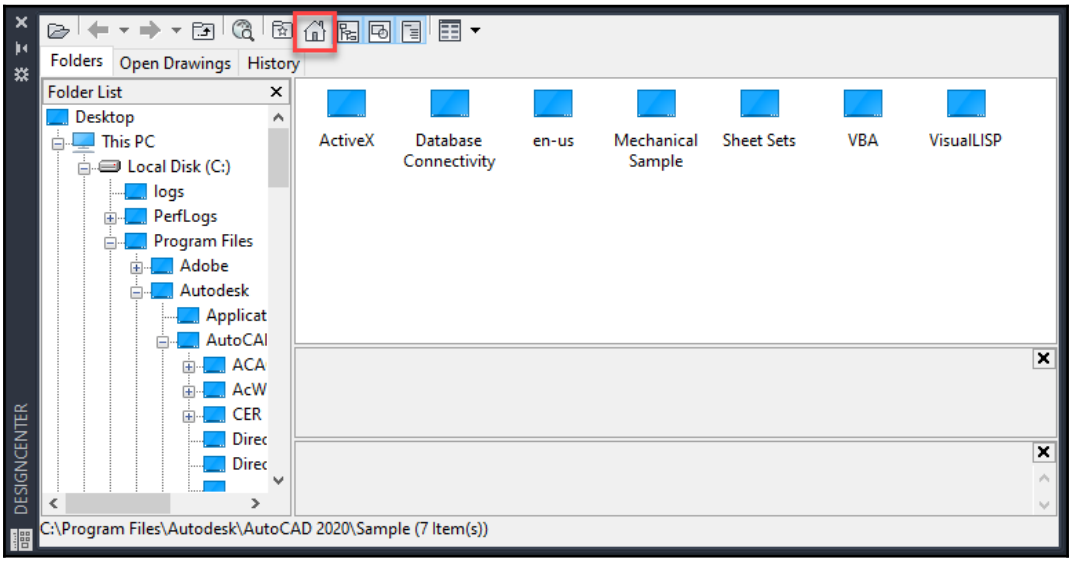


Figure 8.36: Default view of the Design Center palette

3. Design Center is a palette that can be resized just like a layer property manager palette, and you can also dock it to the left or to the right by grabbing the handle, which is to the left, and then moving it to the side where you want to dock it.
4. To access the folder containing our blocks, double-click the **en-us** folder in the top panel, which contains other folders like **ActiveX**, **Database Connectivity**, and **Mechanical Sample**. These folders are available by default and will be available in your Design center palette as well.
5. Double-click the **Design Center** folder and you will see a list of DWG files.
6. These DWG files contain all the blocks of Design Center, and these drawing files are logically created. For example, the `House Designer.dwg` file contains all the blocks you will need when designing the interior of a house, `Kitchens.dwg` contains blocks for kitchens, and the `Basic Electronics.dwg` file contains blocks or basic electronic components.
7. Double-click on the `Home Space Planner.dwg` file and you will see a list of named objects such as **Blocks**, **Dimstyles**, and **Layers**.

8. Double-click on **Blocks**, and now we have all the blocks from the `Home Space Planner.dwg` file, as shown in *Figure 8.37*:

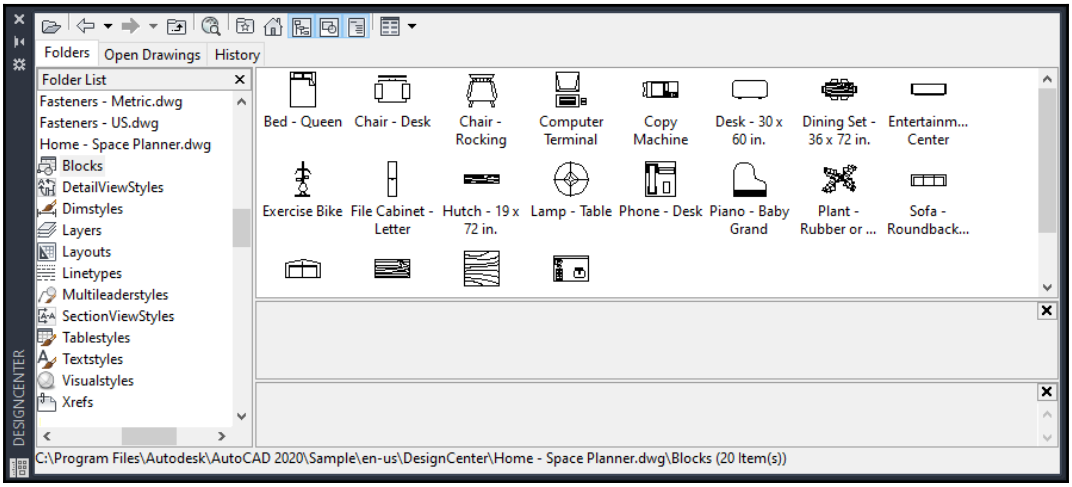


Figure 8.37: Blocks from the Home - Space Planner drawing in the Design Center

The preview contains the blocks of the selected DWG file, and in a similar way, you can select other drawing files from Design Center to see its blocks. When you select any block in Design Center, its preview will appear on the preview panel and a description of the block will show in the description panel. To show or hide the description and **Preview** panels, click the relevant icon, as shown in *Figure 8.38*. You can also click the back button from the heads-up bar of Design Center and it will take you to the previous folder:

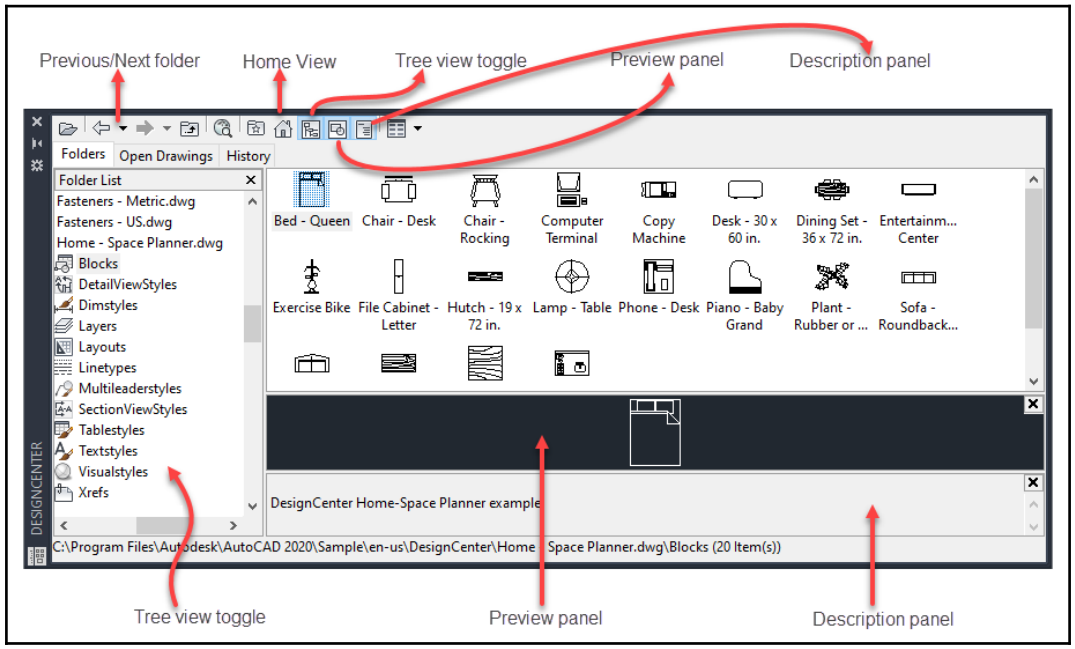


Figure 8.38: Different parts of the Design Center palette

Now, that you know where to find the ready-made blocks, let's learn how to add these blocks to a drawing.

Inserting blocks from Design Center

Once you have a drawing open in the Design Center palette, you are all set to insert a block. Follow these steps to insert the blocks from the Design Center into the drawing:

1. Select the block from the palette, then drag and drop it in the drawing area.
2. The block will be added in its original scale, and if you find it is too large or small for your drawing you can use the `scale` command to make it bigger or smaller.
3. To insert these blocks with more options, double-click or right-click on the block and select **Insert Block** from the **Context** menu.
4. You will now see the **Insert** window, which has options such as **Scale** and **Rotation angle**, and then click **OK** to insert the block.

Once you insert the block from Design Center to your drawing, the block will be added to the local library of blocks as well, and you no longer need to go to the Design Center to insert the block again. You can now go to the Insert block option of the Block panel to insert the same block.

Inserting named objects from other drawings

One of the great advantages of Design Center is that you can insert any named object, such as a block, layer, text style, or dimension style, from one drawing to other using Design Center. To do this, use a drawing that contains a custom dimension style, a custom layer, and a block. In the following example, we will learn to insert named objects from one drawing to another and for this example, I am using a drawing which has all these named objects.

1. Open the Design Center palette using `ADC` command.
2. Click the folder icon on the top of the heads-up bar of Design Center, as shown in *Figure 8.39*, locate the drawing that contains these named objects, select it, and click the **Open** button:

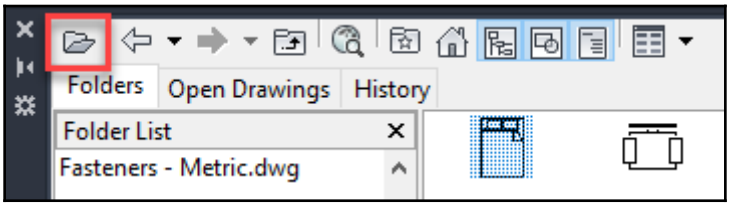


Figure 8.39: Folder icon on the top left of the heads up bar of the Design Center palette

3. The drawing will open in Design Center, and you will see a list of named objects in the panel, as shown in *Figure 8.40*:

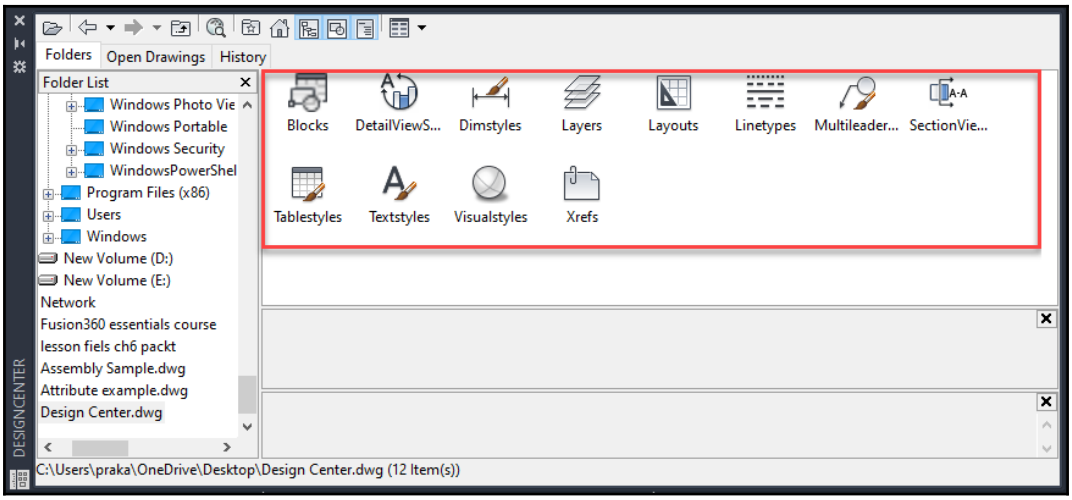


Figure 8.40: List of named objects in the selected drawing

4. Double-click the **Layers** option, and then you will see list of all the layers of the selected drawing.
5. Drag and drop the layer you want to insert from this drawing to the current one and the layer will be added to our drawing.
6. Similarly, click the back button and then open other named objects, such as dimension style, and then in a similar way drag and drop the dimension style you want to insert in the current drawing, and it will be added.
7. The blocks can also be added in a similar way from any selected drawing.

So, now you know that Design Center is not only used for adding library blocks, but it is also useful when you want to extract named objects from other drawings. Just like Design Center, there is also another tool in AutoCAD that has a library of blocks, and this tool is called the tool palette. This tool also has a unique set of features, which we will discuss in the next section.

Using tool palettes

The tool palette is another great source of AutoCAD blocks. Tool palettes not only have simple blocks, but also a special kind of block called dynamic blocks. These dynamic blocks can change shape as per the settings specified in them. Tool palettes also contain blocks containing attributes. In this section, we will learn about using tool palettes to insert the blocks that we need, and we will also learn how to create our own tool palette and then save our custom blocks on these palettes. So, let's begin by exploring the tool palettes.

Inserting blocks from a tool palette

In the following example, we will use a tool palette to insert blocks of different types, and we will also modify those blocks:

1. Open a tool palette using the **Tool Palettes** icon in the **Palettes** panel on the **View** tab, as shown in the following screenshot, or use its command alias, `TP`. You can also press *Ctrl* + *3* on your keyboard:

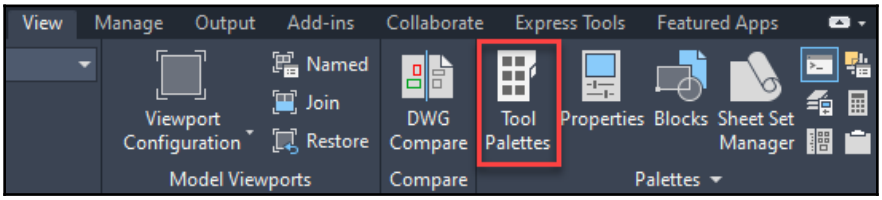


Figure 8.41: Tool palettes on the Palettes panel of the View tab

2. The **Tool palettes** palette will open, and you will see a list of palettes, such as **Architectural**, **Mechanical**, and **Civil**. Select the **Architectural** palette, and you will find a list of blocks on it divided into **Imperial samples** and **Metric samples**, as shown in Figure 8.42:

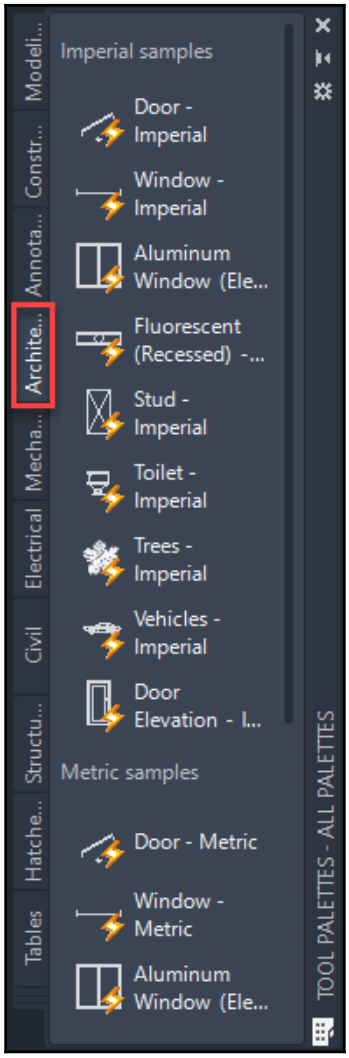


Figure 8.42: Architectural palette with blocks on the Tool palette

3. The imperial category has blocks in Imperial units, and the metric has blocks in Metric units. Depending on the units you are using in your drawing, you can select either of the blocks. All the blocks in this **Architectura**l palette contain a yellow lightning symbol near block thumbnail. These lightning symbols are for the dynamic blocks, and it indicates that the blocks are all dynamic in nature.
4. To insert any block from the **Architectural** palette, simply drag and drop the block into the drawing area. In this example, I will drag and drop the **Door-Imperial** block into the drawing area. If the block is very small or big, then zoom in or out to scale it properly.

When you click on the door block from the drawing area, many grips show up. These grips allow you to change the geometry of the Door block. As an example, when you click the grip marked A as shown in the following image, you get a menu that changes the open-angle of the door, and the grip marked B changes the length of the door block.

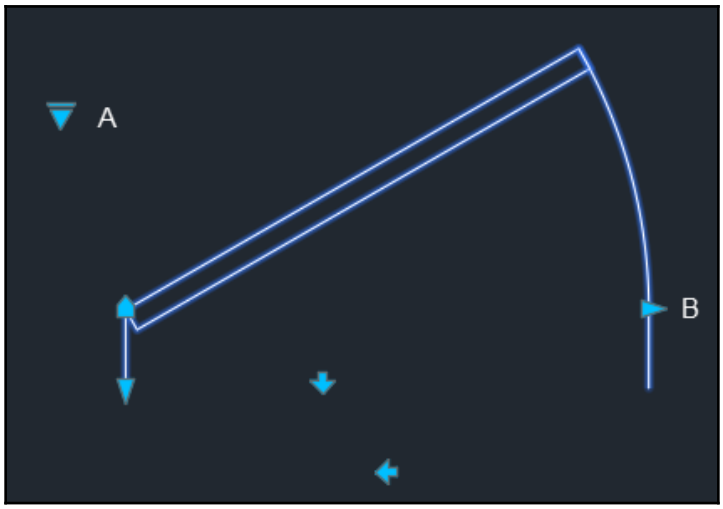


Figure 8.43: Door dynamic block added from the tool palette with multiple grips

These grips are added using the dynamic block properties, which we will discuss in the next chapter.

In a similar way, you can switch to other palettes and add blocks from them. As an example, switch to the Mechanical palette and then select the Shoulder screw – Imperial block at the top of the list and insert it into the drawing. Here, you will also find grips that allow you to change the length or type of this screw.

Other tool palettes can be used to insert other blocks as well. If you, however, want to make your own palette with your own custom blocks, then you can do that as well, and we will discuss this feature in the next section.

Adding custom blocks to tool palettes

In this section, we will learn how to add a custom set of blocks to the tool palette. I will use a drawing containing our custom blocks, as shown in *Figure 8.44*:

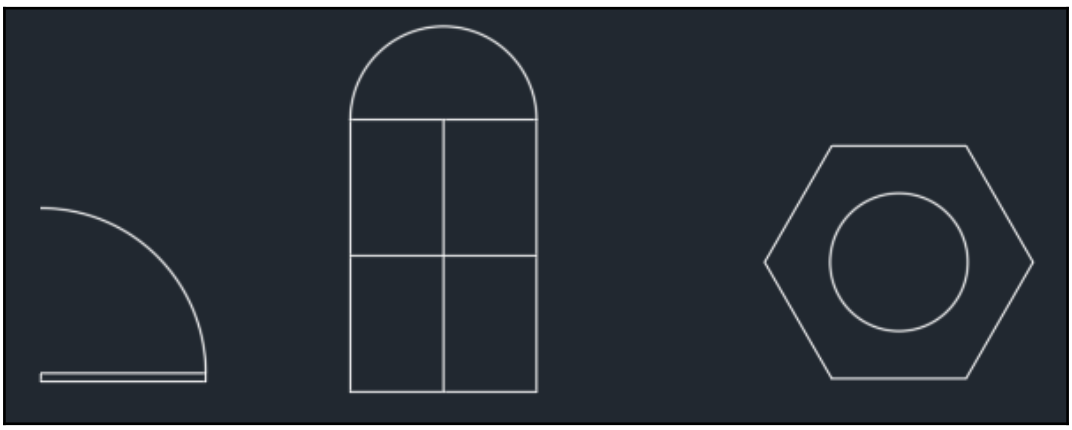


Figure 8.44: Custom blocks made for a tool palette

Here, we have three blocks: a Door, a Window and a Hex Bolt. These blocks are not made as to scale, and they are made only for this example. Also, note that these are not simple drawings; rather, they are blocks and they are named Door, Window, and Hex Bolt respectively.

In the following example, we will learn to create our own tool palette and we will add our custom set of blocks on it:

1. To start making the custom tool palette, open the palette first using the `TP` command, or use the keyboard shortcut *Ctrl + 3*.

2. Right-click on any existing palette name and select **New Palette** from the menu, as shown in *Figure 8.45 A*. Give this palette a name and press the *Enter* key. For this example, name this palette `Sample palette`:

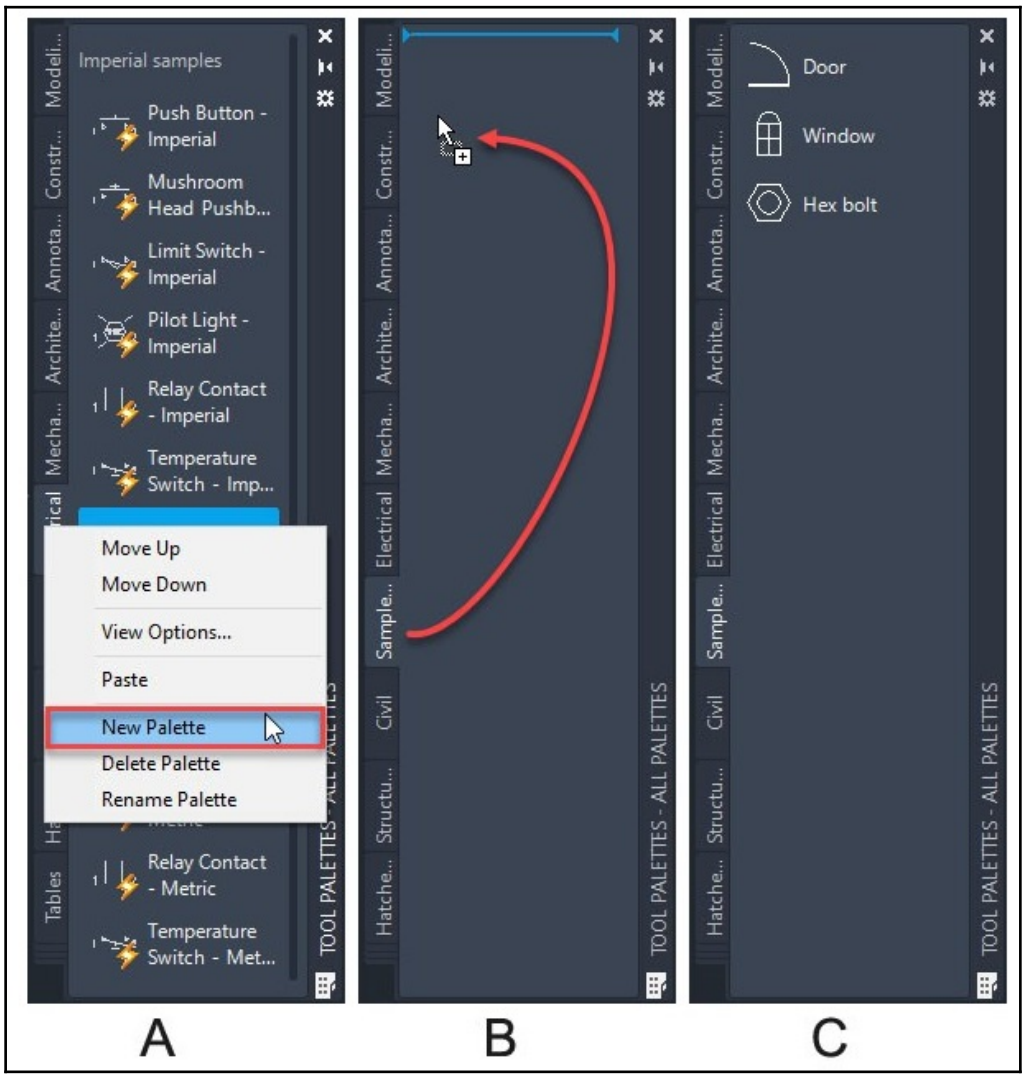


Figure 8.45: Sample palette created and added to the tool palette

3. Now, you will have a completely blank palette. Before adding the blocks, save the drawing somewhere on your local drive or network drive. This step is essential because AutoCAD won't allow you to add blocks to the palette if your drawing is not saved.
4. After saving the drawing, just drag and drop the blocks from the drawing area to the blank palette and the blocks will be added to the palette, as shown in Figure 8.45 B.
5. In a similar way, add the remaining blocks to this custom palette by simply dragging and dropping them into the blank palette. After adding all the blocks to the custom palette, the palette should look like *Figure 8.45 C*.

Now, that we have our custom tool palette with our own set of blocks, we can use these blocks in any drawing, not just in the drawing from which we inserted it into the palette. This palette will also remain available permanently in your software. Even when you close AutoCAD and restart it with a new drawing, you will still find `Sample Palette` when you start the tool palettes command.

The only limitation with the tool palette is that the drawing from which the blocks were added to the palette should be saved in a secure location, and if the file is deleted or renamed or the folder containing the drawing file is renamed, then the tool palette will not be able to get the blocks and will no longer work.

Although Design Center and tool palettes are great sources of ready-made blocks, sometimes you may not find the required type of block in these places, and when that happens you can look elsewhere for the blocks. The first place where you might want to look for blocks is `bimobject.com`, which is a free repository of blocks, and the best thing is that these blocks are uploaded by actual product manufacturers. As the blocks are uploaded by manufacturers, you can be assured that the sizes will be consistent for these blocks. Apart from Bimobject, there are other free and paid AutoCAD blocks websites too from where you can get blocks for your project. If you are downloading blocks from any online source, be sure to check them for any malware before you use them in your project.

So, now that you know about using tool palettes and customizing them to save your own blocks it's time we moved onto a completely different topic, and that's using polylines and splines. Although you already know about polylines and splines, in the next section we will explore these tools in greater detail.

Working with complex polylines and splines

We have learned about polylines and splines in previous chapters, but that was just the basics, and both these tools have a lot more to offer than the basic drawing feature. In this section, we will discuss all the interesting features of polyline and line tools, and we will start with polylines.

Complex polylines

To make a polyline, select the **Polyline** tool from the **Draw** panel of the **Home** tab or use its command alias, `PL`. Start the `polyline` command, and you will notice many options on the command line, as shown in *Figure 8.46*:

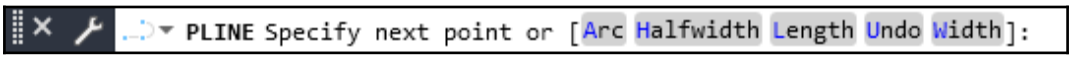


Figure 8.46: Subcommands of the polyline command on the command line

Select the `Halfwidth` option from the command line, or type `H`, and press *Enter*. Now, the command line will prompt you to specify the starting halfwidth value. Type `1` and press *Enter*. The command line will again prompt you to specify the ending halfwidth value. Type `1` again and press *Enter*. Now, click on three points in the drawing area so that you end up with a triangle. Press *Enter* when you are done making the triangle with the polyline, and you will notice that the triangle is made with a thick line with a width of 2 units. In my example, the triangle looked like this:



Figure 8.47- Triangle made with a 2-unit wide polyline

In this case, the total width of the polyline is `2` units, as we specified the halfwidth value as `1` unit, which makes the full width `2` units. This is an example where you can use a polyline to make thick lines without using a solid Hatches pattern to fill the area. Another option on the `polyline` command is `width`, as shown in *Figure 8.46*. This `width` option also lets you make a similar polyline, but the only difference is that you need to specify the total width of the polyline, rather than the halfwidth.

Another interesting example of this polyline feature is explained here:

1. Start the `polyline` command and then select the `Width` option from the command line.
2. Now, specify `0` as the starting and end width of the polyline and make a polyline of length `5` unit towards the right side of the screen.
3. With the `polyline` command active once again, select the `Width` option from the command line, specify the starting width as `1` unit and the ending width as `0` units, and add the line again towards the right side, with a length of `2` units.
4. This time, you will end up with an arrow-shaped polyline, as shown in *Figure 8.48*, where the straight-line part is `5` units long and the thick part is `2` units long. Also, the arrowhead starts with a width of `1` unit and ends at a point as the ending width as `0` unit:



Figure 8.48: Arrow shaped polyline made with varying widths at different points

In this case, we ended up making this interesting shape without the use of multiple commands and as a result, we have a very simple geometry made with the polyline command.

Once you add width to the polyline, it tends to make other polylines with that same width. So, if you want to return to the normal polyline, select the `polyline` command, click the `Width` or `Halfwidth` option, and then change the starting and ending width to `0` units. When you make a polyline this time it will be made with the default thickness.

Editing polylines

Polyline edit, or the `PEDIT` option, allows you to edit existing polylines in your drawing. To edit a polyline, select the `PEDIT` command and then click the polyline you want to edit. Alternatively, you can also double-click the polyline and the polyline edit command will activate. You will see a lot of options on the command line, as shown in *Figure 8.49*:



Figure 8.49: Subcommands of the polyline edit command

Let's look at a few of these options:

- `Close`: This option will close the open polyline into a closed loop.
- `Join`: This option will allow you to join other open geometries, such as a line, with the existing polyline.
- `Width`: Using this option, you can add width to an existing polyline.
- `Fit`: This option turns the polyline into 2D polyline that looks like a Spline Fit but is still a polyline. In this case, you can modify the converted polyline like a spline, but when you hover your cursor over the spline it will still show that it is a polyline.
- `Spline`: This option turns the polyline into a 2D polyline that looks like a Spline CV, that is, a spline with control vertices, but it is still a polyline. In this case, the converted polyline will have control vertices, and you will be able to modify the polyline like splines with control vertices.



Converting a polyline into a true spline

To convert a polyline into true spline double, click on the polyline and then select the `Spline` option from the `PEDIT` options of the command line. The polyline will be converted into a 2D polyline. Type `SPE` for the spline edit command, select the 2D polyline we just created, and press *Enter*. Now, we have a true spline, and if you hover your cursor over the geometry, it will show Spline on the tooltip.

So, these were the polyline tools. Now, let's look at the spline and spline edit tools in the next section.

Complex splines

Just like polylines, we have many options for making splines that allows us to make splines in a controlled way. As you know, there are two types of spline that you can make in AutoCAD, namely Spine Fit and Spline with control vertices. Let's start with the Spline Fit as it has the fewer options of the two.

Spline tolerance in Spline Fit

We will start with the tolerance option of the Spline Fit, which allows us to control the curvature of the spline:

1. Select the Spline Fit tool from the expanded **Draw** panel of the **Home** tab, or use the `SPL` command, then select the `Method` option from the command line, and select `Fit` from the options:

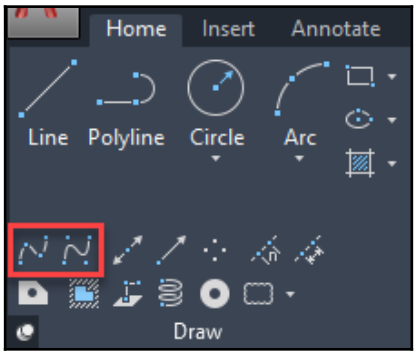


Figure 8.50: The Spline Fit and Spline CV tools on the expanded Draw panel of the Home tab

2. Click on a point to start the spline, and keep clicking on other points so that you end up with spline that looks like *Figure 8.51 A*.
3. Now, start the `Spline Fit` command again and click on a point to start the spline command, and now click on the `tolerance` option of the command line.

4. The default option of tolerance will be `0`. Change it to `1` and press *Enter*. Now, keep clicking on other points so that you end up with spline that looks like *Figure 8.51 B*:

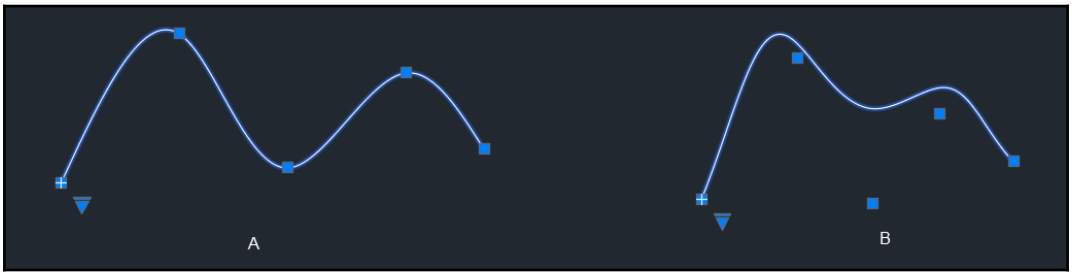


Figure 8.51: Spline Fit made with a tolerance of 0 in spline A and a tolerance of 1 in spline B

Now, you may notice that spline **A** has knots or grips exactly on the spline because the tolerance was `0` in case **A**. With spline B, the knots are away from the spline, except the start and end knots, and this is happening because of the tolerance `1`.

So, the greater the tolerance, the farther the spline will be from the knot. The knot is the point you use to create the spline.

Start and end tangency in Spline Fit

Just like spline tolerance, you can also control the curvature of a spline tightly using the start and end tangency options discussed in the following example:

1. Start the `Spline Fit` command again and click on a point to start this spline. Now, select the `start Tangency` option from the command line:

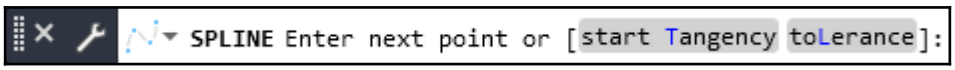


Figure 8.52: Start Tangency subcommand in the SPLINE command

2. Click on another point in the drawing area to specify the direction of tangency.
3. Keep clicking on other points in the drawing area to make the spline, you will notice that the starting segment of the spline will become tangent to the direction we specified in the previous step. The tangent direction is shown by the green dash line in the following diagram.

4. In a similar way, you can specify tangent direction at the end point of the spline. Select `end Tangency` from the command line and then click at a point to specify the tangent direction. The spline command will end automatically after you specify the end direction.

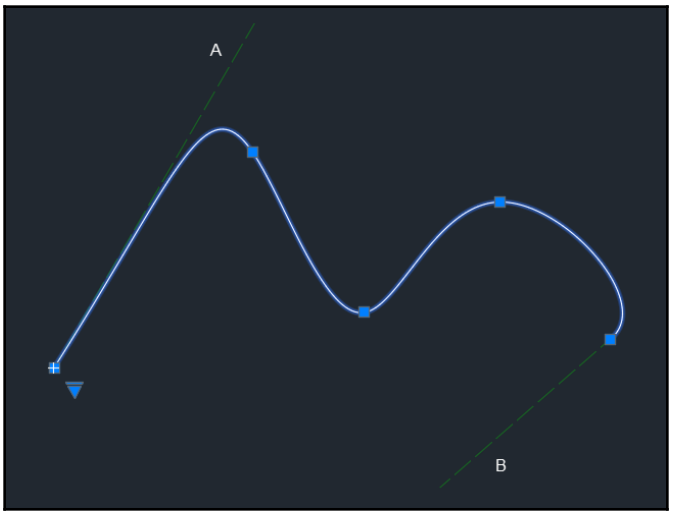


Figure 8.53: Starting and end tangency directions defined by the green dashed lines on the spline

In our example, we started from the first point of spline and then clicked along the green dashed line **A** for the starting direction of the spline tangent. For the end tangent direction, we clicked along line **B** and the end segment will become tangent to the green dashed line **B**. The dashed line **A** and **B** are not part of the spline and they are added only for representation purposes.

So, these were some options related to Spline Fit. Now, let's now discuss the options related to Spline CV.

Making Spline CV

Spline CV, or spline with control vertices, is another way of making splines with more control over its geometry. In the following example, we will make a simple Spline with CV and explore its options:

1. To make a Spline CV, select the **Spline CV** tool from the expanded **Draw** panel of the **Home** tab or start the `SPL` command and then select the Method option from the command line and select `CV`.

2. Now, click on a point to start the spline, and then keep clicking on other points to complete the spline.
3. You will notice that this spline will be made with respect to the control vertices, you will have a control polygon, and the knot of this spline will also show up on the vertices of the control polygon, as shown in *Figure 8.54*:

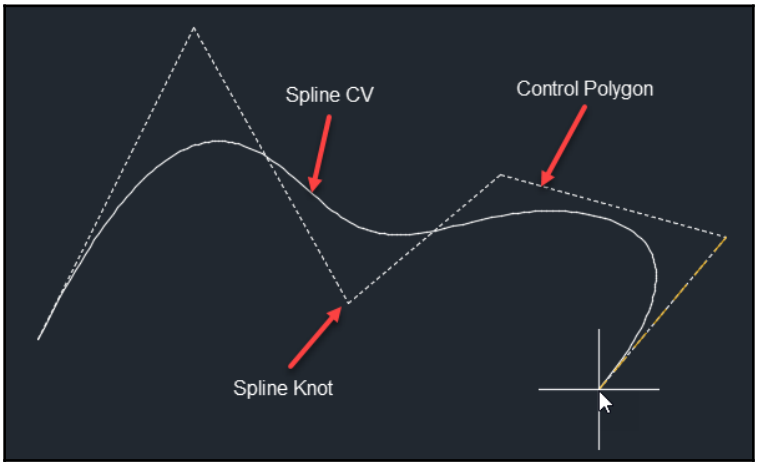


Figure 8.54: Spline CV made with respect to the control vertices

4. Press *Enter* when you are done making the spline, and when you select the spline you will see the control polygon and the knots on it. You can further modify the spline using those knots.

Now, we have our Spline CV, let's talk about modifying the spline using knots and the control polygon.

Modifying a Spline CV

Spline CVs have more modification options, and they also give you greater control over the spline curvature. We will modify the spline we made in the previous example in the following steps:

1. Select the spline so that the fit points become visible.
2. Hover your cursor over the knot, and you will see a menu with options, as shown in *Figure 8.55*:

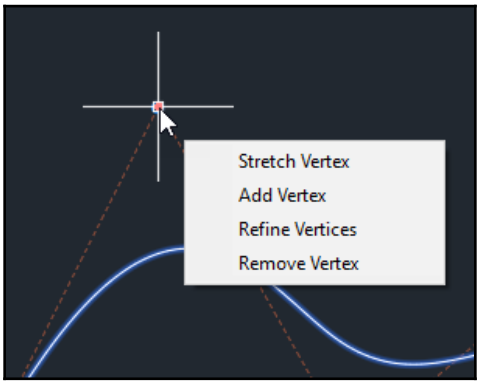


Figure 8.55: The modify spline menu when the cursor hovers over a knot

3. If you select the **Add Vertex** option, it will add additional knots in the spline and make it more refined. When you select **Add Vertex**, you get the option to place the new vertex in your spline.
4. You can also select the **Refine Vertices** option, which again adds a new vertex, but it will do it automatically without requiring you to add a point to add the vertex.
5. The **Remove Vertex** option will remove the selected vertex, and the adjoining vertices will then join to complete the spline.

Just like polylines, you can also modify splines and convert them into different forms. In the following example, we will learn how to modify a spline using the `SPLINEEDIT` command.

Modifying a spline using SPLINEEDIT

You can modify a spline using the `SPLINEEDIT` command, and you can also convert a spline into a polyline. In the following examples, we will learn how to do make modifications in a spline using the `SPLINEEDIT` command:

1. To start the `SPLINEEDIT` command, double-click on the spline or type `SPE` and press *Enter;* then click on the spline you want to modify.
2. Now, you will see lots of options on the command line. When you select the **Close** option from the command line, the spline will close from the open ends and will make a smooth closed spline. This option is especially good when you want to make a closed spline without any sharp points.

3. The **Convert to Polyline** option, as the name suggests, will convert the spline into a polyline. To do this, select the **Convert to Polyline** option and then specify a tolerance value. The higher the tolerance, the denser the polyline will be, and the more polyline segments will be used to convert the spline into a polyline. Also, a higher tolerance value will ensure that your polyline resembles the original spline more closely, but you will end up with a polyline containing lots of line segments.
4. A comparison of the polyline obtained from the same spline with a tolerance of 1 and a tolerance of 20 is shown in *Figure 8.56*:

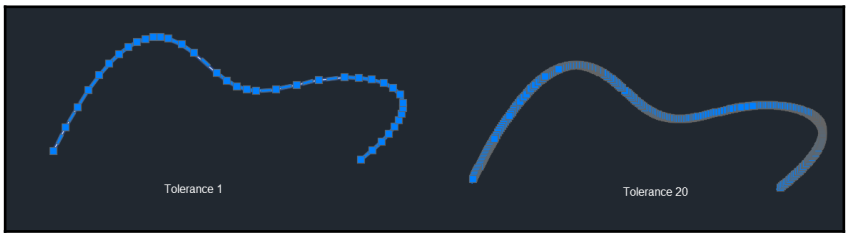


Figure 8.56: A spline modified using the SPLINEEDIT option using different tolerance values

So, this topic concludes our chapter. We have seen a lot of advanced tools and customization features in this chapter, and finally, let's have a look at the things we have covered so far in this chapter and the tools we will cover in the next chapter.

Summary

This chapter was mostly about customizations. We learned how to customize the user interface of AutoCAD and how to create custom objects. The customization tools that we discussed help personalize the user interface and make us more efficient in our drawing workflow. The custom linetype and Hatches are other great features available in AutoCAD, and they are especially useful when you need a project-specific linetype or Hatches pattern. Design Center, tool palettes, and external sources of blocks reduce our dependency on making blocks every time we need them and make the process of making drawing much easier and faster. Finally, we discussed the polyline and spline tools in detail.

In the next chapter, we will discuss external references and dynamic blocks, and both of them are interesting topics. We have already seen dynamic blocks in action in the tool palette section, and in the next chapter we will learn how to make our very own dynamic block. So, I will see you in the next chapter.

9
External References and Dynamic Blocks

In a real-world situation, you will often find many people working on a single project. This will be a big issue if everyone is required to work on a single AutoCAD file because once a file is opened by someone, if you open it again, the file will be read-only and won't allow modifications. Fortunately, we have a tool in AutoCAD that allows many people to work on separate files of a project, and then these files can be grouped together in the main drawing using the External References tool, which we will discuss in this chapter. We will also discuss the dynamic block tool, which lets us make blocks with dynamic features, and finally, we will learn about cleaning our drawing and fixing drawing-related issues.

These are the topics that we will discuss in this chapter:

- Working with **External Referencess** (XRefs)
- Understanding dynamic blocks
- Cleaning and fixing drawing issues

Let's get started with an overview of External Referencess, or XRefs.

Working with External Referencess

When working on a large project, it becomes almost impossible to use only a single drawing for everything. As an example, if you are working on the plans for a large building, you can assign different team members working on the project to different parts of the drawing. A team member can work on the main plan, someone else can work on exteriors, another can work on electrical plans, and another person can work on heat and ventilation drawings. All these drawings can then be clubbed together into a single file using XRefs.

Using this method, you can have multiple people work on a single project seamlessly. Another advantage of using XRefs is that the people working on a project can focus just on their drawing and the drawing progress can be shared in real time.

In the following examples, we will learn everything about making and managing XRefs.

Inserting XRefs

For this example, I am using some drawings of a simple office plan, and these drawings are the work area, the washroom, the meeting room, and the lift area. You can use your own set of drawings for this section as well.

In the following example, we will learn how to add our to a drawing as an XRef:

1. Open a blank drawing, and then type `UN` and press *Enter*. Set the unit of this drawing to inches and the length type to decimal in the **Drawing Units** window.
2. Click the **Attach** button in the **Reference** panel of the **Insert** tab, as shown in Figure 9.1:

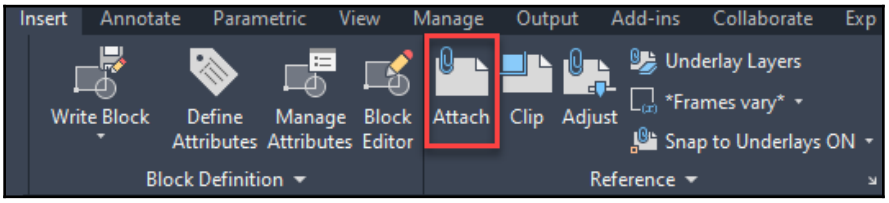


Figure 9.1: The Attach button in the Reference panel of the Insert tab

3. The **Select Reference File** window will open. Change the file type to `DWG` in this window, as shown in Figure 9.2, then locate the drawing you want to insert and click **Open**. In this case, I am inserting the `Work area` drawing:

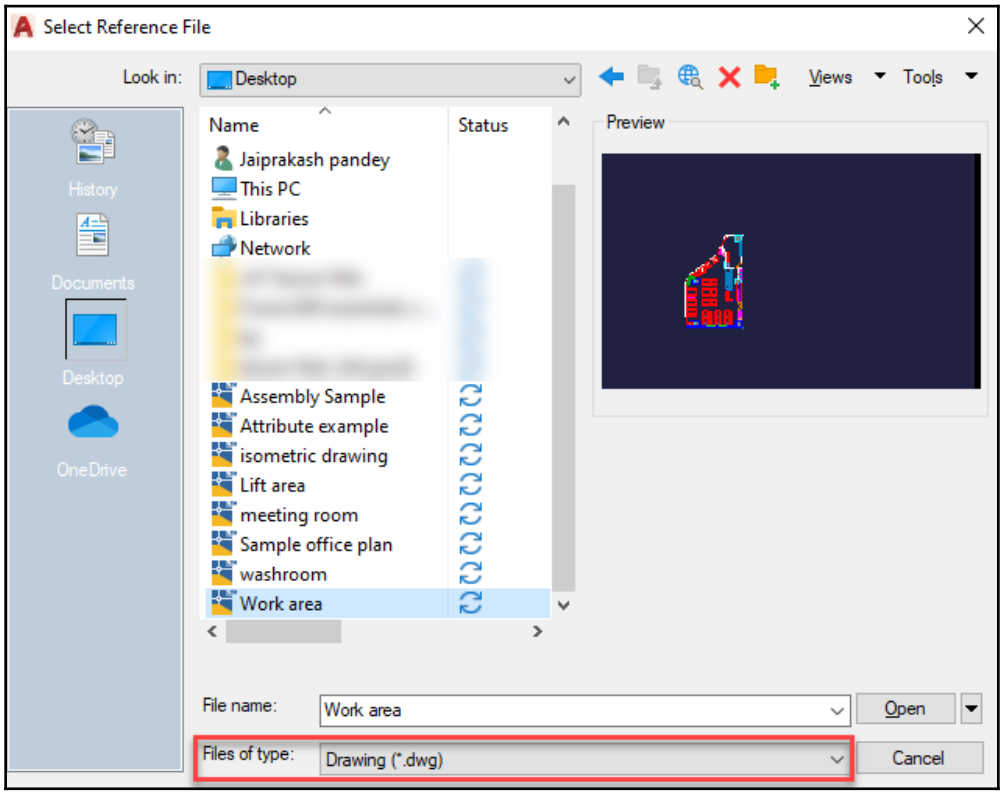


Figure 9.2: Change the file type to DWG and then select the file to insert as XRef

4. Now, the **Attach External References** window will open, with lots of options. We will explore all the options in this panel in a moment, but for now, select **Relative path** as the **Path Type** and **Attachment** in the **Reference Type** panel. Leave the other options at their defaults.
5. Click **OK**, and then click on any point in the drawing area. We now have our `Work area` drawing added to the current drawing as an XRef.
6. If you don't see the drawing, or if it's too large or small, double-click your middle mouse wheel to fit the XRef in the drawing area.

5. The XRef added to the current drawing will appear with 50% transparency, as shown in Figure 9.3. This is one of the characteristics that will help you identify it in a drawing:

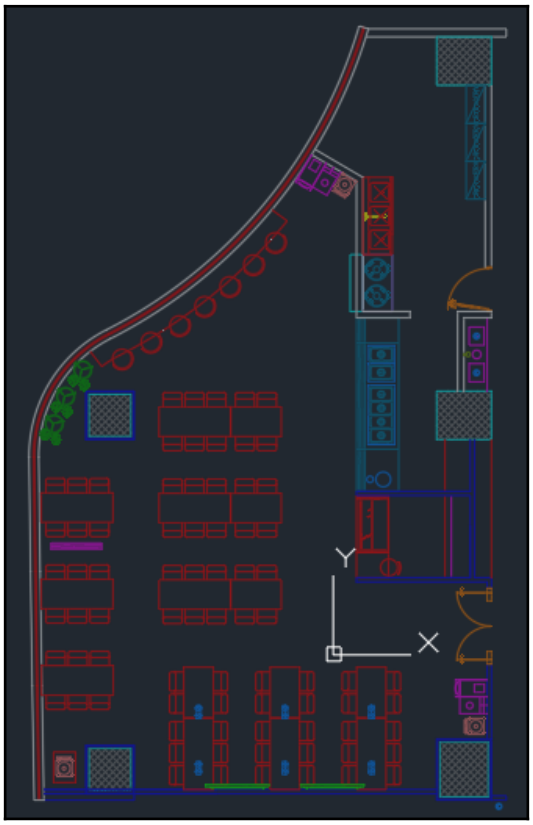


Figure 9.3: The DWG file added to the current drawing as an XRef with 50% transparency

The `Work area` XRef added here is like a link with a preview. The `Work area` drawing looks like it's added to the current drawing, but it's not entirely like that. The drawing is added here only as a linked object. To understand this properly, we will modify the original `Work area` drawing and see its effects on the current drawing:

1. Save the drawing in which the `Work area` XRef has been added and give it a name. I will name it `main XRef` for this example.
2. Open the `Work area` drawing, delete the pillar next to the plants, and then save your drawing. If you are using your own drawing, then make any change in it and save it. After saving, close the drawing.

3. Now, go the `main XRef` drawing, and you will notice an exclamation mark right next to the manage XRef icon of the status bar, as shown in Figure 9.4:

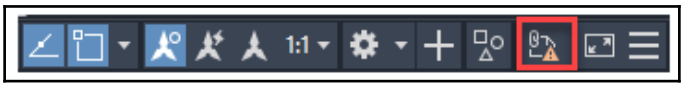


Figure 9.4: Exclamation mark on the manage XRef status bar icon

4. Right-click on the icon and select **Reload DWG XRefs** from the context menu, and you will notice that the attached XRef will update in the current drawing as well, and we will no longer have a pillar in our attached XRef.

So, this is how XRefs work. Once attached, they become part of the drawing, but they can still be modified from the original drawings and the changes will appear in the drawing where they are attached. Before we learn more about XRefs, let's talk about the **Attach External References** window and its options in the next section.

Attach External References window options

Click the Attach icon again in the **Reference** panel of the **Insert** tab and then locate the **Meeting room** drawing and click **Open**. You will once again get the **Attach External References** window with lots of options, as shown in Figure 9.5:

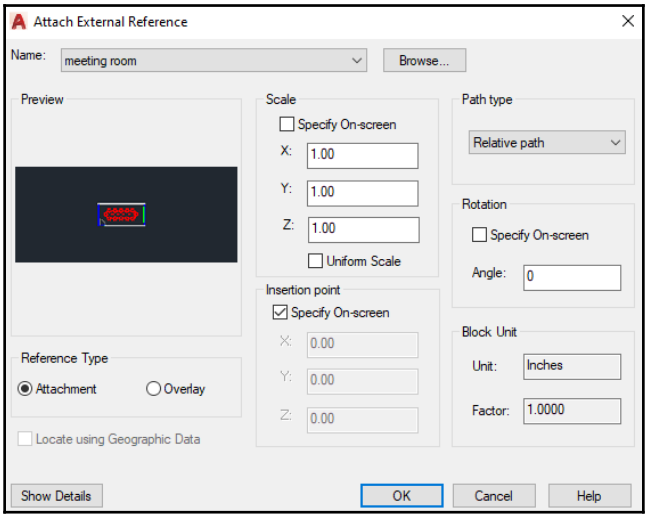


Figure 9.5: The Attach External References window

At the top of this window, we have the **Name** field, which shows the name of the file we are attaching. You can click the **Browse** button to change the file. On the left, just underneath the **Name** field, you will see a **Preview** panel that shows the preview of the drawing we are attaching.

Scale

The scale panel has options for changing the scale along the *X*, *Y*, and Z axes separately. You can use this scale feature to change the scale of an XRef along different axes separately. If you want to change the scale of the overall XRef uniformly, then click the checkbox underneath the axes fields and then specify the uniform scale for the entire XRef:

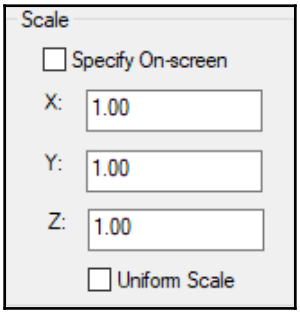


Figure 9.6: The scale panel on the Attach External References window

If you don't want to specify the scale here and want to specify the value in the drawing, then click the **Specify On-screen** checkbox and the prompt for the XRef scale will show up in the drawing area while attaching the XRef.

Path type

There are three path options you will get in the **Path type** panel of the **Attach External References** window. These **Path type** are **No path**, **Full path**, and **Relative path**, as shown in Figure 9.7:

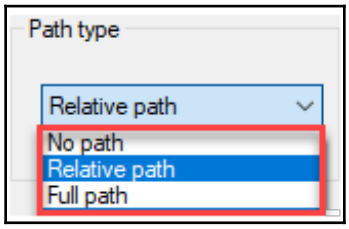


Figure 9.7: Path types in the Attach External References window

Here is a description of each path type:

- **No path**:

  When you select the **No path** option, AutoCAD attaches the XRef without any path information. The XRef should be in the folder of the host drawing, otherwise AutoCAD won't be able to resolve the XRef.

- **Full path**:

  When you select the **Full path** option, AutoCAD saves the complete path of the XRef file. This option offers very little flexibility when it comes to XRef management because if the location of the XRef changes, or if you rename the directory of the XRef or even the drive containing the XRef, then AutoCAD won't be able to resolve the XRef.

- **Relative path**:

  This is the most flexible path type option available in the Attach External References window. This option saves the location of the XRef with respect to the host drawing location and, in this case, even when the location of the XRef changes, or the drive letter of the directory containing XRef changes, AutoCAD will still be able to resolve the XRef as long as the location of the XRef with respect to the host file is unchanged.

In all these path type options, if the XRef is in the same directory as the host file or in the support folder, then AutoCAD will be able to resolve the XRef. The location of the support folder is usually `C:\Program Files\Autodesk\AutoCAD 2021\support`.

Rotation

This option lets you specify a rotation angle for the XRef you are attaching, as shown in the following screenshot:

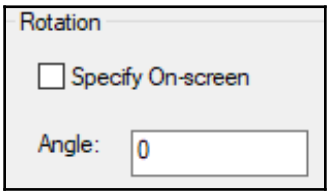


Figure 9.8: Rotation panel on the Attach External References window

You can specify a rotation angle for your XRef or check the box that says **Specify On-screen**, as shown in Figure 9.8, to specify the rotation angle in the drawing area while inserting the XRef.

Insertion point

The **Specify On-screen** option in the **Insertion point** panel is checked by default, and this ensures that you are prompted for a point to attach the XRef. You can uncheck the Specify On-screen option and then add a specific *x*, *y*, and *z* coordinate value for your XRef, and the XRef will be attached at this specific location. Generally, we would want to visually locate the XRef in the drawing area, so it's better to keep the **Specify On-screen** checkbox selected in this panel.

Reference Type

You can attach your External References file as an attachment or as an overlay, and the option for selecting the type of your XRef is in the **Reference Type** panel, as shown in Figure 9.9:

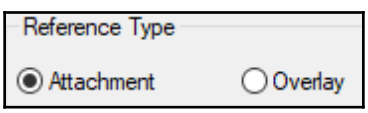


Figure 9.9: External References type panel

- **Attachment**:

  When you select an attachment for an XRef, the file gets nested the next time you attach the host file. To explain this, let's assume you are attaching file B as an XRef in file A. Later, when you decide to attach file A to another drawing as an XRef, then file A, as well as its attached XRef B, will be added to the host file because you selected **Attachment** as the reference type for file B.

- **Overlay**:

  When you select **Overlay** in the reference type, the file is attached as an overlay, and when you attach the host file in another drawing, its nested references are not added. To explain this clearly, let's use the analogy we used earlier. When you attach file B as an XRef and use **Overlay** for the reference type in file A and later attach file A in another drawing as an XRef, then only the contents of file A will be added, and the overlaid XRef will not be inserted in the host drawing.

  So, in a nutshell, the **Attachment** option allows you to add an XRef as well as its nested reference files, whereas **Overlay** only allows you to add the top-level XRef and ignores all the nested XRefs in the drawing. When you want to add drawings as XRefs, then you can select **Overlay** as your default option if you want to avoid the issue of circular or nested references.

Now that you know what all the options in the **Attach External References** window mean, it's time to explore other External References tools.

External References tab

In our `main XRef` drawing, we currently have only one XRef, and that is `Work area`. Let's add other drawings as well to the `main XRef` drawing as XRefs. Attach the Meeting room, Washroom, and Lift area drawing files as XRefs. If you don't have these files, then you can use your own drawings instead. To attach these files, set the path type as **Relative Path**, and set the **Reference Type** as **Overlay**. After adding the XRefs to my drawing, it looked like this:

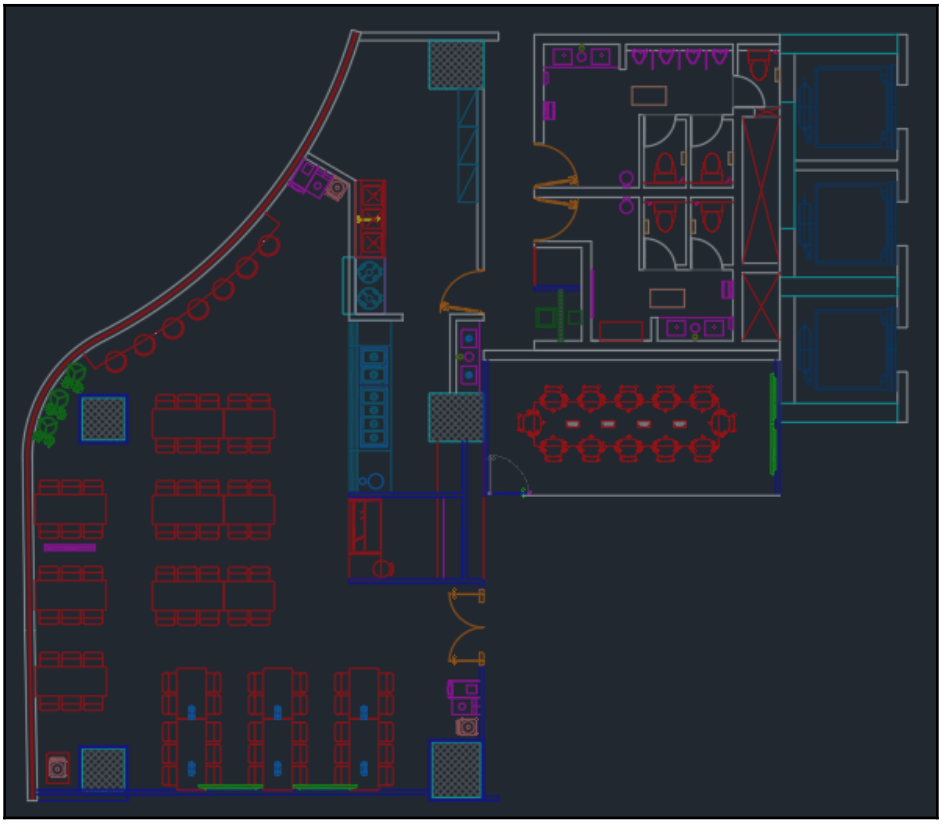


Figure 9.10: Drawing with all XRefs attached

Select any External References from the drawing area and you will get an **External References** tab with **Edit**, **Clipping**, and **Options** panels, as shown in Figure 9.11:

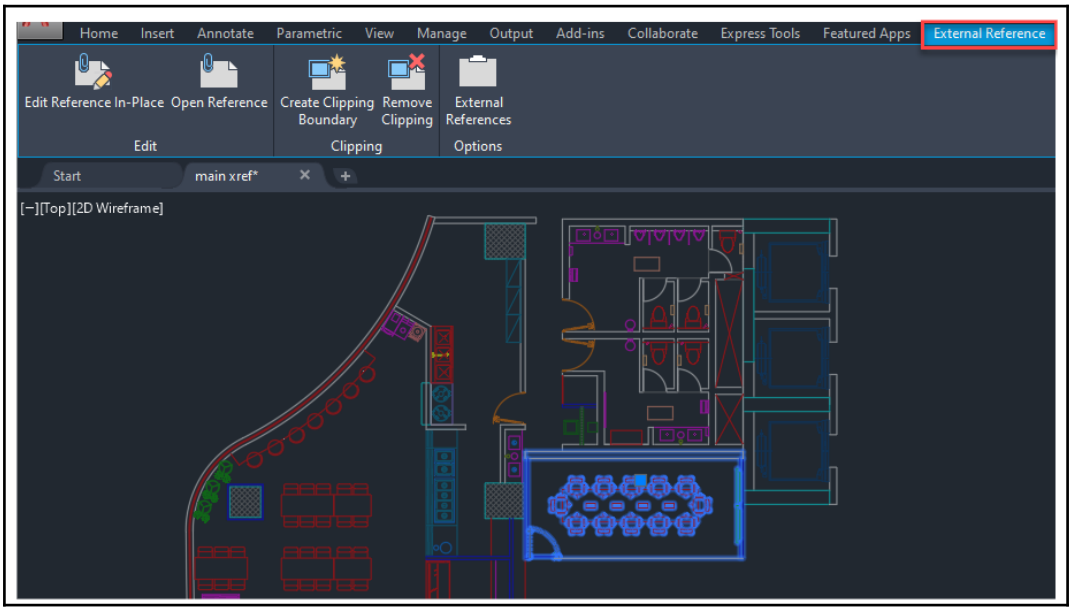


Figure 9.11: External References tab when an XRef is selected

In the next section, we will start by discussing the tools that help modify XRefs.

Modifying XRefs

To modify an XRef, you can select the **Edit Reference In-Place** or **Open Reference** options from the **Edit** panel of the **External References** tab.

Click the **Edit Reference In-Place** option, click **OK** in the next window, and the XRef will be highlighted in the current drawing. You can now go to the **Home** tab or any other tab to modify the External References file. You can add objects to, or remove objects from, the XRef file, and when you are done, click the **Save Changes** option in the **Edit Reference** panel, as shown in Figure 9.12:

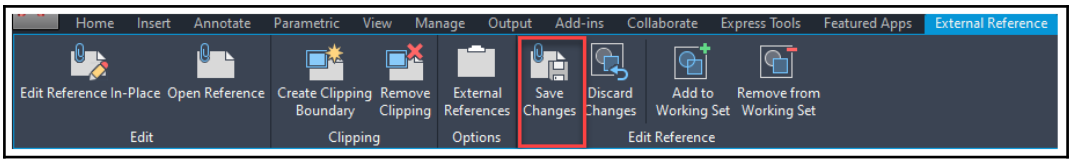


Figure 9.12: Saving an External References after making changes using the Save Changes option on the Edit Reference panel

Using this process, you can modify the External References file without even opening the XRef file. In this case, the file opened in the same window as your host drawing, and once you are done making changes and have saved the drawing, the changes are saved in the original XRef file as well.

If you, however, want to open the file, then select the XRef from the drawing area and then select the **Open Reference** option from the **Edit** panel and the XRef will open in a new tab. You can make changes to this file, save the XRef drawing, and return to the host drawing and update it to see the changes.

When you modify an XRef in the 2021 version of AutoCAD, you will get a new pop-up bubble that looks like Figure 9.13. Here, not only will you be notified that the XRef has been modified, but you will also get a checkbox that will let you compare the modified and the original versions of the XRef. Keep the **Compare the changes** checkbox selected and click on the Reload link of the bubble. The updated XRef will appear:

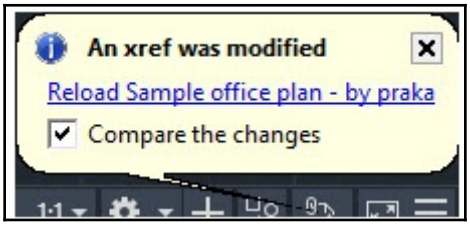


Figure 9.13: This bubble appears when an XRef is modified

Not only that, you will also see a drawing with a comparison between the modified and original version of the drawing. Once you are done checking the difference, click the green checkbox on the **XRef Compare** tab, as shown in Figure 9.14:

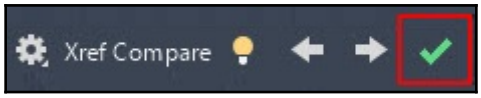


Figure 9.14: XRef Compare palette

As mentioned earlier, this is a new feature in AutoCAD 2021. You won't find the XRef Compare feature in older versions of AutoCAD.

Clipping XRefs

Sometimes, we do not need the entire XRef. We might need only a part of it. But when you insert a drawing as an XRef, all its objects are inserted in the drawing. Once it is inserted in the drawing, you can clip it to remove the parts that you don't need. Clipping an XRef does not remove the part from the host drawing, it only hides it from the drawing area.

In the following example, I will clip the `Work area` section in our example drawing containing multiple XRefs so that only the desks and chairs are visible in the drawing and the rest of the objects from the `Work area` drawing are hidden:

1. Click on the `Work area` drawing in the AutoCAD workspace.
2. Click on the **Create Clipping Boundary** option in the **Clipping** panel of the **External References** tab.
3. Lots of options will now show up on the command line. Select `Polygonal`, as shown in Figure 9.15:



Figure 9.15: The Polygonal option of the Create Clipping Boundary option

4. Click on different points so that you end up with a polygon that includes the desks and chairs, as shown in Figure 9.16. Press *Enter* once you are done making your clipping polygon:

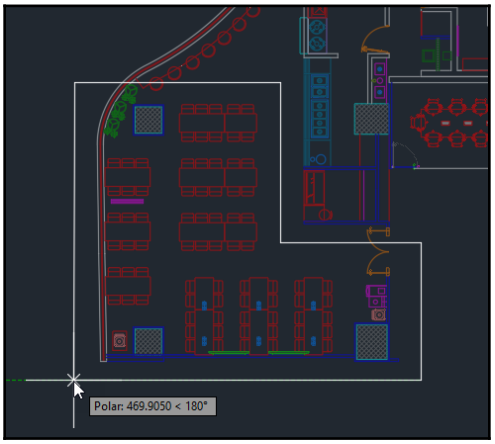


Figure 9.16: Clipping boundary made in the drawing containing an XRef

5. Now, the `Work area` XRef will be clipped, and either the part inside the clipping boundary or outside the boundary will appear in the drawing area. In my case, the part inside the clipping polygon showed up.
6. To reverse the direction of the clipping, click the arrow grip, as shown in Figure 9.17:

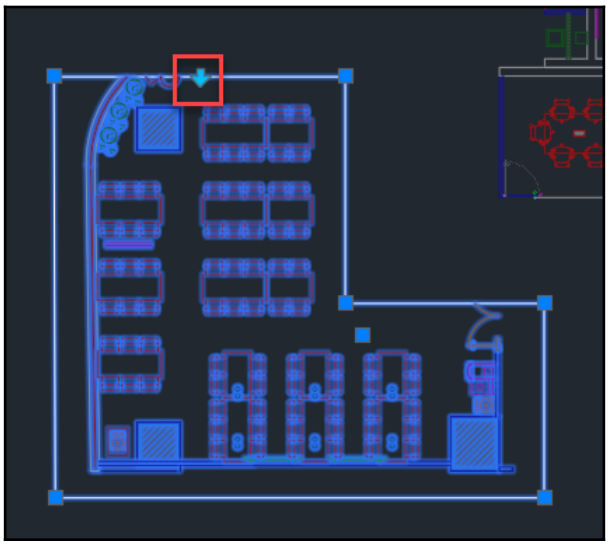


Figure 9.17: Arrow grip for reversing the clipping direction of the XRef

7. Press the *Esc* key once you are done clipping the XRef. As mentioned earlier, this only hides the remaining part of the XRef. You can remove the clipping to reveal the hidden part of the XRef.
8. To remove the clipping, select the clipped XRef or click on the clipping polygon boundary and click on the **Remove Clipping** option in the **Clipping** panel of the **External References** tab.
9. The clipping will be removed from the selected XRef, and the entire XRef will now be displayed in the drawing area.

That was all about clipping XRefs. In the next section, we will talk about the **External Referencess** palette, which is like the control panel of XRefs, and you can use this palette to modify many settings related to inserted XRefs.

External Referencess palette

The XRef palette allows you to insert more XRefs, change them into blocks, change their path type, and a lot more. You can open the XRef palette using the XRef command, or select an XRef from the drawing area and then click the **External Referencess** tool from the **Options** panel of the **External References** tab. The **External Referencess** palette looks like this:

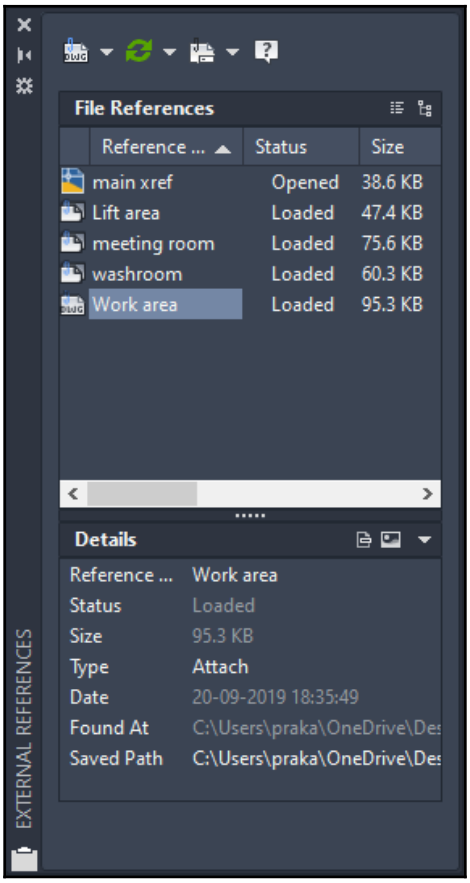


Figure 9.18: External Referencess palette

In the example of the **External Referencess** palette shown in Figure 9.18, we have the host file called `main XRef` and four External References files attached to it, all listed on the **File References** panel. Select any External References from the **File References** panel and the details of that selected XRef will appear on the **Details** panel.

In Figure 9.16, the `Work area` XRef file is selected in the **File References** panel. Its details, such as file size, reference type, and path, are shown along with other properties in the **Details** panel. Also, when you select an XRef from the **External Referencess** palette, it will highlight the XRef in the drawing area as well.

To attach the DWG and other files as XRefs, click the down arrow next to the DWG icon, as shown in Figure 9.19, and you will find a list of formats compatible with XRefs:

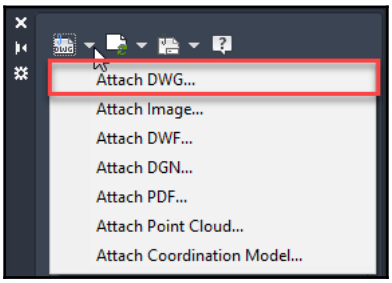


Figure 9.19: List of formats that can be attached as XRefs

From this list, you can select DWG, Image, DWF, PDF, and other formats and attach them in your drawing as XRefs. You can use this option to attach new XRefs as well as copies of existing XRefs in the drawing area.

When you select any XRef from the **External Referencess** palette and right-click, you will get a bunch of options, as shown in Figure 9.20:

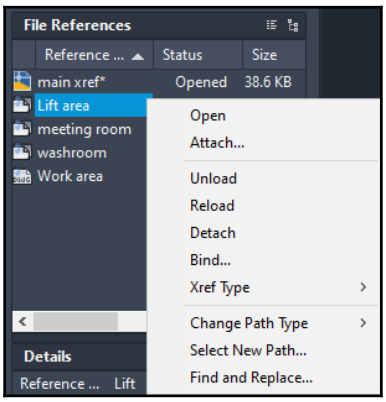


Figure 9.20: XRef options on the right-click menu of External Referencess

Here is a brief description of these options.

Open

This option will open the XRef in a separate tab. You can modify the XRef and save it, and the XRef will be modified in the host drawing as well.

Attach

The second option in the right context menu of Figure 9.20 is Attach, which will once again let you attach the selected XRef in the drawing.

This will essentially create a duplicate copy of XRef in the drawing, but a new XRef will not be added to the **File References** panel of the **External Referencess** palette as there can only be a single instance of an XRef in the XRef palette.

Unload and Reload

These options will hide the XRef from your drawing area. When an XRef is unloaded, a red arrow will appear next to the XRef in the XRef palette. To load the XRef again and make it visible, select it from the XRef palette and then right-click and select the **Reload** option.

Detach

When you select **Detach** from the XRef palette, the External References is deleted from the drawing area as well as from the **External Referencess** palette. This change is permanent, and to bring back the XRef you need to once again attach it in the drawing.

Bind

This option lets you convert the XRef into a block. When you select the **Bind** option, you will get a palette with two more options, **Bind** and **Insert**, as shown in Figure 9.21. Selecting either of these options will convert the selected XRef into a block in the host drawing. The name of the block will be the name of the XRef.

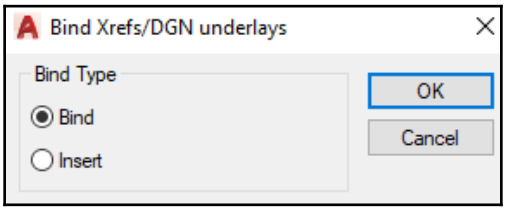


Figure 9.21: The Bind and Insert options of the XRef

The **Bind** option here converts the XRef into a block, and named objects, such as the layers, are also added separately for the blocks with a `name of block $a$` prefix, where `a` could be any number string starting with `0`.

The **Insert** option also converts the XRef into a block, but in this case, the named objects are merged into the host drawing and no prefix is added to the name of named objects.

XRef type

This option lets you change the type of XRef from attach to overlay, and vice versa.

Changing the path type

This option lets you change the path type of the XRef. When you select this option, the current path type will be grayed out and the remaining two path types will remain selectable, as shown in Figure 9.22:

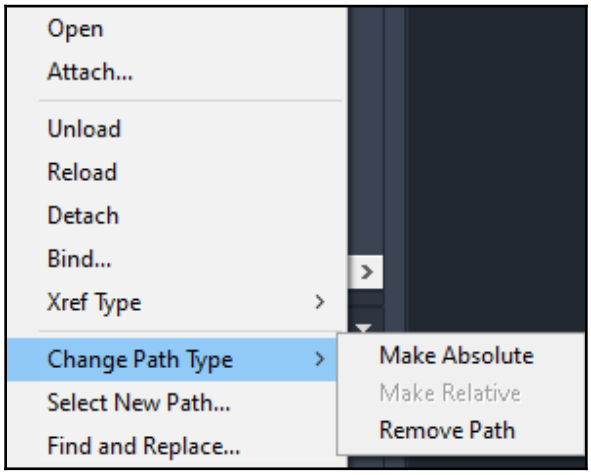


Figure 9.22: Changing the XRef path type options

You can select the path type from the available options of the **Change Path Type** menu.

Selecting a new path

If, for any reason, the location of the attached XRef is changed, when you open the host file, an error message will prompt you to either locate the file and set a new path or ignore the XRef, as shown in Figure 9.23:

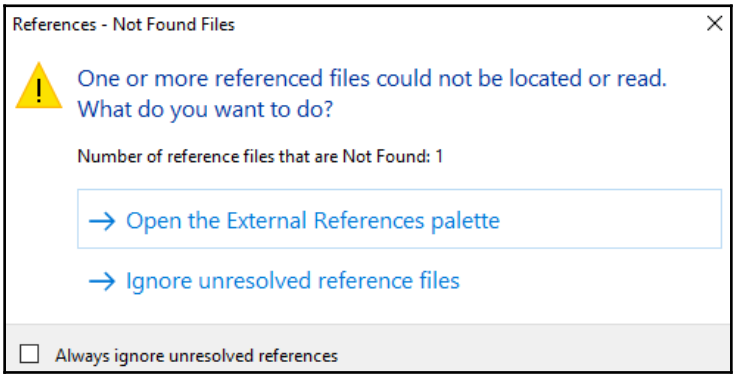


Figure 9.23: References not found pop up when AutoCAD is unable to locate the XRef(s)

When you select the **Ignore unresolved reference files** option, the host file will open, but the unresolved XRef will not show up in the drawing area. You will also get an exclamation mark next to the name of the XRef in the **File References** panel of the XRef palette, as shown in Figure 9.24:

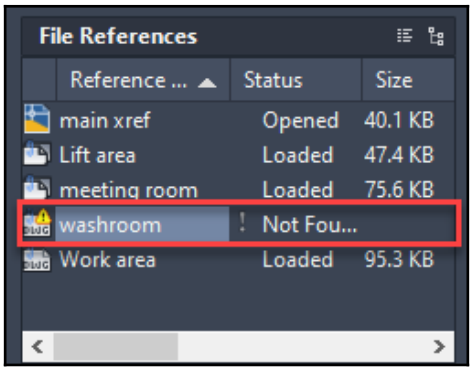


Figure 9.24: Exclamation mark next to the unresolved XRef in the External Referencess palette

If you want to resolve the XRef and specify its new location, then right-click on the unresolved XRef in the XRef palette, select the **Select New Path** option, locate the XRef in a new path, and click **Open**. The XRef will be loaded and the new path will be refreshed in the XRef palette as well. Once the XRef is loaded in the drawing, the exclamation mark will also disappear from the XRef palette.

Once you are done making the changes in the XRef palette, click the X mark in the top left of the XRef palette to close it. Transferring XRefs directly can be challenging because if you leave any XRef file, then the host file won't be able to resolve those missing files and will show an incomplete drawing. Fortunately, there is a tool in AutoCAD that makes the process of transferring XRef files very convenient. This tool is called **eTransmit**. We will discuss this tool in the next section.

Using eTransmit for XRefs

Transferring drawings containing XRefs is not as easy as sending the main host file. You need to send the host file as well as its dependent XRefs with the same folder structure as the main drawing. Doing this manually is time-consuming and prone to error, but we can use eTransmit to do this quickly and with lots of other customizations too.

In the following example, we will learn how to create the `eTransmit` package for the drawing we have created so far with multiple attached XRefs:

1. Open the `main XRef` host drawing and then activate the `eTransmit` command using the application button in the top left of the **AutoCAD** window, as shown in Figure 9.23, or use the `ETRANSMIT` command:

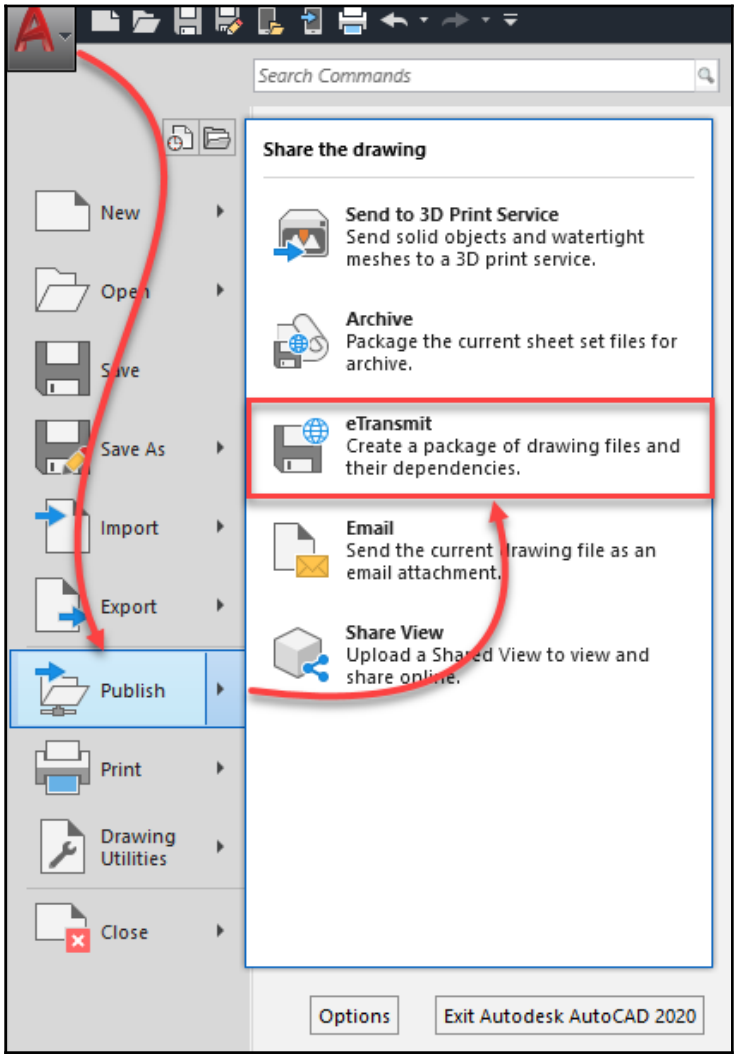


Figure 9.25: The eTransmit option in the application menu

2. The **Create Transmittal** window will now open with a list of all the External Referencess included in the current drawing, as shown in block **A** of Figure 9.26:

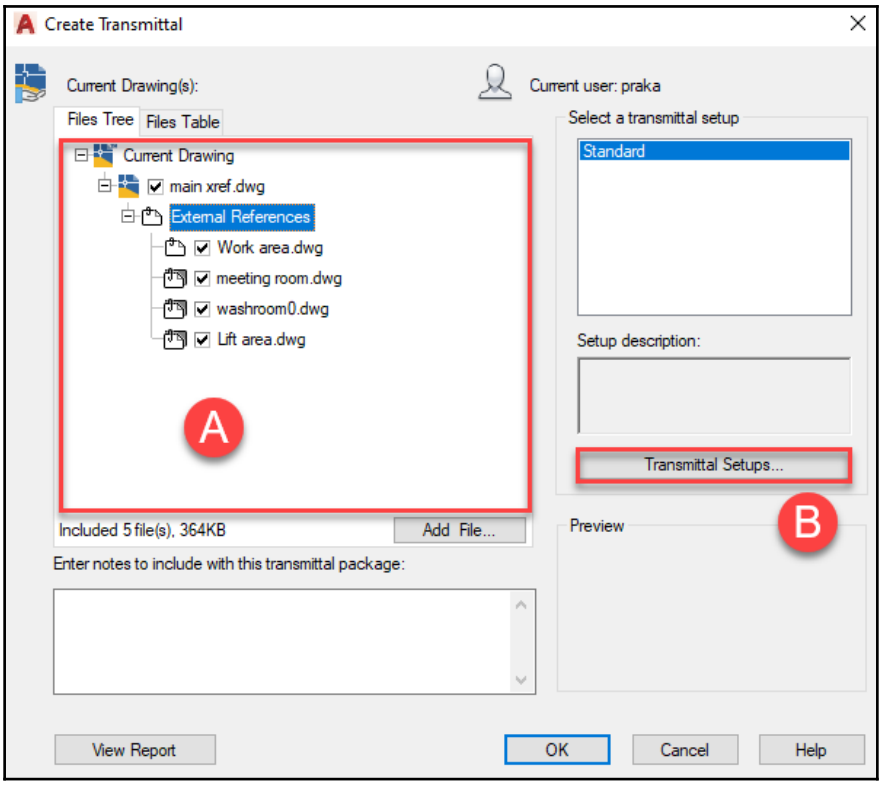


Figure 9.26: List of XRefs in block A and the Transmittal Setups button in block B of the Create Transmittal window

3. If you want to add more files to the final `eTransmit` package, then click the **Add File** button underneath block **A** of Figure 9.26. To change the settings of this `eTransmit` package, click the **Transmittal Setups** button, as shown in Block **B** of Figure 9.26.
4. Click the **Modify** button in the next window, and the **Modify Transmittal Setup** window will open with lots of options, as shown in Figure 9.27:

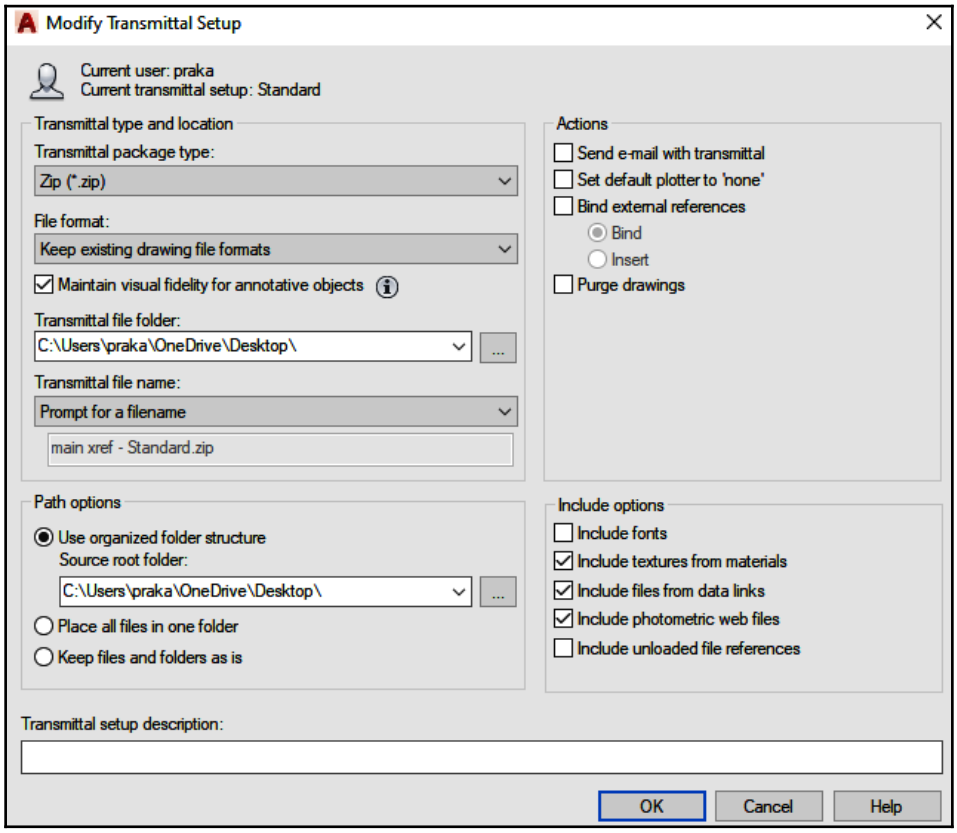


Figure 9.27: Modify Transmittal Setup window

5. In the first panel, **Transmittal type and location**, click the **Transmittal package type** drop-down menu and select **Zip (*.zip)** from the list. You can also select a folder if you want to send all the XRef files to a folder.
6. The next drop-down menu is for the file format, and in this menu you can select the format in which you want to save the XRef files. This feature is great if you want to send the XRef to people who are working with older versions of AutoCAD. From this list, select the **AutoCAD 2013** file format for our example.
7. The next field is the `Transmittal file` folder, and this is basically the location where your transmittal file will be saved. For our example, I have selected the desktop as the location of the transmittal file.

8. The next panel, **Path options**, has three options for controlling the folder structure of the transmittal package. The first option, **Use organized folder structure**, will keep the relative path folder structure, and if you are not sure which folder structure to use, this is the best option you can use by default.
9. The second option will place all the XRef files in the root folder, which is like no path option of XRef, and the last option, **Keep files and folders as is**, will keep the original folder structure in which you saved the XRef. For our example, I will select the first option.
10. The next panel, **Actions**, has options for converting the XRef into blocks using the **Bind external references** option, and you can use **Bind** or **Insert** for the converted XRef. The **Purge drawings** checkbox will clean all the XRef drawings of unused named objects such as layers or blocks. You can keep the **Purge drawings** option checked if you want to clean unused named objects from the XRefs.
11. The last panel, **Include Options**, has options that let you include or exclude some common objects, such as fonts, textures, and data links. If you have used custom fonts in the host drawing or XRef, then it is recommended to keep the **Include fonts** option checked. If you have used other objects too, then keep their respective checkboxes selected as well.
12. After setting all of these settings, click **OK** and then click **Close** in the **Transmittal Setups** window. Then, click **OK** again in the **Create Transmittal** window.
13. Specify the name of the transmittal package from the next window and click the **Save** button. AutoCAD will take a moment to create the transmittal package, and you will have the eTransmit file with all the XRefs, fonts, and other files inside a single zip file or folder.

Now that you have all the XRef files collected at a single location, you can easily send it via email, FTP, or any other method, and you can be sure that this XRef package contains all the files required for the project.

Now that we are done with XRefs, let's move on to another interesting topic, which is dynamic blocks. We have already seen blocks in action in previous chapters, and in the next section we will learn about dynamic blocks, which are a flexible type of block that can be modified even after being adding to a drawing. You can add different types of geometrical properties to these blocks.

Working with dynamic blocks

Dynamic blocks are blocks with dynamic properties, and you can change the shape of dynamic blocks to suit the location where it's used. We have already seen a few dynamic blocks in action in the previous chapters, but here we'll insert a ready-made dynamic block to see how it works.

Type `TP` and press the *Enter* key. The tool palette will open. Select the **Architectural** tab from the list of tabs and drag and drop the Door – Imperial block in the drawing area. This block is a dynamic block that is available for use inside AutoCAD, and when you select the Door – Imperial block, you will get many grips, as shown in Figure 9.28:

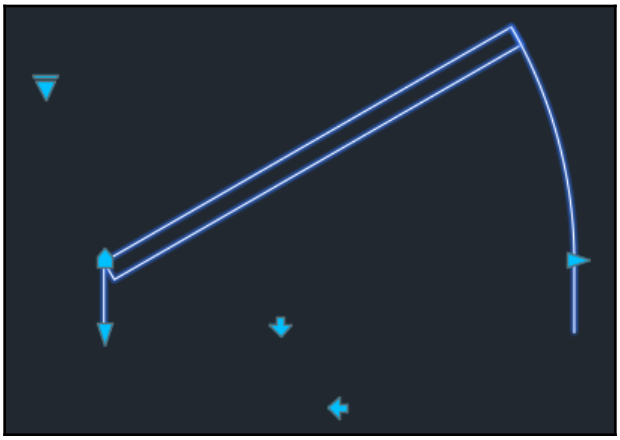


Figure 9.28: Door dynamic block with multiple grips

You can click the grip on the top left of the door to change the open angle of the door. The arrow type grips in the middle of the Door block will flip the direction of the Door block in the horizontal and vertical directions. The arrow to the right will change the length of the door, and the arrow pointing down towards the bottom left of the door is for the wall thickness.

With a combination of all these geometry customization grips, the Door block here has become truly versatile. It can be used in many different locations in your drawing. In the next few examples, we will learn how to make these kinds of dynamic blocks with many different tools and options. So, let's get started with creating our first dynamic block.

Making a dynamic block using constraints

In this example, we will make a dynamic block using the constraints and parameters that we saw in the parametric drawing section of this chapter. For this example, I will use the drawing shown in Figure 9.29. The rectangle in this drawing is `10` units in length and `5` units in width, and the circle has a radius of `1.2` units:

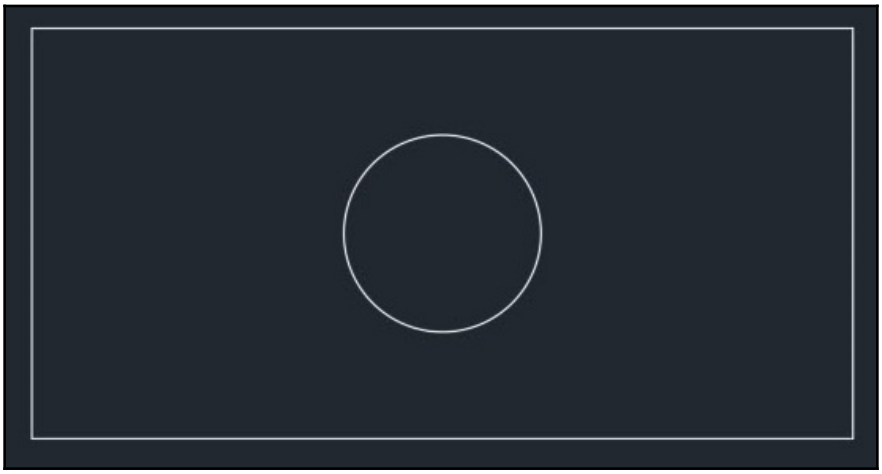


Figure 9.29: Sample drawing for the first dynamic block

In the following example, we will convert this drawing into a dynamic block in such a way that the length can be stretched in both directions and the circle remains at the center of the rectangle:

1. Click the create block icon in the **Block** panel of the **Home** tab, or type `B` and press *Enter* to start the **Block Definition** window.
2. Give this block a name. For this example, I am naming it `Test Block`.
3. Click the **Pick point** box, select the center of the circle as the pick point of the block, click the **Select objects** box, and then select the rectangle and circle and press *Enter*.
4. Make sure **Convert to block** is selected from the options underneath the **Select objects** box.
5. From the **Behavior** panel, uncheck the **Annotative** and **Scale uniformly** options and select the **Allow exploding** option from the list.
6. Check the box that says **Open in block editor** in the lower-left corner of the **Block Definition** window, as shown in Figure 9.30:

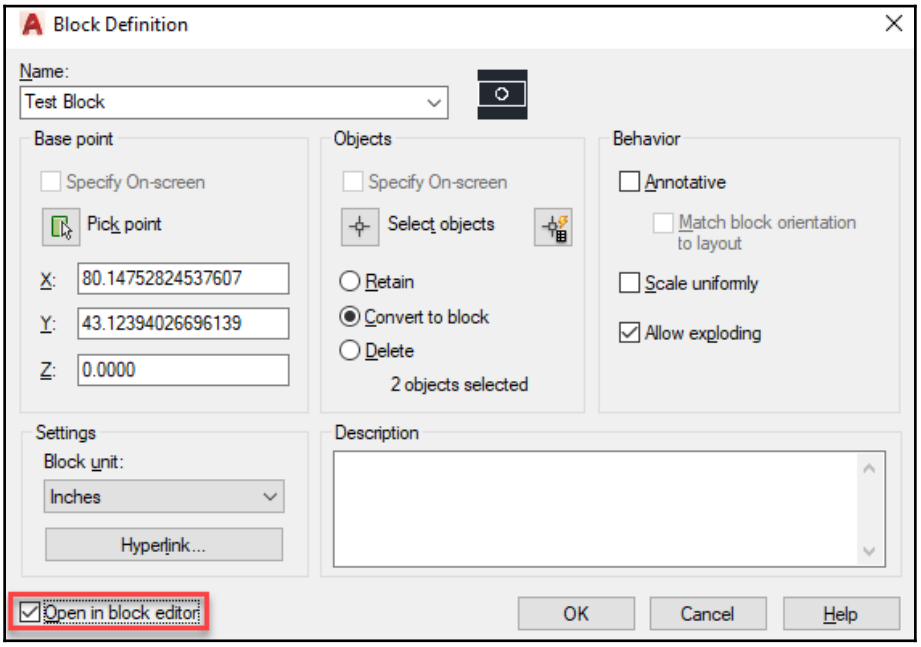


Figure 9.30: The Open in block editor checkbox selected in the Block Definition window

7. Click **OK** after making these changes and the block will now directly open in the **Block Editor** environment, where we can add dynamic properties to this block.
8. You will find the familiar **Geometric** and **Dimensional** constraints panels in the **Block Editor** environment, as shown in Figure 9.31. Select **Auto Constrain** from the **Geometric** panel, select the rectangle and the circle, and press *Enter*. All the required geometric constraints will be automatically applied to the drawing:

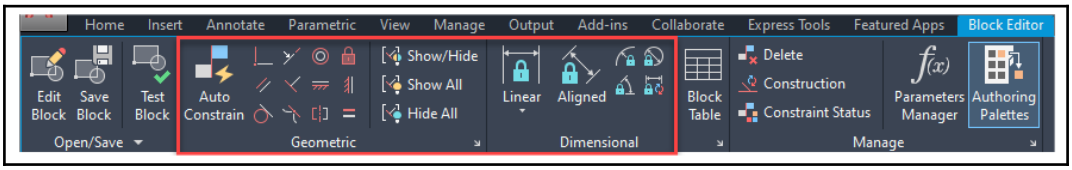


Figure 9.31: Geometric and Dimensional constraint panels under the Block Editor tab

9. Select the **Linear** constraint from the **Dimensional** panel and click close to the midpoint of the left vertical line. Then, click close to the midpoint of the right vertical line and click again on the top of the rectangle to place the dimensional constraint.
10. Select the **Linear** constraint again from the **Dimensional** panel and then click on the midpoint of the left vertical line. Then click on the circle and then click below the rectangle to place this dimensional constraint.
11. As we want the circle to remain exactly at the center of the rectangle, we need to control the second dimensional constraint using a formula that depends on the first dimensional constraint, **d1**.
12. Double-click the **d2** dimensional constraint and change its value to `d2=d1/2`, as shown in Figure 9.32:

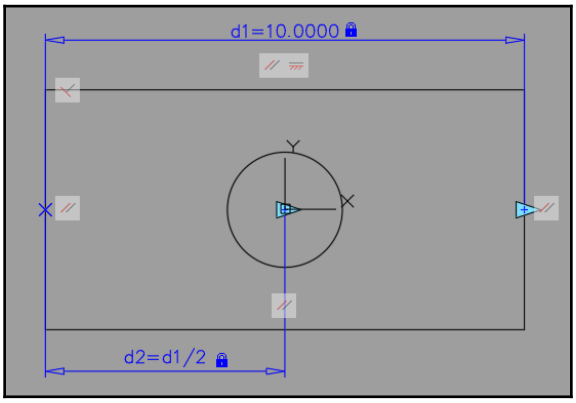


Figure 9.32: Value of the d2 dimensional constraint changed to d1/2

13. Now we are done with our first dynamic block. Click the **Close Block Editor** checkmark at the end of the **Block Editor** tab and save the changes.
14. The drawing will return to the model space. Select the block you created, and you will see a blue arrow grip to the right or to the left of the drawing. Select it, then move it in either direction and the block will stretch, and the circle will also remain centered, as shown in Figure 9.33:

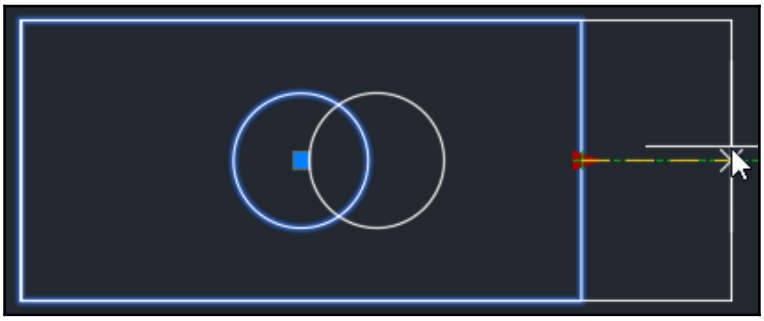


Figure 9.33: Circle remaining centered when the rectangle is stretched

Now you know how constraints can be used to make simple blocks, so let's talk about making blocks with even more properties using parameters and actions. For this example, I will select the Door block, and we will learn how to add a couple of parameters and actions to it.

Making a dynamic block using parameters and actions

Constraints can be used for simple blocks, but for blocks containing lots of objects, it simply becomes too difficult to manage all the geometries using constraints. For this kind of block, you need a specialized set of dynamic block tools called parameters and actions.

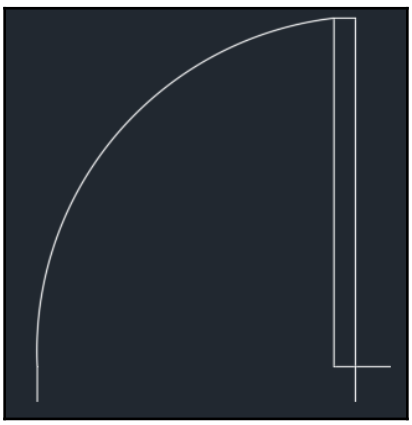


Figure 9.34: Sample Door block to which we will add Rotate, Flip, and Align parameters

In the following example, we will add the Rotate, Flip, and Align parameters to the Door block shown in Figure 9.34:

1. Select the **Create block** option from the **Block** panel of the **Home** tab, or type `B` and press *Enter* to start the create block window.
2. Give this block a name. For this example, let's call it `Sample Door`. Click the **Pick point** box and select the lower-right corner of the door as the base point.
3. Click the **Select Objects** box and then select all the objects that make this door. Make sure **Convert to block** is selected underneath the Select Objects box.
4. From the **Behavior** panel, uncheck the **Annotative** and **Scale uniformly** options and select the **Allow exploding** option. Also, make sure that **Open in Block Editor** in the bottom left of the **Block Definition window** is checked.
5. Click **OK**, and the block will again open in the **Block Editor** environment. In the drawing area, you will also find the **BLOCK AUTHORING PALETTES** option, as shown in Figure 9.35. We will use this palette to add the dynamic properties:

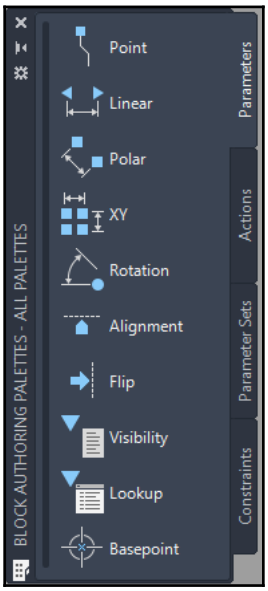


Figure 9.35: The Parameters tab under the BLOCK AUTHORING PALETTES

6. If you don't see the **BLOCK AUTHORING PALETTES** option, then select the **Authoring Palettes** tool from the **Manage** panel of the **Block Editor** tab.

Now that we have everything in place, let's start by adding the **Flip** parameter. This parameter will allow you to flip the direction of the block with respect to the reflection line.

Flip parameter

We will apply two **Flip** parameters to this drawing; one for the horizontal direction and another for the vertical direction:

1. Select the **Flip** parameter from the **Parameters** tab of **BLOCK AUTHORING PALETTES**, as shown in Figure 9.35.
2. Click somewhere near the midpoint of the **AC** line. This will ensure that the arrow type grip will be added at the point where you click.
3. Now, click on another point along the **AC** line. Now, click again to place **Flip state1** close to the horizontal line. The final drawing with **Flip state1** should look like this:

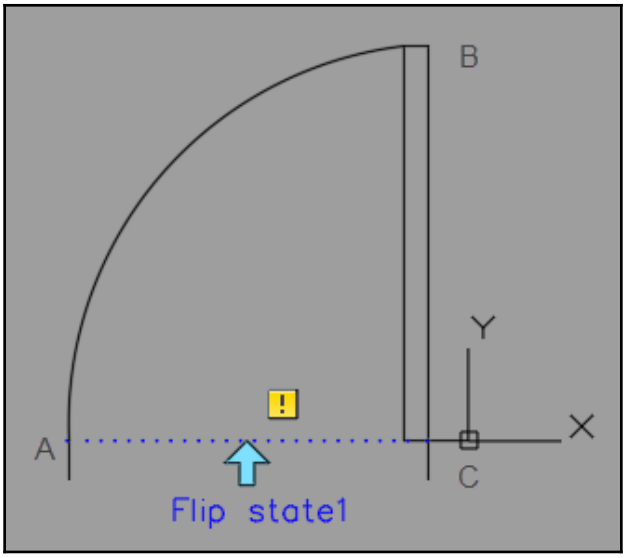


Figure 9.36: Flip parameter added along the Door block

4. Select the **Actions** tab under **BLOCK AUTHORING PALETTES**, select the **Flip** action, click on **Flip state1** from the drawing area, and then all the objects that make the Door block, including **Flip state1**, and press *Enter*:

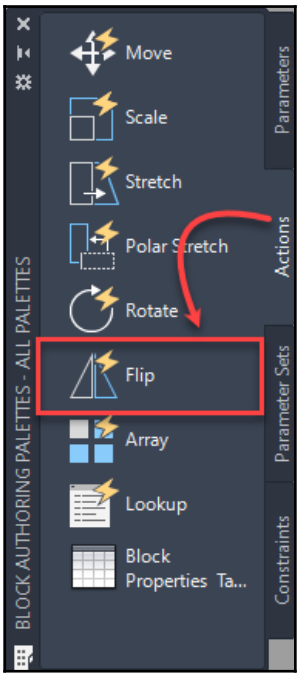


Figure 9.37: Flip action on the Actions tab of BLOCK AUTHORING PALETTE

5. You will notice that the exclamation mark next to the **Flip** state will now disappear because we have added the required action for the **Flip** parameter.
6. To add another **Flip** parameter, once again go to the **Parameters** tab and select the **Flip** parameter under **BLOCK AUTHORING PALETTES**.
7. Click close to the midpoint of the **BC** line for the second flip arrow, and then click again on point **B** or **C** in the vertical direction.
8. Click again close to the vertical reflection line to place **Flip state2**.
9. Select the **Actions** tab, select the **Flip** action under **BLOCK AUTHORING PALETTES**, and click on **Flip state2**. Then, select the entire Door block, including flip states 1 and 2, and press the *Enter* key.

10. The final block should look like Figure 9.38 in the **Block Editor** environment:

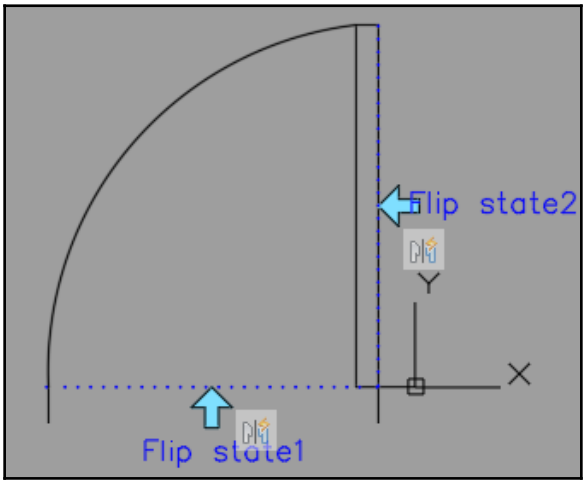


Figure 9.38: Block with two flip states added

11. Now we have two **Flip** parameters added to our block and it's time to test it. To test this block, you don't need to exit the **Block Editor** environment. Click the **Test Block** option in the **Open/Save** panel of the **Block Editor**, as shown in Figure 9.39:

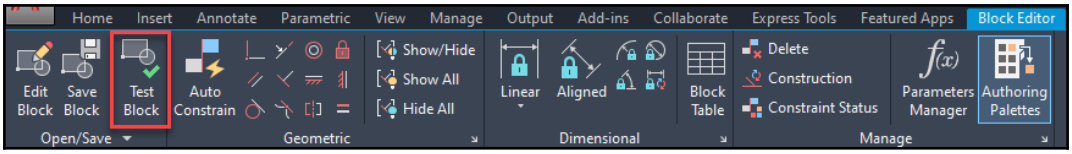


Figure 9.39: Test Block option under the Block Editor tab

12. Now, the block will open in the test environment. Select the block, and the two arrow grips along the horizontal and vertical directions will show up. Click on the arrow grips and the direction of the Door block will flip.
13. When you are done checking the block, click the **Close Test Block** option, which is at the end of the **Home** tab, and you will once again return to the **Block Editor**.

If, for some reason, you exit the **Block Editor**, you can enter it again by selecting the block and then selecting the **Block Editor** option from the right-click menu.

Now, without exiting the **Block Editor** environment, we will again the next parameter, Alignment in our dynamic block.

Alignment parameter

The **Alignment** parameter aligns the selected block toward the alignment direction of the selected line. This tool is helpful in aligning the doors to different directions of the wall, and you can use it in other situations too.

In the following example, we will add the **Alignment** parameter under our Door block:

1. Select the **Alignment** parameter from the **Parameters** tab under **BLOCK AUTHORING PALETTES**.
2. Click on point **C** of the Door block and then click on point **A** along the horizontal line.
3. You will get the alignment grip at point **C** of the Door block. To test this block, once again click on the **Test Block** option from the **Open/Save** panel of the **Block Editor** tab.
4. This block can't be tested without any reference drawing. So, to test this, I will draw a few lines at random angles, as shown in Figure 9.40:

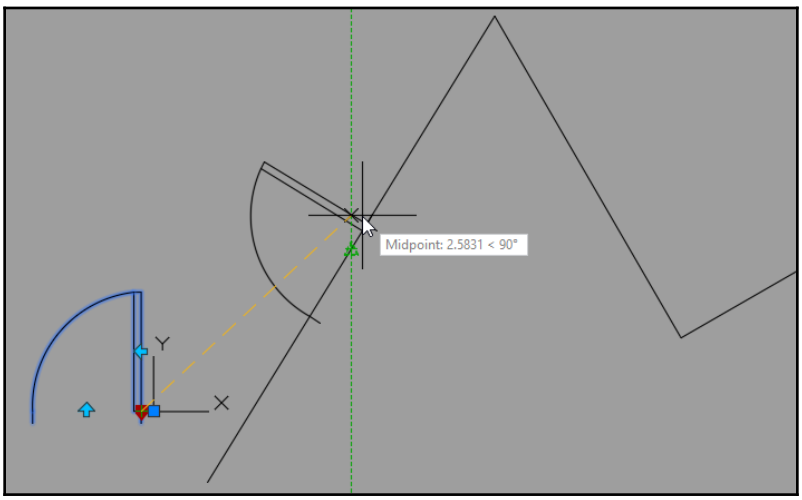


Figure 9.40: Door block aligning to the direction of aligned lines because of the Alignment parameter

5. Click on the Alignment grip at point **C** and then move your Door block close to any line. The Door block will automatically align to the direction of the line, as shown in Figure 9.38. You can move the door to different lines and the alignment will change accordingly.
6. Once you are done checking the block, click the **Close Test Block** option at the end of the **Home** tab and you will return to the **Block Editor**. The reference lines will also disappear from the **Block Editor**.

As you may have noticed, this parameter is very simple and does not require any action; simply adding an Alignment parameter was enough. Now, we have two parameters added to our dynamic block, a **Flip** parameter and an **Alignment** parameter. Let's add one more parameter, **Rotation**, and we will also modify this parameter with a number of extra properties.

Rotation parameter

The **Rotation** parameter will let you rotate the block about a base point. In this case, we need to add the parameter as well as the associated action to make this parameter work.

In the following example, we will add the **Rotation** parameter to our existing Door sample drawing:

1. Start by selecting the **Rotation** parameter from the **Parameters** tab under **BLOCK AUTHORING PALETTES**.
2. Click on point C of the Door block. This point will become the base point about which the Door block will rotate.

2. Now, click horizontally toward the right and then click again in the same horizontal direction to place the **Rotation** parameter. The **Angle1** parameter as shown in Figure 9.41 will be added:

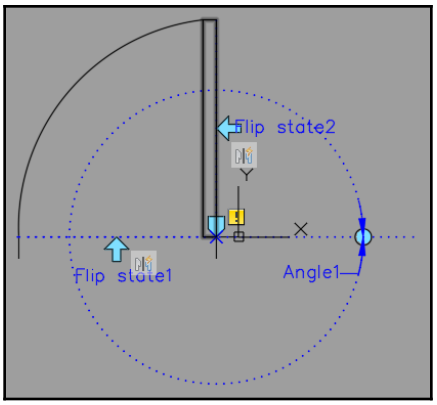


Figure 9.41: Door block with a Rotation parameter called Angle1 added

4. Select the **Rotate** action from the **Actions** tab as shown in Figure 9.42. Now, click the **Angle1** parameter and then make a window to select all the objects, including the **Flip state**, **Alignment**, and **Angle1** parameters, and then press *Enter*:

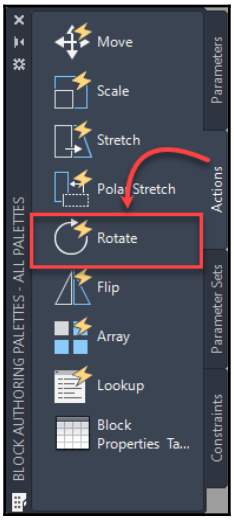


Figure 9.42: Rotate action on the Actions tab

5. Now, we have added the **Rotation** parameter to our block as well. Now, once again, click the **Test Block** option of the **Open/Save** panel on the **Block Editor** tab and the block will open in the test environment.
6. Select the block and you will get a circular grip toward the right of the Door block. Click this grip and move your cursor. You will notice that the Door block will now rotate with respect to the point **C** of the Door block, as shown in Figure 9.43:

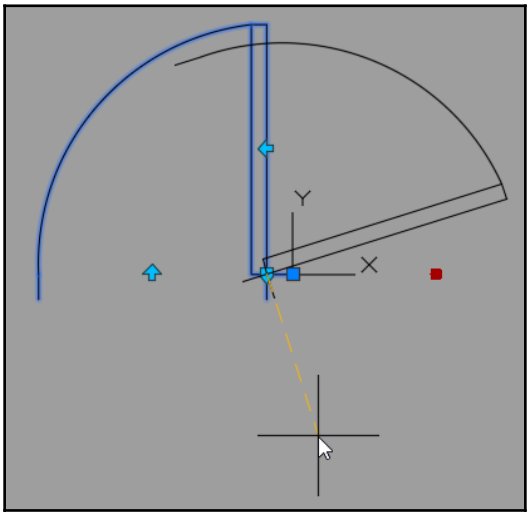


Figure 9.43: Block rotating with respect to point C

7. When you are done testing the block, click the **Close Test Block** button on the **Home** tab and the block will return to the **Block Editor** environment.

The Door block that we have right now is free to move in any direction, but for many practical purposes, you might want to restrict this rotation to a set of angles, such as 30 degrees and its multiples. You can do that as well by changing the properties of the rotation parameter.

In the next example, we will do just that. We will add a rotation value of 30 degrees and its multiples to the Door block:

1. Select the **Angle1** parameter from the drawing area of **Block Editor** and then right-click and select **Properties** from the menu.
2. Scroll down in this **Properties** palette to the **Value Set** panel and select **Increment** from the **Ang type** option.
3. In the next field, **Ang increment**, type `30`. The property palette should look like this once these changes have been made:

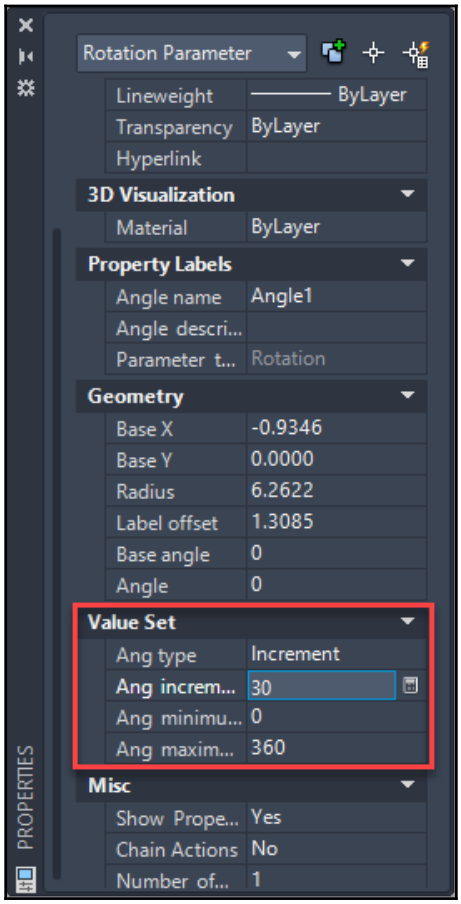


Figure 9.44: Settings in the Value Set panel of the Properties palette

4. Close the **Properties** palette. Now, if you go to the Test Block environment and check the **Angle** parameter, you will find that this time, the angle will only change with an increment of `30` degrees, as shown in Figure 9.45:

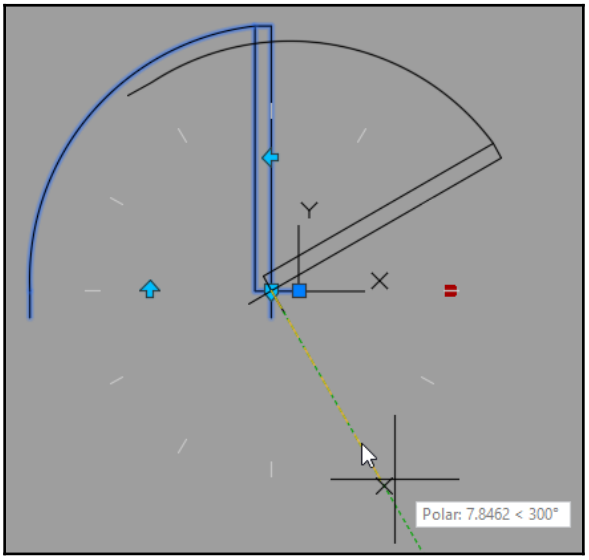


Figure 9.45: Angle of the Door block changing with 30-degree increments

We are done adding all the parameters and actions to this Door block and we can now close the **Block Editor** environment. Click the **Close Block Editor** checkbox at the end of the **Block Editor** tab and make sure you select **Save changes** in the next prompt.

The block will now show up in the model space of the drawing area, and when you select the block, all its dynamic parameters will appear, and just like the test block environment, the block will work the way you want. You can flip this block horizontally and vertically. You can align this block to any line, and finally, you can rotate this block along point **C** in increments of `30` degrees.

This block has lots of parameters and actions, but you can add even more parameters and actions to this block to make it even more useful. In the next section, we will talk about adding the visibility parameter and action, and you can use it to add different open angles to this door.

Visibility parameter

The Door block we made in the previous example now has many parameters and actions and with that it now has many properties, but we can add even more properties to that Door block using the visibility parameter. In the next example, we will add the visibility parameter to the Door block.

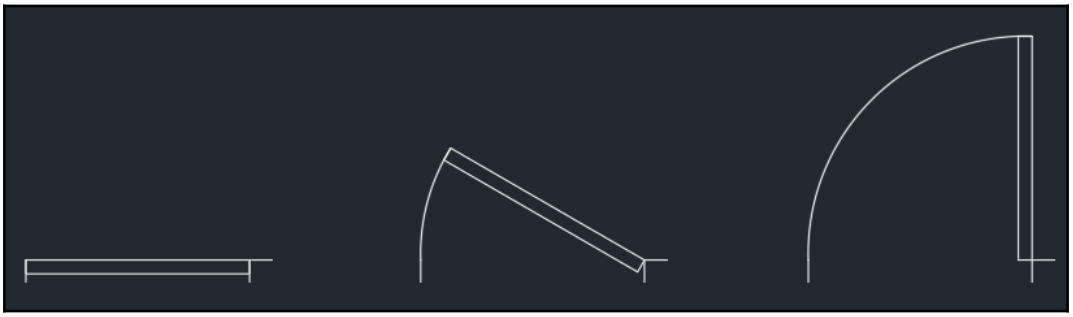


Figure 9.46: Doors with different open angles

In the following example, I will make another Door dynamic block with open angles of 0, 30, and 90 degrees. To do this, make three doors, as shown in Figure 9.46:

1. Click the **Create Block** option in the **Block** panel of the **Home** tab to start the `Block Definition` command, or use the command alias `B` to start this command.
2. Name the block `0 angle`. Click the **Pick point** box and specify the point marked in Figure 9.47 as the base point:



Figure 9.47: Basepoint highlighted in the red circle of the 0 angle block

3. Click **Select objects** and then select all the objects that make the 0-degree open-angle block, as shown in Figure 9.47, and then press *Enter*.
4. Uncheck the **Annotative**, **Scale uniformly**, and **Open in block editor** checkboxes, and check the **Allow exploding** box. The final **Block Definition** window should look like this:

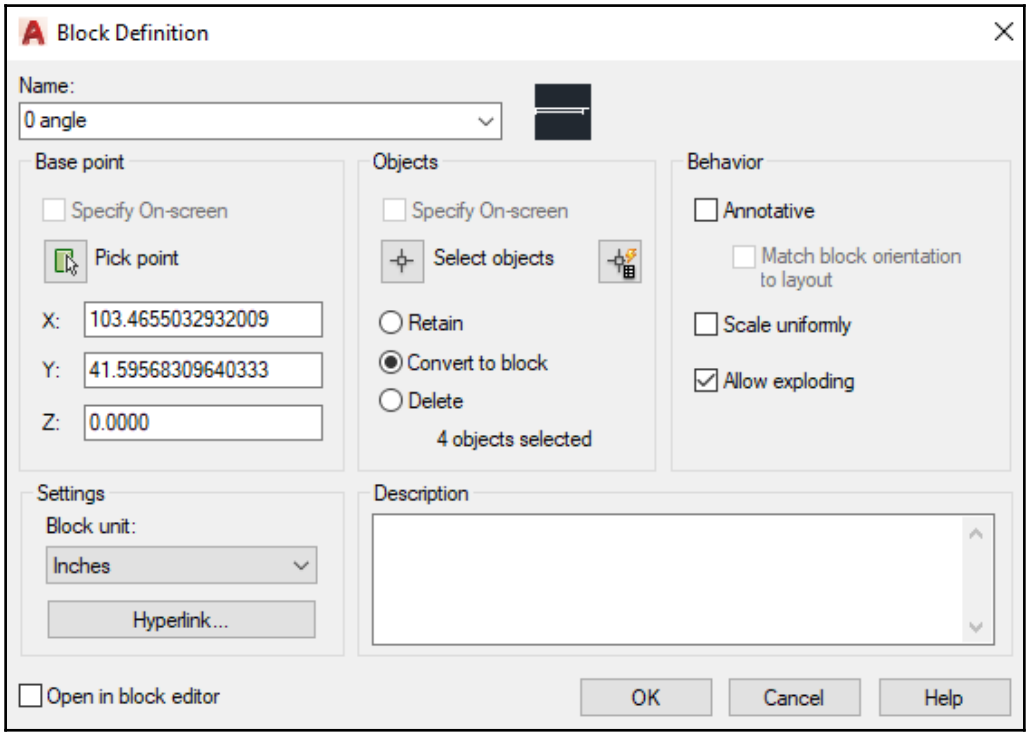


Figure 9.48: The Block Definition window with all the settings in it

5. Click **OK** when you are done making these changes. Now, we have a simple block in the drawing area called `0 angle`.
6. Repeat the process and make two more blocks, called `30 angle` and `90 angle`, and similar block definition parameters. Also, make sure that you select the same base point as we did in Figure 9.47 in other blocks too.

This was the first step toward making a dynamic block with a visibility parameter. In this case, we made three separate blocks, and now we will use all these blocks in our dynamic block:

1. Place all the blocks overlapping one another in such a way that the base points of all three blocks overlap, as shown in Figure 9.49:

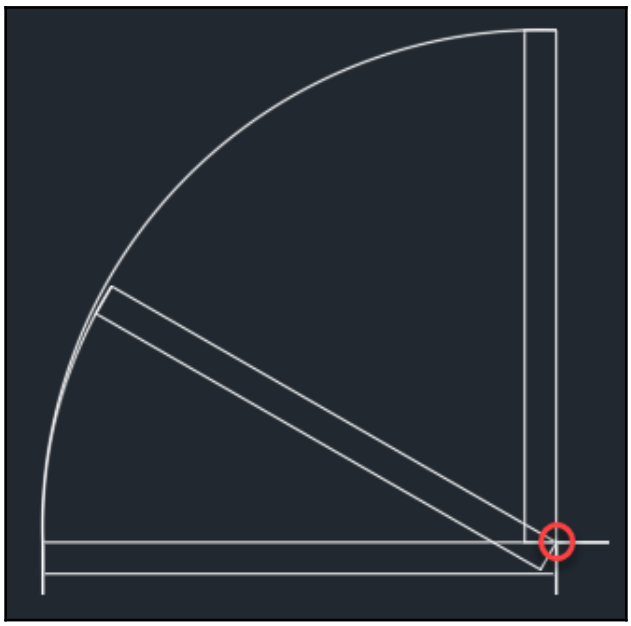


Figure 9.49: All blocks placed over one another with overlapping base points as shown in the red circle

2. Once again, start the **Create block** command from the **Block** panel of the **Home** tab, or use the command alias `B`.
3. Give this block a name. For this example, I will name it `Door Vis`, short for Door Visibility.
4. Click the **Pick point** box and select the common base point of the blocks as the pick point.
5. Click the **Select objects** box and select all three Door blocks from the drawing area. Press *Enter*.

6. Uncheck the **Annotative** and **Scale uniformly** options, and check the **Allow exploding** and **Open in block editor** checkboxes, as shown in Figure 9.50:

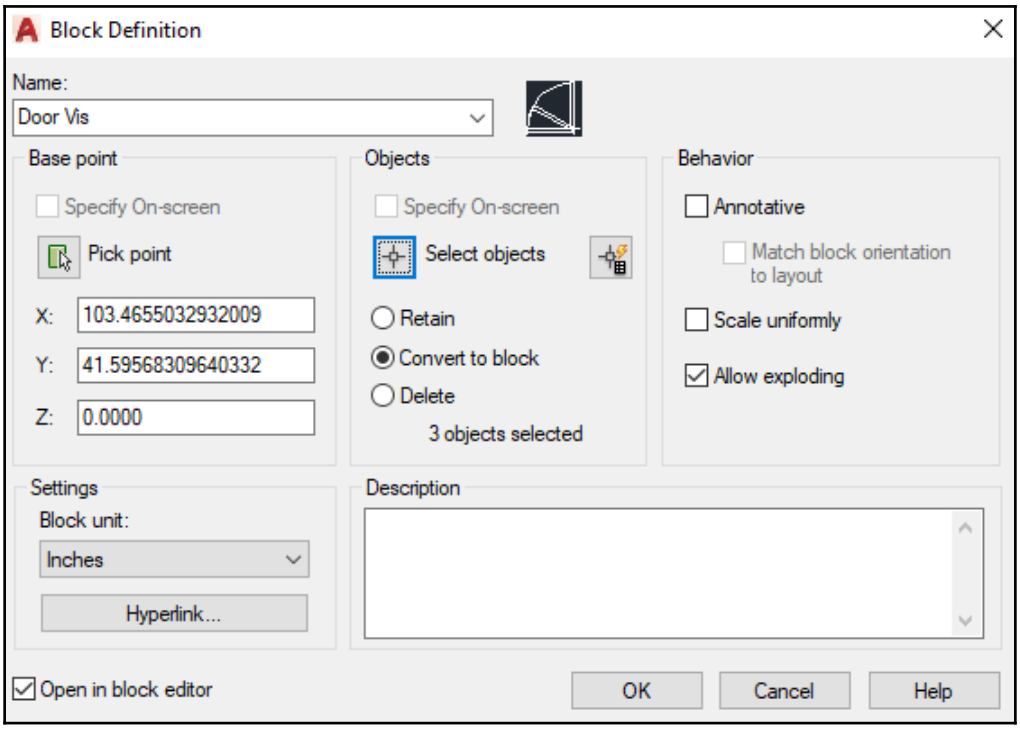


Figure 9.50: Settings in the Block Definition window for the Door Vis block

7. Click **OK** in the **Block Definition** window, and the `Door Vis` block will open in the **Block Editor**.

Now we have a set of blocks in the **Block Editor**. We will add a visibility parameter and action to the set of blocks so that only one of the blocks remains visible in each visibility parameter:

1. From the **BLOCK AUTHORING PALETTE** of the **Block Editor** environment, select the **Visibility** parameter from the **Parameters** tab, as shown in Figure 9.51:

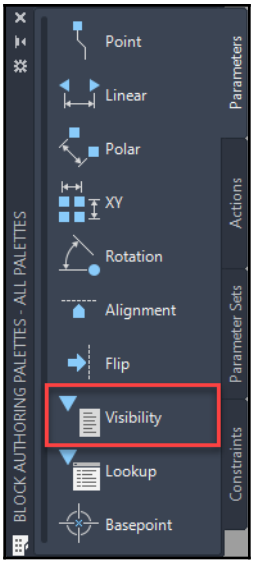


Figure 9.51: The Visibility parameter on the Parameters tab

2. Click on any point close to the set of blocks to place the **Visibility** parameter. For this example, I clicked close to the top-right point of the Door block.
3. A `Visibility1` parameter with an exclamation mark will be added. Now, we need to add actions to make this parameter work.
4. Click the **Visibility States** option under the **Visibility** panel of the **Block Editor** tab, as shown in Figure 9.52:

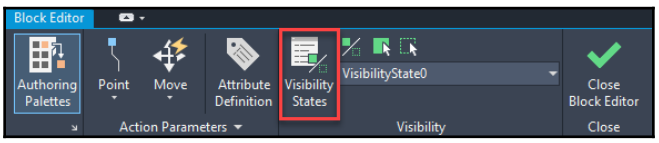


Figure 9.50: Visibility States option under the Visibility panel of the Block Editor tab

5. Click the **New** button and then give this new visibility state a name. For this example, name it `30-angle` and click the **OK** button. Repeat the process and add one more visibility state called `90-angle`.
6. Now, the **Visibility States** window should have three **Visibility States**, as shown in Figure 9.53. Click **OK** when you are done adding all three **Visibility States**:

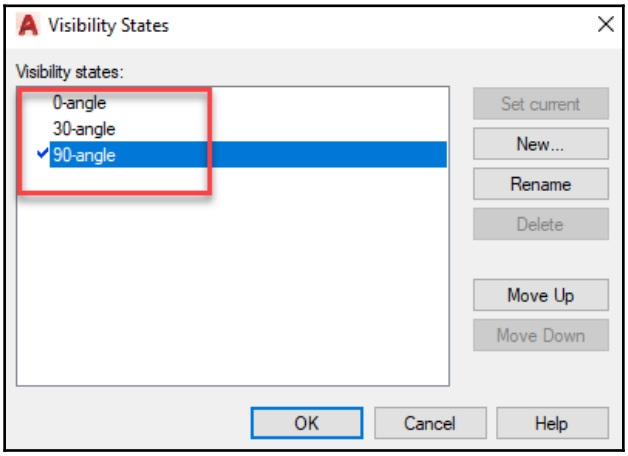


Figure 9.53: Three Visibility States added in the Visibility States window

8. Now, select `0-angle` from the visibility drop-down menu of the **Visibility** panel, and then click the **Make Invisible** option from the **Visibility** panel, as shown in Figure 9.54. Click the `30-degree` and `90-degree` blocks from the **Block Editor** drawing area (leave the `0-degree` block) and press *Enter*:

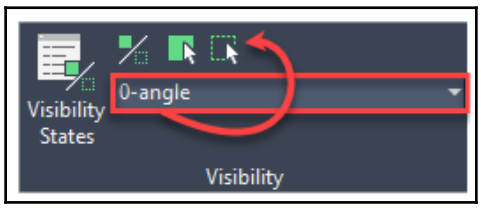


Figure 9.54: The Make Invisible option on the Visibility panel with the 0-angle state selected

9. You will notice that the `30-degree` and `90-degree` blocks will disappear from the drawing area of the **Block Editor**, and for the `0-angle` visibility state, only the `0-degree` block will remain visible.

10. Once again, in the **Visibility** parameter drop-down menu in the **Visibility** panel, select the `30-angle` parameter and then click the Make Invisible icon, as shown in Figure 9.54.
11. All three blocks will again appear. Now, click the `0-degree` and `90-degree` blocks from the drawing area of **Block Editor** and press *Enter*.
12. Again, select the `90-degree` visibility parameter from the dropdown and, in this case, make the `0-degree` and `30-degree` blocks invisible using the same steps we have followed so far.
13. When you are done applying these visibility actions, click **Close Block Editor** at the end of the **Block Editor** tab and save the changes in this block.
14. The block will now appear in the model space, and when you select it, a down arrow grip will appear next to the block. Click on it, and you will get a list of the Visibility States we have applied to this block.
15. Select `0-angle` and the block will change to the `0-angle` block. Similarly, if you select the `30-angle` and `90-angle` blocks, the block will change to the respective angle Door block, as shown in Figure 9.55:

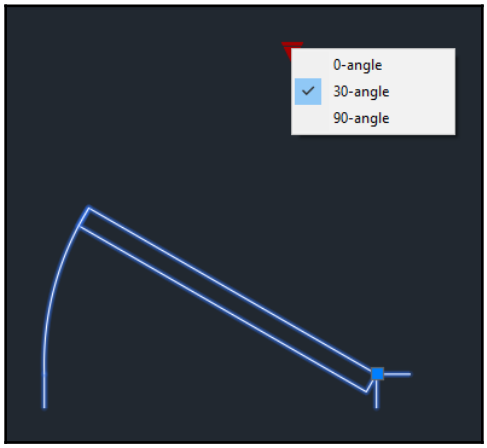


Figure 9.55: Visibility parameter making only one of the blocks visible

So, this was an example of the visibility parameter and action tool of dynamic blocks. As you can see, there are lots of other parameters and actions available in the **BLOCK AUTHORING PALETTE**, and I encourage you to explore them too.

Now we have finished the drawing and editing part of 2D drawing in AutoCAD. Before we move on to the 3D drawings, let's learn a little bit about managing drawing files and keeping them clean in the next section.

Cleaning and fixing drawing issues

Just like any physical object, drawings do tend to get untidy over time, and usually they accumulate a lot of redundant objects, such as zero-length geometries, DGN linetypes, and unused named objects that not only bloat the file size, but also start creating errors in the drawing.

There are many inbuilt tools in AutoCAD that help you keep drawings clean, and even when your drawings seem to show errors, these tools can fix those errors too. In this section, we will talk about four such tools: `PURGE`, `OVERKILL`, `AUDIT`, and `RECOVER`.

Using PURGE to clean drawings

Open any drawing that you suspect has redundant objects, type `PURGE`, and press the *Enter* key. The **Purge** window will open, which looks like this:

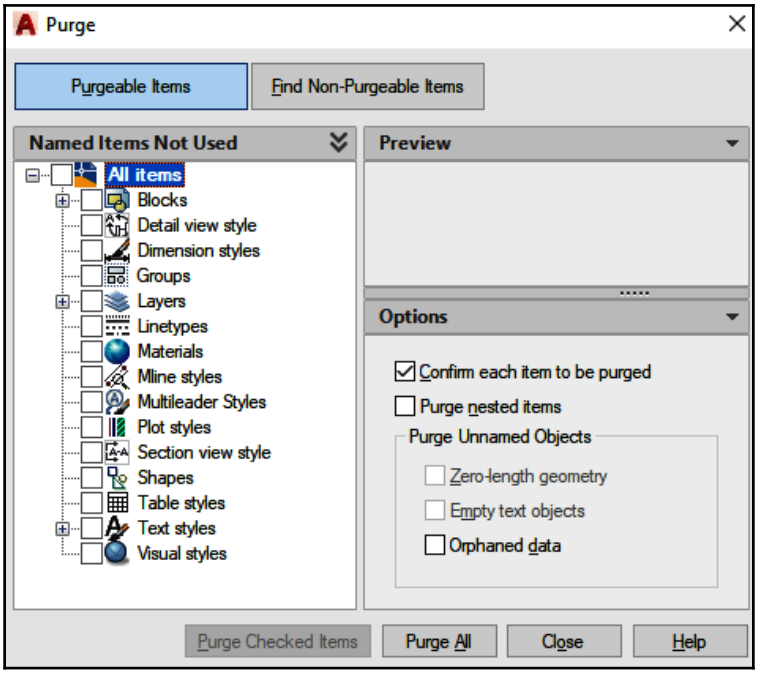


Figure 9.56: The Purge window with the list of objects in the current drawing

In the **Purge** window, you will find two large buttons at the top: **Purgeable Items** and **Find Non-Purgeable Items**. By default, **Purgeable Items** is selected, and when it is selected, the **Named Items Not Used** panel will show a list of items that can be purged. There is an entire list of named objects, such as **Blocks**, **Groups**, and **Layers**, but the redundant named objects will have a plus sign next to them, like the **Blocks**, **Layers**, and **Text styles** shown in Figure 9.56.

When you click the **+** icon, the list will expand to show the redundant objects, and you can select the object, right-click on it, and select **Purge** from the right-click menu to purge only the selected object. You can also click the checkboxes next to the objects to select multiple objects for purging. To purge all the redundant objects, click the **Purge All** button at the bottom of the **Purge** window.

The **Purge** window has options for purging **Zero-length geometry**, **Orphaned data**, and **Empty text objects** too, and, if available, it will be highlighted in the **Options** panel of the **Purge** window, as shown in Figure 9.57. You can select the checkboxes in the **Options** panel and click the **Purge All** button to get rid of this type of object, too:

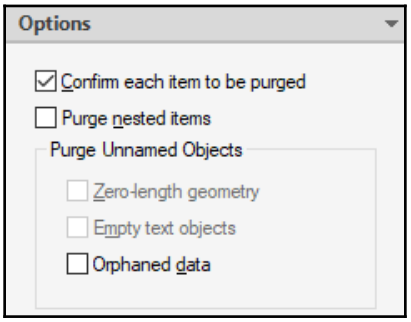


Figure 9.57: Purgeable objects such as zero-length geometry and orphaned data in the Options panel

When you click the **Find Non-Purgeable Items** button, the **Purge** window shows objects that can't be purged. When you select this kind of object, AutoCAD shows the possible reason why they are non-purgeable in the panel, as shown in Figure 9.58:

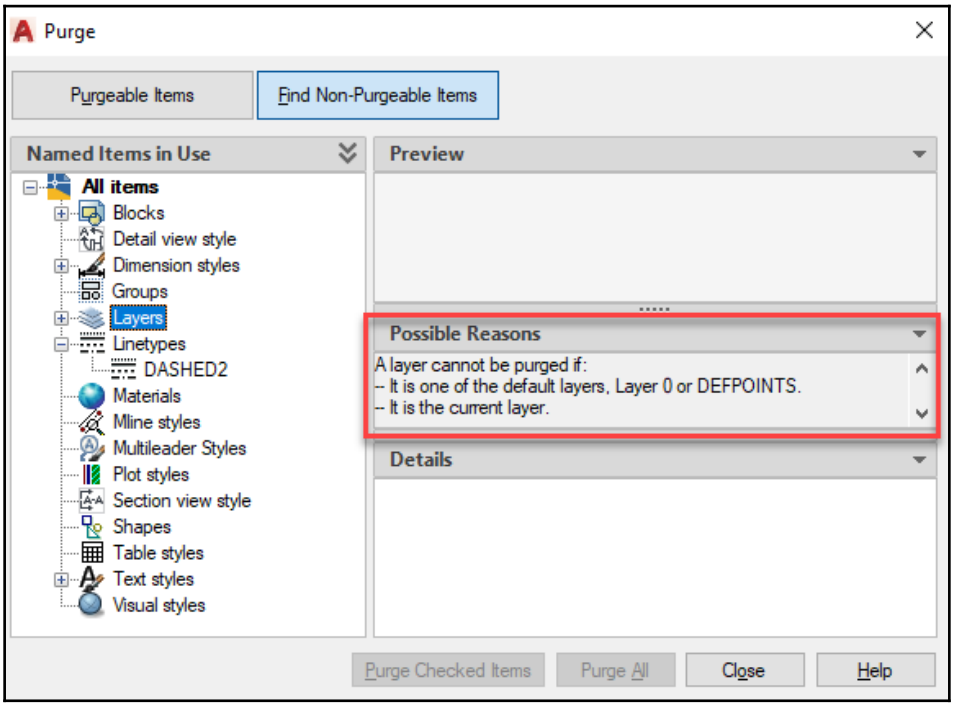


Figure 9.58: The Possible Reasons panel contains reasons why an object is non-purgeable

As you can see, this tool is pretty good at cleaning drawings, and there is no harm in using it a couple of times on the drawings of finished projects. However, there is another drawing clean-up tool just like Purge that is especially good at cleaning drawing objects on the go. We will discuss this tool next.

Using OVERKILL to clean drawings

When making drawings, we often end up making reference geometries that overlap other geometries. Sometimes, we also end up making geometries with several segments that can be made with just one segment. For these kinds of drawings, the `OVERKILL` tool can be used. This tool cleans all overlapping objects from the drawing and merges broken segments into one.

To use this tool, type `OVERKILL` on the command line and press *Enter*; the command will start. Now, select all the objects from the drawing area that you want to clean and press the *Enter* key. The **Delete Duplicate Objects** window will now pop up, as shown in Figure 9.59; don't change any setting in this window. Simply click **OK**, and the `OVERKILL` command will clean the drawing for you:

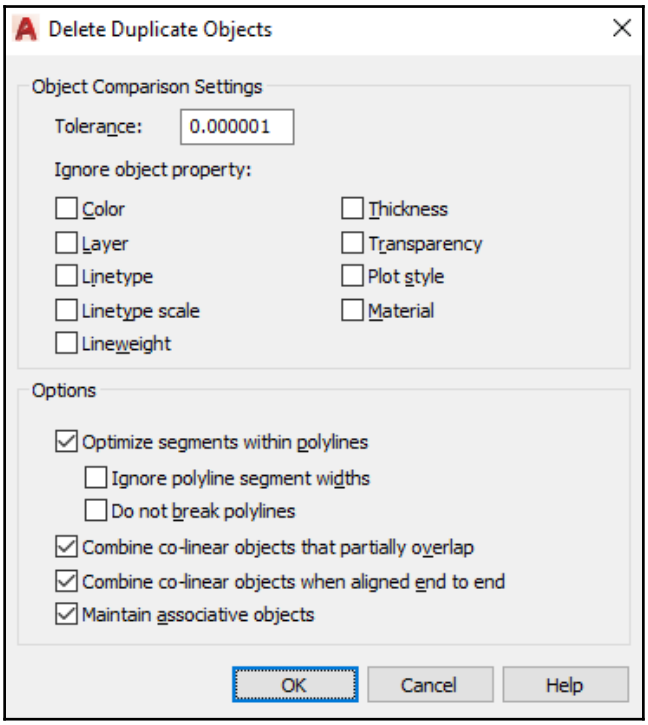


Figure 9.59: The Delete Duplicate Objects window of the OVERKILL command

Now that you know how to clean drawings by getting rid of known issues, it's time to look at tools that help you clean and fix drawings by removing unknown issues.

Using the AUDIT command

This command is helpful for fixing drawings that start showing unexpected behavior, or you can use this on drawings that you suspect contain errors. This tool can also be used to fix corrupted drawings; if you are able to open the drawing, you can use this tool to fix it.

Simply open the drawing in question and type `AUDIT` and press *Enter*, and then select `Yes` on the command prompt that says `Fix any errors detected?`. AutoCAD will run the command, find the errors and fix the issues, or delete the objects causing the issues. The result of the `AUDIT` operation will appear above the command line, as shown in Figure 9.60:

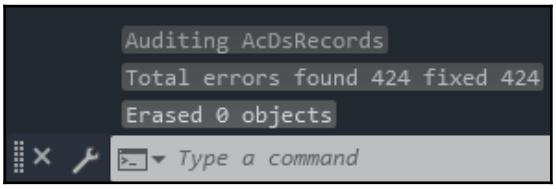


Figure 9.60: Result of the AUDIT command showing the number of errors fixed and objects erased

As you can see, I managed to fix 424 errors in my sample drawing, and once the errors are fixed, we can save this drawing.

Sometimes, the drawings are corrupted to such an extent that they won't even open. In that case, you can use the `RECOVER` tool, discussed next.

Using the RECOVER command

Using the `RECOVER` command, you can open drawings that won't open normally in the AutoCAD window. To use the `RECOVER` tool, open a blank drawing, type `RECOVER`, and press *Enter*. The **Select File** window will open. Select the drawing that needs to be recovered and click the **Open** button. AutoCAD will now attempt to `AUDIT` the file and open it.

Sometimes, this works and sometimes it doesn't. So, the `AUDIT` and `RECOVER` commands are the last option for you when everything else fails to fix the drawing.

So, with this topic, we have finished the 2D part of AutoCAD. From the next chapter onward, we will start learning about the 3D tools of AutoCAD. Before we look further into the 3D topics, let's quickly look at the topics we have covered and the topics we will cover next.

Summary

In this chapter, we have learned about XRefs, which are a great collaboration tool that's built into AutoCAD, and we also explored the dynamic block tool. We used the concepts of parametric drawing to make our dynamic block, and we also made dynamic blocks using parameters and actions, which are the core features of the dynamic block tool. Finally, we learned how to fix broken drawings and clean bloated ones using tools such as AUDIT and PURGE.

From the next chapter, we will start covering AutoCAD's 3D tools, and we will begin by looking at the 3D user interface and the common tools required to understand the workings of the 3D workspace of AutoCAD. Then, we will gradually move to the drawing and modification tools of AutoCAD 3D.

10
Introduction to 3D Modeling

This is an introductory chapter to 3D modeling in AutoCAD, but before we start creating 3D models we need to set up AutoCAD properly for 3D modeling. In fact, AutoCAD has already done that for us by including the workspaces that are designed for working in 3D. We will learn about the concept of the *workspace* and will explore the 3D modeling workspaces offered by AutoCAD. Another mandatory skill that we will cover is how to navigate the model efficiently and how to quickly switch between view angles. In addition to that, we need to learn how to configure the objects' visual styles, as the style and the amount of details used in displaying objects will directly impact both the model visibility and PC performance. After that, we will learn how to use multi-viewport configurations, and how to work with the coordinate systems in AutoCAD.

So in short, in this chapter, we will cover the following topics:

- 3D modeling workspaces
- Navigation and switching views
- Adjusting visual styles
- Configuring multi-viewports
- Exploring the **User Coordinate System** (**UCS**)

Now, let's start our journey with 3D modeling in AutoCAD.

Technical requirements

To start creating 3D models in AutoCAD, there are hardware and software requirements that we must meet.

The general hardware requirements are the same as those discussed in `Chapter 1`, *An Introduction to AutoCAD*. They may vary to according to the version used and the model size, and the recommended requirements for AutoCAD 2021 can be found at `https: //autode.sk/2CTGy2d`.

In terms of software requirements, access to the 3D modeling functionality requires the full version of AutoCAD, as most of the 3D modeling commands and tools are not present in the LT version. In AutoCAD LT, you can only open 3D model files built using the full version and do some small modifications, but it is not possible to create a full model.

3D modeling workspaces

There are many types of projects and models that can be built using AutoCAD, and each application or project type has a specific set of tools and commands that best serve this type of project or application. Workspaces are a feature of AutoCAD that help you to optimize the working area for better display and access to the appropriate tools that best serve your project.

In this section, the concept of workspaces will be introduced, and we will take a quick tour of the preset 3D workspaces that are available in AutoCAD.

Introduction to workspaces

As shown in `Chapter 1`, *An Introduction to AutoCAD*, the commands and tools are organized in tabs and panels. Each tab contains a number of panels, and each panel contains a group of commands and tools. These collections of tabs and panels are a part of bigger groups called workspaces.

The reason for using workspaces is to optimize the user interface to best serve the application or the project. Workspaces help the user to display specific sets of commands and tools that are relevant to the application or the project type.

By default, AutoCAD has three workspaces:

- **Drafting & Annotation**
- **3D Basics**
- **3D Modeling**

The first workspace, **Drafting & Annotation**, is the one that has been used in this book until now, and contains the main 2D drafting and editing tools.

The second and third workspaces in the preceding list are for 3D modeling. They contain the full set of tools and commands that can be used for 3D modeling, including creating and editing solid objects and complex surfaces, and mesh modeling tools.

To switch between workspaces, click on the workspace switching button (the small gear icon) at the bottom-right corner of the screen. This will expand the workspace menu, which is divided into two parts. The upper part shows the available workspaces to choose from, and the lower part displays other options for customizing workspaces, as shown in the following figure:

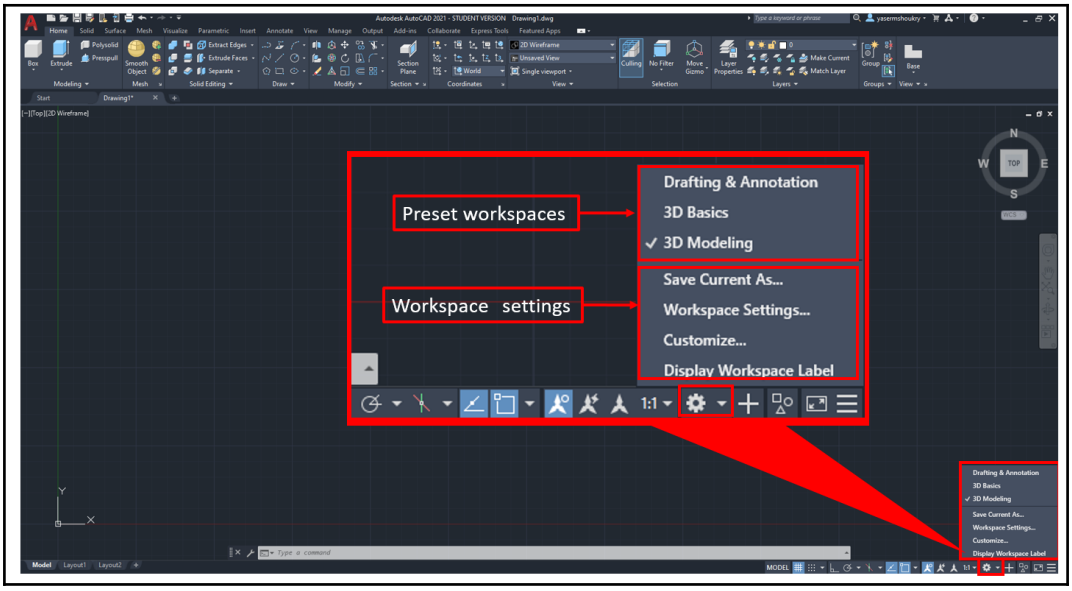


Figure 10.1: Switching between workspaces

By default, the **Drafting & Annotation** workspace is selected, which is indicated by a small checkmark as shown in the figure. To switch to another workspace, you can simply click on the workspace name. An alternative way to change the workspace is by typing `WSCURRENT` anywhere in the software, press *Enter*, then type the desired workspace name, and press *Enter* again.

Now that we know about the different workspaces, let's look at the 3D Basics workspace in more detail.

Exploring the 3D Basics workspace

We will take a quick tour of the 3D modeling related tabs and panels in the **3D Basics** workspace. As shown in the previous section, to switch to the **3D Basics** workspace, click on the workspace switching button and select **3D Basics.**

Starting with the **Home** tab, you can find two panels that are present in the standard **Drafting & Annotation** workspace, the **Draw** panel and the **Modify** panel. The **Layers** panel is also present but it is modified to be **Layers & View**. There are also other panels that contain 3D modeling tools, as shown in the following figure:

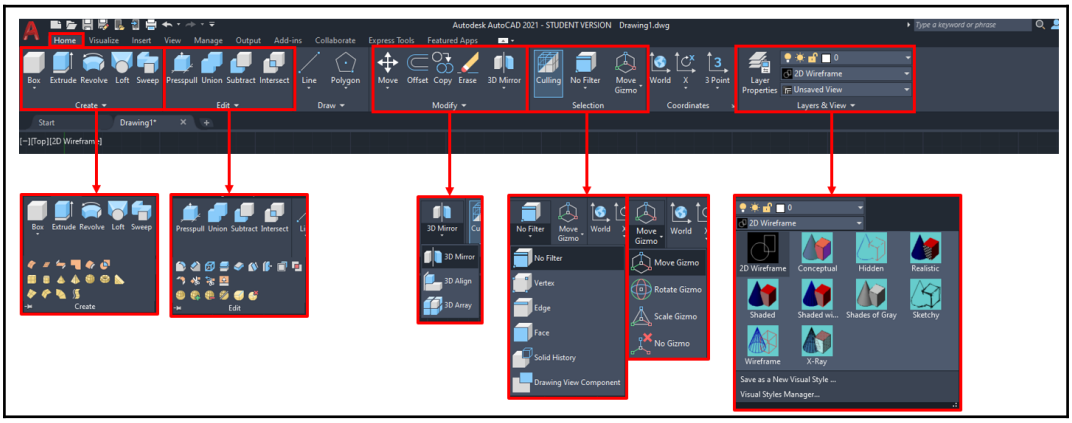


Figure 10.2: Home tab in the 3D Basics workspace

Let's look at the panels in more detail:

- The **Create** panel is the first one from the left. It can be expanded by clicking on the small arrow near the panel name. As shown in the preceding figure, it contains the primitive solid creation commands that will be covered in `Chapter 11`, *Creating Primitive 3D Shapes*. It also contains some 2D-to-3D conversion commands such as **Extrude**, **Loft**, and **Sweep**, which will be covered in `Chapter 12`, *Conversion Between 2D and 3D*. The **Create** panel can be expanded to show a set of additional commands for surfaces and mesh modeling, which will be covered in `Chapter 14`, *Surfaces and Mesh Modeling*.
- The **Edit** panel contains a set of solid editing tools, which will be covered in `Chapter 10`, *Introduction to 3D Modeling*. The panel can be expanded to show additional solid editing tools, as well as surfaces and mesh editing tools that will be covered in `Chapter 13`, *Modifying 3D Objects*.
- The **Modify** panel contains the same modify tools used for 2D drafting. These commands are also usable with 3D objects, in addition to which it contains some 3D variants including **3D Mirror**, **3D Align**, and **3D Array**. These commands will be covered in detail in `Chapter 13`, *Modify 3D Objects*.
- The **Selection** panel contains selection highlight filter commands and the **3D Gizmo**, which will be covered in `Chapter 13`, *Modify 3D Objects*.
- The **Layers & View** panel contains the layer management commands, along with a list of visual styles that will be covered later in this chapter in the *Visual styles* section.

The next tab is called **Visualize**, and contains a collection of commands related to adjusting the viewports and rendering. The following figure shows the different panels contained in the **Visualize** tab:

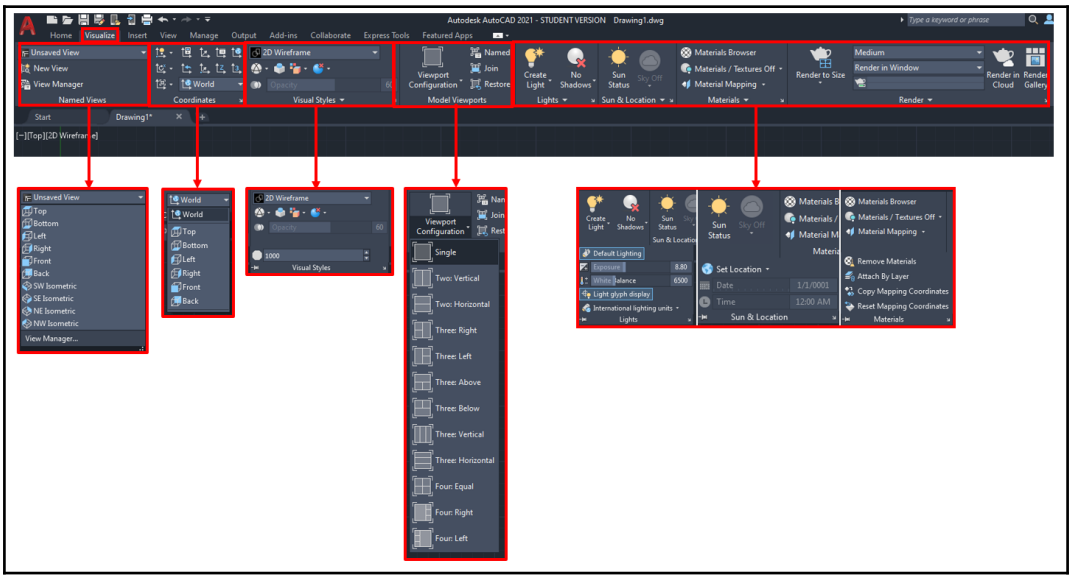


Figure 10.3: Visualize tab in the 3D Basics workspace

Now let's have a deeper look at the commands contained in each panel:

- The **Named Views** panel contains the list of preset views, as well as the view manager, which will be covered later in this chapter in the *Navigation and switching views* section.
- The **Coordinates** panel contains a set of commands to control coordinate systems in the model. It will be covered in detail later in this chapter in the *User Coordinate System (UCS)* section.
- The **Visual Styles** panel contains the list of preset visual styles, as well as visual style controls and visual style manager, which will be covered later in this chapter in the *Visual styles* section.
- The **Model Viewports** panel contains different multi-viewport configurations. It will be covered in detail later in this chapter in the *Configuring multi-viewports* section.
- There is also a group of panels (**Lights**, **Sun & Location**, **Materials**, **Render**) that are dedicated to rendering, which will be covered in `Chapter 16`, *Rendering and Presentation.*

Now that we have explored the 3D Basics workspace, let's take a look at the second workspace offered by AutoCAD for 3D modeling.

Exploring the 3D Modeling workspace

The 3D Modeling workspace is also dedicated to 3D modeling, with commands organized in a different way than the 3D Basics workspace. In the 3D Modeling workspace there is a dedicated tab for each type of 3D object that can be created in AutoCAD; that is, there is a tab for solid objects, a tab for surface modeling, and a tab for mesh modeling with most of the commands repeated in the panels of the **Home** tab, while as shown in the previous section, the 3D Basics workspace has a compact style. All of the commands are available in both of the workspaces and you can use the one you find more intuitive and comfortable to work with. In this section, we will take a quick tour of the workspace, exploring the different tabs and panels. And remember, as shown previously, to switch to the 3D Modeling workspace, click on the workspace switching button and select **3D Modeling.**

The **Home** tab in the 3D Modeling workspace is loaded with panels and commands. It has some commands that are present in the **Drafting & Annotation** workspace, including the **Draw**, **Modify**, **Layers**, and **Groups** panels, along with other panels dedicated to 3D modeling. The following figure shows the panels contained in the **Home** tab:

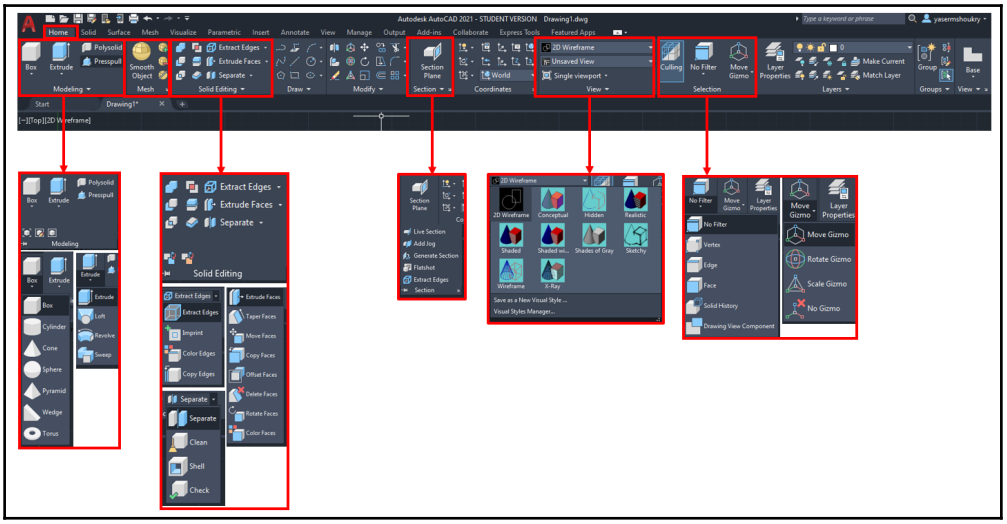


Figure 10.4: Home tab in the 3D Modeling workspace

Now let's have a more detailed look at the buttons contained in each panel:

- The **Modeling** panel contains the set of commands for primitive 3D objects. They can be accessed by expanding the button on the left named **Box**. These commands will be covered in `Chapter 11`, *Creating Primitive 3D Shapes*. In addition to that, it contains the 2D-to-3D conversion commands grouped in the expandable button which is located second from the left side. These commands will be covered in `Chapter 12`, *Conversion Between 2D and 3D*.
- The **Mesh** panel contains the mesh modeling commands, which will be covered in `Chapter 14`, *Surfaces and Mesh Modeling*.
- The **Solid Editing** panel contains the commands dedicated to solid objects editing, which will be covered in `Chapter 13`, *Modify 3D Objects*.
- The **Section** panel contains the section creation and editing tools, which will be covered in `Chapter 13`, *Modify 3D Objects*.
- The **View** panel contains the preset views list, as well as the visual styles, both of which will be covered later in this chapter in *Navigation and switching views* and *Adjusting visual styles* sections .
- The **Selection** panel contains the selection highlight filter commands and the 3D Gizmo, both of which will be covered in `Chapter 13`, *Modify 3D Objects*.

There are three types of 3D objects in AutoCAD: solids, surfaces, and mesh objects. Each type has a dedicated tab in the 3D Modeling workspace, starting with **Solid** tab, which is shown in the following figure:

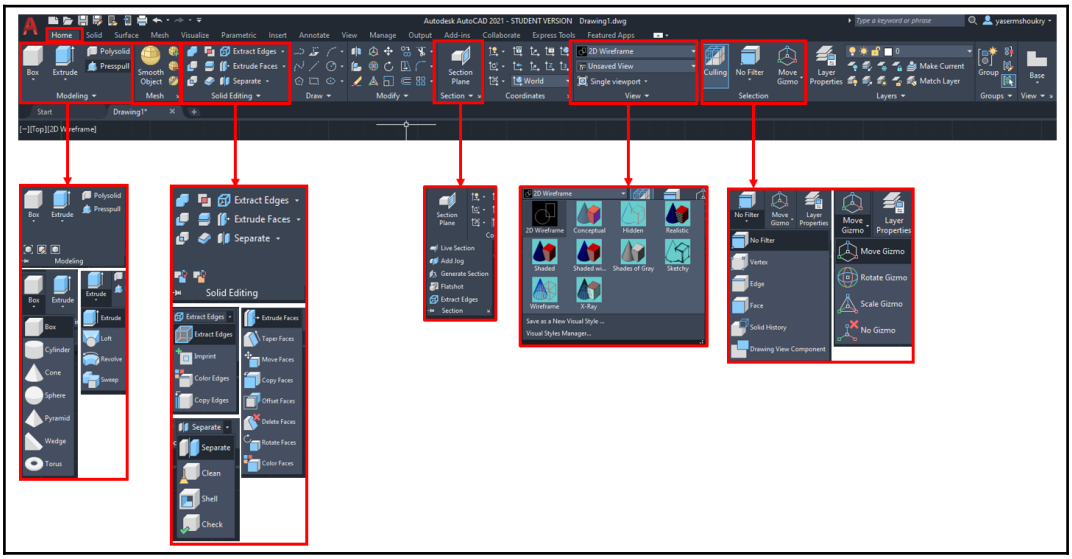


Figure 10.5: Solid tab in the 3D Modeling workspace

Now let's take a deeper look on the panels of the **Solid** tab:

- The **Primitive** panel contains all the commands for creating primitive 3D solid objects (boxes, spheres, pyramids, and so on), which will be covered later in `Chapter 11`, *Creating Primitive 3D Shapes*.
- The **Solid** panel contains other solid creation commands, which will be covered in `Chapter 12`, *Conversion Between 2D and 3D*.
- The **Boolean** and **Solid Editing** panels contain all the solid editing commands. Some of these buttons are expandable. The solid editing commands will be covered in `Chapter 13`, *Modify 3D Objects*.
- The **Section** panel appears again in this tab. It contains the section creation and editing tools, which will be covered in `Chapter 13`, *Modify 3D Objects*.
- The **Selection** panel is also repeated in this tab. It contains the selection highlight filter commands and the 3D Gizmo, which will be covered in `Chapter 13`, *Modify 3D Objects*.

The next tab is called **Surface**. This tab houses the commands for creating and editing surfaces. The panels of the **Surface** tab are shown in the following figure:

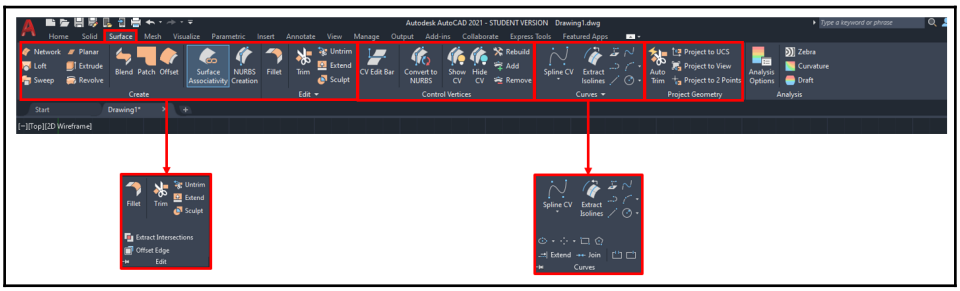


Figure 10.6: Surface tab in the 3D Modeling workspace

Let's take a closer look at the tool buttons contained in each panel of the **Surface** tab:

- The **Create** and **Edit** panels contain the tools for creating and modifying surface objects. The surface object creation and edit commands will be covered in `Chapter 14`, *Surfaces and Mesh Modeling*.
- The **Control Vertices** panel contains the commands related to the usage of control vertices.
- The **Curves** panel is a comprehensive panel to all the 3D and 2D curve creation and editing commands, and is also expandable.
- The **Project Geometry** panel contains the projection-related commands.

The next tab is called **Mesh**, where you can find all the tools and commands related to working with mesh objects. The following figure shows the panels contained in the **Mesh** tab:

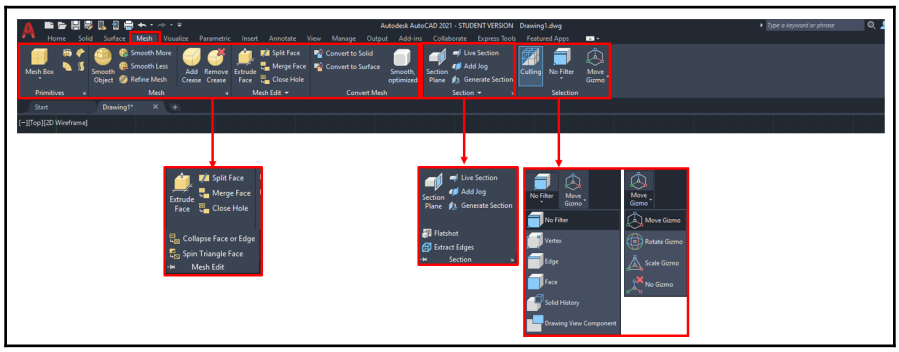


Figure 10.7: Mesh tab in the 3D Modeling workspace

Now let's take a deeper look at the contents of each panel:

- The **Mesh** objects creation and modification panels include **Primitives**, **Mesh**, **Mesh Edit**, and **Convert Mesh**. The **Mesh Edit** tab is expandable. Mesh modeling will be covered in `Chapter 14`, *Surfaces and Mesh Modeling*.
- The **Section** panel appears again in this tab. It contains the section creation and editing tools, which will be covered in `Chapter 13`, *Modify 3D Objects*.
- The **Selection** panel is also repeated in this tab. It contains selection highlight filter commands and the 3D Gizmo, which will be covered in `Chapter 13`, *Modify 3D Objects*.

Now that we have looked at the tabs related to the creation of different types of 3D objects, the next tab, which is called **Visualize**, contains a collection of commands related to adjusting the viewports and rendering. The panels contained in the **Visualize** tab are shown in the following figure:

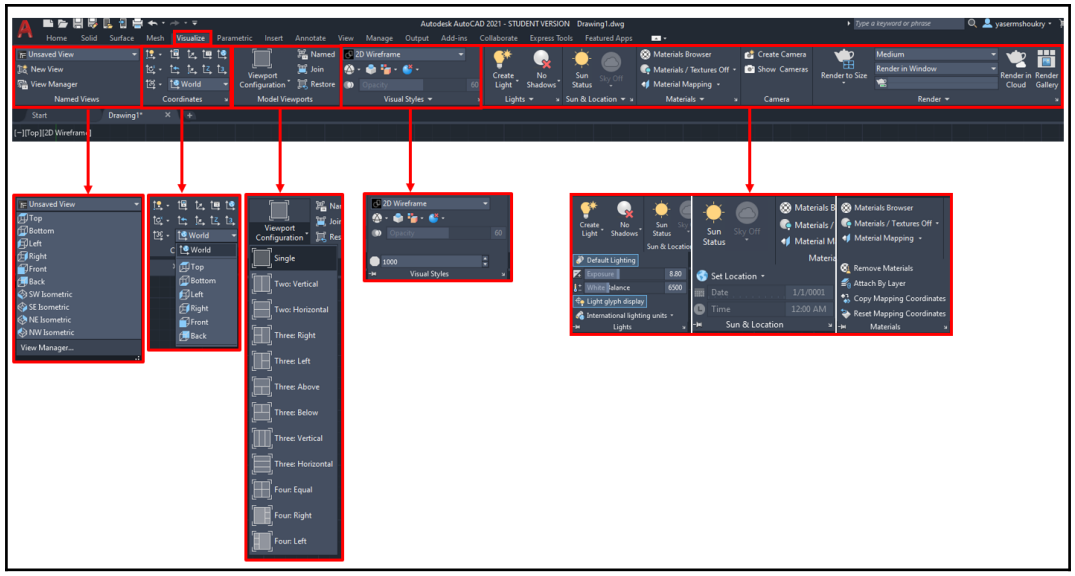


Figure 10.8: Visualize tab in the 3D Modeling workspace

Now let's have a look at the tool buttons contained in each panel:

- The **Named Views** panel contains the list of preset views, as well as the view manager, which will be covered later in this chapter in the *Navigation and switching views* section.
- The **Coordinates** panel contains a set of commands to control coordinate systems in the model. It will be covered in detail later in this chapter in the *User Coordinate System (UCS)* section.
- The **Model Viewports** panel contains different multi-viewport configurations. It will be covered in detail later in this chapter in the *Multi-viewport configurations* section.
- The **Visual Styles** panel contains the list of preset visual styles, as well as visual style controls and visual style manager, which will be covered later in this chapter in the *Visual styles* section.
- A group of panels (**Lights**, **Sun & Location**, **Materials**, and **Render**) that are dedicated to rendering will be covered in `Chapter 16`, *Rendering and Presentation*.

We have now finished introducing the different workspaces and have had a quick tour of the 3D Basics and 3D Modeling workspaces. Now we will look at how to navigate the drawing area and switch between different views.

Navigation and switching views

In 2D drafting, the whole drawing exists on one plane, so the drawing area view was normal to that plane and the only navigation tools needed were panning and zooming. In 3D modeling, you will need to create details that may exist on any angle of the three main axes, and even creating a single part or detail of a model may require switching back and forth between different viewing angles. Therefore, an understanding of navigation and switching between different views is essential for efficient 3D modeling, so in this section, the basic navigation and view switching tools will be presented.

Basic navigation

The navigation tools used in the normal 2D drafting mode are still usable when in 3D modeling. Starting with zooming, the mouse wheel can be used to zoom in and zoom out, as well as the zoom and zoom extend commands that work exactly the same as in the 2D drafting mode. For more details on these commands, please refer to `Chapter 1`, *Basic Drawing Tools* of this book.

To pan the view, press and hold the middle mouse button and the mouse cursor will turn into a black hand, as shown in the following figure. Move the mouse to pan the view then release the button, again just like in the 2D drafting mode.

The middle mouse button can also be used to rotate the view. This can be done by holding down *Shift*, and then holding down the middle mouse button. The cursor icon will change to the rotate icon, as shown in the following figure. Move the mouse to rotate the view as desired then release both buttons:

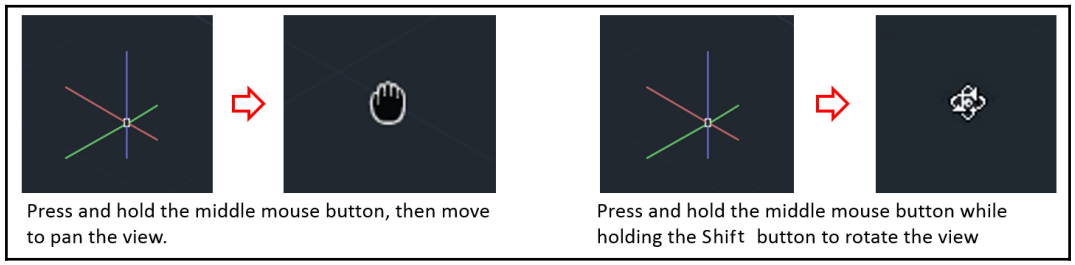


Figure 10.9: Using the middle mouse button to rotate and pan the view

In addition to the middle mouse button, AutoCAD also features a navigation tool called the ViewCube, which will be discussed in the next sub-section.

The ViewCube

The ViewCube can be found at the top-right corner of the drawing area. It can be used to easily switch between different standard plan views, the standard isometric views, or to rotate the view to a general angle as desired.

The six standard plan views (**Top**, **Bottom**, **Right**, **Left**, **Front**, and **Back**) are represented on the six faces of ViewCube, as shown in the following figure:

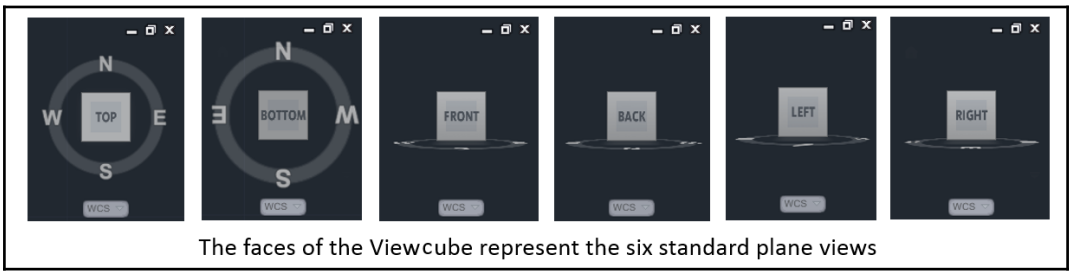


Figure 10.10: The six standard views

To switch to any of the standard plan views, click on the face with the desired view name. In addition to that, the standard plan views can be rotated by 90 degrees using the small arrows that appear when you move the cursor near the ViewCube as shown in the following figure:

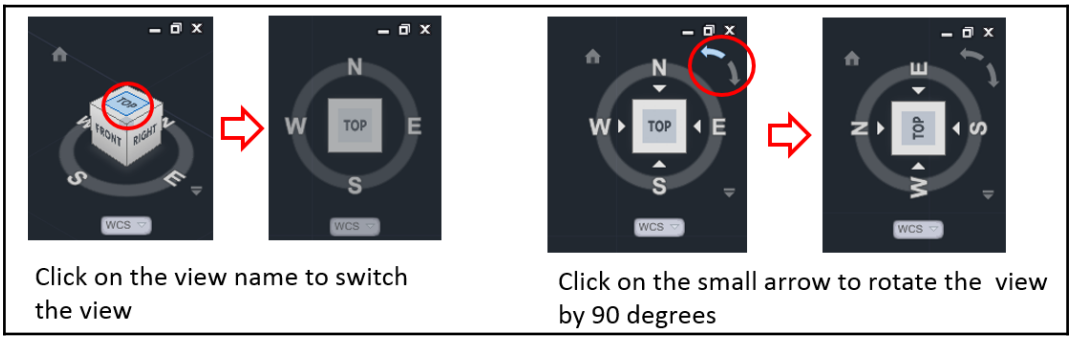


Figure 10.11: Switching to one of the standard views and rotating the view

The 8 corners and the 12 edges of the ViewCube represent a total of 20 standard isometric views. To switch to any of them, just click on the corner or the edge representing the desired view, as shown in the following figure:

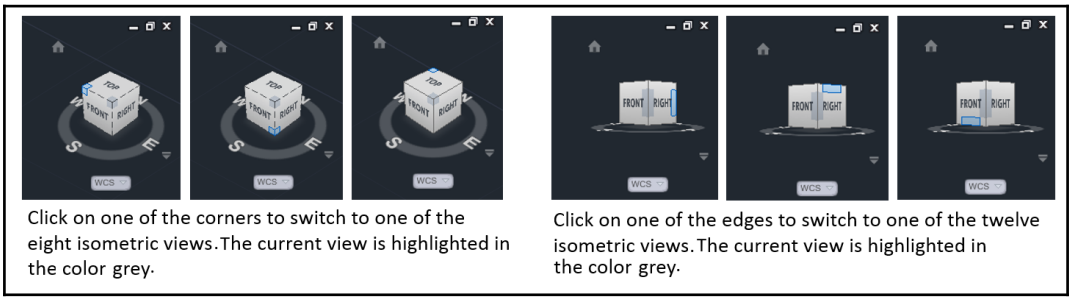


Figure 10.12: Switching to the standard isometric views

In addition to the standard views, the Viewcube can be used to set the angle to a general or nonstandard view. You can click and drag the ViewCube to generally rotate the view as desired.

The compass in the following figure of the ViewCube shows the four main directions. By default, the north, south, east, and west directions coincide with the **back**, **front**, **left**, and **right** views respectively. To switch to the standard plan view of any of the four main directions, click on the desired cardinal direction letter on the compass in the ViewCube. The compass ring can also be clicked and dragged to rotate the view with a general angle, as shown in the following figure:



Figure 10.13: Using the compass

Now that we have learned how to rotate the view using the middle mouse button and the ViewCube, let's take a look at another alternative method to switch views: the preset views list.

Preset views

An alternative method to switch between standard views is to use the preset views list, which can be found in multiple locations. Starting with the 3D Basics workspace, it located in the **Layers & View** panel in the **Home** tab, as shown in the following figure:

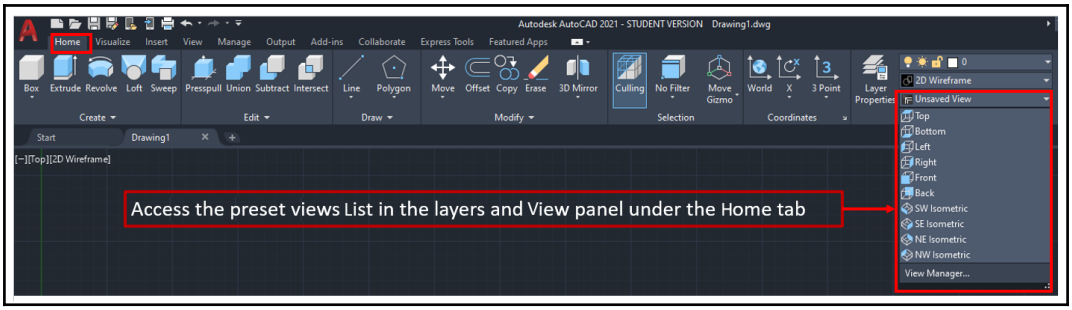


Figure 10.14: Location of the preset view list in 3D Basics workspace

The same list can also be found in the 3D Basics workspace on the **Named Views** panel in the **View** tab. And if you are on the 3D Modeling workspace, you can find the preset views list at three different locations. The first location is in the **View** panel in the **Home** tab, as shown in the following figure:

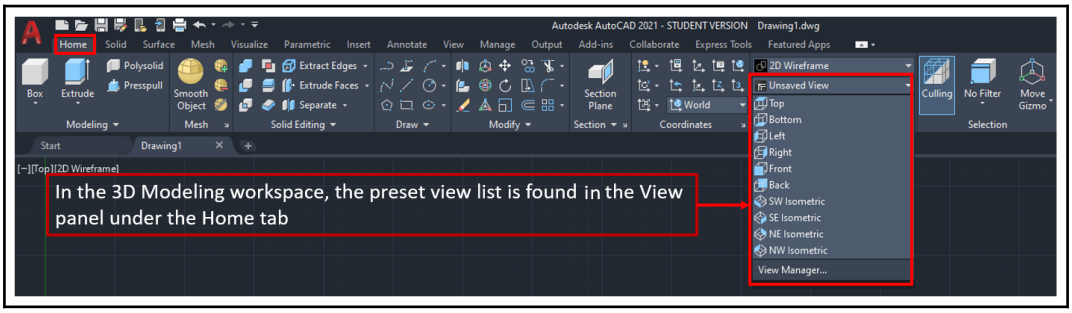


Figure 10.15: Location of the preset view list in 3D Modeling workspace

The last two locations of the preset view list are in the **Named Views** panel in the **Visualize** tab, or in the **Named Views** panel in the **View** tab.

The preset view list contains the six standard plan views (**Top**, **Bottom**, **Front**, **Back**, **Left**, and **Right**), in addition to which you have four standard isometric views named using the cardinal direction letters (**NW**, **NE**, **SW**, and **SE**).

As you can see, the preset views list is an alternative method to quickly switch between the standard views that are already found in the ViewCube, as discussed in the previous sub-section. The additional option that can't be implemented using the ViewCube is the creation and use of custom view angles, which we will look at now.

Creating a custom named view

During the creation of 3D models, you may come across a part of a detail in your model where the best view angle to work from is a general angle that is not one of the standard views available in the ViewCube or the list of preset views. To make things worse, you may also need to switch back and forth between two different general angles during the model creation, which would be a hideous process of adjusting the view angle using the middle mouse button or dragging the ViewCube every time you want to change the view. To avoid this, you can add your own general angle view that is the best for your model as an entry to the views list, and just click on it for a quick view switch. An easy way to do this is to set the desired view in the drawing area, and after that, do as shown in the following step-by-step example:

1. Click on the **View Manager** button at the end of the preset views list.
2. This will open the **View Manager** window. Click on the **New** button.
3. Type a name for the view (for example, `The New View`), make sure that the current display option is selected, and press **OK**.
4. The created view will be added to the preset view list.

The following figure shows an illustration of the previous steps:

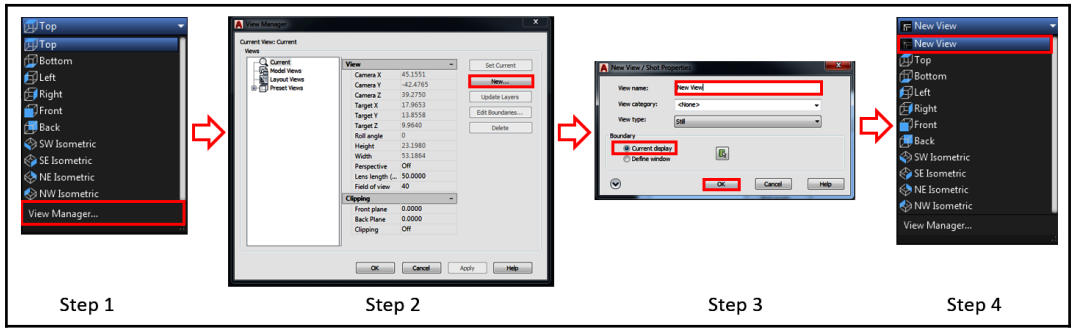


Figure 10.16: Steps for creating a custom view

Now that we have learned how to efficiently rotate the view, switch between standard views, and how to add custom views, let's take a look at how to control the appearance of the objects by adjusting visual styles.

Adjusting visual styles

3D objects can be displayed in various ways or styles in AutoCAD with varying degrees of details and complexity, ranging from simple outline edges up to shaded surfaces with realistic materials. A number of preset visual styles are available to choose from, as well as the ability to tweak different aspects in each visual style or even create a new one.

Preset visual styles

There are a number of preset visual styles that are available to choose from. You can access this list of visual styles by clicking on the visual style button located in the **Layers & View** panel in the **Home** tab, or in the **Visual Styles** panel in the **Visualize** tab in the **3D Basics** workspace, as shown in the following figure:

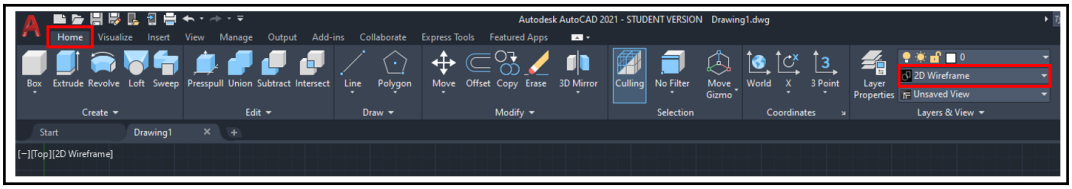


Figure 10.17: Location of the preset visual styles list in the 3D Basics workspace

If you are using the **3D Modeling** workspace, you can find the visual style button in the **View** panel in the **Home** tab, or in the **Visual Styles** panel in the **Visualize** tab, as shown in the following figure:

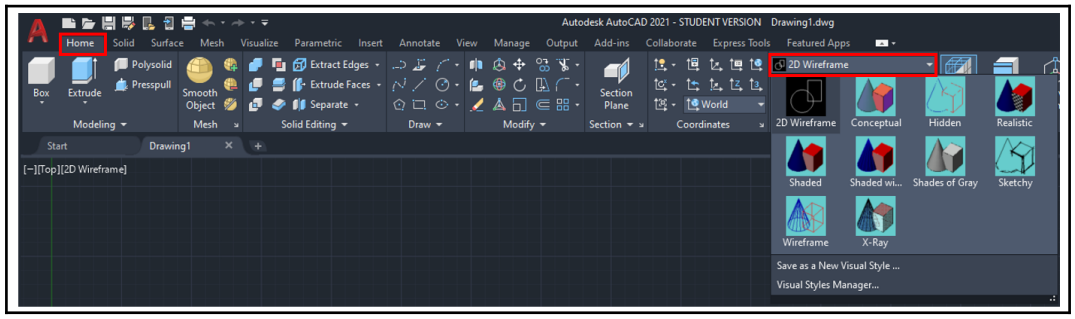


Figure 10.18: Location of the preset visual styles list in the 3D Modeling workspace

The preset visual styles list has 10 different options, which vary in the amount of details and demand for computing resources. You can choose the one that gives you the best results in terms of model visibility and reliable performance, or the one that best suits your presentation style. The different available visual styles are as follows:

- **2D Wireframe** is the simplest – it displays the 3D objects using the edges only with no surfaces, and is the least demanding in terms of performance. A preview of the object's appearance using the **2D Wireframe** style is shown in the preceding figure.
- **Conceptual**, which uses Gooch face style and smooth shading, is not a realistic-looking style but it provides good model visibility. A preview of the object's appearance using the **Conceptual** style is shown in the preceding figure.
- **Hidden** is similar to **2D visual style** but only shows the front-facing edges. A preview of the object's appearance using the **Hidden** style is shown in the preceding figure.
- **Realistic** shows the surfaces of the object using materials, textures, and smooth shading. A preview of the object's appearance using the **Realistic** style is shown in the preceding figure.
- **Shaded** shows the surfaces of the object. It uses smooth shading but without applying materials, which makes it less demanding than **Realistic**. A preview of the object's appearance using the **Shaded** style is shown in the preceding figure.
- **Shaded with edges** displays the surfaces of the objects and highlights the edges, improving the model visibility. It also uses smooth shading. A preview of the object's appearance using the **Shaded with edges** style is shown in the preceding figure.

- **Shades of Gray** is similar to the **Shaded** visual style, but it uses monochromatic shades of gray. A preview of the object's appearance using the **Shades of Gray** style is shown in the preceding figure.
- **Sketchy** is similar to the **Hidden** visual style but has sketch-like edges. This visual style can be used for concept presentations. A preview of the object's appearance using the **Sketchy** style is shown in the preceding figure.
- The **Wireframe** visual style is similar to the **2D Wireframe** style, the difference being that it doesn't regenerate the model when changing the view, which makes it even less demanding in terms of performance. A preview of the object's appearance in the **Wireframe** style is shown in the preceding figure.
- **X-Ray** is very similar to the **Shades of Gray** style, adding transparency to the surfaces. A preview of the object's appearance in **X-Ray** style is shown in the preceding figure.

Now that we have seen the available preset visual styles, let's have a look at how to customize their settings and how to create a new visual style.

Adjusting visual style settings

In addition to the preset visual styles, different aspects of an object's visuals can be controlled using the tools located in the **Visual styles** panel in the **Visualize** tab. Let's start with edges, which have three different settings:

- **No Edges**: Will set them to be completely hidden
- **Isolines**: Shows only iso-lines
- **Facet Edges**: Shows facet edges

You can choose not to display the hidden lines, or only show the front-facing edges and surfaces using the **Hide** button. Also, there are four different options regarding face colors:

- **Normal**: This does not apply any modifications to the colors of objects.
- **Monochrome**: This uses only one color to display all objects. You can set which color to use with the `VSMONOCOLOR` system variable.
- **Tint**: This uses different settings for the hue and saturation of one color to display all objects. You can choose the color using the `VSMONOCOLOR` system variable.
- **Desaturate**: Colors are softened by reducing saturation by 30%.

You can see all of these settings in the following figure:

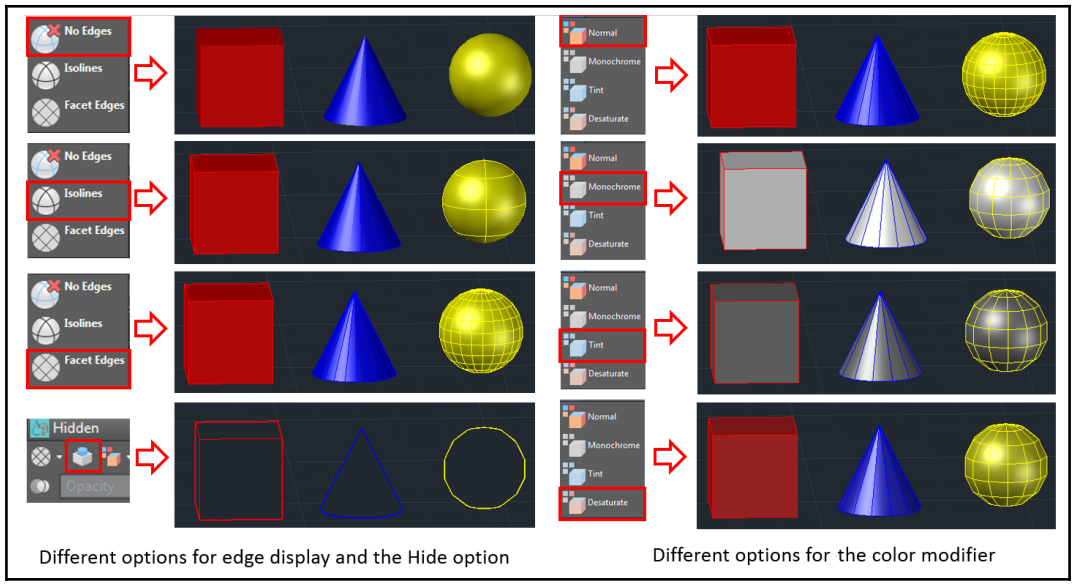


Figure 10.19: Modifying the visual style

Besides face color, you can also control the transparency of objects by clicking on the **Transparency** button, then dragging the slider to adjust the opacity percentage. In addition to that, you have three face styles to choose from:

- **No Face Style**
- **Realistic Face Style**
- **Warm-Cool Face Style**

The three face styles can be seen in the following figure:

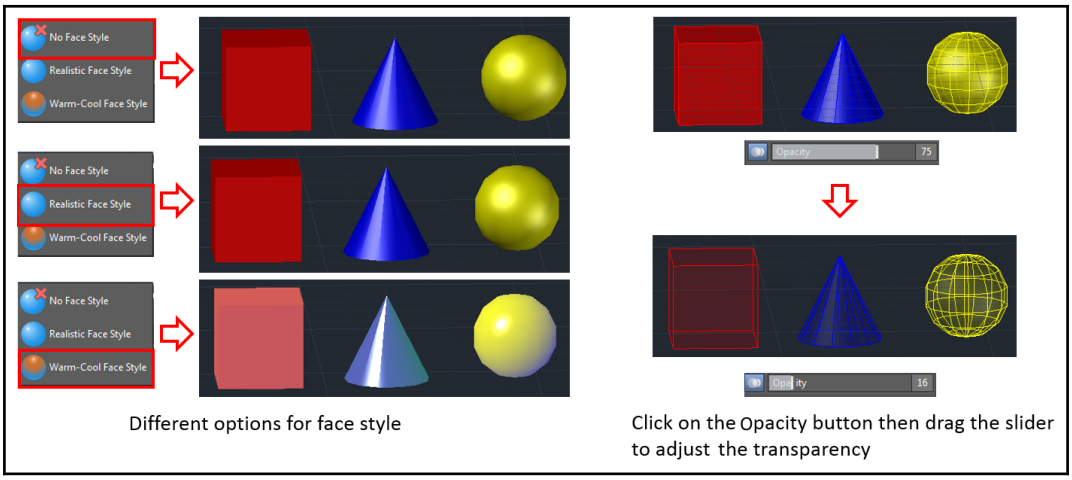


Figure 10.20: Modifying the visual style, part 2

When the settings of the visual style are changed, the visual style name will change to **Current**. You can create a new visual style to save the changes using the **Save as New Visual Style** button as shown in the following figure:

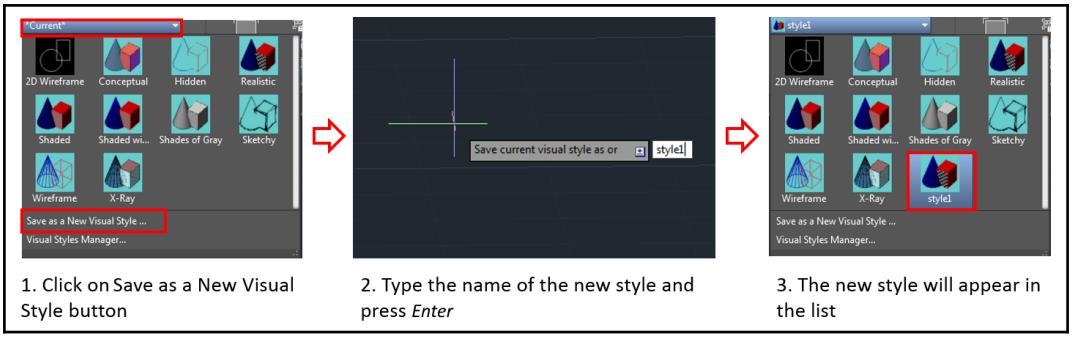


Figure 10.21: Creating a custom visual style

Now that we have learned how to tweak the visual style for the best visibility and performance, let's take a look at another important feature: the use of multi-viewport configurations.

Configuring multiple viewports

When working in 3D, it is a good practice to use multiple viewports. In AutoCAD you can divide the drawing area into multiple viewports with different configurations. An easy and quick way to switch to the multi-viewport is to click on the **Single viewport** button located in the **View** panel in the **Home** tab in the **3D Modeling** workspace, as shown in the following figure. The button will expand to a small list with two entries, the second of which is the **Multiple viewports** option:

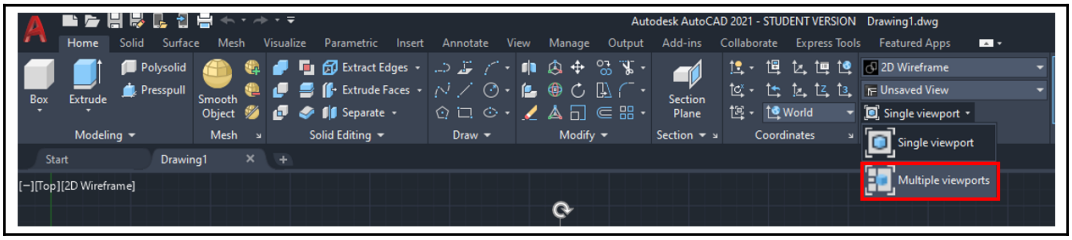


Figure 10.22: Location of Multiple viewports tool button in the 3D Modeling workspace

When clicking on the **Multiple viewports** button, the drawing area will automatically be divided into four equal viewports. Each viewport can have a different view angle and visual style, in addition the boundaries can be clicked and dragged to resize the viewports, as shown in the following figure:

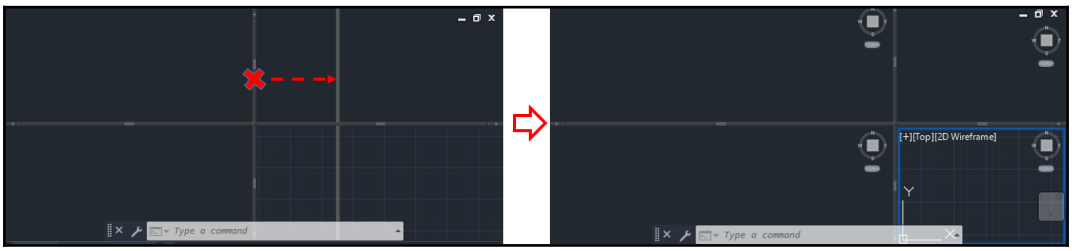


Figure 10.23: Resizing the viewports' boundaries

In the top-left corner of every viewport you will find quick information and controls for different aspects of each viewport :

- On the left, you will find a plus sign, which you can click to expand a menu where you can maximize the viewport for better visibility and restore back the multi-viewport configuration. You can also control the visibility of the ViewCube, the navigation bar, and the steering wheels, along with changing the viewport configuration.
- In the middle, you will find the name of the preset view used. If you are not using a preset view, it will display **Custom view**. You can click it and switch to any of the other preset or user-created views. You can also switch between parallel and perspective orientations and access the view manager.
- On the right, the name of the visual style is displayed. You can click this to switch to a different visual style, and can also open the Visual Style manager.

The multi-viewport configuration in AutoCAD is not limited to just four equal viewports; there are other multi-viewport configurations to choose from, and they can also be modified. To access the other viewport configurations, you can click on the **Viewport Configuration** button located in the **Model Viewports** panel in the **Visualize** tab in the **3D Basics** workspace. The same button is available in the **Model Viewports** panel in the **Visualize** tab in the **3D Modeling** workspace. This will expand a list of the available multi-viewport configurations, ranging from one up to four viewports with different configurations. The **Viewport Configuration** button's location is shown in the following figure:

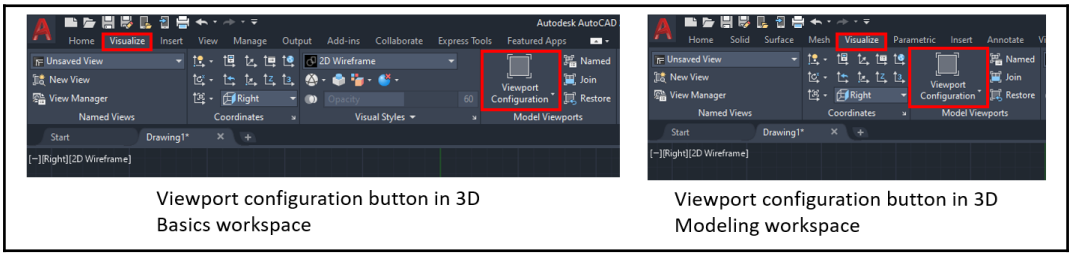


Figure 10.24: Viewport configuration button locations

Next to the **Viewport Configuration** button you will find the **Join** button, which can be used to modify the viewport configuration as follows:

1. Click on the **Join** button
2. Choose two adjacent viewports to be joined
3. The viewport configuration will be modified

The following figure is an illustration of the previous steps:

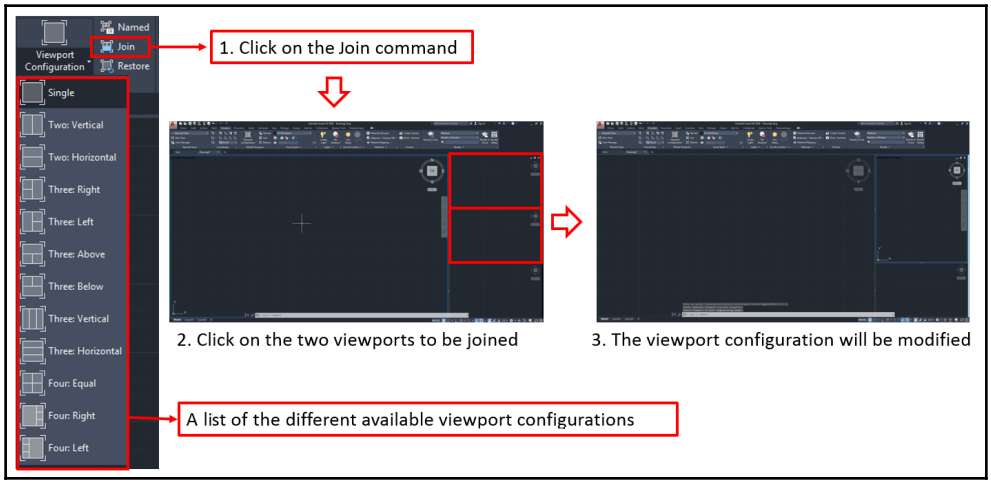


Figure 10.25: Customizing the viewport configuration

Now that we have learned how to set up the visual style and viewport configurations, and how to effectively navigate and rotate the view angle, let's have a look at the coordinate system in AutoCAD.

User Coordinate System (UCS)

Many of the drafting commands in AutoCAD will only work in the horizontal plane, or in other words in the **XY** plane of the coordinates system. Even some parts of the 3D solid creation commands also have the same restriction (such as the base of the object when creating a pyramid or a wedge). While in the process of creating 3D models, you may need to draw a 2D shape on an elevated XY plane, a vertical plane, or even on the surface of another object. AutoCAD offers a set of tools that allows you to manipulate the coordinate system so you can align the **XY** plane as desired and hence the restrictions are eliminated.

In AutoCAD, there is an immovable reference coordinate system called the **World Coordinate System** (**WCS**). In addition to that, there is a **User Coordinate System** (**UCS**) that can be moved or rotated as desired. In this section, the tools related to manipulating the coordinate system will be discussed starting with the options for the UCS icon appearance, after which we will have a look at how to manipulate the UCS. At the end of the section, the Dynamic UCS function will be presented.

Appearance of the UCS icon

AutoCAD allows for different preferences in the appearance and the position of the **UCS** icon. The position can be controlled using the button at the top-left corner of the **Coordinates** panel (with the little lamp icon). If you click on it you will find three different options.

As shown in the following figure, the first option will lock the **UCS** icon at the origin point of the drawing. The second option will always show the **UCS** icon at the bottom-left corner of the drawing area, and the last option is to hide the **UCS** icon. All of this is shown in the following figure:

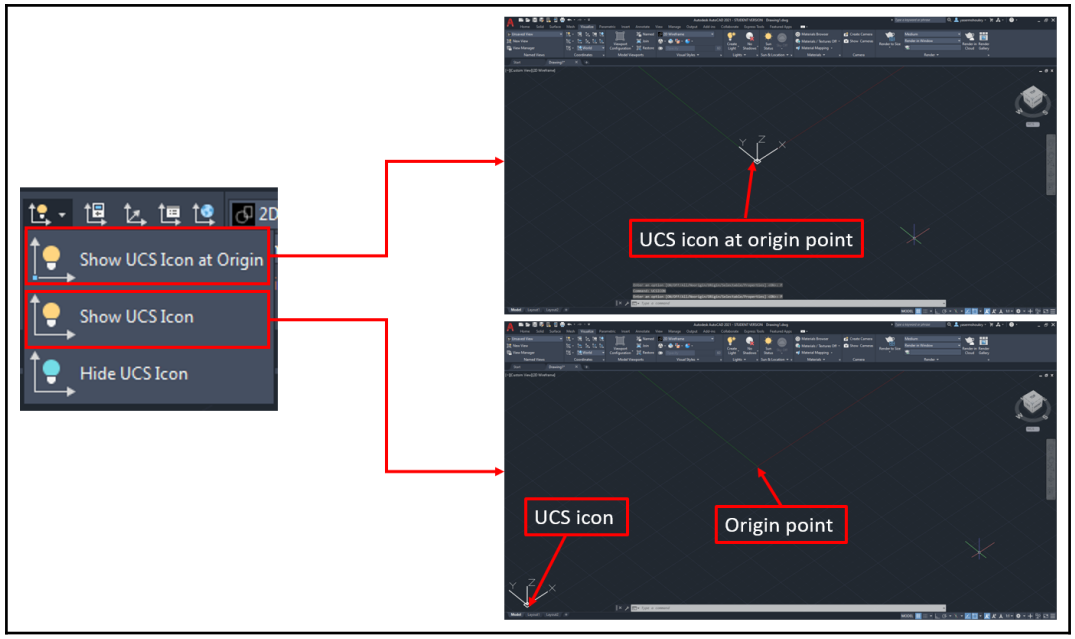


Figure 10.26: Different options for displaying the UCS icon

In addition to controlling the position, the appearance of the **UCS** icon can also be adjusted using the **UCS** icon properties button found in the **Coordinates** panel. You can open the **UCS** icon window, which contains all the different options for controlling the **UCS** icon appearance, as shown in the following figure:

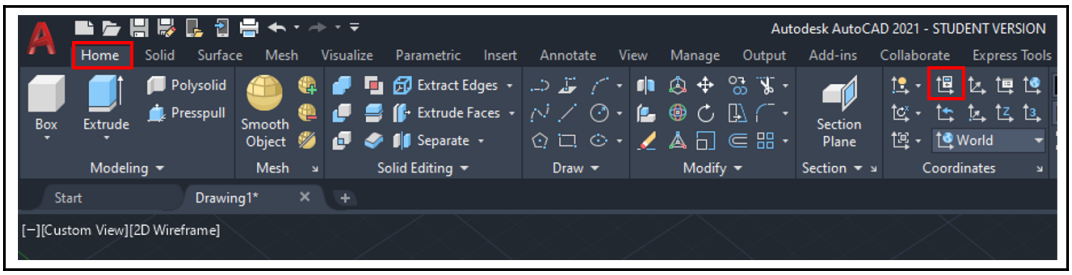


Figure 10.27: UCS icon properties tool's button location in the 3D Modeling workspace

As shown in the following figure, the **UCS** icon window is divided into four sections:

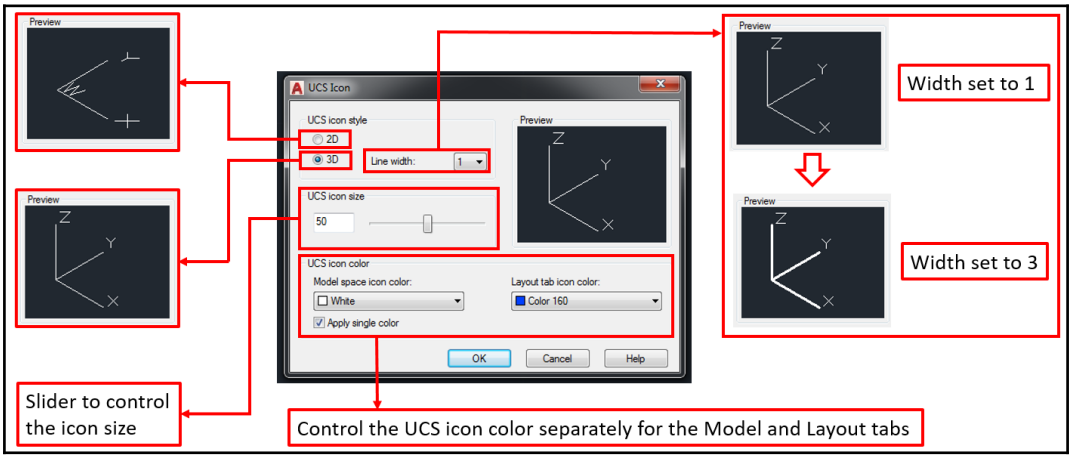


Figure 10.28: Different options for controlling UCS icon appearance

At the top right, you will find a preview for the **UCS** icon according to the currently selected settings.

At the top left, you will find the **UCS icon style** section. Here, you can choose between two different UCS icon styles, as shown in the preceding figure – one is 2D and the other is a 3D icon. In addition to that, you can also control the **Line width**, as shown in the preceding figure, with settings ranging from **1** for the thinnest and **3** for the maximum width.

You can also control the **UCS icon size** in the drawing area using the slider on the **UCS icon size** section in the middle of the window, and finally, at the bottom you will find the **UCS icon color** section, where you can control the color for both the **Model** tab and the **Layout** tabs.

Now that we have seen how to control the UCS appearance, let's take a look at how to manipulate its position and orientation.

Rotating the UCS

As discussed at the beginning of the section, during the modeling process, you might need to draw a 2D shape (that will be extruded or swept, for example) on a generally inclined **XY** plane. Any transformation can be broken down to rotation and translation motions, so in this section and the next one, we will discuss how to rotate and translate the UCS.

You can rotate the UCS around any of the three main axes by using the UCS rotation buttons found in the **Coordinates** panel. The following figure shows the location of the buttons:

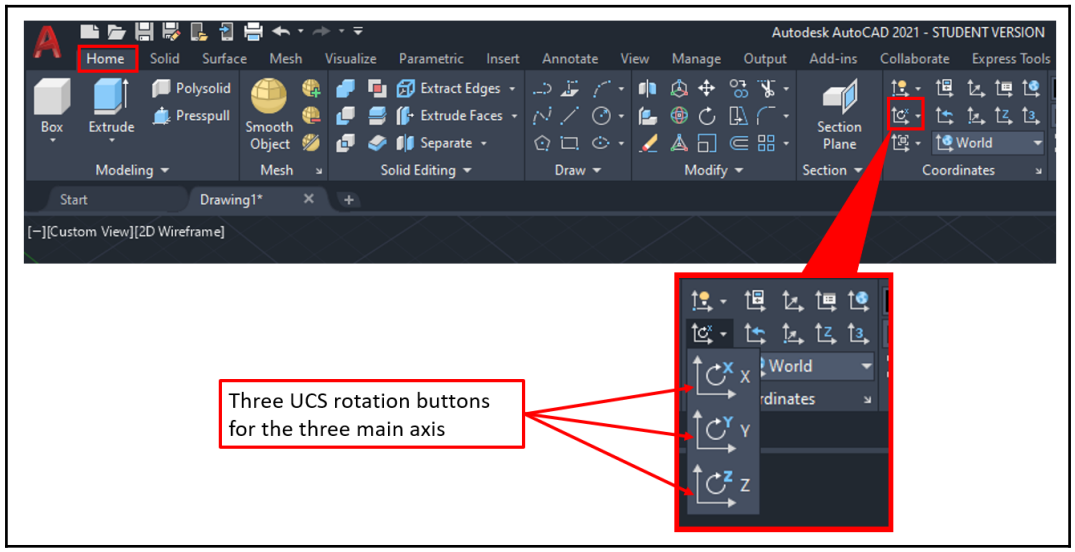


Figure 10.29: UCS rotation tools

To rotate the UCS, simply click on the button with the desired rotation axis as shown in the following step-by-step example:

1. Click on the desired UCS axis rotation button (for example, rotate around the *z* axis).
2. Use mouse movement to specify the rotation angle; alternatively, you can type the desired rotation angle using the keyboard then press *Enter*.
3. When you arrive at the desired angle click again. The command will terminate and the UCS will be rotated as shown in the following figure.

The following figure shows an illustration of the previous steps:

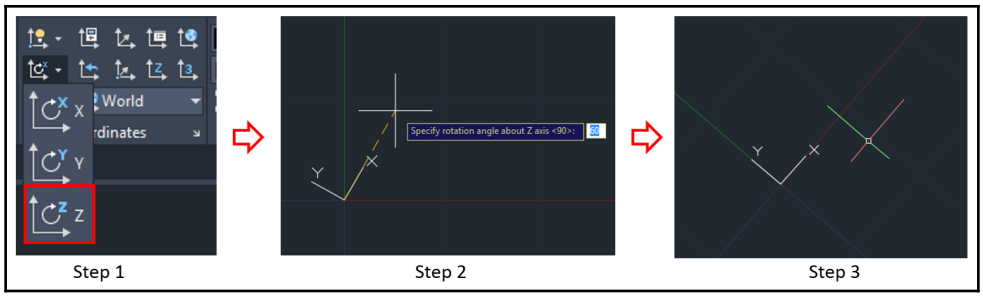


Figure 10.30: Steps for rotating the UCS

Now that we have seen how to rotate the UCS, let's take a look at how to translate it.

Translating the UCS

The UCS can be translated (meaning to change the origin point) to any other location as desired using the **Origin** button found in the **Coordinates** panel, as shown in the following figure:

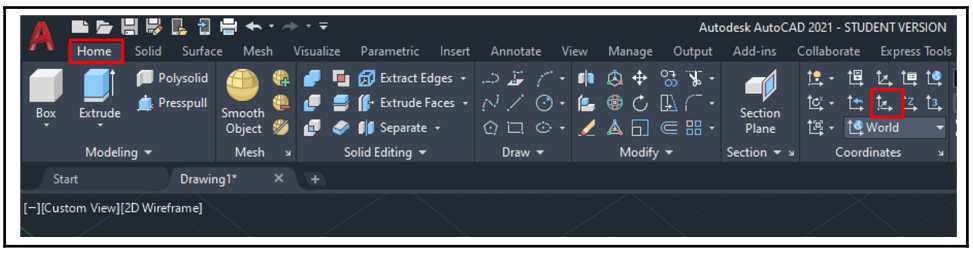


Figure 10.31: UCS origin tool button location in the 3D Modeling workspace

The origin of the UCS can be modified as shown in the following step-by-step example:

1. Start the **UCS Origin** command.
2. Use the mouse movement to specify the new origin location, then click to finish. Alternatively, you can type the new origin coordinates then press *Enter*, as shown in the following figure:

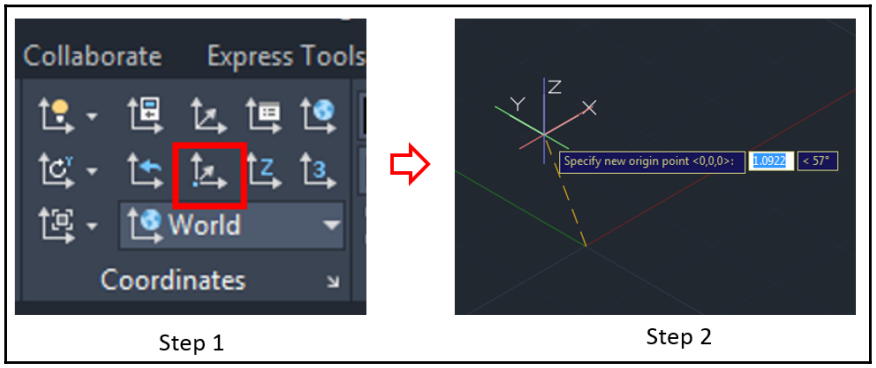


Figure 10.32: Changing the UCS origin location

Now that we've learned how to generally manipulate the position and the orientation of the UCS, let's take a look at an alternative method that aligns the UCS with objects in the model.

Aligning the UCS with objects

In many cases, you may need to place a 2D object to be extruded on top of another existing object. To do this, you can move and rotate the UCS as shown in the previous sub-sections, or you can just align it with the desired object in one step.

You can do this by using the button called **Object**, found in the **Coordinates** panel as shown in the following figure:

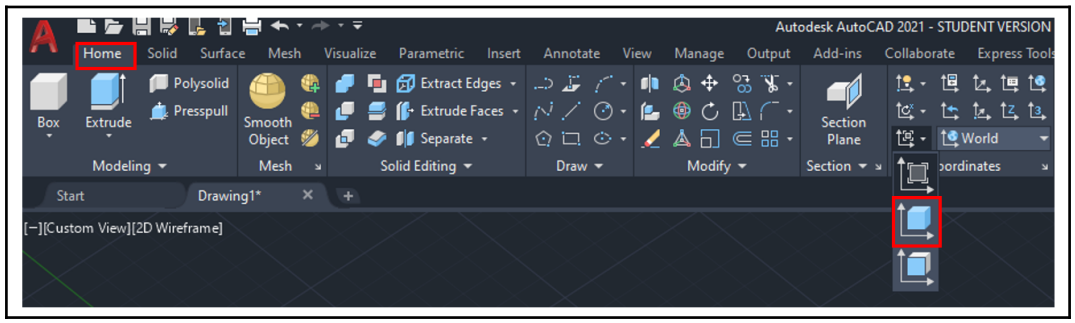


Figure 10.33: Object tool button location in the 3D Modeling workspace

To align the UCS you simply click on the **Object** button, then select the object you want to align the UCS to. This command can be used, for example, to align the UCS to a corner of a cube, to the tip of a cone, or to the center point of a cylinder face. In the following figure are some examples of using the command to align the UCS to different objects:

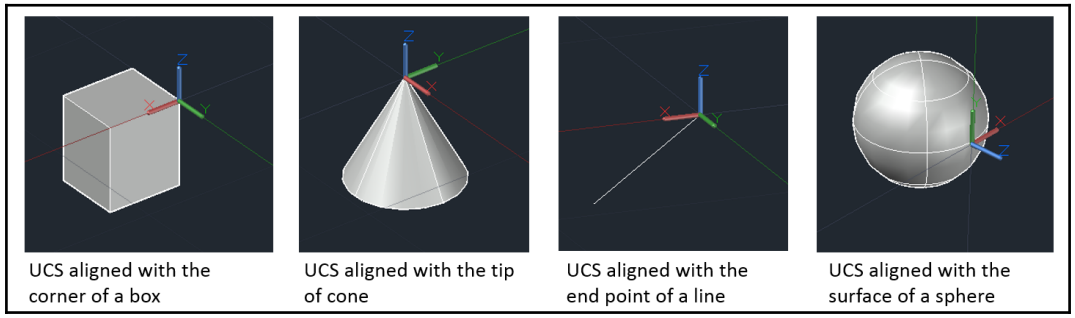


Figure 10.34: Examples for using the Object tool

In addition to the **Object** command, there is also command called **Face** found in the **Coordinates** panel, as shown in the following figure. This command works in a similar way to the **Object** command, the difference being that the **Face** command is limited to aligning the UCS to only faces of 3D objects:

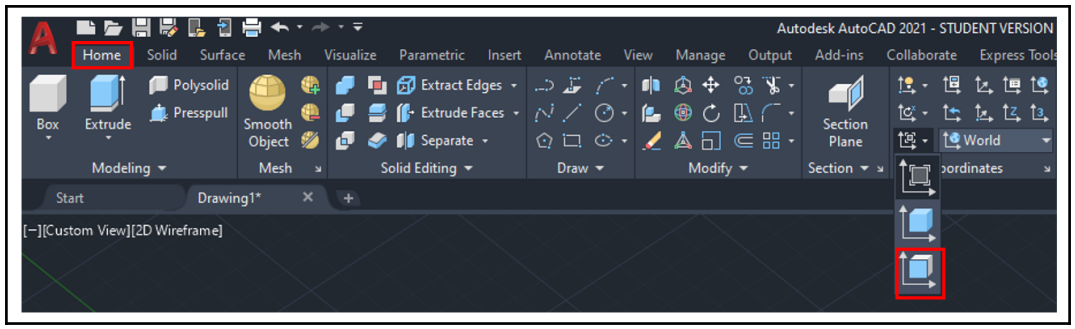


Figure 10.35: Face tool button location in the 3D Modeling workspace

Now that we have learned how to generally manipulate the UCS, and how to align it with existing objects in the model, let's have a look at a different approach to define the UCS, and that is by defining points as shown in the next sub-section.

Creating a UCS by specifying points

The UCS can also be defined by specifying the locations of three points defining the **XY** plane. This can be done using the **3 Point** button found in the **Coordinates** panel (the button with the number **3** written over the axis). You simply click on the button then specify the locations of the following using the mouse:

- The origin point
- A point on the *x* axis
- A point on the *y* axis

These three points will define the **XY** plane of the UCS, as shown in the following figure.

In addition to the **3 Point** option, you can also create the UCS by specifying a two-point vector representing the *z* **axis**. The first of these two points will be the origin point of the new UCS. The tool button for using this option is called *Z* **axis Vector**, and is located in the **Coordinates** panel (the button with the small **z** letter) as shown in the following figure.

And finally, you can use the current view angle as the **XY** plane of the UCS using the **View** button in the **Coordinates** panel, as shown in the following figure:

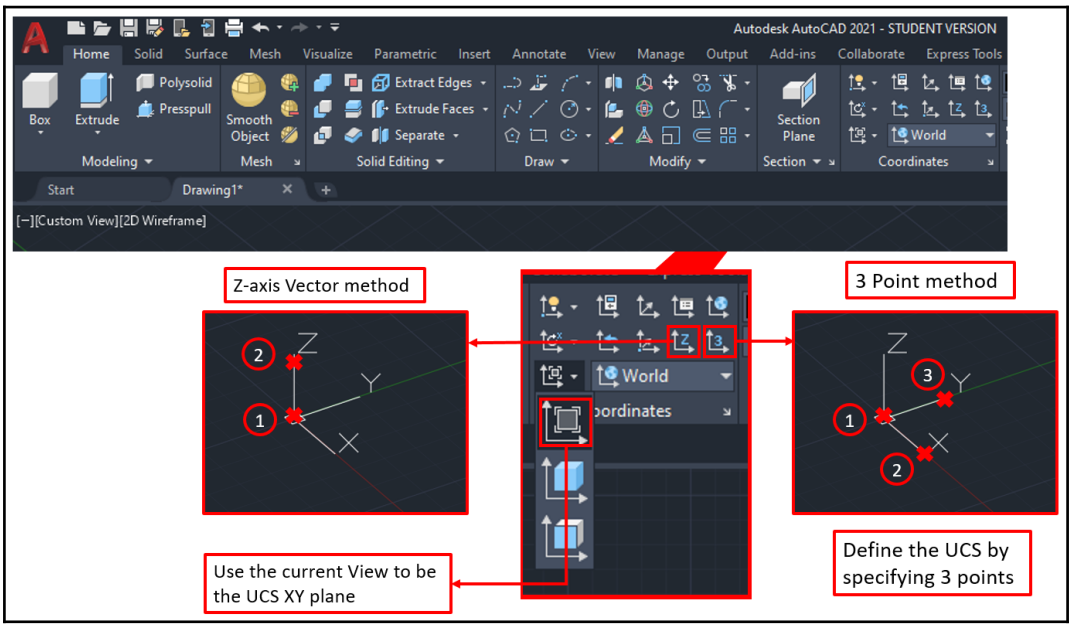


Figure 10.36: Z-axis Vector and 3 Point UCS alignment tools

We have learned the different alternative methods to manipulate the UCS as needed. Now, let's see how to save the manipulated UCS.

Saving the modified UCS

When starting a new drawing file, a reference coordinate system is set for the file, which is called the WCS. When you manipulate the coordinate system, a new UCS is temporarily created and is called **unnamed**. You can save this UCS setting so you can quickly get back to it, and you can always return back to the original WCS.

To quickly return to the WCS, you can click on the named UCS combo control tool button and select **World** as shown in the following figure:

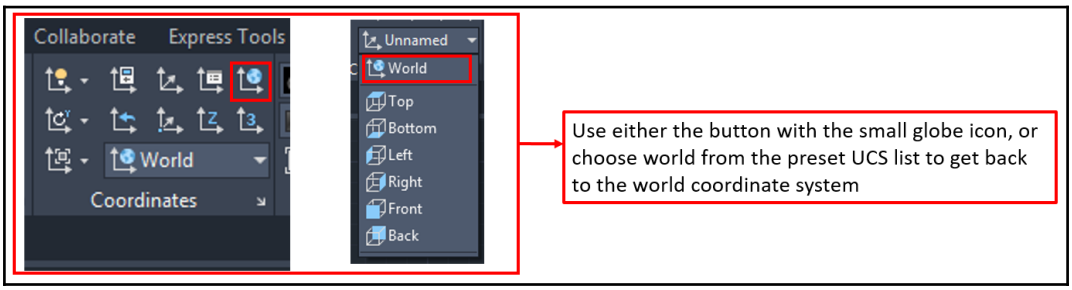


Figure 10.37: Resetting the UCS

To save a UCS, you can click on the **UCS Manager** button found in the **Coordinates** panel as shown in the following figure. This will open a window called **UCS** – double-click on the unnamed view, type a name as desired, and then press **OK**:

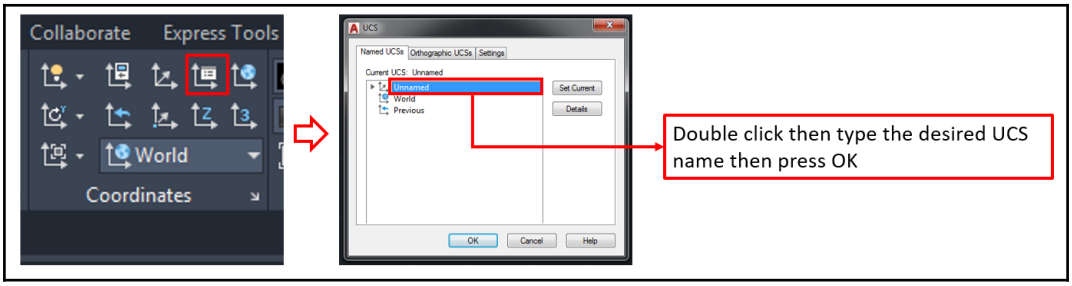


Figure 10.38: Saving a custom UCS alignment

Now that we've learned about the tools for manipulating the UCS in different ways, and have seen how to create and save custom UCS configurations, in the next sub-section we will talk about a feature in AutoCAD that can be used to dynamically change the UCS on the go.

Dynamic UCS functions

As discussed earlier, creating a 3D model may require continuous changes in the UCS to efficiently draw/create different parts of the 3D model, and despite having many tools for changing the UCS in AutoCAD, it may remain a tedious thing to do.

The Dynamic UCS feature addresses this problem. It allows the user to change the UCS temporarily on the go in a very easy and intuitive way. First of all, it can be activated/deactivated by pressing the *F6* key on the keyboard. When pressed, the command bar will show whether the Dynamic UCS function is **on** or **off**, as shown in the following figure where the *F6* key was pressed twice to turn **Dynamic UCS off** then **on** again:

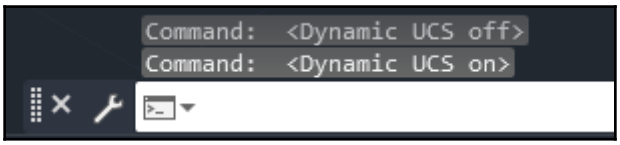


Figure 10.39: Switching the Dynamics UCS feature on

Now let's suppose we have a Box and we want to draw a line on one of its vertical surfaces. You can do this by aligning the UCS with the desired surface, then start drawing. Alternatively, if the UCS function is active, you can do the same in a more time-efficient way, as shown in the following step-by-step example:

1. Instead of changing the UCS, simply start the **LINE** command.
2. Move the cursor near the Box's desired surface. As shown in the following figure, the surface will be highlighted, indicating that the UCS is now temporarily aligned with that surface.
3. Start drawing normally.

The following figure is an illustration of the preceding task:

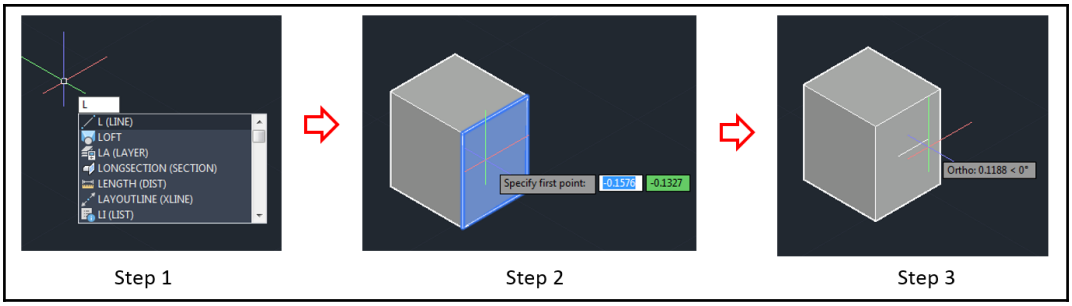


Figure 10.40: Using Dynamics UCS to draw a line on an object's face

Now we know how to properly set AutoCAD to start working in 3D. We have explored the workspaces offered by AutoCAD for 3D modeling, and learned how use different navigation tools, how to adjust the visual styles for model visibility and performance, how to use multi-viewport configurations, and how to work with the coordinate system tools. Let's move to the next step in the next chapter, which is creating 3D objects.

Summary

In this chapter, we learned how to properly set up AutoCAD to start doing 3D modeling. Starting with setting up our working environment to better serve the 3D modeling process, we learned about the concept of workspaces and explored the dedicated 3D workspaces in AutoCAD 2021. We also looked at how to divide the drawing area into multiple viewports to view the model from different angles simultaneously.

The next step was learning how navigate and change the view effectively and quickly. We learned about the different alternative navigation tools available in AutoCAD, how to switch between the standard views, and how to create our own set of views that better suits our project.

Another important aspect is the style by which the objects are displayed. 3D objects can be displayed in many different styles, ranging from something as simple as wire frames up to a realistic models with lighting, materials, and reflections. Each style has advantages and disadvantages regarding the model visibility and the impact on PC performance. We explored the different available visual styles in AutoCAD 2021, seeing the differences between them, and learned how to tweak the settings to create a new custom visual style that better suits our work.

And finally, we learned about the UCS and how to manipulate it as desired to facilitate the modeling process.

In the next chapter, we will be creating 3D solid objects, and we will start by exploring the set of primitive solid shapes. The different commands used for this will be presented along with the details of the different options and alternative methods for each command.

11
Creating Primitive 3D Shapes

AutoCAD features a variety of commands to create 3D solid objects. In this chapter, we will discover the first set of creation commands, which is a set of primitive 3D objects. The purpose of these commands is to provide a shortcut to creating primitive 3D shapes, and even if the desired model is a complex shape, they act like a starting point from which you can start modifying and adding details to create the desired shape. The primitive 3D solid commands that will be covered are as follows:

- Creating Boxes
- Creating Cylinders
- Creating Cones
- Creating Spheres
- Creating Pyramids
- Creating Wedges
- Creating a Torus
- Creating polysolids

Each command will be presented in detail, including the alternative options and methods included therein.

Creating primitive 3D shapes

The 3D primitives are a set of commands that can quickly create simple, basic 3D objects, such as a box, a sphere, or a pyramid. Among these, you can find the `Polysolid` command, which is a powerful tool that can be used to create 3D walls. In the following sections, each command will be presented in detail, including the alternative options and settings of each command. Let's now begin with the first primitive shape, which is the Box.

Creating Boxes

To start the `Box` command, click on the **Box** button in the **Create** panel under the **Home** tab in the **3D Basics** workspace. Alternatively, you can type the command name `BOX` and then press *Enter*, as shown in the following screenshot:

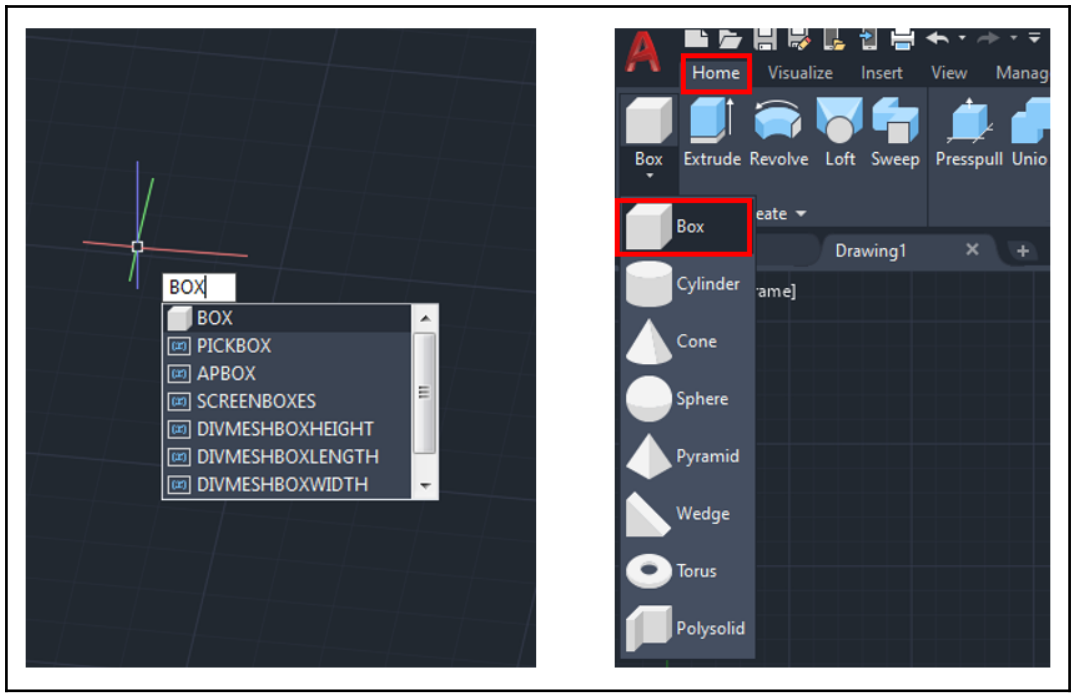


Figure 11.1: Alternative methods to start the Box command

If you are using the **3D Modeling** workspace, you can find the **Box** command button in the **Modeling** panel under the **Home** tab, or in the **Primitive** panel under the **Solid** tab.

To start creating a Box object, perform the following steps:

1. Choose the location and click to specify the first corner of the Box base. Alternatively, you can type the desired coordinates of the base corner and then press *Enter*.
2. Specify the location for the second corner of the Box base.
3. Move the mouse in the **z** direction and click to specify the height of the Box. Alternatively, you can type the Box height value and then press *Enter.*

The following screenshot is an illustration of the previous steps:

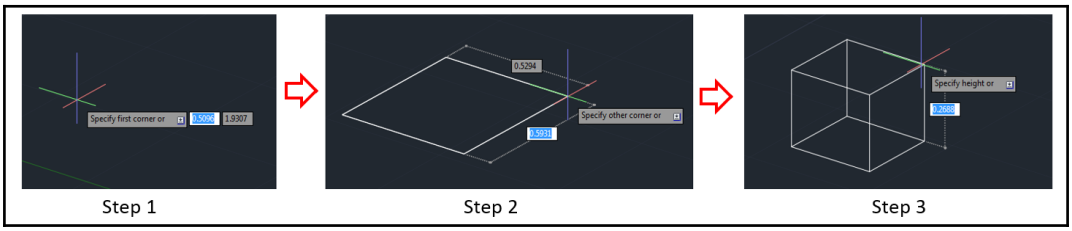


Figure 11.2: Steps for using the Box command

If you select the Box object, you will find a collection of different types of grab points, which can be divided into four categories as follows:

- At the Box base, you will find four endpoints at the four corners; these will be highlighted with a blue square shape.
- On the midpoint of each side of the Box base, there are four small arrows.
- At the center point of the bottom and top faces, there is a small blue arrow indicating (normal to) the face.
- In the middle of the bottom face, you find the Box base point, highlighted as a small square:

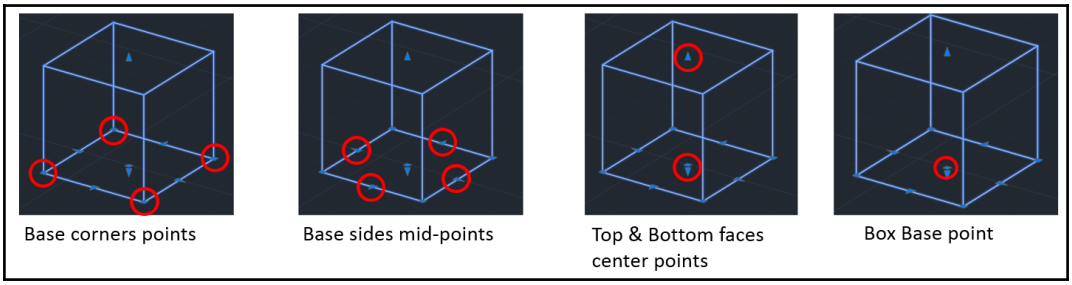


Figure 11.3: Box's grab points

Corner points can be used to change the shape of the Box base area as follows:

1. Use the mouse cursor and click to pick one of the corner points.
2. Move the corner point to the new location. Observe that the shape of the cube will change dynamically as you move the corner point.
3. Click again to set the new position of the cube corner.

The following screenshot is an illustration of the previous steps:

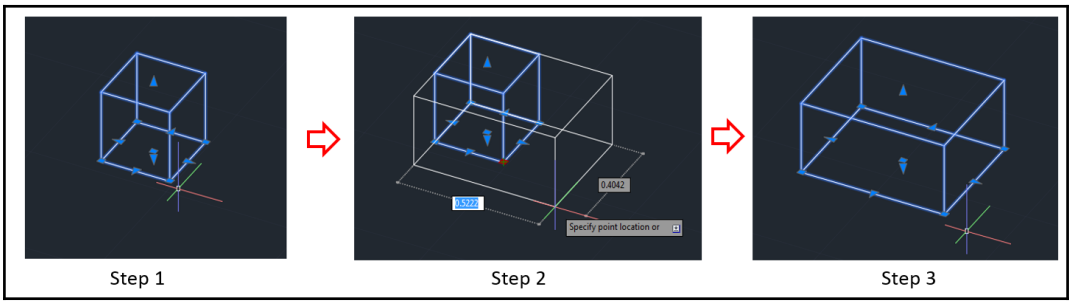


Figure 11.4: Resizing the Box using corner grab points

The sides' midpoints can also be used to modify the base of the Box as follows:

1. Pick one of the midpoints of the base sides (the small blue arrow).
2. Move the side to the new location. Observe that the shape of the cube will change dynamically as the mouse moves.
3. When you arrive at the desired location, click to finish. Alternatively, you can type the new side length and then press *Enter*.

The following screenshot is an illustration of the previous steps:

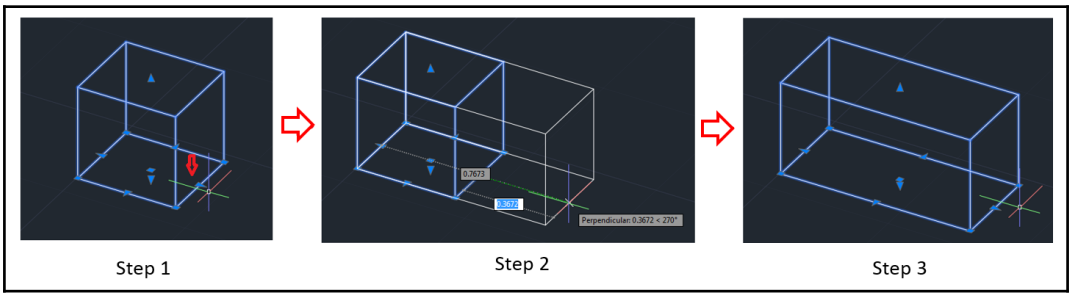


Figure 11.5: Resizing the Box using side grab points

The top and bottom face center points can be used to change the height of the Box as follows:

1. Pick one of the face center points (the blue arrow pointing outside of the Box).
2. Move the face to the new desired location. Observe that the shape of the Box will change dynamically as the mouse moves.
3. Click again to set the new height of the cube. Alternatively, you can type the new desired cube height and then press *Enter*.

The following screenshot is an illustration of the previous steps:

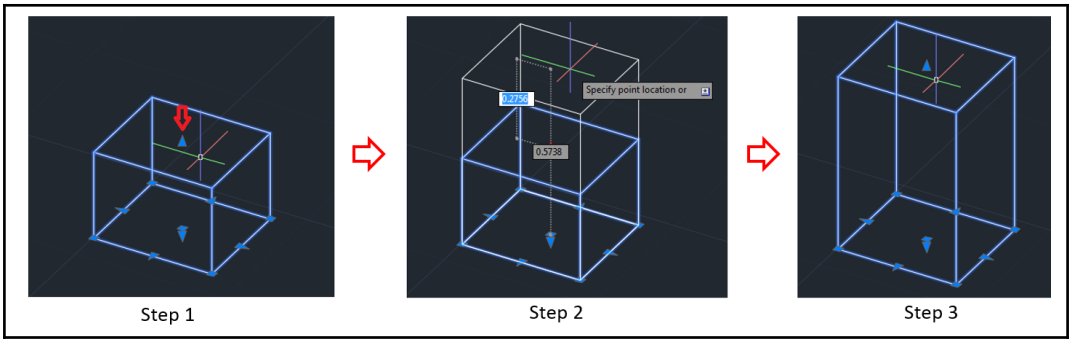


Figure 11.6: Changing the Box height using grab points

And finally, the base point can be used to pick and move the Box, and paste it at the desired location.

There is a set of options and settings associated with the `Box` command. These give you different creation methods during the Box creation.

The first option is called **Center**. This option appears in the command bar right after you start the `Box` command:

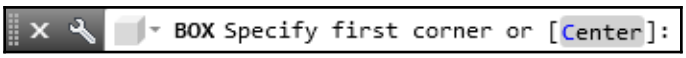


Figure 11.7: Additional options in the Box command

To select this option. simply click on the word `Center` in the command bar, or you can type `C` (the letter in blue), and then press *Enter*. Basically, this is the same as the default method, but in this case, you will begin by specifying the center point of the Box, and it goes as follows:

1. Specify the center point (instead of the corner in the default method).
2. Move and click to specify the corner location. Notice that all the base sides are moving outward or inward as you change the corner location and all will have the same length.
3. Move in the **z** direction and click to specify the height. Notice that both the top and bottom faces will move the same distance as you change the height.

The following screenshot is an illustration of the previous steps:

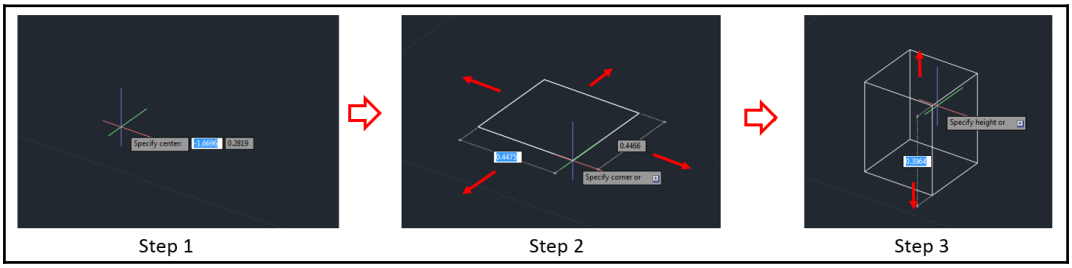


Figure 11.8: Using the Center option

The second set of options will appear after you specify the first corner (or the center point in case you choose the **Center**):

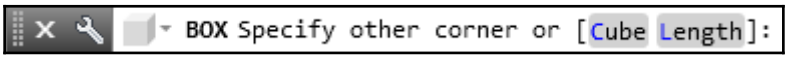


Figure 11.9: Second set of additional options in the Box command

1. To select the **Cube** option, click on the option name in the command bar or just type `C` (the letter in blue color) and then press *Enter*. This option will restrict the Box sides to all have the same length, so the only parameter you will need to enter is the side length. Use your mouse to specify the side length, or you can enter the value using the keyboard. You can also move your mouse to change the cube orientation around the **z-axis**.

2. To select the **Length** option, click on the option name in the command bar, or type the letter `L` and then press *Enter*. After selecting this option, you will be able to enter separately the values of the length, width, and height of the Box, and you can do this by using your mouse or keyboard:

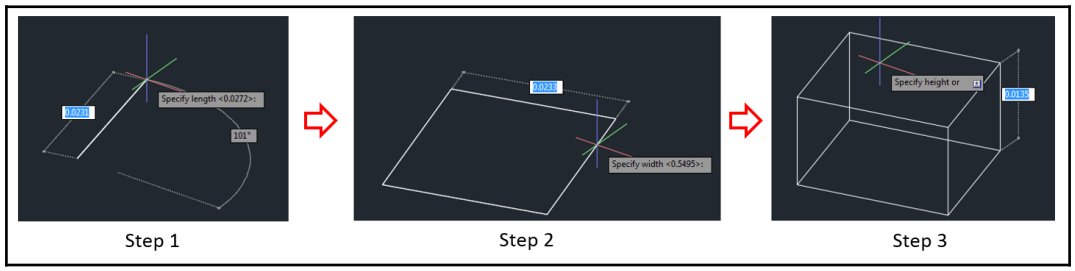


Figure 11.10: Using the Cube option

The last option in the `Box` command is called **2Point**. This option appears in the command bar after you draw the base of the Box, as shown in the following screenshot:

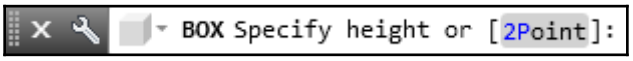


Figure 11.11: Third set of additional options in the Box command

You can select the **2Point** option by clicking on the option name in the command bar, or you can type `2P` and then press *Enter*. This option will allow you to use the `2Point` method used in other AutoCAD commands to specify the height of the Box:

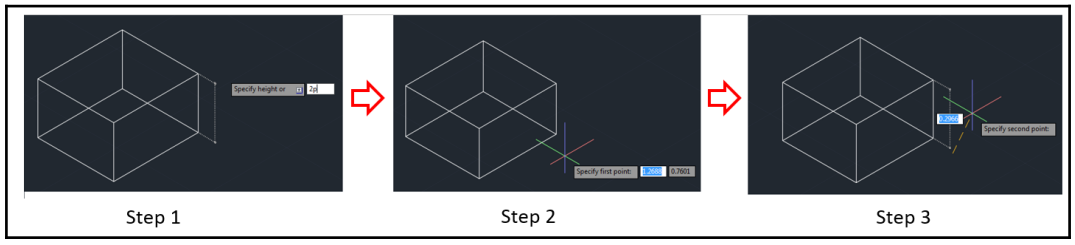


Figure 11.12: Using the 2Point option

Now that we have learned about the different methods and options for creating a Box object, let's have a look at the next shape, which is a cylinder.

Creating Cylinders

To start the `Cylinder command`, you can type the word `CYLINDER` (or just type `CYL`) and then press *Enter*, or you can click on the **Cylinder** button found in the **Create** panel in the **3D Basics** workspace. If you are using the **3D Modeling** workspace, you will find the **Cylinder** button in the **Modeling** panel under the **Home** tab, or in the **Primitive** panel under the **Solid** tab, as shown in the following screenshot:

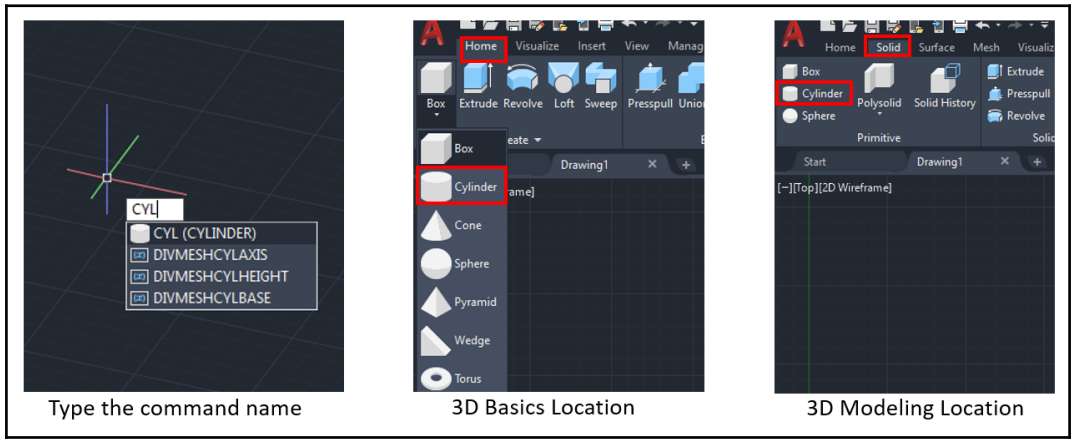


Figure 11.13: Alternative methods to star the Cylinder command

To create a cylinder using the default settings, perform the following steps:

1. Choose a location and click to specify the center point of the cylinder base.
2. Move the mouse and click to specify the radius of the cylinder. Alternatively, you can use the keyboard to enter the radius value and then press *Enter*.
3. Move the mouse cursor to specify the height and click to finish. Alternatively, you can use the keyboard to enter the height value and then press *Enter*.

The following screenshot is an illustration of the previous steps:

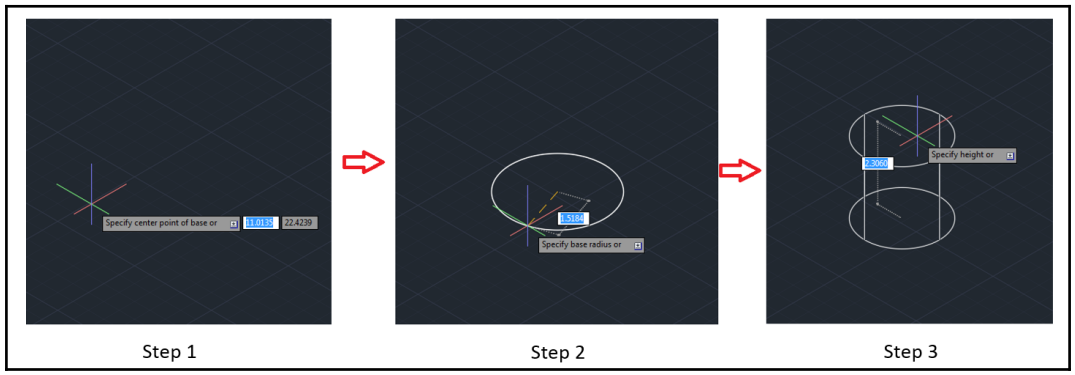


Figure 11.14: Steps for using the Cylinder command

If you select the cylinder object, you will find a collection of different types of grab (control) points. These points can be divided into three categories as follows:

- On the circumference of the cylinder base, there are four small arrows to control the radius.
- At the center point of the bottom and top faces, there is a small blue arrow pointing out (normal to) the face.
- In the middle of the bottom face, you find the cylinder base point, highlighted as a small square:

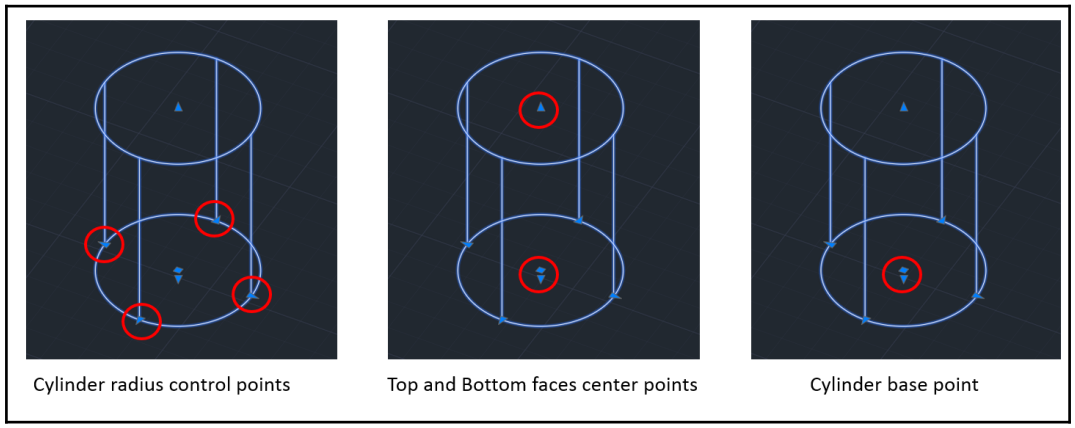


Figure 11.15: Cylinder's grab points

Small arrows on the base circumference can be used to modify the radius as follows:

1. Use the mouse cursor to pick one of the circumference small arrows.
2. Move the arrow to modify the radius. Observe that the shape of the cylinder will change dynamically as the mouse moves. You can also enter the desired value using the keyboard.
3. Click again to set the new radius.

The following screenshot is an illustration of the previous steps:

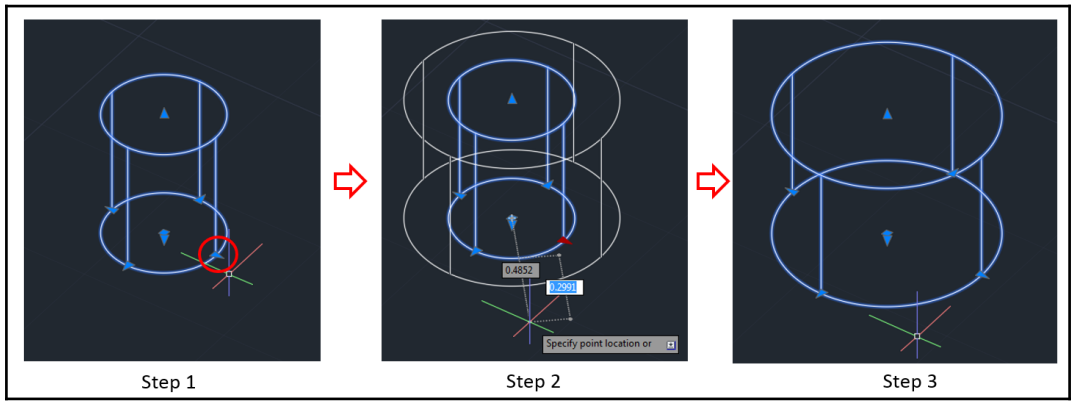


Figure 11.16: Modifying the cylinder radius using grab points

The top and bottom face center points can be used to change the height of the cylinder as follows:

1. Pick one of the face center points (the blue arrow pointing outside the cylinder).
2. Move the face to the new desired location. Observe that the shape of the cylinder will change dynamically as the mouse moves. You can also enter the desired value using the keyboard.
3. When you arrive at the desired location, click to finish.

The following screenshot is an illustration of the previous steps:

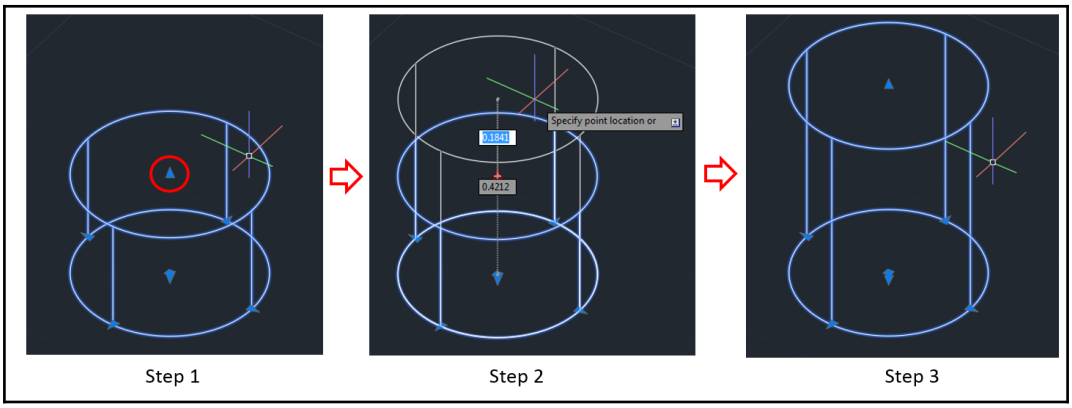


Figure 11.17: Modifying the cylinder height using grab points

And finally, the base point (highlighted as a square in the bottom face center point) can be used to pick and move the cylinder, and then paste it at the desired location.

There are a number of options associated with the **Cylinder command**. These options make provision for alternative methods in creating the Cylinders. The first set of options appears in the command bar right after you start the command:

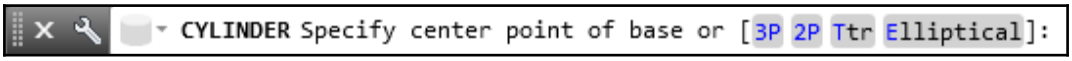


Figure 11.18: Additional option in the Cylinder command

The first three options in this set are alternative methods for creating the base circle, followed by an option to create the cylinder with an elliptical base:

- The first option is called **3P**, which stands for three points. To choose this option, you can click on the option name in the command bar, or you can type `3P` and then press *Enter*. When this option is selected, you will draw the base circle by specifying the location of three points situated on the base circle circumference:

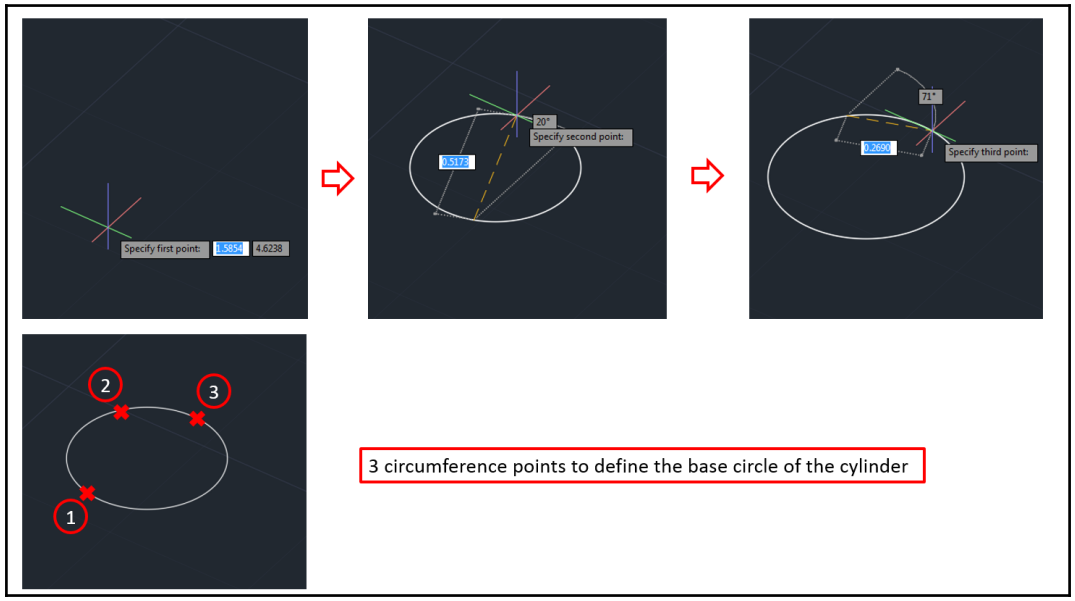


Figure 11.19: Using the 3P option

- The second option is called **2P**, which stands for two points. To choose this option, you can click on the option name in the command bar, or you can type `2P` and then press *Enter*. When this option is selected, you will draw the base circle by specifying two points located on the diameter in the base circle, as shown in the following screenshot:

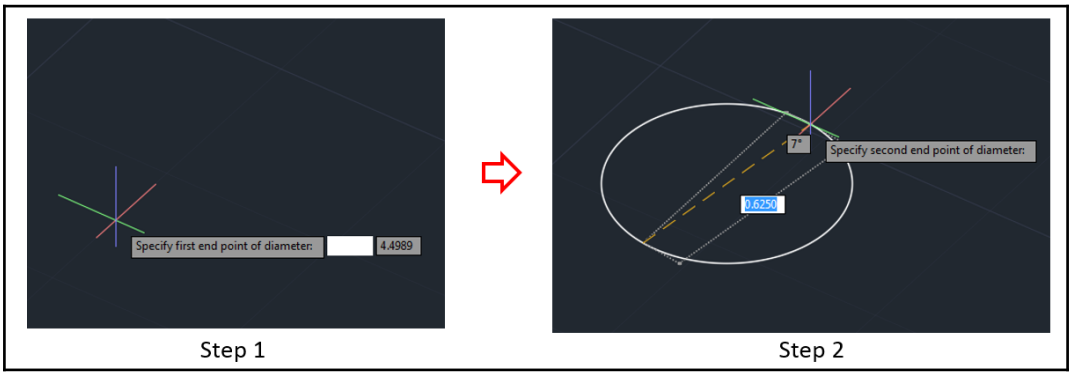


Figure 11.20: Using the 2P option

- The third option is called **Ttr**, which stands for Tangent Tangent Radius. This option can be selected by clicking on the option name in the command bar, or by just typing `T` and then pressing *Enter*. When you select this option, you can create the base circle to be tangent to any two other curves in the drawing, as shown in the following step-by-step example:
    1. Move the cursor near the first curve. The tangent icon will appear. Click to specify the first tangent curve.
    2. Move the cursor near the second curve. The tangent icon will appear. Click to specify the second tangent curve.
    3. Type the radius value and then press *Enter*. Alternatively, you can use the mouse to specify the radius.

The following screenshot is an illustration of the previous steps:

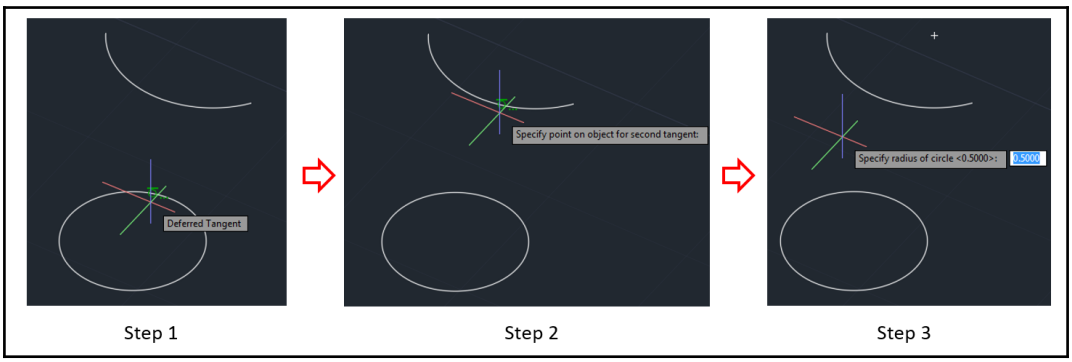


Figure 11.21: Using the Ttr option

- The last option is called **Elliptical**. This option can be selected by clicking on the option name in the command bar, or by just typing `E` and then pressing *Enter*. This option allows you to draw the elliptical base instead of a circle, as shown in the following step-by-step example:
    1. Click to specify the location of the ellipse's first axis endpoint. You also have the option to specify the center by selecting the **Center** option from the command bar.
    2. Move and click again to specify the second point of the first axis.
    3. Move to specify the orientation of the ellipse and the length of the second axis, and then click to confirm.
    4. Specify the height of the cylinder as usual.
    5. The command terminates and the cylinder is created.

The following screenshot is an illustration of the previous steps:

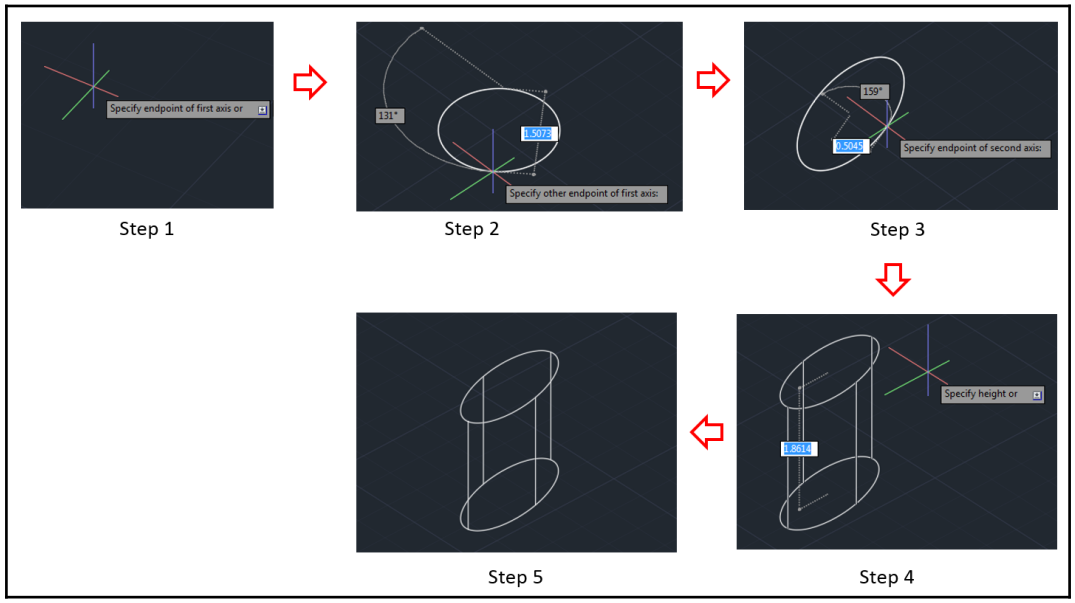


Figure 11.22: Using the Elliptical option

The second set of options appear after you determine the center of the cylinder. This set contains one option called **Diameter**, and this allows you to enter the diameter of the cylinder instead of the radius in the default case, as shown in the following screenshot:

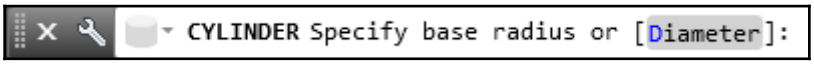


Figure 11.23: Second set of options in the Cylinder command

The third set of options appears after you draw the base. This contains two options, as shown in the following screenshot:

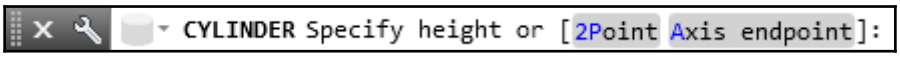


Figure 11.24: Third set of options in the Cylinder command

- The first one is called **2Point**, which is an alternative method for specifying the height of the cylinder. It is exactly the same as the `2P` method in the `Box` command (for more details on the `2P` method, please refer to the *Creating Boxes* section).
- The second option in this set is called **Axis endpoint.** To select this option, you can click on the option name in the command bar, or you can type the letter `A` and then press *Enter*. The axis endpoint allows you to change the orientation of the cylinder, as shown in the following step-by-step example:
    1. Activate the **Axis endpoint** option.
    2. Move the mouse cursor to change the orientation of the cylinder axis.
    3. When you reach the desired orientation, click again.

The following screenshot is an illustration of the previous steps:

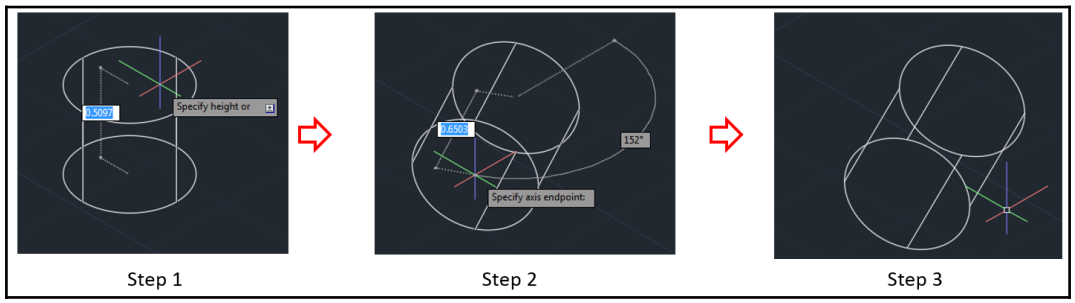


Figure 11.25: Using the Axis endpoint option

We have learned about the creation of two shapes. Now, let's take a look at the next 3D object, which is a cone.

Creating Cones

To start the `Cone` command, you can type the word `CONE` and then press *Enter*. Alternatively, you can click on the **Cone** button found in the **Create** panel in the **3D Basics** workspace. If you are using the **3D Modeling** workspace, you will find the **Cone** button in the **Modeling** panel under the **Home** tab, or in the **Primitive** panel under the **Solid** tab, as shown in the following screenshot:

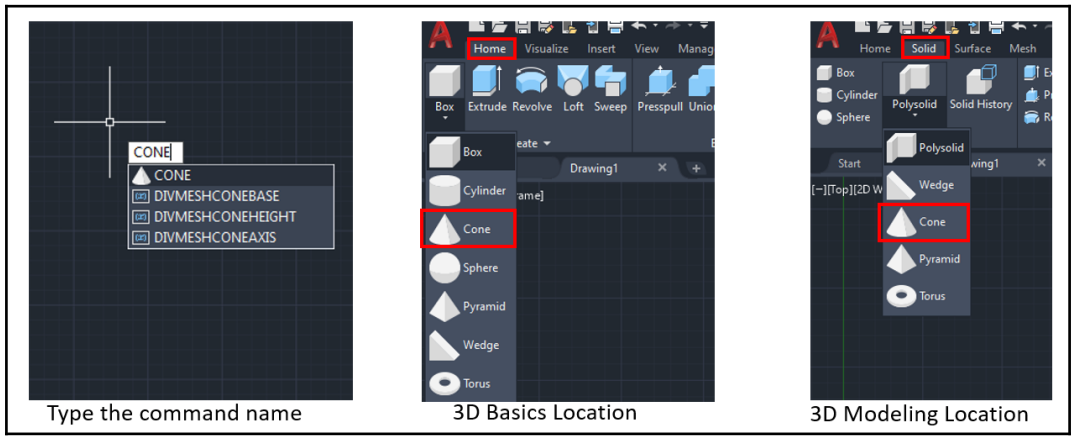


Figure 11.26: Alternative methods to start the Cone command



If you are using the **3D Modeling** workspace, you will find the **Cone** button in the **Modeling** panel under the **Home** tab, or in the **Primitive** panel under the **Solid** tab.

To create a cone using the default settings, perform the following steps:

1. Choose a location and click to specify the center point of the cone base.
2. Move the mouse and click to specify the radius of the cylinder. Alternatively, you can use the keyboard to enter the radius value and then press *Enter*.
3. Move the mouse cursor to specify the height and click to finish. Alternatively, you can use the keyboard to enter the height value and then press *Enter* to finish.

The following screenshot is an illustration of the previous steps:

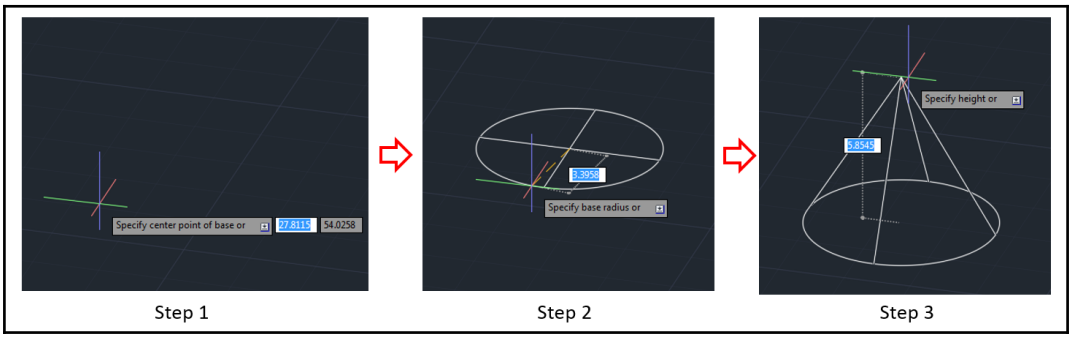


Figure 11.27: Steps for using the Cone command

If you select the cone object, you will find a collection of different grab (control) points, these points can be divided into four categories as follows:

- On the circumference of the cone base, there are four small arrows to control the radius.
- Near the tip of the cone, there is a blue arrow to create/control the top face radius.
- At the center point of the bottom face and at the tip of the cone, there is a small blue arrow pointing out (perpendicular to) the face.
- In the middle of the bottom face, you will find the cone base point, highlighted as a small square:

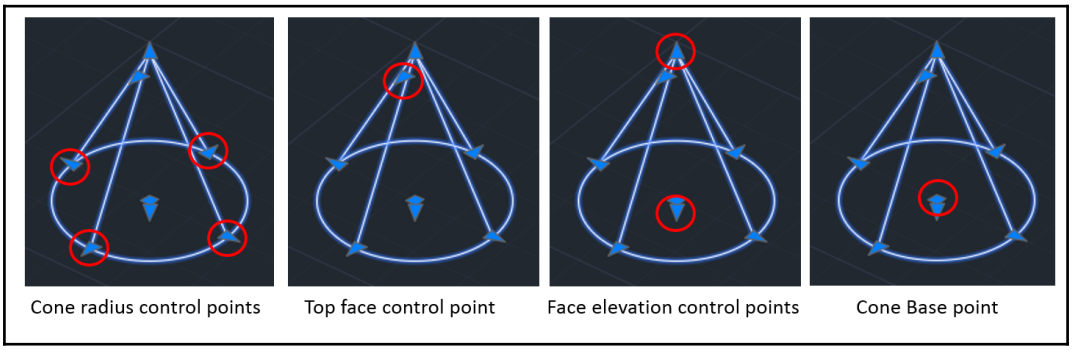


Figure 11.28: Cone's grab points

The small arrows on the base circumference can be used to modify the radius, as shown in the following step-by-step example:

1. Use the mouse cursor to pick one of the circumference small arrows.
2. Move the arrow to modify the radius. Observe that the shape of the cone will change dynamically as the mouse moves. You can also enter the desired value using the keyboard.
3. Click again to set the new radius.

The following screenshot is an illustration of the previous steps:

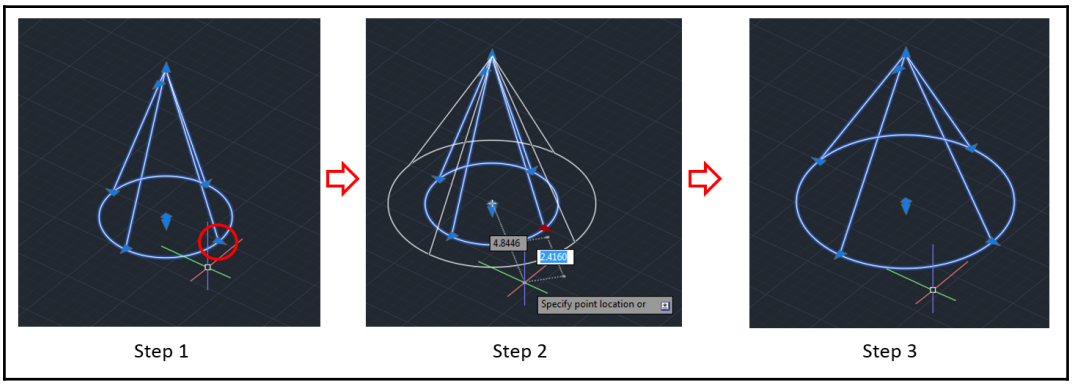


Figure 11.29: Modifying the cone's radius using grab points

The arrow near the tip of the cone can be used to create/modify the radius of the upper face, as shown in the following step-by-step example:

1. Pick the arrow near the tip of the cone.
2. Move to modify the radius of the top face. You can also enter the desired value using the keyboard.
3. When you arrive at the desired location, click to finish.

The following screenshot is an illustration of the previous steps:

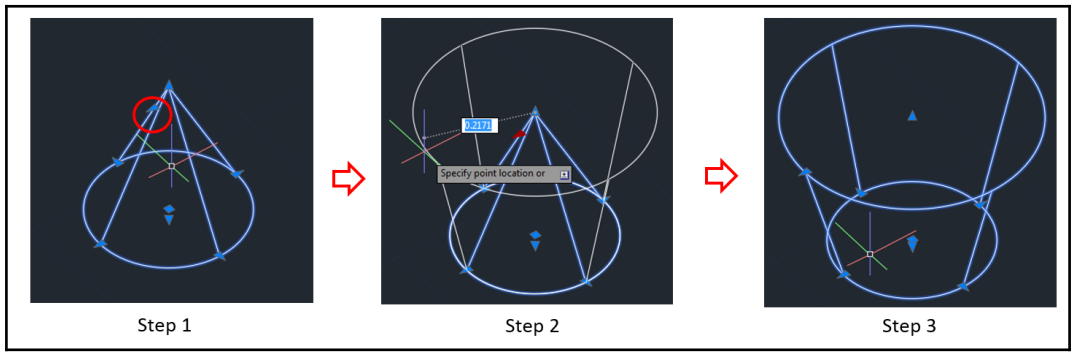


Figure 11.30: Adding a top face to the cone using grab points

The arrows at the bottom face, and at the tip of the cone, can be used to change the height of the cone, as shown in the following step-by-step example:

1. Pick the arrow at the tip of the cone (or the arrow in the bottom face center point).
2. Move the face to the new desired location. Observe that the shape of the cone will change dynamically as the mouse moves. You can also enter the desired value using the keyboard.
3. When you arrive at the desired location, click to finish.

The following screenshot is an illustration of the previous steps:

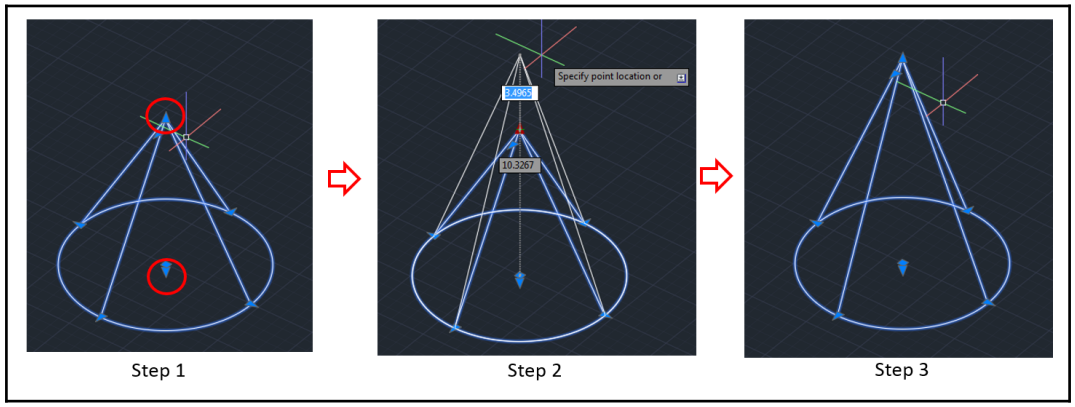


Figure 11.31: Modifying the height of the cone using grab points

And finally, the base point (highlighted as a square in the bottom face center point) can be used to pick and move the cone.

As shown in the previous screenshot, the first two sections of the **Properties** palette (**General & 3D Visualization**) contain the general properties that are present for any AutoCAD object, including **Color**, **Layer**, and **Material**. The section entitled **Geometry** contains the properties specific to the cone object, such as the Cartesian coordinates of the base point location that can be modified, as well as the cone dimensions (**Base radius, Height**) that can also be modified. In addition, there is an option called **elliptical** that has two settings (**Yes** or **No**). This option can be used to quickly convert the cone base from circular to elliptical, as shown in the example under the *Creating Cylinders* section.

There are a number of options associated with the `Cone` command. These options make provision for alternative methods in creating the cone. The first set of options appears in the command bar right after you start the command:

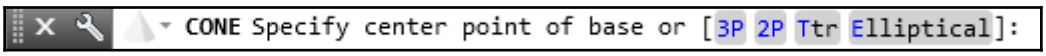


Figure 11.32: Additional options in the Cone command

The first three options in this set are alternative methods to create the base circle, while the last option is for creating a cone with an elliptical base. The four options are identical to the first set of options in the `Cylinder` command. Please refer to the *Creating Cylinders* section for details of these four options.

We have learned how to quickly create a simple cone shape. Let's now look at the next primitive solid, which is the sphere.

Creating Spheres

To start the `Sphere` command, you can type the command name `SPHERE` (or just type `SPH`) and then press *Enter*, or you can click on the **Cylinder** button found in the **Create** panel in the **3D Basics** workspace, as shown in the following screenshot:

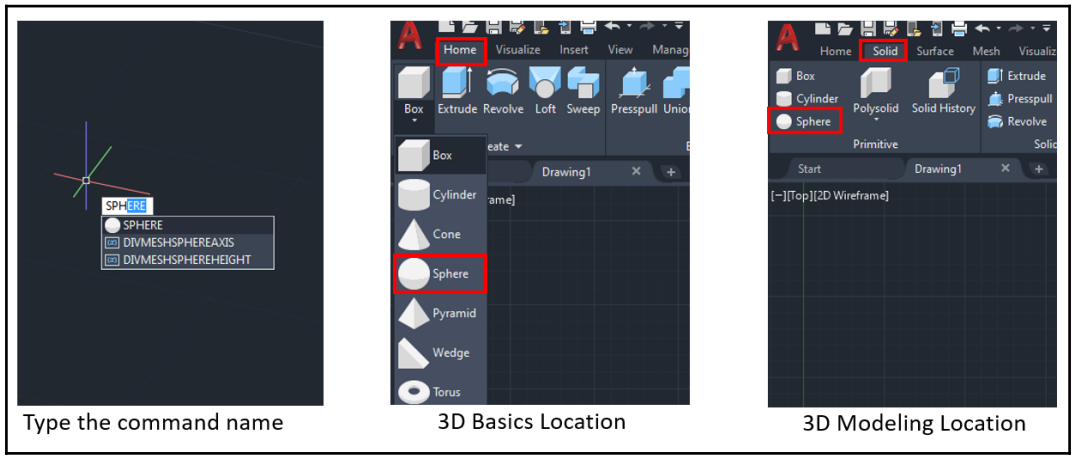


Figure 11.33: Alternative methods to start the Sphere command

If you are using the **3D Modeling** workspace, you will find the **Sphere** button in the **Modeling** panel under the **Home** tab, or in the **Primitive** panel under the **Solid** tab, as shown in the preceding screenshot:

To create a sphere using the default settings, after you start the command, perform the following steps:

1. Choose a location and click to specify the center point of the sphere.
2. Move the mouse, click to specify the radius of the cylinder, and then click to finish. Alternatively, you can use the keyboard to enter the radius value and then press *Enter*, as shown in the following screenshot:

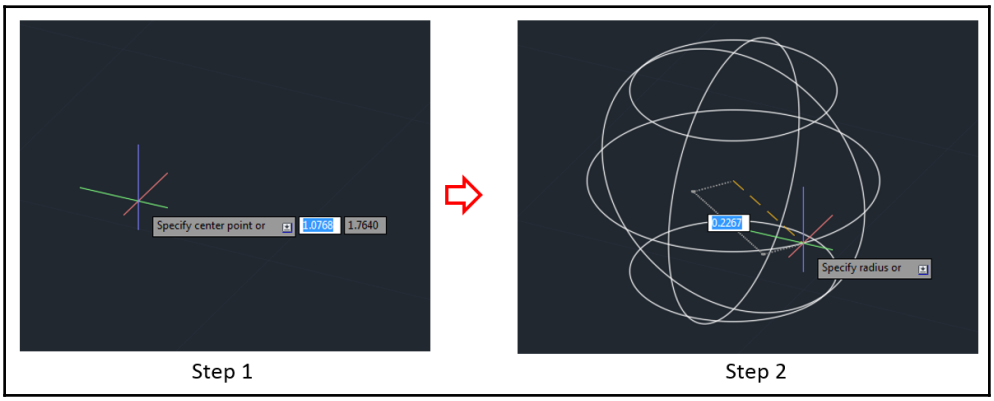


Figure 11.34: Using the Sphere command

If you select the sphere object, you will find a collection of different types of control points, and they can be divided into two categories as follows:

- On the circumference of the sphere, there are four small arrows to control the radius, as shown in the following screenshot.
- At the center of the sphere you can find the base point, highlighted as a small square in the following screenshot:

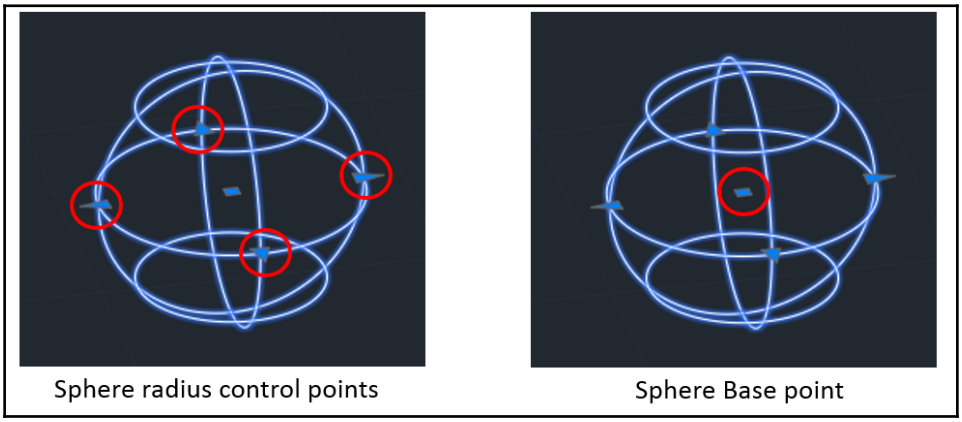


Figure 11.35: Sphere's grab points

The small arrows on the sphere circumference can be used to modify the radius, as shown in the following step-by-step example:

1. Use the mouse cursor to pick one of the circumference small arrows.
2. Move the arrow to modify the radius. Observe that the shape of the sphere will change dynamically as the mouse moves. You can also enter the desired value using the keyboard.
3. Click again to set the new radius, as shown here.

The following screenshot is an illustration of the previous steps:

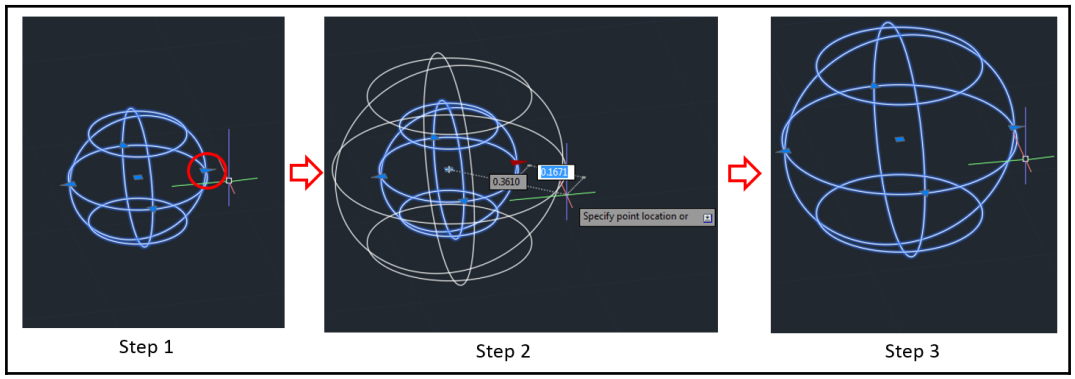


Figure 11.36: Modifying the sphere's radius using grab points

And finally, the sphere base point (highlighted as a square in the center) can be used to pick and move the sphere.

There are a number of options associated with the `Sphere` command. These options make provision for alternative methods in creating the sphere. The first set of options appears in the command bar right after you start the command, as shown in the following screenshot:

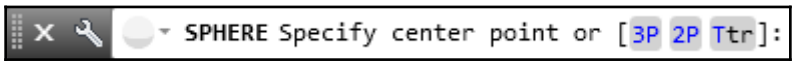


Figure 11.37: Additional options in the Sphere command

The options in this set are alternative methods for creating a circle. They are identical to the first set of options in the *Cylinder command* section. Please refer to the *Creating Cylinders* section for details of these three options.

The second set of options appears after you determine the center of the sphere, as shown in the following screenshot. This set contains one option called **Diameter**, and this allows you to enter the diameter of the sphere, instead of the radius in the default case:

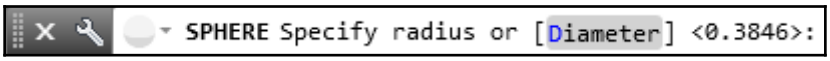


Figure 11.38: Second set of additional options in the Sphere command

After we have learned about sphere creation in AutoCAD, let's take a look at creating Pyramids.

Creating Pyramids

To start the `Pyramid` command, you can type the command name `PYRAMID` (or just type `PYR`) and then press *Enter*. Alternatively, you can click on the **Pyramid** button found in the **Create** panel in the **3D Basics** workspace. If you are using the **3D Modeling** workspace, you will find the **Pyramid** button in the **Modeling** panel under the **Home** tab, or in the **Primitive** panel under the **Solid** tab, as shown in the following screenshot:

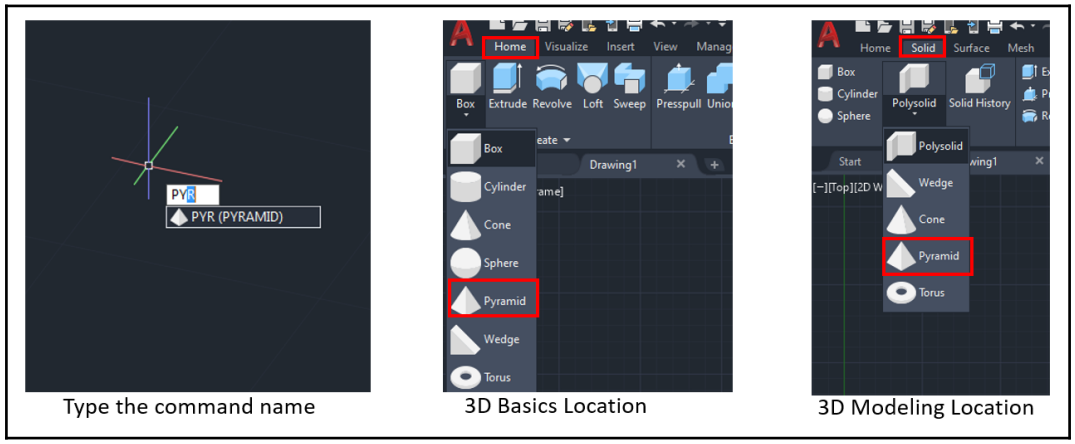


Figure 11.39: Alternative methods to start the Pyramid command

After starting the command, to create a pyramid using the default settings, perform the following steps:

1. Choose a location and click to specify the center point of the pyramid base.
2. In the default method, you specify a radius for a circle, and a square base is created, circumscribed about that circle. You can specify the radius using the mouse cursor. Alternatively, you can use the keyboard to enter the radius value and then press *Enter*.
3. Move the mouse cursor in the **z** direction to specify the height and click to finish. Alternatively, you can use the keyboard to enter the height value and then press *Enter* to finish.

The following screenshot is an illustration of the previous steps:

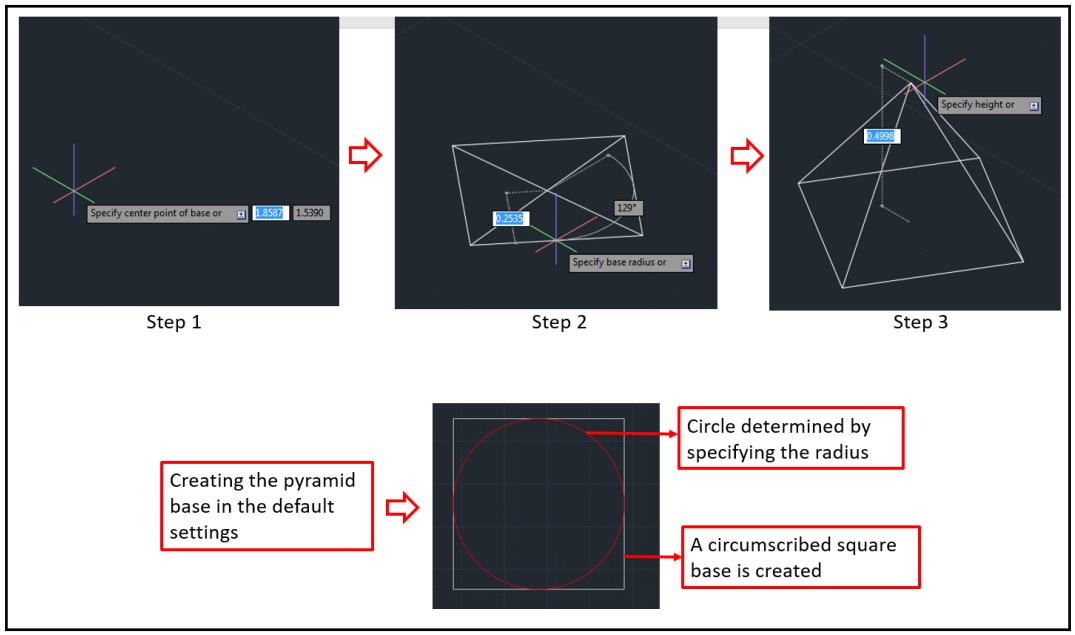


Figure 11.40: Steps involved in using the Pyramid command

If you select the pyramid object, you will find a collection of different types of control points, and these can be divided into five types as follows:

- On the corners of the pyramid's base, there are four small arrows to control the base shape.
- On the midpoints of the sides of the pyramid's base, there are four small arrows to control the base shape.
- At the center point of the bottom face and at the tip of the pyramid, there is a small blue arrow pointing out (normal to) the face.
- Near the tip of the pyramid, there is a blue arrow to create/control the top face.

- In the middle of the bottom face, you find the pyramid base point, highlighted as a small square in the following screenshot:

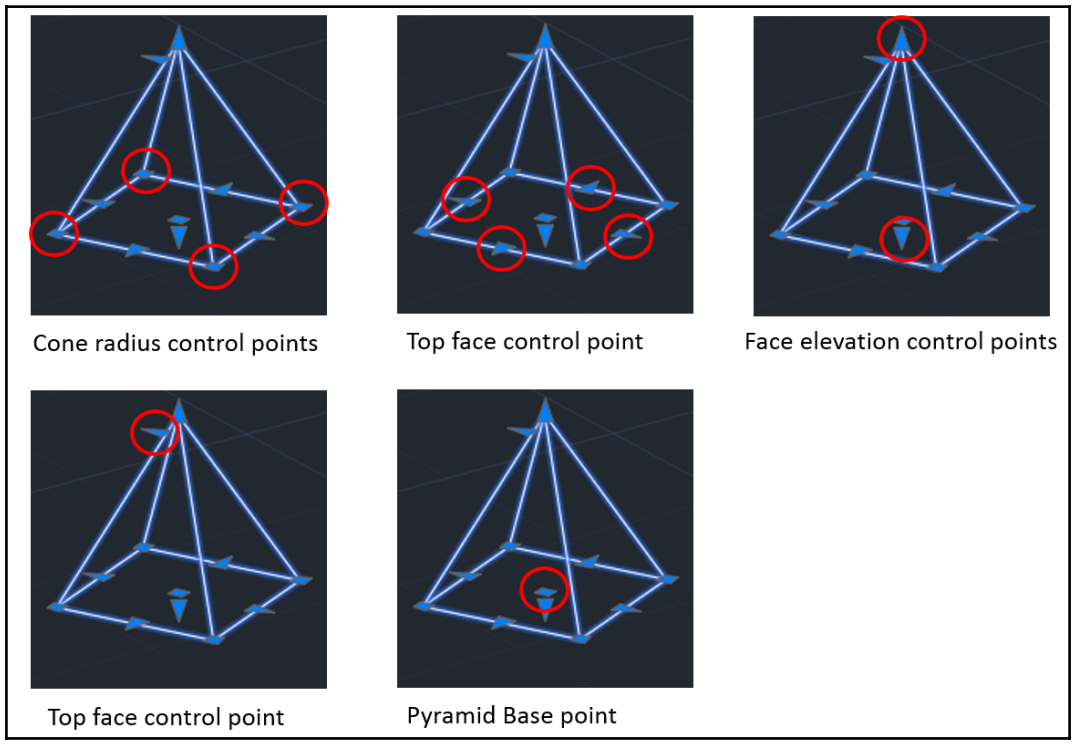


Figure 11.41: Pyramid's grab points

The small points on the pyramid's base corners can be used to modify the base shape, as shown in the following step-by-step example:

1. Use the mouse cursor to pick one of the base corners.
2. Move the arrow to modify the base shape. Observe that the shape of the pyramid will change dynamically as the mouse moves.
3. Click again to finish.

The following screenshot is an illustration of the previous steps:

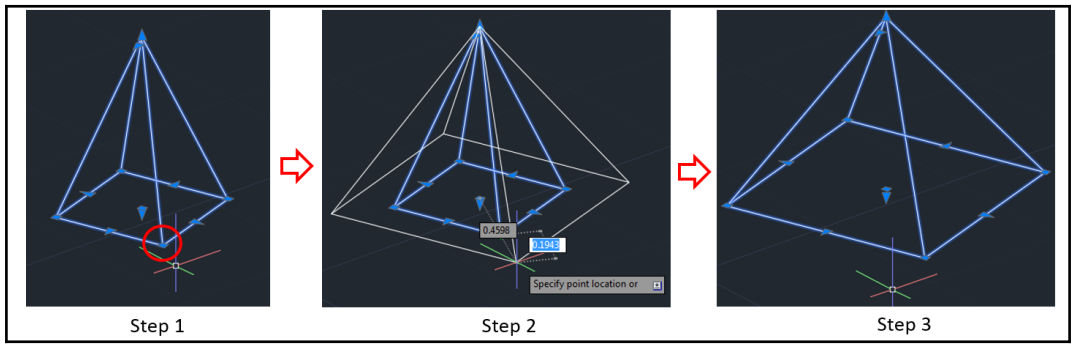


Figure 11.42: Modifying the pyramid's base using corner grab points

The small arrows on the pyramid's base sides can be used to modify the base shape, as shown in the following step-by-step example:

1. Use the mouse cursor to pick one of the midpoints on the base sides.
2. Move the arrow to modify the base shape. Observe that the shape of the pyramid will change dynamically as the mouse moves.
3. Click again to finish.

The following screenshot is an illustration of the previous steps:

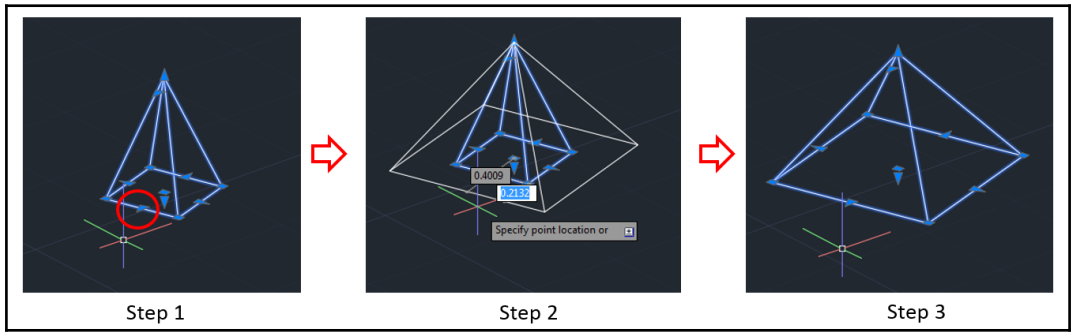


Figure 11.43: Modifying the pyramid's base using side grab points

The arrow near the tip of the pyramid can be used to create/modify the upper face, as shown in the following step-by-step example:

1. Pick the arrow near the tip of the pyramid.
2. Move the mouse to modify the shape of the top face. You can also enter the desired value using the keyboard.
3. When you arrive at the desired location, click to finish.

The following screenshot is an illustration of the previous steps:

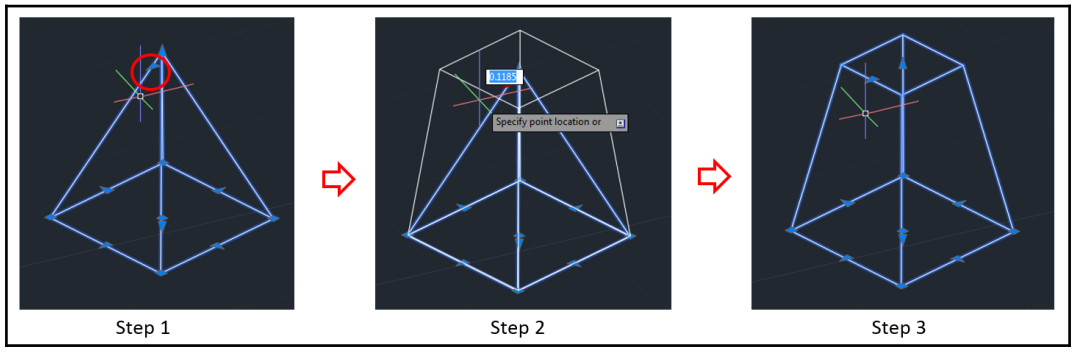


Figure 11.44: Adding a top face using grab points

The arrows at the bottom face and at the tip of the pyramid can be used to change the height of the pyramid, as shown in the following step-by-step example:

1. Pick the arrow at the tip of the pyramid or the arrow in the bottom face center point.
2. Move the face to the new desired location. Observe that the shape of the pyramid will change dynamically as the mouse moves. You can also enter the desired value using the keyboard.
3. When you arrive at the desired location, click to finish.

The following screenshot is an illustration of the previous steps:

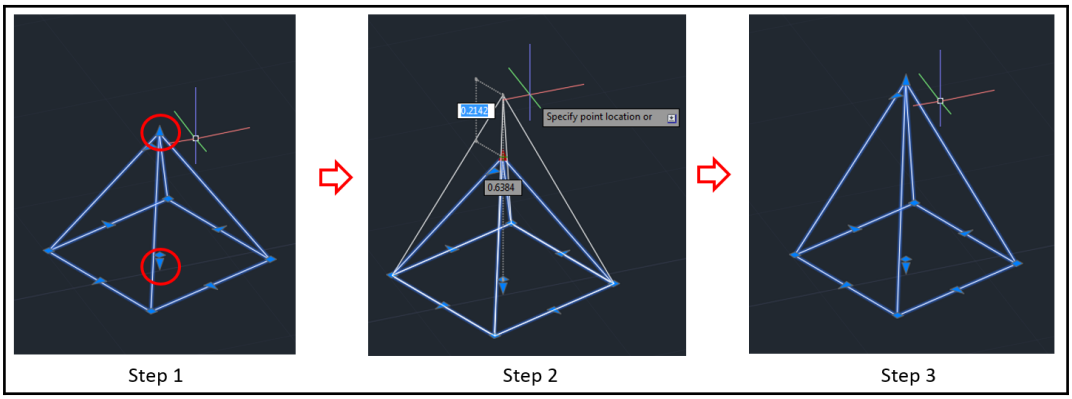


Figure 11.45: Modifying the pyramid's height using grab points

And finally, the base point (highlighted as a square in the bottom face center point) can be used to pick and move the pyramid.

There are a number of options associated with the `Pyramid` command. These options make provision for alternative methods in creating the pyramid. The first set of options appears in the command bar right after you start the command, as shown in the following screenshot.

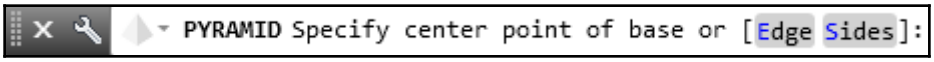


Figure 11.46: Additional options in the Pyramid command

- The first option in this set is called **Edge**. This can be selected by clicking on the option name in the command bar, or you can just type `E` and then press *Enter*. This option will allow you to specify the location of one of the base corners instead of specifying the center, as in the default method.

- The second option is called **Sides**. This can be selected by clicking on the option name in the command bar, or you can just type `S` and then press *Enter*. This option is used to determine the number of sides in the pyramid base. After you select the option, enter the number of sides in the pyramid base and then press *Enter*, and then continue creating the pyramid as usual. The created base will be an equilateral shape with a number of sides that were entered as shown in the example in the following screenshot:

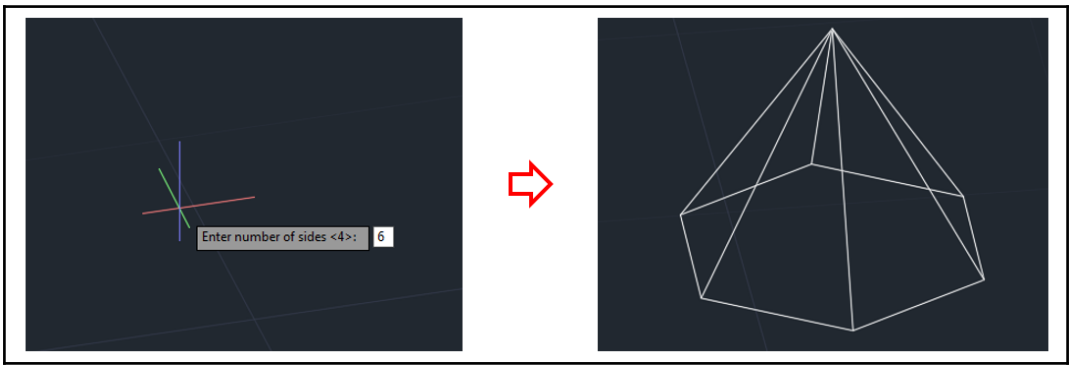


Figure 11.47: Using the Sides option to create a pyramid with a hexagonal base

The second set of options appears after you specify the center point of the pyramid base. This set contains one option called **Inscribed**. It can be selected by clicking on the option name in the command bar or you can just type `I` and then press *Enter*. After you select this option, the base of the created pyramid will be Inscribed inside the circle radius instead of circumscribed in the default setting:

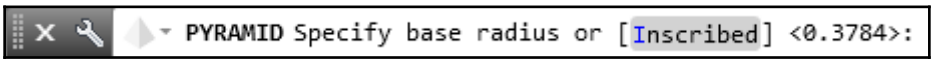


Figure 11.48: Second set of additional options in the Pyramid command

The difference between **circumscribed**, which is the default base creation method, and the optional **Inscribed** method is illustrated in the following screenshot:

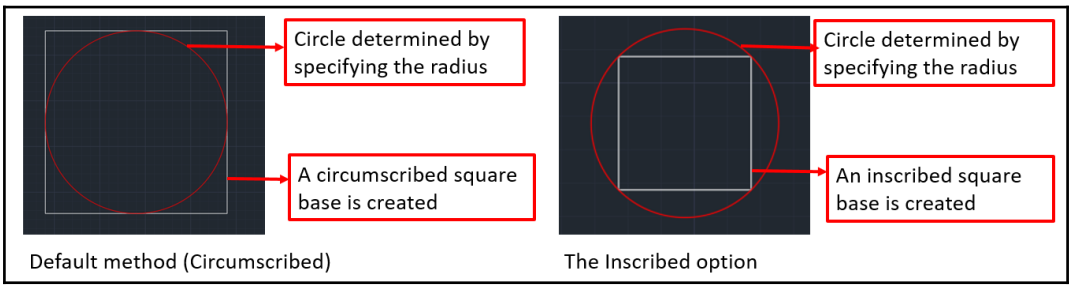


Figure 11.49: Circumscribed versus the Inscribed option

The third set of options appears once the base is created. It contains three different options, as shown in the following screenshot:

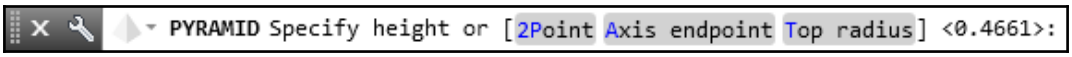


Figure 11.50: Third set of additional options in the Pyramid command

The first one is called **2Point**. This option is an alternative way to create the height of the pyramid. The second one is called **Axis endpoint**. This option allows you to change the pyramid height direction. The third one is called **Top radius**. This option can be used to create a top face instead of the pyramid tip. These three options work exactly like the `Cone command`. Please refer to the *Creating Cones* section for the details of these options.

Now that we have learned how to create five different 3D shapes, let's take a look at the next one in the primitives list, which is the Wedge.

Creating Wedges

To start the `Wedge` command, you can type the command name `WEDGE` (or just type `WED`) and then press *Enter*. Alternatively, you can click on the **Wedge** button found in the **Create** panel in the **3D Basics** workspace. If you are using the **3D Modeling** workspace, you will find the **Wedge** button in the **Modeling** panel under the **Home** tab, or in the **Primitive** panel under the **Solid** tab, as shown in the following screenshot:

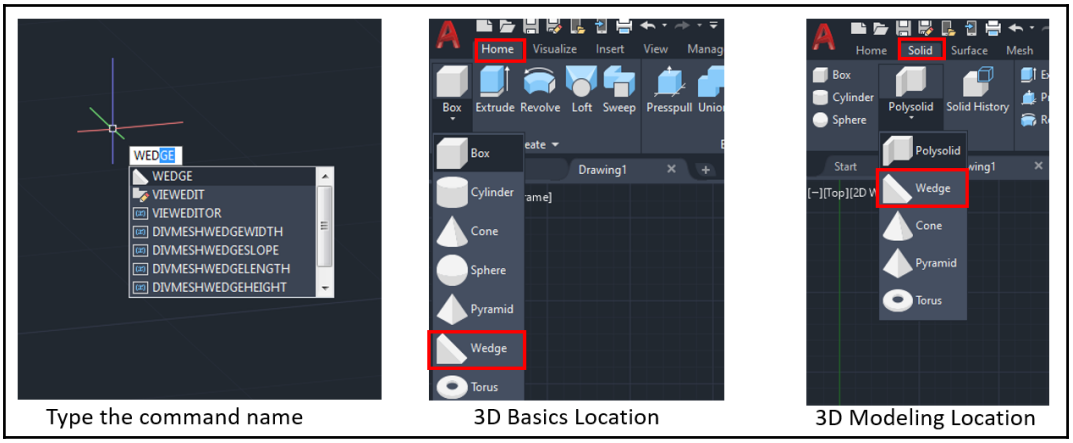


Figure 11.51: Alternative methods to start the Wedge command

The Wedge command is used to quickly and easily create 3D solid Wedges, as shown in the following step-by-step example:

1. Choose a location and click to specify the first corner in the Wedge's base.
2. Move the mouse and click to specify the location of the opposite corner in the Wedge base.
3. Move the mouse cursor to specify the height and click to finish. Alternatively, you can use the keyboard to enter the height value and then press *Enter* to finish.

The following screenshot is an illustration of the previous steps:

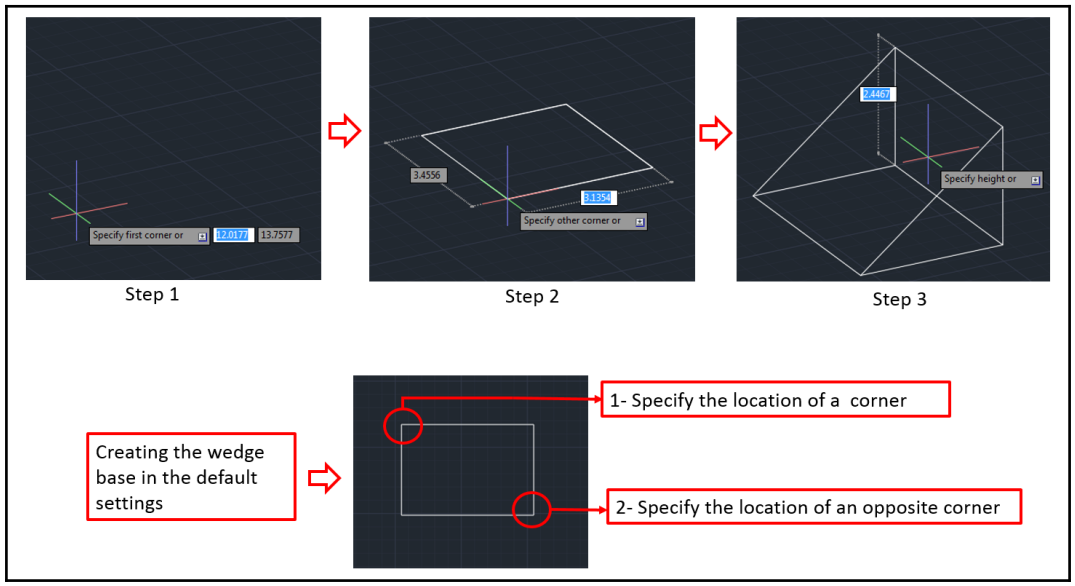


Figure 11.52: Steps for using the Wedge command

If you select the Wedge object, you will find a collection of different types of grab (control) points, and these can be divided into four categories as follows:

- On the corners of the Wedge's base, there are four small arrows to control the base shape.
- On the midpoints of the sides of the Wedge's base, there are four small arrows, also to control the base shape.
- At the midpoints of the Wedge back face, there are two small blue arrows pointing upward and downward. These can be used to control the Wedge height.

- In the middle of the bottom face, you find the pyramid base point, highlighted as a small square, as shown in the following screenshot:

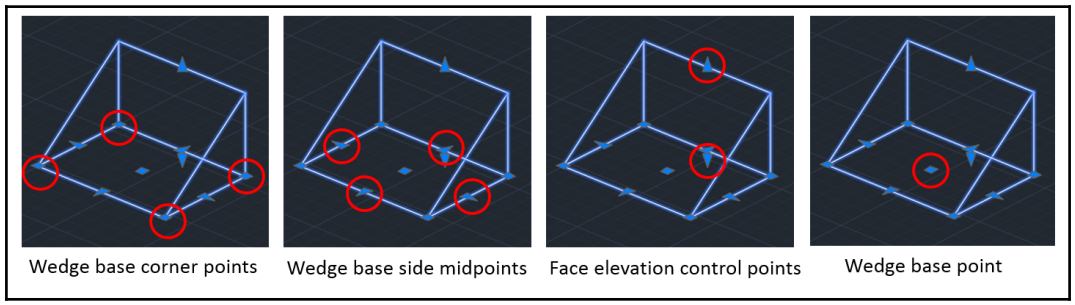


Figure 11.53: Wedge's grab points

The small points on the Wedge's base corners can be used to modify the base shape as follows:

1. Use the mouse cursor to pick one of the base corners.
2. Move the corner point to modify the base shape. Observe that the shape of the Wedge will change dynamically as the mouse moves.
3. Click again when you arrive at the desired location.

The following screenshot is an illustration of the previous steps:

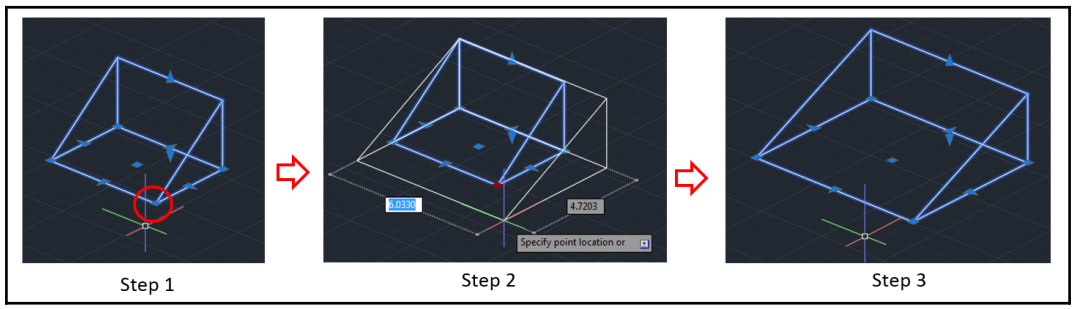


Figure 11.54: Modifying the Wedge's base using corner grab points

The small arrows on the Wedge's base sides can be used to modify the base shape as follows:

1. Use the mouse cursor to pick one of the midpoints on the base sides.
2. Move the arrow to modify the base shape. Observe that the shape of the Wedge will change dynamically as the mouse moves.
3. Click again when you arrive at the desired location.

The following screenshot is an illustration of the previous steps:

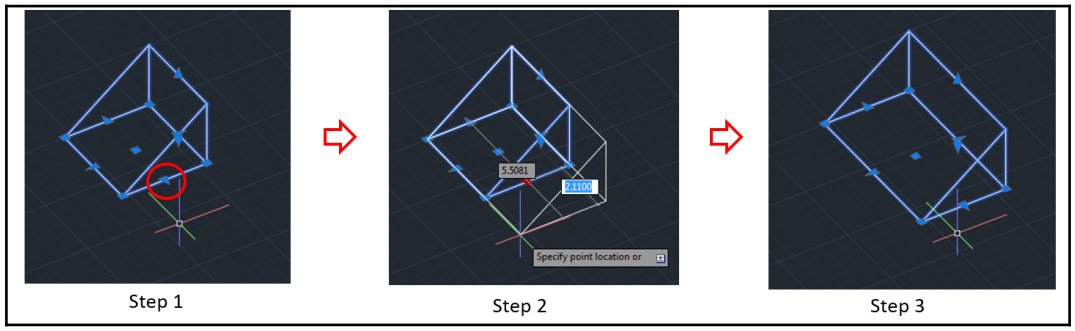


Figure 11.55: Modifying the Wedge's base using side grab points

The arrows at the bottom face, and at the midpoint of the upper Wedge edge, can be used to change the height as follows:

1. Pick the arrow at the midpoint of the upper edge, or the arrow in the bottom face center point.
2. Move the face to the new desired location. Observe that the shape of the pyramid will change dynamically as the mouse moves. You can also enter the desired value using the keyboard.
3. When you arrive at the desired location, click to finish.

The following screenshot is an illustration of the previous steps:

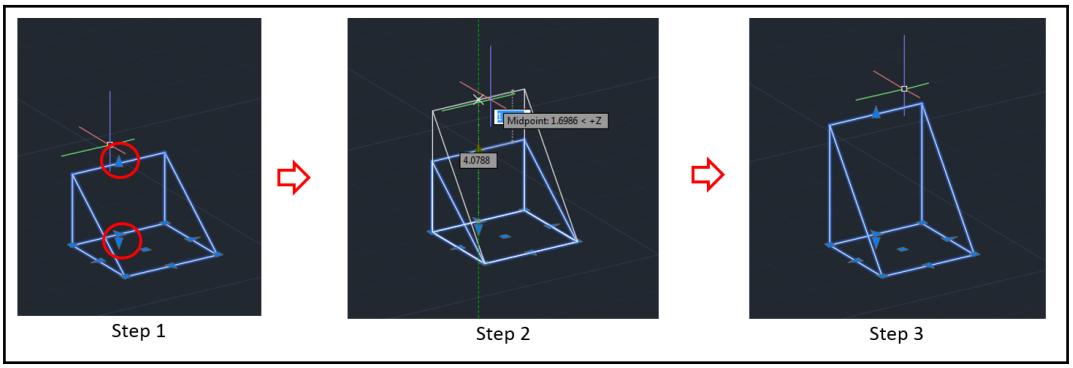


Figure 11.56: Modifying the Wedge's height using grab points

And finally, the base point (highlighted as a square in the bottom face center point) can be used to pick and move the pyramid.

There is a number of options associated with the `Wedge` command. These options make provision for alternative methods in creating the Wedge. The first set of options appears in the command bar right after you start the command, as shown in the following screenshot:

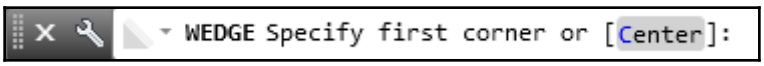


Figure 11.57: Additional options in the Wedge command

This set contains one option called **Center**. It can be selected by clicking on the option name in the command bar, or you can just type `C` and then press *Enter*. This option allows you to specify the location of the Wedge center instead of specifying a base corner in the default method. This option works exactly like the **Center** option in the `Box` command. Please refer to the *Creating Boxes* section for further details regarding this option.

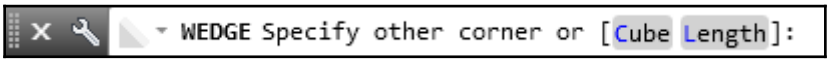


Figure 11.58: Second set of additional options in the Wedge command

The second set of options appears after you specify the first corner point of the Wedge base. This set contains two options:

- The first one is called **Cube**. This will restrict the length, width, and height of the Wedge to be the same. After selecting this option, you just specify one distance to draw the Wedge, similar to the Cube option in the Box command.
- The second one is called **Length**. This option allows you to individually enter a value for the length, the width, and the height of the Wedge, similar to the length option in the Box command. For further details, please refer to the length option in the *Creating Boxes* section.

The third set of options appears after the base is created. It has only one option, called **2Point**. This option is an alternative method for specifying the height of the Wedge; it is exactly the same as the **2Point** option in the `Box` command. For further details, please refer to the *Creating Boxes* section:

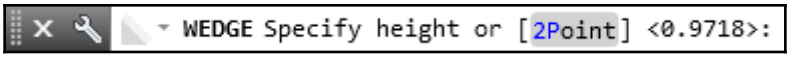


Figure 11.59: Third set of additional options in the Wedge command

We have learned how to create and quickly modify six different primitive shapes. Now, let's take a look at the last one, which is the Torus.

Creating a Torus

To start the `Torus` command, you can type the command name `TORUS` (or just type `TOR`) and then press *Enter*. Alternatively, you can click on the **Torus** button found in the **Create** panel in the **3D Basics** workspace, as shown in the following screenshot. If you are using the **3D Modeling** workspace, you will find the **Torus** button in the **Modeling** panel under the **Home** tab, or in the **Primitive** panel under the **Solid** tab, as shown in the following screenshot:

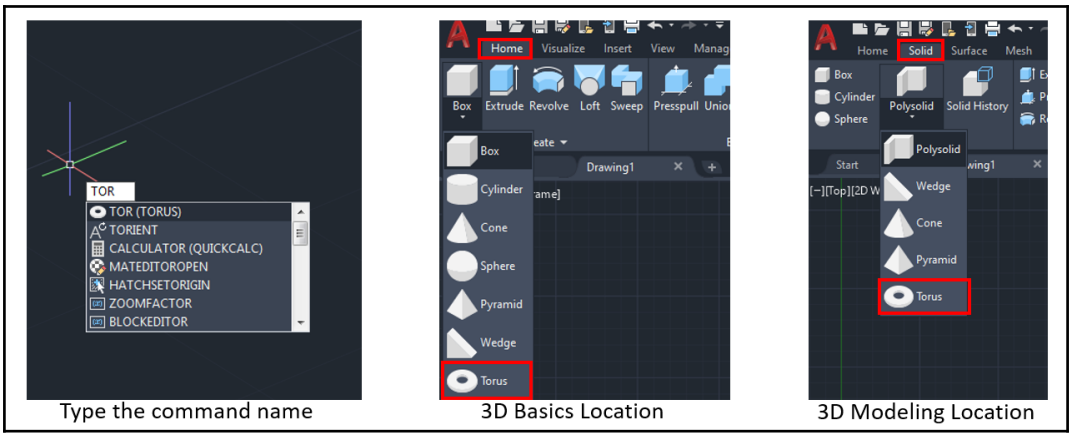


Figure 11.60: Alternative methods to start the Torus command

The `Torus` command can be used to quickly and easily create 3D solid Torus objects, as shown in the following step-by-step example:

1. After starting the command, use the mouse to specify the center point location of the Torus. Alternatively, you can enter the coordinates using the keyboard.
2. Move the mouse and click to specify the radius of the Torus. Alternatively, you can enter the radius value using the keyboard.
3. Move the mouse cursor to specify the tube radius and then click to finish. Alternatively, you can use the keyboard to enter the radius value and then press *Enter* to finish.

The following screenshot is an illustration of the previous steps:

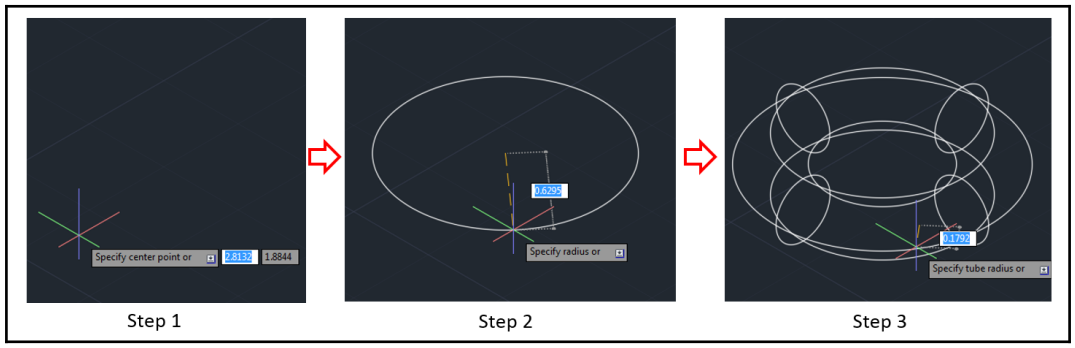


Figure 11.61: Steps for using the Torus command

If you select the Wedge object, you will find a collection of different types of grab (control) points, and these can be divided into three types as follows:

- On the tube circumference, there are four small arrows to control the tube radius.
- On the center line of the tube, there is a small arrow to control the Torus radius.
- At the center point, you will find the Torus base point, highlighted as a small square, as shown in the following screenshot:

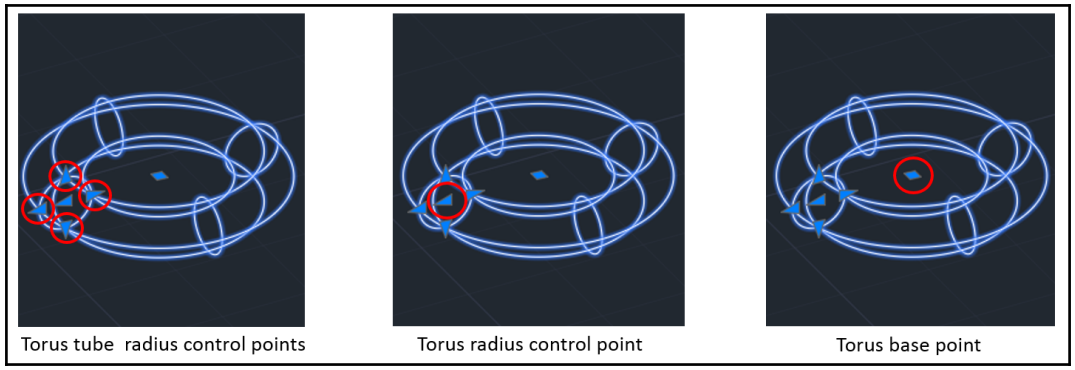


Figure 11.62: Torus grab points

The small arrows found on the tube circumference can be used to modify the tube radius, as shown in the following step-by-step example:

1. Use the mouse cursor to pick one of the tube radius control points.
2. Move the arrow to modify the tube radius. Observe that the shape of the Wedge will change dynamically as the mouse moves.
3. Click again to finish.

The following screenshot is an illustration of the previous steps:

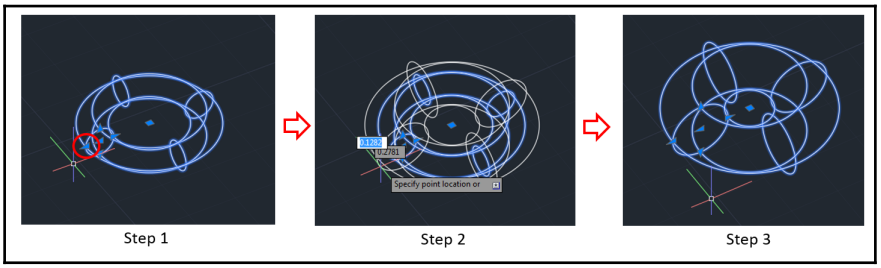


Figure 11.63: Modifying the Torus tube radius using grab points

The small arrow on the tube's center line can be used to modify the Torus radius, as shown in the following step-by-step example:

1. Pick the arrow on the Torus tube center line.
2. Move the arrow to modify the Torus radius. Observe that the shape of the Torus will change dynamically as the mouse moves. Alternatively, you can enter the desired value using the keyboard.
3. Click again to finish.

The following screenshot is an illustration of the previous steps:

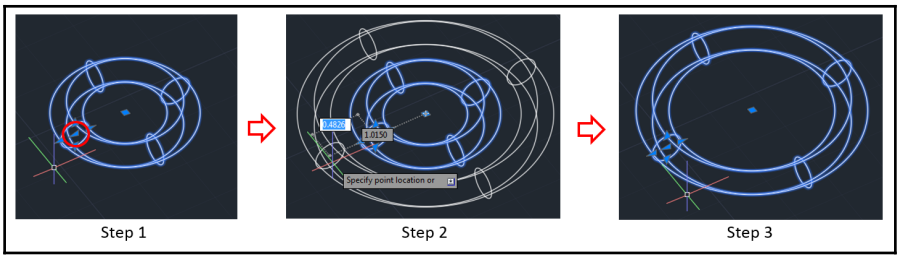


Figure 11.64: Modifying the Torus radius using grab points

And finally, the base point (highlighted as a square in the center point) can be used to pick and move the Torus.

There are a number of options associated with the `Torus` command. These options make provision for alternative methods in creating the Torus. The first set of options appears in the command bar right after you start the command, as shown in the following screenshot:

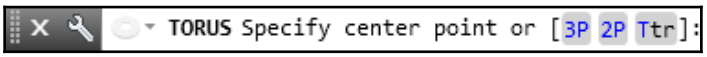


Figure 11.65: Additional options in the Torus command

The Torus is created by drawing a circle, which is the center line of the Torus tube, and then specifying the Torus tube radius. The options in this set are alternative methods for creating that circle. They are identical to the first set of options in the `Cylinder command`. Please refer to the *Creating Cylinders* section for details of these three options:

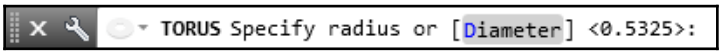


Figure 11.66: Second set of additional options in the Torus command

The second set of options appears after you specify the center point of the Torus. This set contains one option called **Diameter**. By selecting this option, you will specify the diameter of the Torus instead of the radius in the default setting.

The third set of options appears after the tube center line circle is created. This contains two options. The first one is **2Point**. This option is an alternative method for specifying the radius of the Torus tube. It is exactly the same as the **2Point** option in the `Box` command. For further details, please refer to the *Creating Boxes* section.

The second option is called **Diameter**. This can be selected to specify the diameter of the Torus tube instead of the radius in the default setting:

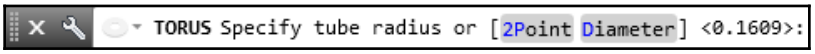


Figure 11.67: Third set of additional options in the Torus command

We have learned how to create the seven primitive 3D solid objects. These simple shapes can be combined together or modified to create more complex 3D models, as will be shown in the following chapters. For now, let's have a look at another type of 3D solid object in AutoCAD, which is known as a Polysolid.

Creating a Polysolid

A Polysolid is the 3D counterpart of the 2D polyline that was covered in `Chapter 2`, *Learning about Modify Commands* the difference being that a Polysolid has a height that makes it perfect for quickly creating solid walls. To start the `Polysolid` command, you can type the command name `POLYSOLID`, or just type `POLYS` and then press *Enter*. Alternatively, you can click on the **Polysolid** button found in the **Create** panel in the **3D Basics** workspace, as shown in the following screenshot. If you are using the **3D Modeling** workspace, you will find the **Polysolid** button in the **Modeling** panel under the **Home** tab, or in the **Primitive** panel under the **Solid** tab, as shown in the following screenshot:

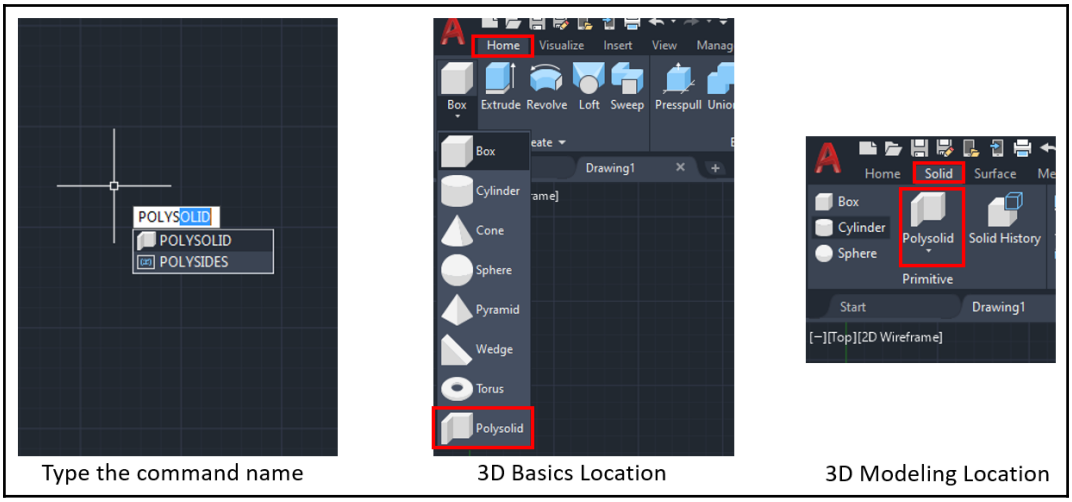


Figure 11.68: Alternative methods to start the Polysolid command

The `Polysolid` command is very similar to the 2D polyline command. You can create connected straight line or curved segments, and you can also create closed loops. It can be used to quickly create 3D walls. The following is a step-by-step example of how to use the `Polysolid` command:

1. After starting the command, choose a location and click to specify the starting point of the first section.
2. Move and click to specify the endpoint of the first section.
3. Another section will be created using the end of the previous section. Move the mouse again and click to create another section:

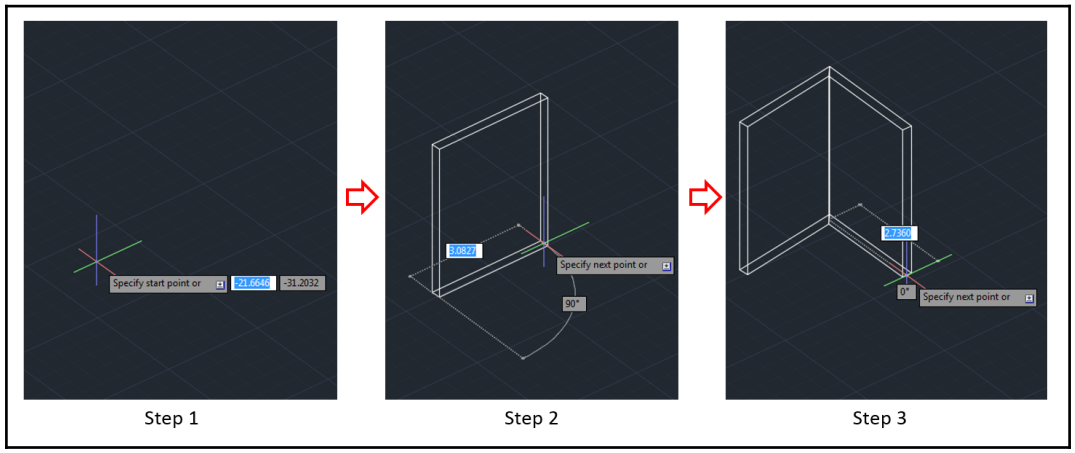


Figure 11.69: Steps for using the Polysolid command (1/3)

4. To create curved sections, click on the **arc** option in the command bar, or just type `A` and then press *Enter*.
5. Move and click to specify the endpoint of the arc section.
6. You can also create multiple curved sections:

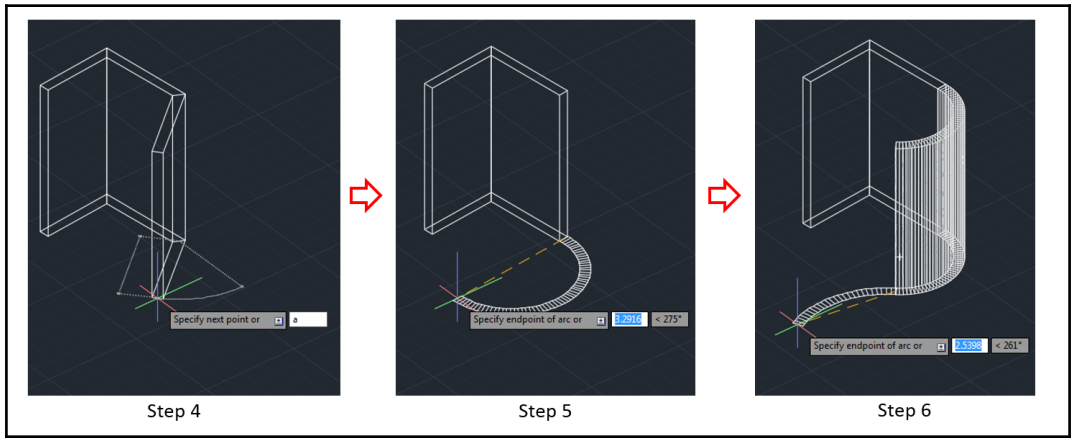


Figure 11.70: Steps for using the Polysolid command (2/3)

7. To return back to straight line sections, click on the **line** option in the command bar, or just type `L` and then press *Enter*.

8. To terminate the command, right-click and then press *Enter*:

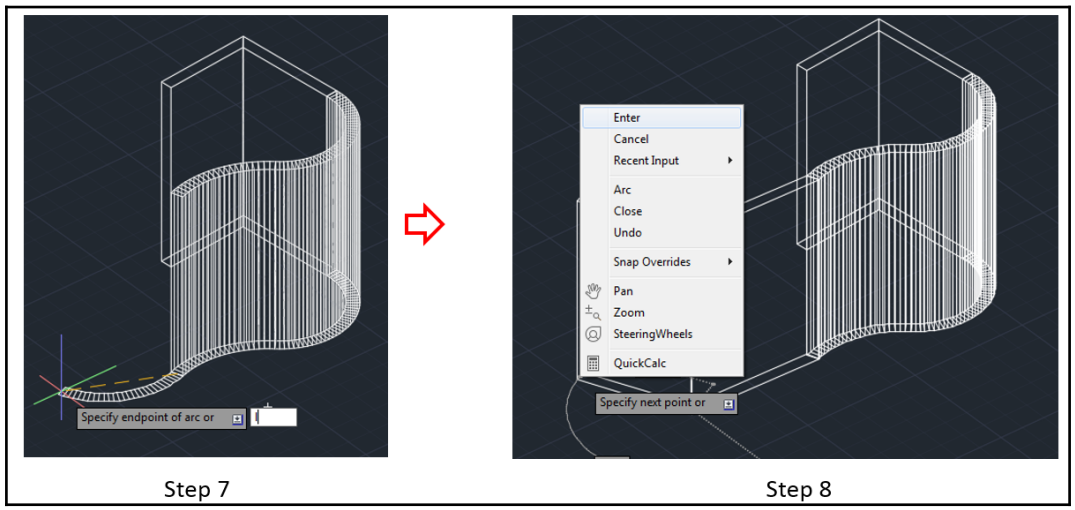


Figure 11.71: Steps for using the Polysolid command (3/3)

If you select the Polysolid object, you will find a collection of different types of control points, and these can be divided up as follows:

- On the center line of the bottom surface of the Polysolid, there is a control point at the ends of each section.
- On the center line of the bottom surface of the Polysolid, there is a control point at the midpoint of each section:

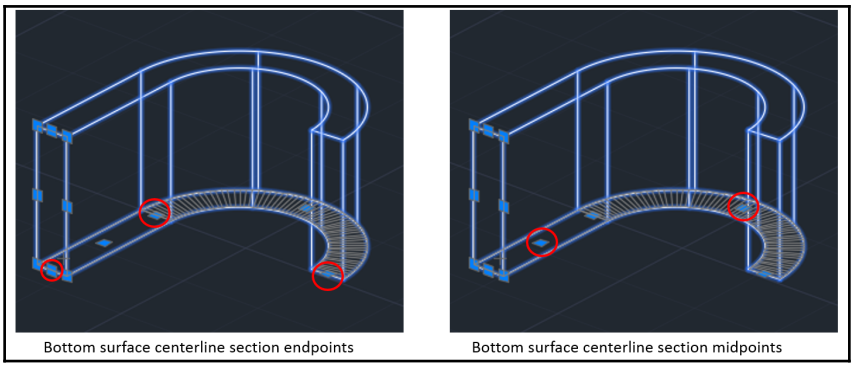


Figure 11.72: Polysolid's grab points (1/2)

- On the vertical surface of the starting section, there are four points at the corners.
- On the vertical surface of the starting section, there are two points at the midpoint of the Polysolid height:

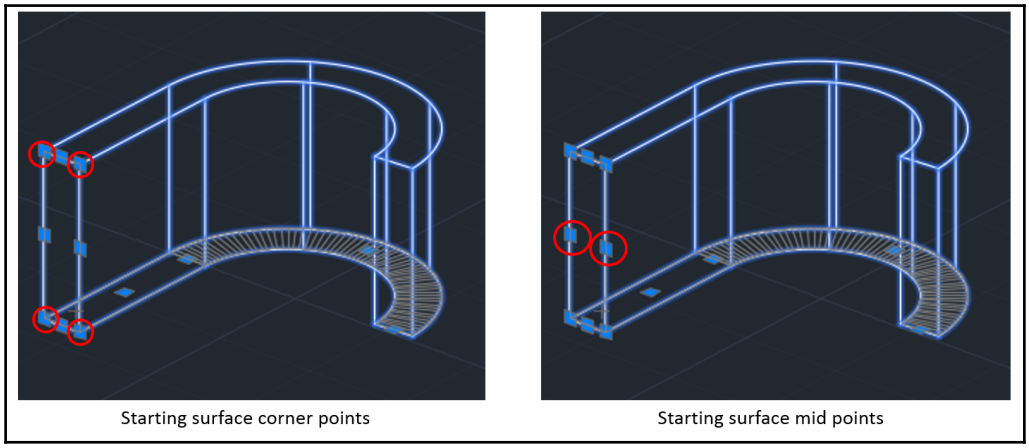


Figure 11.73: Polysolid's grab points (2/2)

Just like the normal 2D polyline, the section endpoint locations of the Polysolid can be modified as shown in the following step-by-step example:

1. Use the mouse cursor to pick one section endpoint.
2. Move the endpoint to the desired location. Observe that the shape of the Polysolid will change dynamically as the mouse moves.
3. Click again to finish.

The following screenshot is an illustration of the previous steps:

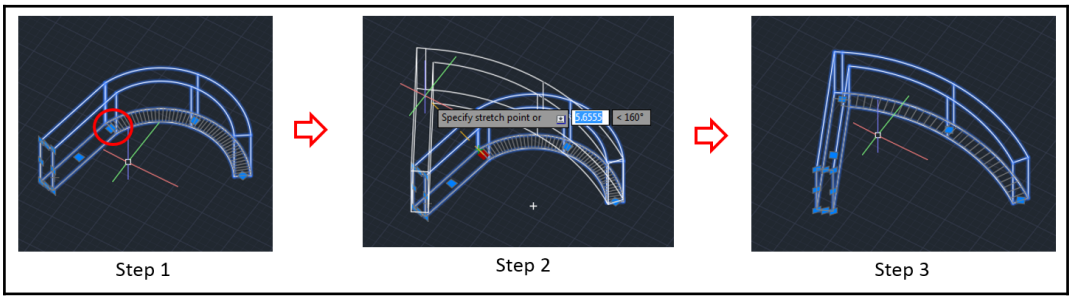


Figure 11.74: Modifying the Polysolid path using grab points .

Just like the normal 2D polyline, the section midpoints of the Polysolid can be modified as shown in the following step-by-step example:

1. Use the mouse cursor to pick one of the section midpoints.
2. Move to modify the location. For the straight section, the straight line sections will move without any stretching, and in the case of the curved sections, this will change the radius of the curve. Observe that the shape of the Polysolid will change dynamically as the mouse moves.
3. Click again to finish.

The following screenshot is an illustration of the previous steps:

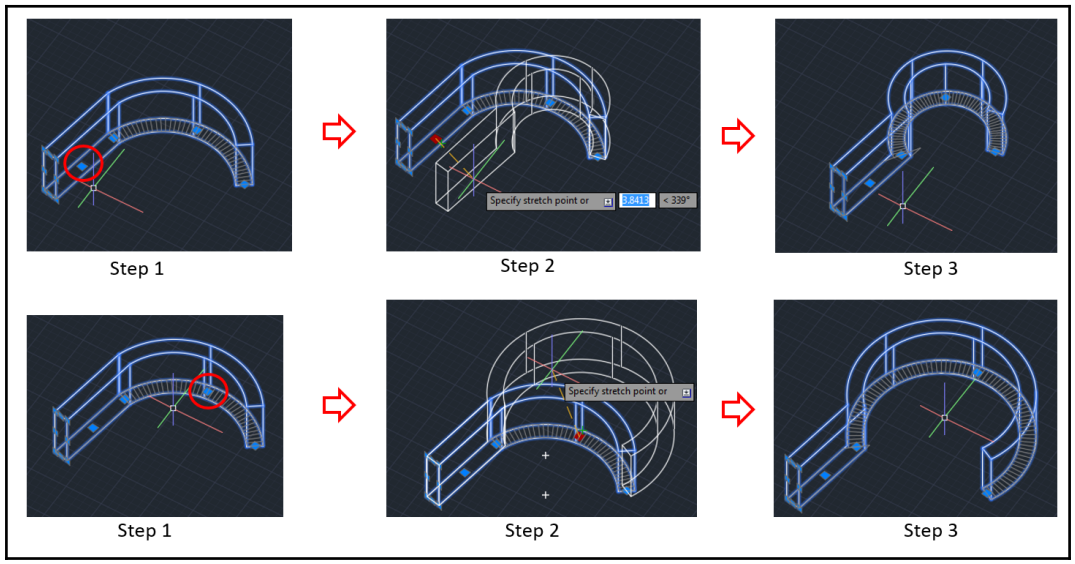


Figure 11.75: Modifying the Polysolid path using grab midpoints

The starting section corner points can be used to modify the Polysolid cross-section, as shown in the following step-by-step example:

1. Pick one of the starting section corner points.
2. Move to modify the shape of the Polysolid cross-section.
3. When you arrive at the desired location, click to finish.

The following screenshot is an illustration of the previous steps:

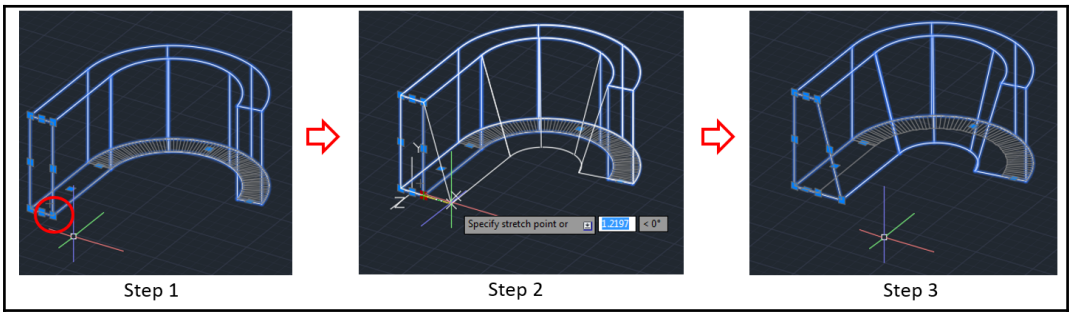


Figure 11.76: Modifying the Polysolid cross-section using grab points

The starting section midpoints can be used to change the Polysolid width, as shown in the following step-by-step example:

1. Pick one of the starting section midpoints.
2. Move to modify the Polysolid width. Observe that the shape of the Polysolid will change dynamically as the mouse moves. You can also enter the desired value using the keyboard.
3. When you arrive at the desired location, click to finish.

The following screenshot is an illustration of the previous steps:

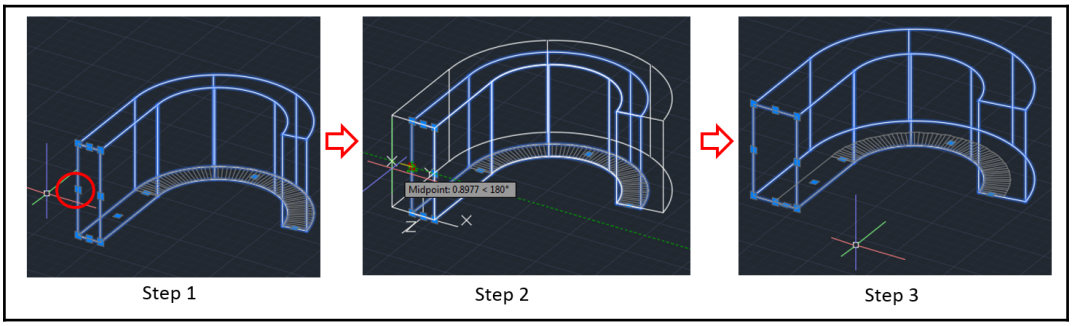


Figure 11.77: Modifying the Polysolid cross-section using grab midpoints

This brings us to the end of this chapter. We have now completed the set of primitive 3D solids in AutoCAD.

Summary

We have completed our journey of creating primitive shapes in AutoCAD. These shapes can be used as a starting point to create the more complex lifelike 3D models for your projects. As we saw, AutoCAD offers a wide variety of these shapes with many different alternatives and options for each one.

In the next chapter, we will look at another method for creating 3D solids, which is to create them from a 2D shape, and we will also look at how to perform this operation in reverse, which is to create a 2D shot or section from an existing 3D model.

12
Conversion between 2D and 3D

It is very common during the process of 3D modeling to break down the creation process into drawing 2D shapes (representing cross-sections or bases), then extruding or revolving to create the 3D object. Another possible scenario is to have an entire 2D drawing that needs to be converted to a 3D model. In this chapter, the following topics will be discussed in detail (these are the 2D-to-3D conversion commands):

- Using the `Extrude` command
- Using the `Revolve` command
- Using the `Sweep` command
- Using the `Loft` command
- Using the `Presspull` command

In addition to that, the opposite scenario might be the case, where a 3D model is present and we want to create a 2D shape from it. This task can easily be done using either of the following:

- The `Flatshot` command
- The `Section` tool

Both of these command tools will also be demonstrated in detail in this chapter. Let's now start with the `Extrude` command.

Using the Extrude command

Extrude is the first of the group of commands that are used to convert 2D to 3D objects, and it is a mandatory operation for any 3D modeling software, as many 3D bodies can simply be created by drawing a 2D base then extruding it. To start the `Extrude` command, you can type the command name `EXTRUDE`, or just type `EXT` then press *Enter*. Alternatively, you can click on the **Extrude** button found in the **Create** panel in the **3D Basics** workspace, and if you are using the **3D Modeling** workspace, you will find the **Extrude** button in the **Modeling** panel under the **Home** tab, or in the **Solid** panel under the **Solid** tab as shown here:

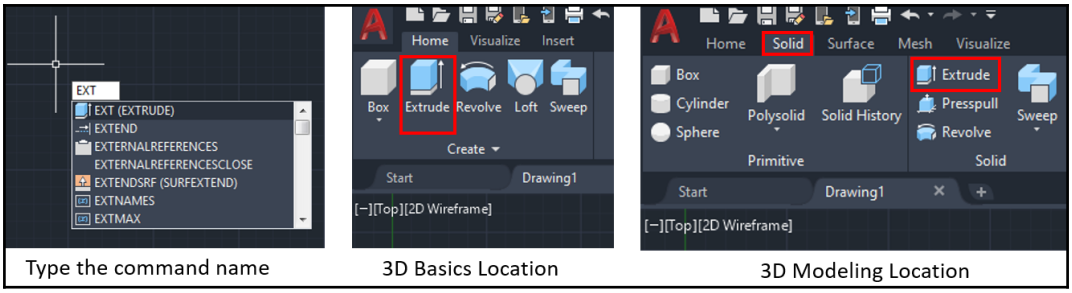


Figure 12.1: Alternative methods of starting the Extrude command

Depending on the selected object, the `Extrude` command will create a 3D solid or a surface. Generally, if you select a surface or a 2D closed shape (such as a circle, a polygon, or a closed polyline), the `Extrude` command will create a 3D solid, else if the selected object is an open 2D shape (like a line or an open curve), the extruded object will be a surface, as shown in the following step-by-step example:

1. Start the `Extrude` command.
2. Select one or multiple 2D shapes or surfaces, then press *Enter* to end the selection stage.
3. Move the cursor to specify the height. Alternatively, you can type the height using your keyboard.
4. When you arrive at the required height, click again or press *Enter* to finish.

Here's a pictorial representation for the previous steps:

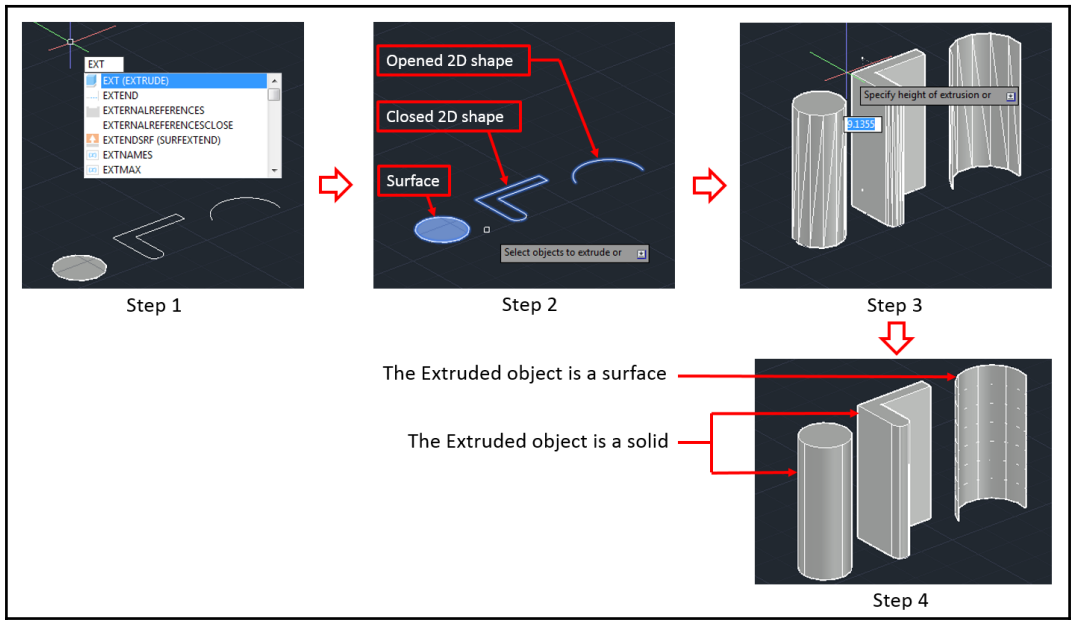


Figure 12.2: Steps for using the Extrude command

There are a number of options associated with the `Extrude` command. These options allow for alternative methods and settings. The first set of options appear in the command bar right after you start the command as shown here:

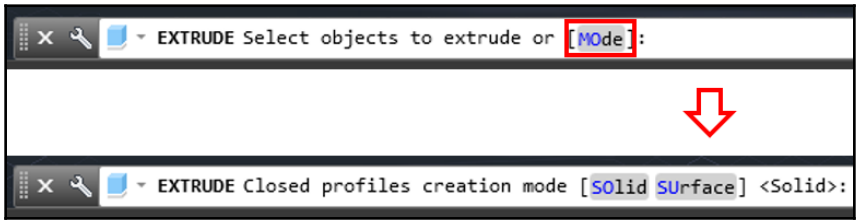


Figure 12.3: Using the Mode option

This set contains only one option called **Mode**. To select this option, you can just type `MO` and press *Enter* or you can click on the option name in the command bar. This option is used to determine the type of the extruded object, either to be a solid or a surface. By default, when you select a closed shape or a surface the extruded object will be a solid. This option allows you to create a surface even if the extruded object is a closed shape or a surface.

The second set of options appear after you select the objects to be extruded. This set contains four different options as shown in the following screenshot:

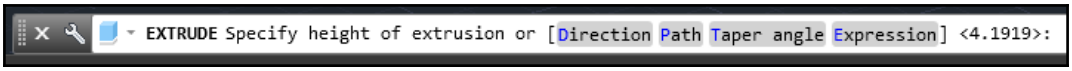


Figure 12.4: Additional options in the Extrude command

The first one is called **Direction**. To select it you can just type `D` and press *Enter*, or you can click on the option name in the command bar. This option is used to change the extrusion direction. By default, the extrusion direction is normal to the 2D shape or surface selected. By using the direction option you can specify any direction for the extrusion as shown in the following step-by-step example:

1. After starting the `Extrude` command, select the surfaces or 2D shapes to be extruded, then press *Enter*.
2. Type `D` and press *Enter* to activate the direction option. Notice that before you activate the direction option, the extrusion was perpendicular to the selected objects, although they had different orientations.
3. The extrusion direction is determined by specifying two points. Choose a location and click to specify the first point.
4. Click to specify the second point, and this will create the extruded object.

The following figure shows an illustration of the previous steps:

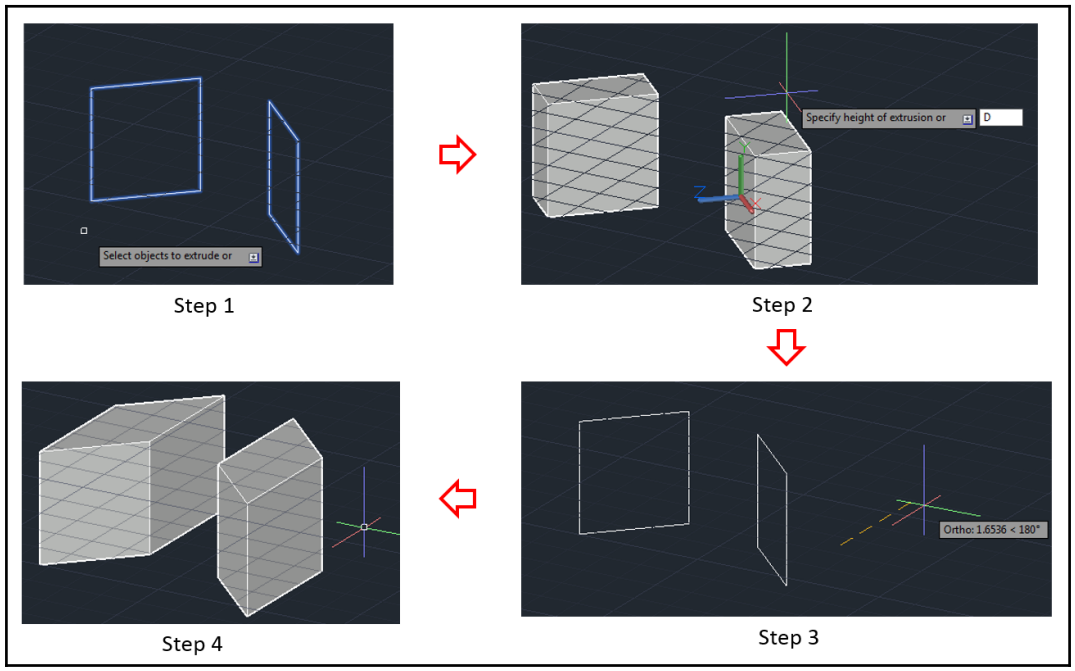


Figure 12.5: Steps for using the Direction option

The next option in the second set is called **Path**. To select this option, type `P` and then press *Enter*, or you can click on the option name in the command bar. This option allows you to use a line or curve as an extrusion as shown in the following step-by-step example:

1. Start the `Extrude` command, select the objects to be extruded, and then press *Enter*.
2. Type `P` then press *Enter* to activate the path option.
3. Click on the line, polyline, or curve that will be used as a path. (Notice that the path is not necessarily attached to the object.)
4. Once you click, the objects will be extruded along the selected path.

The following figure shows an illustration of the previous steps:

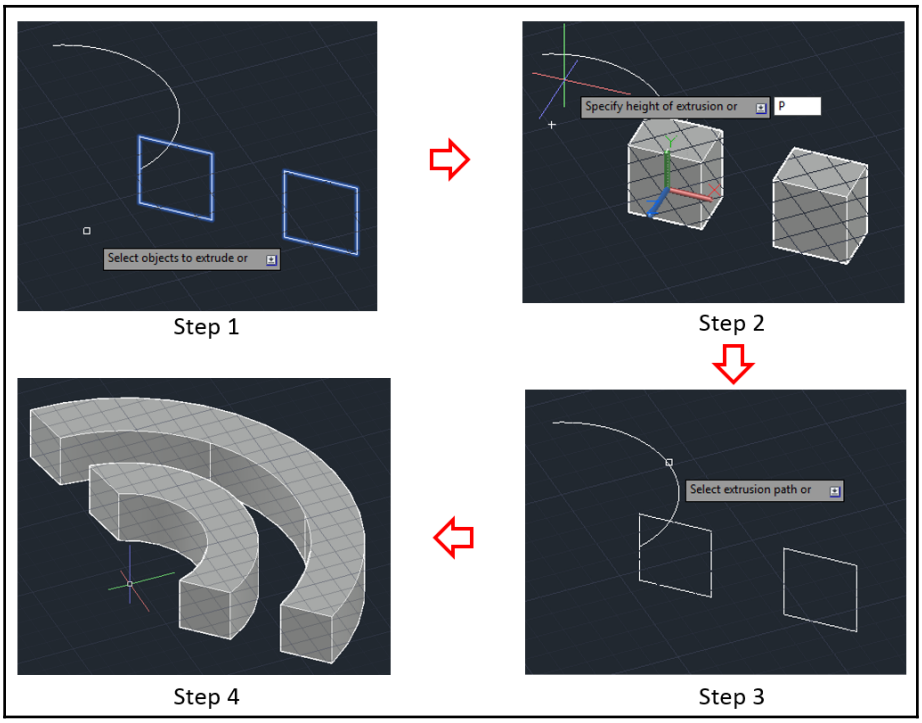


Figure 12.6: Steps for using the Path option

The third option is called **Taper**, to select this option you just type `T` then press *Enter*, or you can click on the option name in the command bar. This option allows you to add a taper angle to the extrusion as shown in the following step-by-step example:

1. Start the `Extrude` command, select the objects to be extruded, then press *Enter*.
2. Type `T` and press *Enter* to activate the Taper option.
3. Use the mouse movement to specify the required taper angle, or type the angle value using the keyboard, then press *Enter*. The positive angle values will taper inward while the negative angle values will taper outward.
4. After that, specify the height using the mouse movement or by typing the value using the keyboard. Notice that the taper angle value is maintained while you change the height.
5. When you arrive at the desired height, click or press *Enter* to finish.

The following figure shows an illustration for the previous steps:

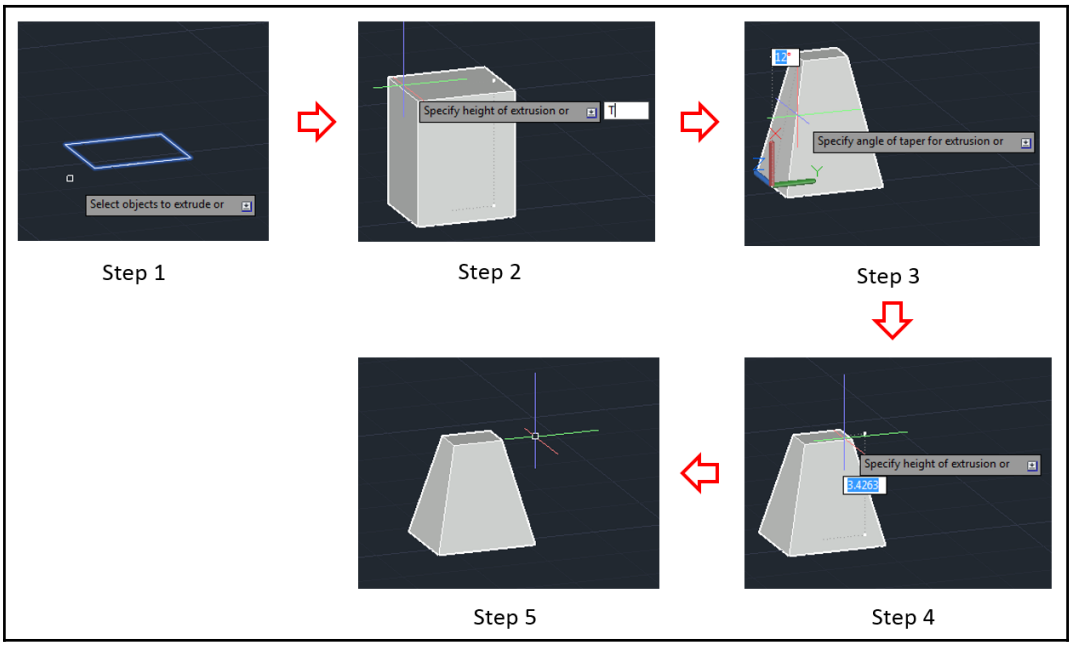


Figure 12.7: Steps for using the Taper option

The last option is called **Expression**. To select this option, you just type `E` and then press *Enter*, or you can click on the option name in the command bar. This option allows you to enter a formula or an expression that will be calculated and the result will be used as the extrusion height.

For example, to create an extrusion with a height of `10`, instead of directly typing the value `10` you just can enter the expression `(5+5)`, which have the same result. Another example is to click on a defined length constrain in the model, and the extrusion height will be set to the same value automatically.

By default, the extruded objects and the extrusion paths will be deleted after the `Extrude` command ends. You can control whether to delete these objects or not using the **DELOBJ** system variable. To access that, just type `DELOBJ` and press *Enter*, the current value of the **DELOBJ** system variable will be displayed, then you can type the new value and press *Enter* as shown in the following figure:



Figure 12.8: Changing the DELOBJ value

The following table shows the different values for the **DELOBJ** system variable and the corresponding action for each value:

Value	Function
0	Nothing will be deleted.
1	Deletes the profile curves or cross-sections, but keep the path curves.
2	Deletes all the paths and cross-sections for the `SWEEP` and `LOFT` commands.
3	All defining geometries is deleted for the `SWEEP` and `LOFT` commands, and in the case of creating a solid object using `CONVTOSOLID`, `CONVTOSURFACE`, `CONVTONURBS`, and `CONVTOMESH` commands.
-1	Prompts are displayed before deleting the defining geometries, including paths and guide curves used with the `SWEEP` and `LOFT` commands. The geometries used in the `CONVTOSOLID`, `CONVTOSURFACE`, and `CONTOMESH` commands are removed directly with no prompting.
-2	Prompts are displayed before deleting all defining geometries, including paths and guide curves used with the `SWEEP` and `LOFT` commands. Original geometries for `CONVTOSOLID`, `CONVTOSURFACE`, `CONVTONURBS`, and `CONVTOMESH` commands are removed without prompting.
-3	Prompts are displayed before deleting all defining, original geometries are removed without prompting for `CONVTOSOLID`, `CONVTOSURFACE`, `CONVTONURBS`, and `CONVTOMESH` commands.

We have learned about using the Extrude command, which is the first one in the 2D-to-3D group of commands. Now let's have a look at the second one in our list, which is the `Revolve` command.

Using the Revolve command

The `Revolve` command is conceptually the same as the `Extrude` command: you have a 2D base shape but instead of extruding it, it is revolved around a specified axis and with a given angle. The same results can be created using Extrude but you will have to draw the exact path, and it is much easier to just specify the axis of rotation and the angle, as will be shown in this section. To start the `Revolve` command you can type the command name `REVOLVE`, or just type `REV` then press *Enter*. Alternatively, you can click on the **Revolve** button found in the **Create** panel in the **3D Basics** workspace, and if you are using the **3D modeling** workspace, you will find the **Revolve** button in the **Modeling** panel under the **Home** tab, or in the **Solid** panel under the **Solid** tab, as shown in the following screenshots:

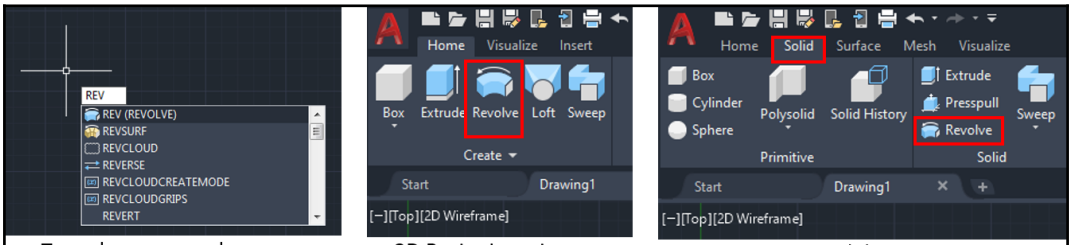


Figure 12.9: Alternative methods to start the Revolve command

Depending on the selected object, the `Revolve` command will create a 3D solid or a surface. Generally, if you select a surface or a 2D closed shape (like a circle, a polygon, or a closed polyline), the `Revolve` command will create a 3D solid, else if the selected object is an open 2D shape (such as a line or an open curve), the revolved object will be a surface, as shown in the following step-by-step example:

1. After starting the `Revolve` command, select the object or the objects that will be revolved, then press *Enter* to end the selection stage.
2. Specify the revolving axis by specifying two points. You can use the mouse or type the coordinates of the two points using the keyboard.

3. Move the mouse specify the revolve angle. Alternatively, you can enter the angle value using the keyboard.
4. When you arrive at the required revolve angle, click or press *Enter* to finish.

The following figure shows an illustration for the previous steps:

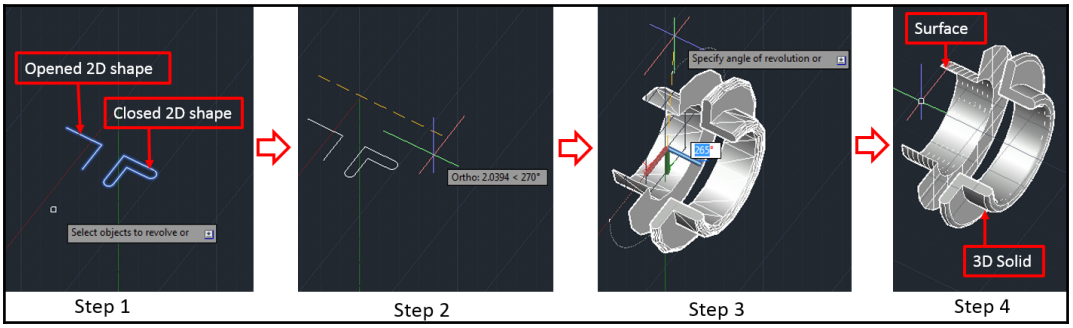


Figure 12.10: Steps for using the Revolve command

There is a number of options associated with the `Revolve` command. These options allow for alternative methods and settings. The first set of options appears in the command bar right after you start the `Revolve` command, as shown in the following figure:

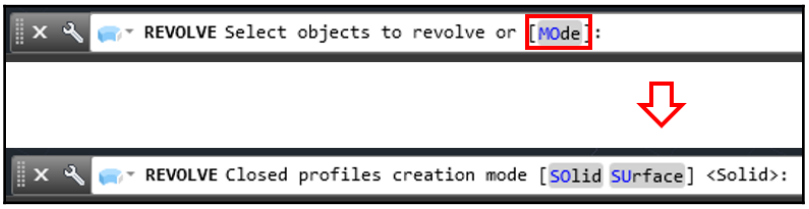


Figure 12.11: Different modes for the Revolve command

This set contains only one option called **Mode**. To select this option, you can just type `MO` and press *Enter*, or you can click on the option name in the command bar. This option is used to determine the type of the Revolved object, either to be a solid or a surface. By default, when you select a closed shape or a surface, the `Revolve` command will produce a solid. This option allows you to create a surface even if the revolved object is a closed shape or a surface.

The second set of options appear after you select the objects to be revolved. This set contains four different options as shown in the following figure:

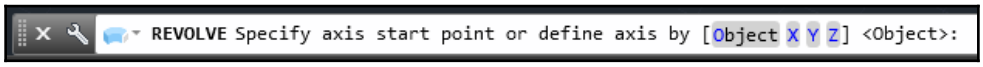


Figure 12.12: The second set of additional options in the Revolve command

The first one is called **Object**. To select it, you can just type `O` and press *Enter*, or you can click on the option name in the command bar. This option allows you to select an object to be the revolve axis as shown in the following step-by-step example:

1. Start the `Revolve` command, select the object to be revolved, and press *Enter* to end the selection stage.
2. Type `O` then press *Enter* to activate the Object option.
3. Select an object to be the revolve axis (for example, the small line).
4. Move to determine the revolve angle, then click to finish.

The following figure is an illustration of the previous steps:

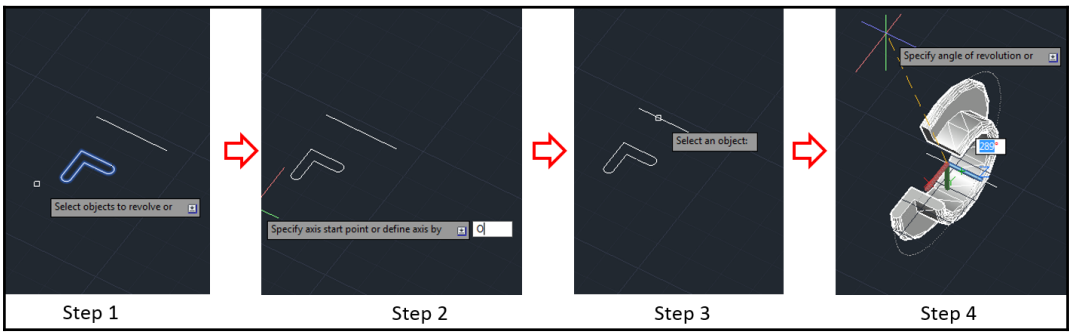


Figure 12.13: Using the Object option

The next options in the second set are called **X**, **Y**, and **Z**. To select any of them, just type the option name then press *Enter*, or you can click on the option name in the command bar. These options allow you to use the **user coordinate system** (**UCS**) axis as the revolve axis.

The third option set appears after you specify the revolve axis. This set contains three options as shown here:

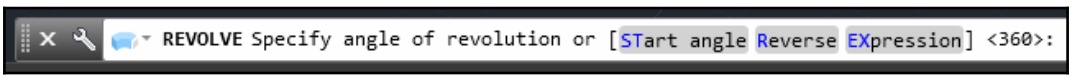


Figure 12.14: The third set of additional options in the Revolve command

The first one is called **Start angle**. To select it you can just type `ST` and press *Enter*, or you can click on the option name in the command bar. By default the `Revolve` command will start the revolution from the selected object. This option allows you to set an angle between the object you selected and the start of the revolved objects as shown in the following step-by-step example:

1. Start the `Revolve` command, select an object to be revolved, then specify the revolve axis.
2. After you specify the revolve axis, type `ST` and press *Enter* to activate the **STart angle** option.
3. Move the mouse cursor to set the required start angle then click. Alternatively, you can type the desired start angle using the keyboard then press *Enter*.
4. Specify the revolve angle using the mouse movement or the keyboard. Notice that the angle will start from the specified start angle.
5. After you arrive at the required angle, click again or press *Enter* to finish.

The following figure shows an illustration for the previous steps:

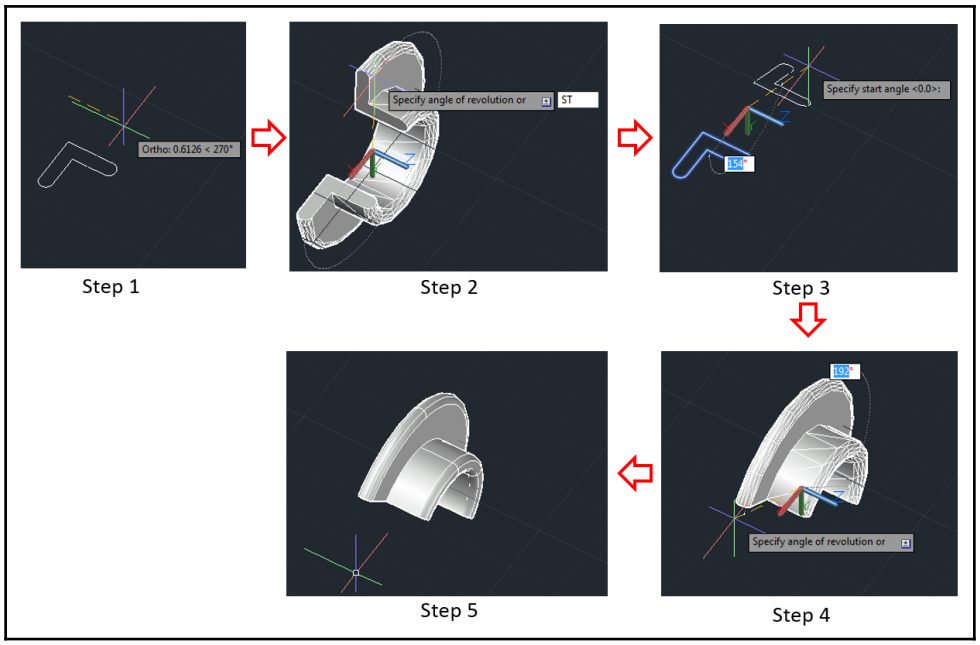


Figure 12.15: Steps for using the Start angle option

The next option is called **Reverse**. To select this option, you can type `R` and press *Enter*, or you can click on the option name in the command bar. During the specification of the revolve angle, activate this option to reverse rotation angle direction as shown in the following figure:

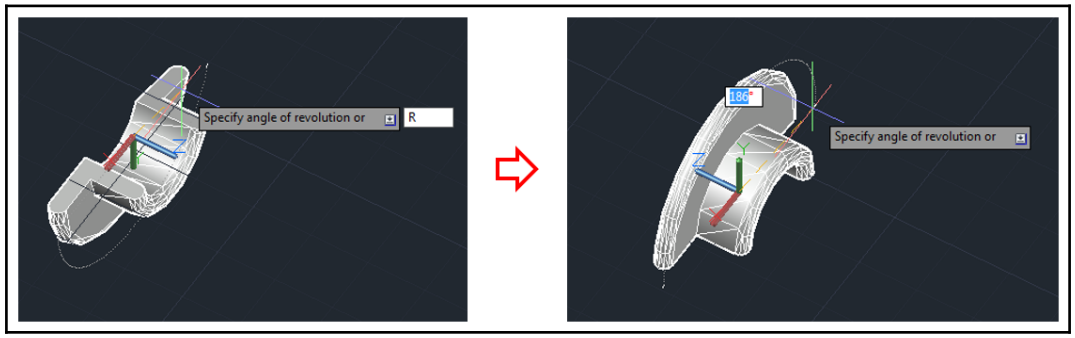


Figure 12.16: Using the Reverse option

The last option is called **Expression**. To select this option you just type `E` then press *Enter*, or you can click on the option name in the command bar. This option allows you to enter a formula or an expression that will be calculated and the result will be used as the revolve angle.

By default, the revolved objects will be deleted after the `Revolve` command ends. You can control whether to delete these objects or not using the DELOBJ system variable, which you can access just by typing `DELOBJ` and pressing *Enter*, and the current value of the DELOBJ system variable will be displayed for you to type the new value and press *Enter* again. For more details about the DELOBJ system variable, please refer to the *Using the Extrude command* section of this chapter.

Now that we have learned how to use two commands of the 2D-to-3D conversion group, let's take a look at the third one, which is the Sweep command.

Using the Sweep command

`Sweep` is very similar to the `Extrude` command—a 2D-cross section shape is extruded (or swept) along a specified path, but the difference here is that if the 2D base shape and the path are separated, the `Extrude` command will create the object at the 2D location, while the `Sweep` command will create it at the path location. In addition to that, the `Sweep` command has more options for the behavior of the base shape along the path such as the Alignment and the Twist angle options, which are not present in the `Extrude` command. To start the `Sweep` command you can type the command name `SWEEP`, or just type `SW` then press *Enter*. Alternatively, you can click on the **Sweep** button found in the **Create** panel in the **3D Basics** workspace, and if you are using the **3D modeling** workspace, you will find the **Sweep** button in the **Modeling** panel under the **Home** tab, or in the **Solid** panel under the **Solid** tab, as shown in the following screenshots:

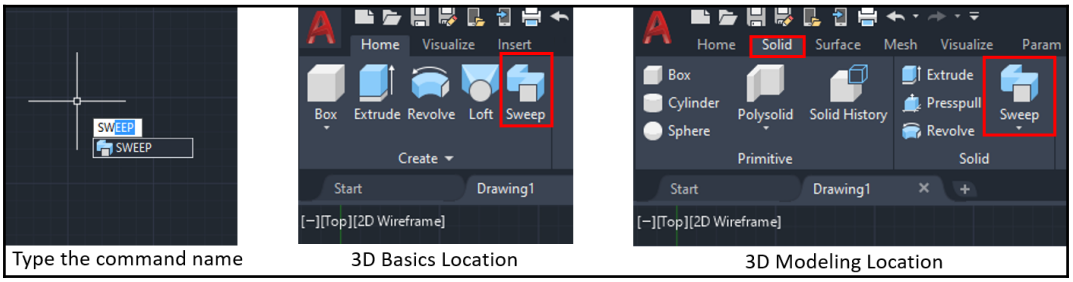


Figure 12.17: Alternative methods for starting the Sweep command

Depending on the selected object, the `Sweep` command will create a 3D solid or a surface. Generally, if you select a surface or a 2D closed shape (such as a circle, a polygon or a closed polyline), the `Sweep` command will create a 3D solid, else if the selected object is an open 2D shape (such as a line or an open curve), the extruded object will be a surface, as shown in the following steps:

1. After starting the `Sweep` command, select the object that will be swept, then press *Enter* to end the selection stage.
2. Click to select a sweeping path; the sweep path can be a line, a polyline, or a curve.
3. Once you click, the swept volume will be created.

The following figure shows an illustration for the previous steps:

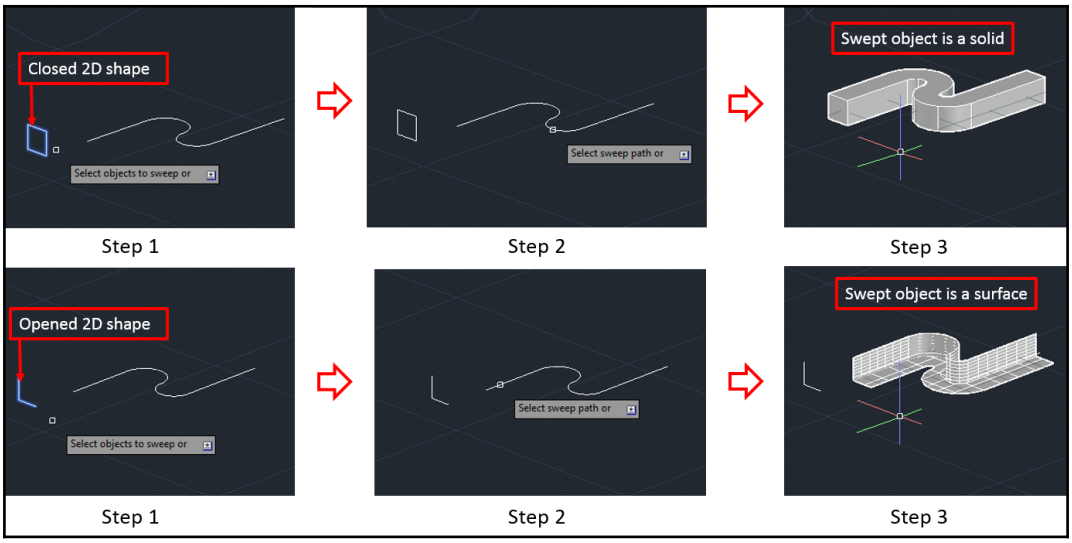


Figure 12.18: Using the Sweep command

There are a number of options associated with the `Sweep` command. These options allow for alternative methods and settings.

The first set of options appear in the command bar right after you start the `Sweep` command, as shown in the following figure:

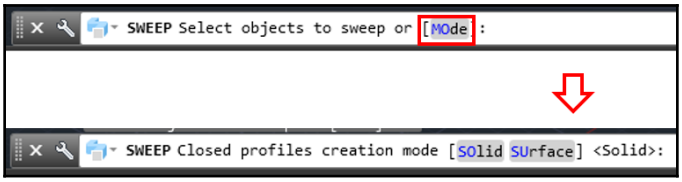


Figure 12.19: Changing the mode of the Sweep command

This set contains only one option called **Mode**. To select this option, you can just type `MO` and press *Enter* or you can click on the option name in the command bar. This option is used to determine the type of the Swept object, either to be a solid or a surface. By default when you select a closed shape or a surface, the `Sweep` command will produce a solid. This option allows you to create a surface even if the Swept object is a closed shape or a surface.

The second set of options appear after you select the objects to be swept. This set contains four different options as shown here:

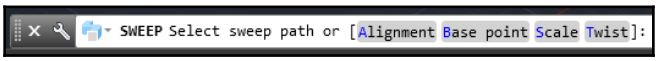


Figure 12.20: Second set of additional commands in the Sweep command

The first one is called **Alignment**. To select it, you can just type `A` and press *Enter*, or you can click on the option name in the command bar, then you choose whether to set it to **Yes** or **No**. **Yes** is the default and by choosing that, the swept object is automatically aligned to be perpendicular to the sweep path.

If you select **No**, the sweep object keeps its relative angle with the sweep path as shown in the following figure:

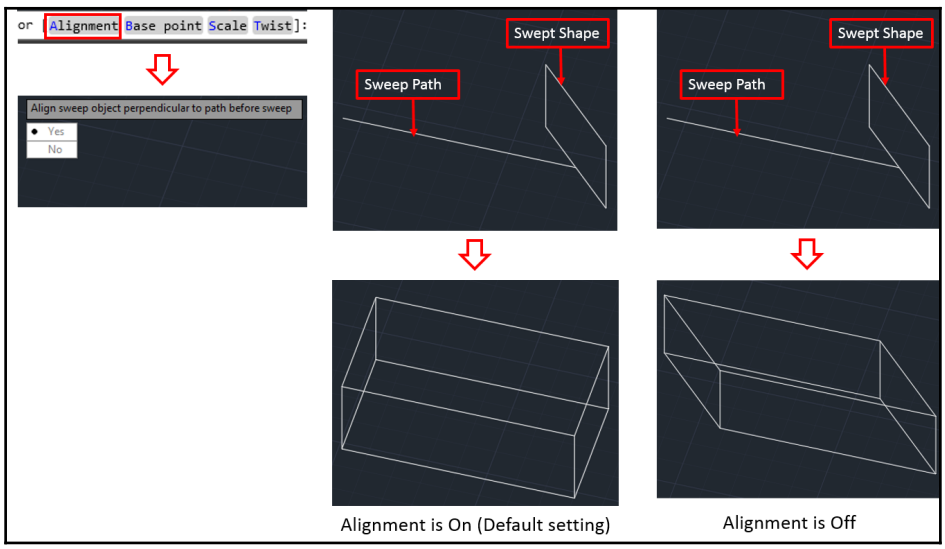


Figure 12.21: Using the Alignment option

The next option in the second set is called **Base point**. To select this option, type `B` then press *Enter*, or you can click on the option name in the command bar. This option allows you to set the sweeping base point. By default, the swept shape is automatically moved so that its center point is on the sweep path, or in other words the center point of the selected shape is used as the sweep base point. When you activate the base point option, you can click to specify or type the coordinates of a new sweep base point.

The third option is called **Scale**. To select this option, type `S` then press *Enter*, or you can click on the option name in the command bar. This option allows you to apply a scaling factor for the swept shape along the sweep path as follows:

1. Start the `Sweep` command, select the object to be swept, and press *Enter* to end the selection stage.
2. Type `S` to activate the Scale option.
3. Type the scaling factor and press *Enter*.
4. Click to select the sweep path, and the swept volume will be created. Notice that the scaling factor is applied at the side where the object was.

The following figure is an illustration of the previous steps:

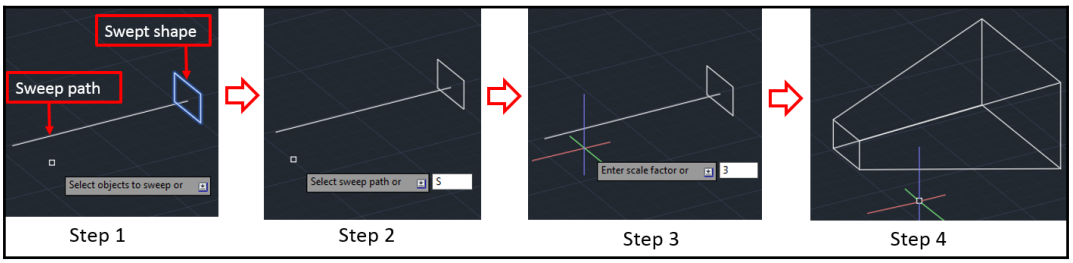


Figure 12.22: Steps for using the Scale option

The fourth option is called **Twist**. To select this option, you just type `T` and then press *Enter*, or you can click on the option name in the command bar. This option allows you to apply a gradual twist angle along the sweep path as follows:

1. Start the `Sweep` command, select the objects to be swept, and then press *Enter*.
2. Type `T` and press *Enter* to activate the **Twist** option.
3. Type the required twist angle and press *Enter*.
4. Select the twist path, and the swept volume will be created with the specified twist angle.

The following figure shows an illustration of the previous steps:

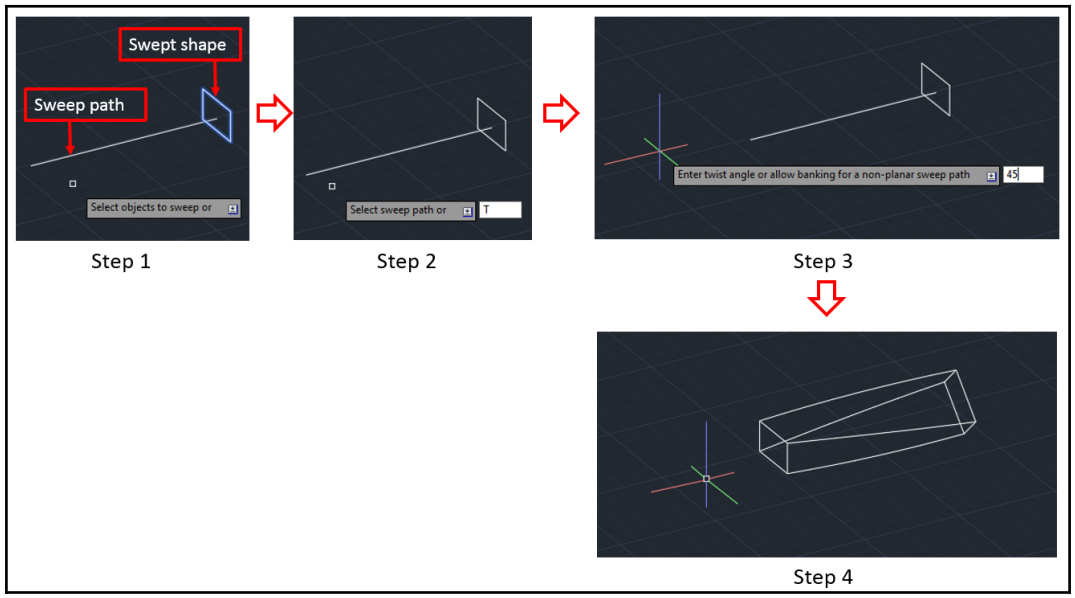


Figure 12.23: Steps for using the Twist option

By default, the swept objects and the paths will be deleted after the `Sweep` command ends. You can control whether to delete these objects or not using the DELOBJ system variable. To access that, just type `DELOBJ` and press *Enter*, and the current value of the DELOBJ system variable will be displayed, then you can type the new value and press *Enter*. For more details about the DELOBJ system variable please refer to the *Using the Extrude command* section of this chapter.

We have learned about three commands in the 2D-to-3D conversion group, so let's now take a look at the fourth one, which is the `Loft` command.

Using the Loft command

Unlike the previous three commands, where one base shape is selected and then extruded to create the 3D body, Loft allows you to create a 3D body that has different cross-section shapes, which opens up the possibilities and the complexities of the 3D bodies that can be created. To start the `Loft` command, you can type the command name `LOFT`, or just type `LOF` then press *Enter*. Alternatively, you can click on the **Loft** button found in the **Create** panel in the **3D Basics** workspace, and if you are using the **3D modeling** workspace, you will find the **Revolve** button in the **Modeling** panel under the **Home** tab, or in the **Solid** panel under the **Solid** tab as shown in the following figure:

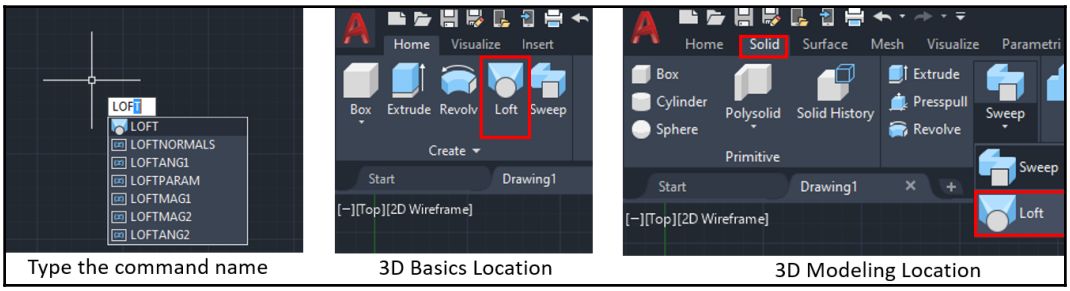


Figure 12.24: Alternative methods to start the Loft command

The `Loft` command is similar to the `Sweep` command, but instead of using one profile or cross-section to be swept along a path, you can use multiple profiles or cross-sections as shown in the following steps:

1. After starting the `Loft` command, select the objects that will be used as the profiles or the cross-sections. You should select them in the order you intend them to be lofted.
2. A preview of the lofted object is shown as you select more profiles. To undo a selection, just press *Ctrl* + *Z*.
3. Press *Enter* after you select all the profile shapes. This will open a list of different Loft options.
4. Click on **Cross sections only**. The command will terminate and the lofted object will be created.

The following figure shows an illustration of the previous steps:

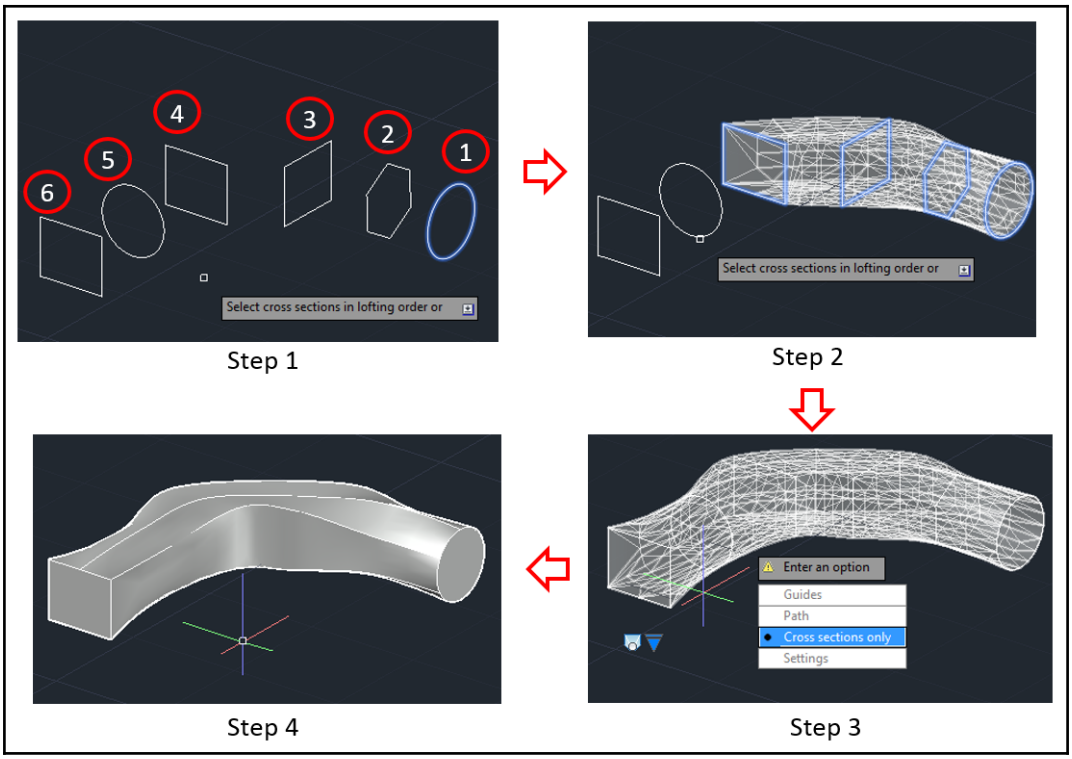


Figure 12.25: Steps for using the Loft command

There is a number of options associated with the `Revolve` command. These options allow for alternative methods and settings. The first set of options appear in the command bar right after you start the `Loft` command. This set contains three different options as shown here:

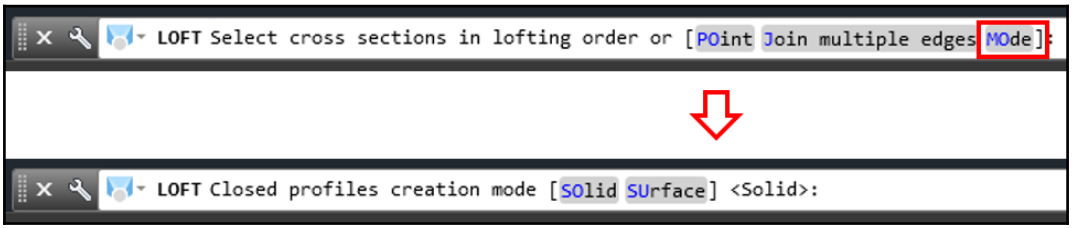


Figure 12.27: Additional options in the Loft command

This first option called **POint**. To select this option you can just type `PO` and press *Enter* or you can click on the option name in the command bar. This option allows you to start the Loft from a point, end it in a point, or both, as shown in the following step-by-step example:

1. Start the `Loft` command, and before selecting any profile shape, type `PO` and press *Enter* to activate the **Point** option.
2. You will be prompted to specify the Loft starting point, choose the location using the mouse and click. Alternatively, you can enter the coordinates using the keyboard and press *Enter*.
3. Select the profile shapes in order as usual, then type `PO` and press *Enter* to activate the **Point** option.
4. You will be prompted to specify the Loft end point. Choose the location using the mouse and click, or alternatively you can type the coordinates using the keyboard.
5. Choose one of the Loft options.
6. Once you click on the desired Loft option, the object will be created.

The following figure is an illustration of the previous steps:

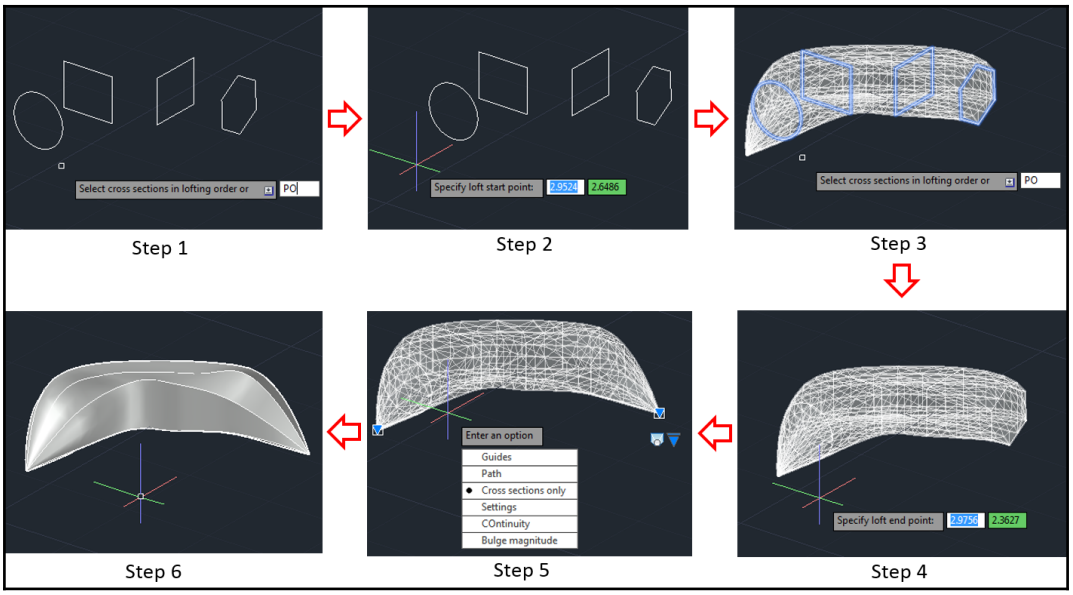


Figure 12.28: Using the Point option

The next option is called **Join multiple edges**. To select this option you can just type `J` and press *Enter* or you can click on the option name in the command bar. This option allows you to take a set of existing edges that form a cross-section and use them in the Loft command.

The last option in the first set is called **Mode**. To select this option you can just type `MO` and press *Enter* or you can click on the option name in the command bar. This option is used to determine the type of the object produced by the `Loft` command, either to be a solid or a surface. By default when you select a closed shape or a surface, the lofted object will be a solid. This option allows you to always create a surface even if the lofted object is a closed shape or a surface:

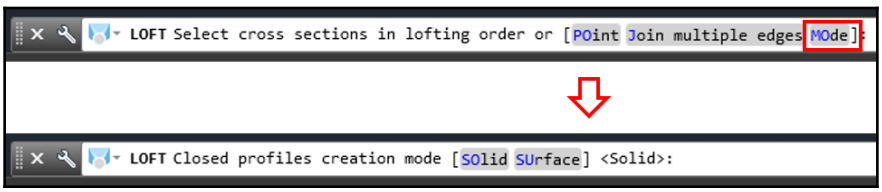


Figure 12.29: Using the Mode option

The second set of options appear after you finish selecting the cross-sections. It is displayed in the command bar and as a drop-down menu near the mouse cursor. This set contains four different options as shown in the following figure:



Figure 12.30: Second set of additional options in the Loft command

The first option in this set is called **Guides**. To activate this option, just type `G` and press *Enter*, or you can click on the option name in the command bar. By default, the lofted body created between the cross-sections will take a smooth path. This option allows you to modify how the lofted object is shaped by adding guides between the cross-sections as follows:

1. Start the `Loft` command, select the cross-section shapes, and then press *Enter*.
2. A preview of the lofted object will be displayed. Noticing that it is a straight, smooth path between the two selected cross-sections, click on the **Guides** option.
3. Select the two guide arcs. A preview of the Loft will be displayed as you select.
4. After you finish selecting the guide arcs, press *Enter* and the lofted object will be created.

The following figure shows an illustration of the previous steps:

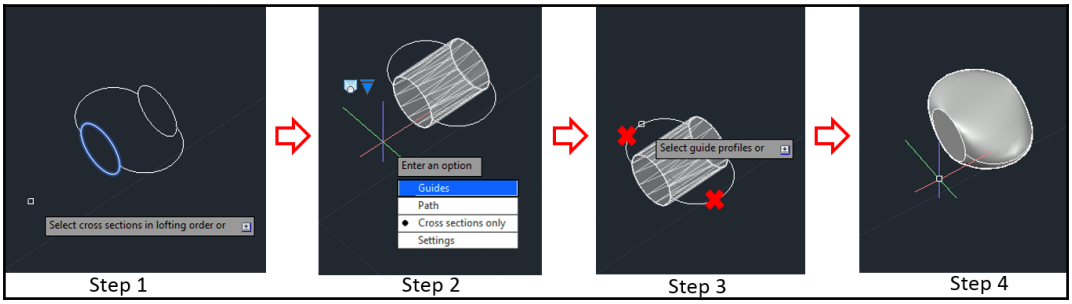


Figure 12.31: Using the Guides option

The second option is called **Path**. To activate this option, just type `P` and press *Enter*, or you can click on the option name in the command bar. By default, the lofted body created between the cross-sections will take the shortest path. This option allows the use of a different path between the cross-sections as follows:

1. Start the `Loft` command, select the cross-section shapes, then press *Enter*.
2. A preview of the lofted object will be displayed. Notice it is a straight, smooth path between the two selected cross-sections. Click on the **Path** option.
3. Select the outer arc as a path. A preview of the Loft will be displayed as you select.
4. Once you click on the path, the lofted object will be created.

The following figure is an illustration of the previous steps:

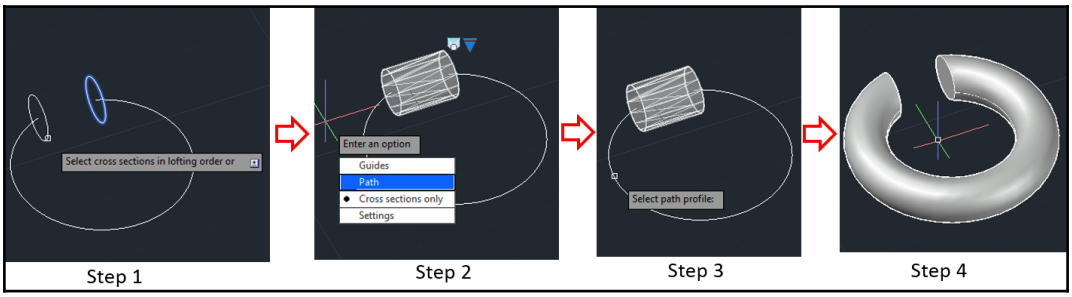


Figure 12.32: Using the Path option

By default, the lofted objects will be deleted after the `Loft` command ends. You can control whether to delete these objects or not using the DELOBJ system variable. To access that just type `DELOBJ` and press *Enter*, and the current value of the DELOBJ system variable will be displayed, then you can type the new value and press *Enter*. For more details about the DELOBJ system variable, please refer to the *Using the Extrude command* section of this chapter.

That's it for the `Loft` command. Now let's have a look at the final command in our 2D-to-3D conversion group, which is called `Presspull`.

Using the Presspull command

The last command in our 2D-to-3D conversion group is called `Presspull`. It is used to directly create an extrusion (or cut into the material) out of the face of a solid body. In addition to that, it can also directly extrude an enclosed 2D shape as we will see in this section. To start the `Presspull` command, you can type the command name `PRESSPULL` (or just type `PRES`) then press *Enter*, or alternatively you can click on the **Presspull** button found in the **Create** panel in the **3D Basics** workspace, and if you are using the **3D modeling** workspace, you will find the **Presspull** button in the **Modeling** panel under the **Home** tab, or in the **Solid** panel under the **Solid** tab, as shown in the following figure:

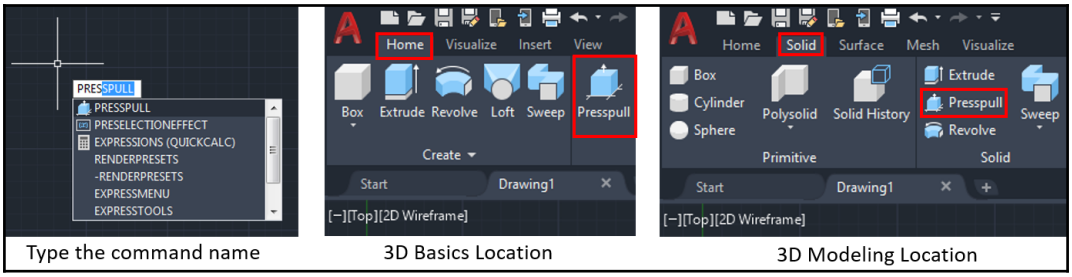


Figure 12.33: Alternative methods to start the Presspull command

You can use the `Presspull` command to quickly extrude faces of existing 3D solid objects as follows:

1. After starting the `Presspull` command, move to the object face that will be extruded. Notice that the face edges will be highlighted in blue. Click to confirm the selection.
2. Move to specify the extrusion height, or alternatively, you can enter the required height using the keyboard.
3. When you arrive at the desired height, click or press *Enter* to finish.

The following figure shows an illustration of the previous steps:

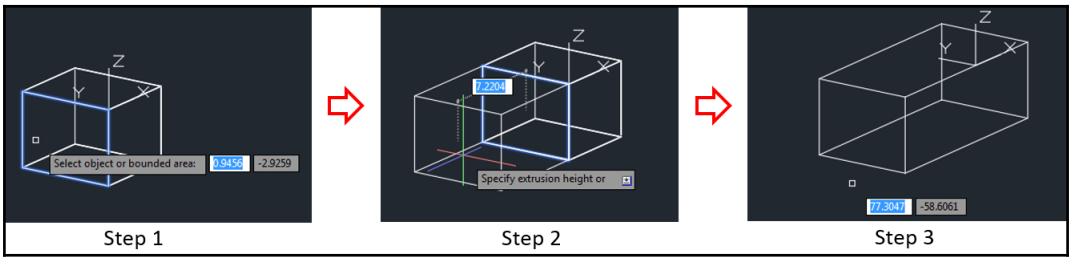


Figure 12.34: Steps for using the Presspull command

Notice that after **Step 3**, the command will not terminate, letting you select another face to extrude, or to end the command press *Esc* or right-click.

The previous example showed how to extrude a face of an existing solid body, but the `Presspull` command can also be used to directly extrude an enclosed region, as shown in the following step-by-step example:

1. Move the mouse cursor to the middle of an enclosed region. The borders of the region will be highlighted as shown in the first panel of the following figure. Click to confirm the selection.
2. Move the cursor vertically to specify the extrusion height, or alternatively, you can type the height value then press *Enter*.
3. After the height specification, you will be prompted to select another face (or enclosed region) to extrude. Press *Enter* to end the command.

The following figure is an illustration of the previous steps:

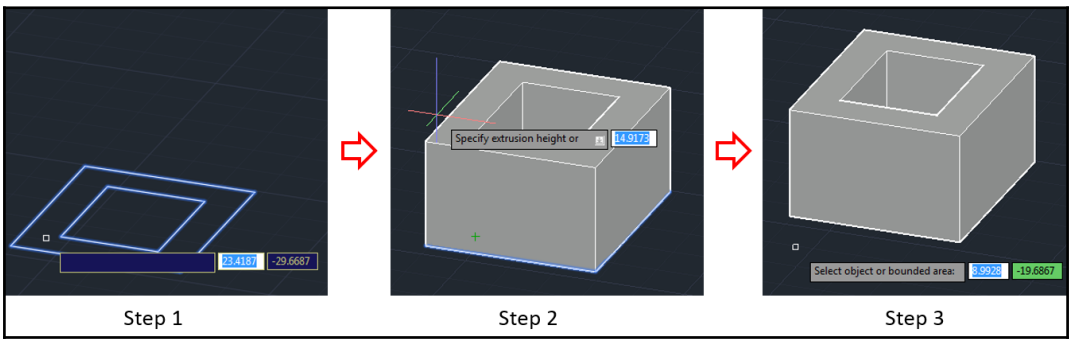


Figure 12.35: Using the Presspull command to create an enclosure

As shown in the previous example, the enclosed open region between the two rectangles was automatically detected by the `Presspull` command. The same can't be done using the `Extrude` command, where you will create two solids then subtract the inner one to create the same shape in the previous example.

The `Presspull` command has one option that appears in the command bar after you select the face to be extruded. The option is called **Multiple**, and to activate this option you can type `M` and press *Enter* or you can click on the option name in the command bar. The **Multiple** option allows you to select multiple objects to be extruded at once as shown in the following step-by-step example:

1. After starting the `Presspull` command and selecting one face, type `M` and press *Enter* to activate the **Multiple** option.
2. Select the other faces to be extruded.
3. Move the mouse to specify the height of all the extrusions simultaneously. Alternatively, you can enter the value using the keyboard.

The following figure shows an illustration of the previous steps:

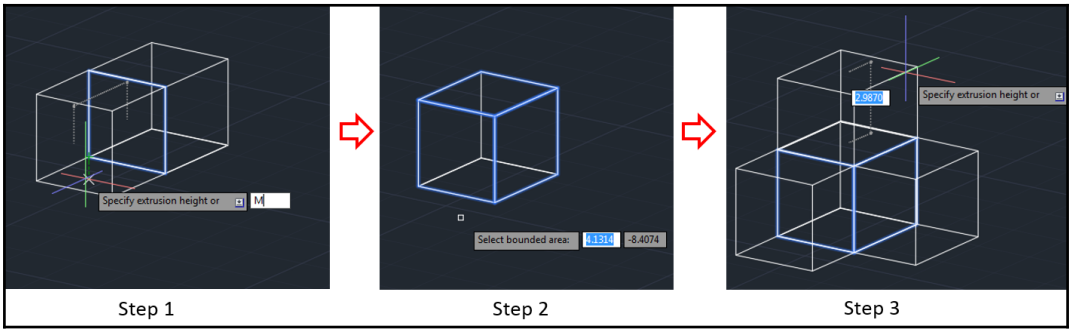


Figure 12.36: Using the Multiple option

Now that we learned about the 2D-to-3D conversion commands, let's have a look at the `Flatshot` command, which is used to create a 2D drawing from a 3D model.

Using the Flatshot command

To start the `Flatshot` command, you can type the command name `FLATSHOT`, or just type `FL` then press *Enter*. Alternatively, you can click on the **Flatshot** button found in the **Section** panel under the **Home** tab, or on the **Section** panel under the **Solid** tab, in the **3D modeling** workspace, as shown in the following:

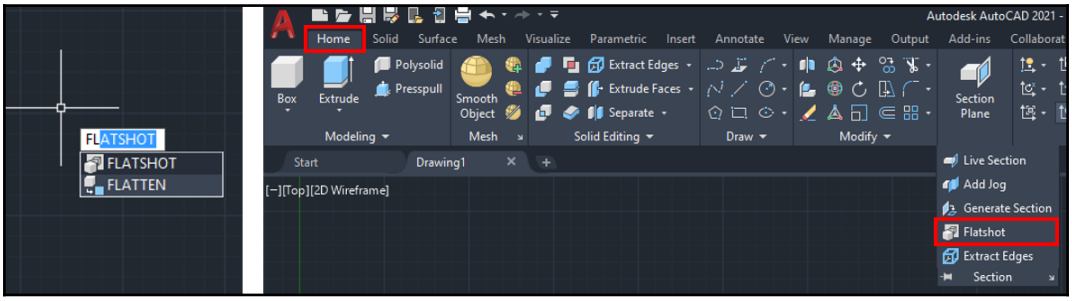


Figure 12.37: Alternative methods to start the Flatshot command

The `Flatshot` command can be used to create 2D drawings by taking a shot of an existing 3D model. You just adjust the view angle as desired, and then use the command to create the 2D shot. The created 2D drawing is inserted to the model as a block in the XY plane as follows:

1. Start by adjusting the view angle as desired, and since the block will be inserted in the **XY** plane, it is a good practice to align the coordinate system's **XY** plane with the view angle. You can quickly do this using the **View** tool in the **Coordinates** panel.
2. Notice that the UCS icon now shows that the **XY** plane is aligned with the view angle, and the UCS title has changed to **Unnamed**. Activate the `Flatshot` command as shown in the beginning of this section.

3. The **Flatshot** window appears, which contains different options and settings. We will get back to it, but for now, press **Create**:

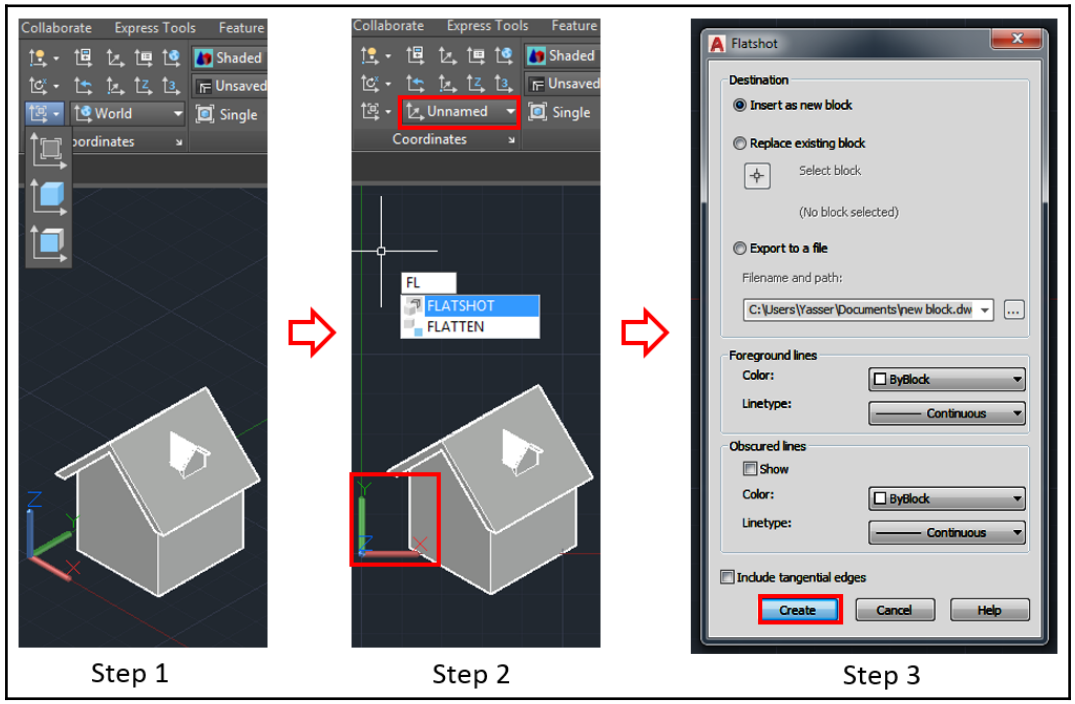


Figure 12.38: Steps for using the Flatshot command (1/2)

4. You will be prompted to specify the insertion point for the block containing the 2D shot. Notice that the base point of the block is the UCS origin point.
5. Use the mouse movement or enter a value using the keyboard to specify the **X** scale factor for the inserted block. After that, you will be prompted to do the same for the **Y** scale factor.
6. Use the mouse movement to specify the rotation angle. Alternatively, you can enter the rotation angle using the keyboard.

7. After you specify the rotation angle, the command terminates and the 2D shot is inserted:

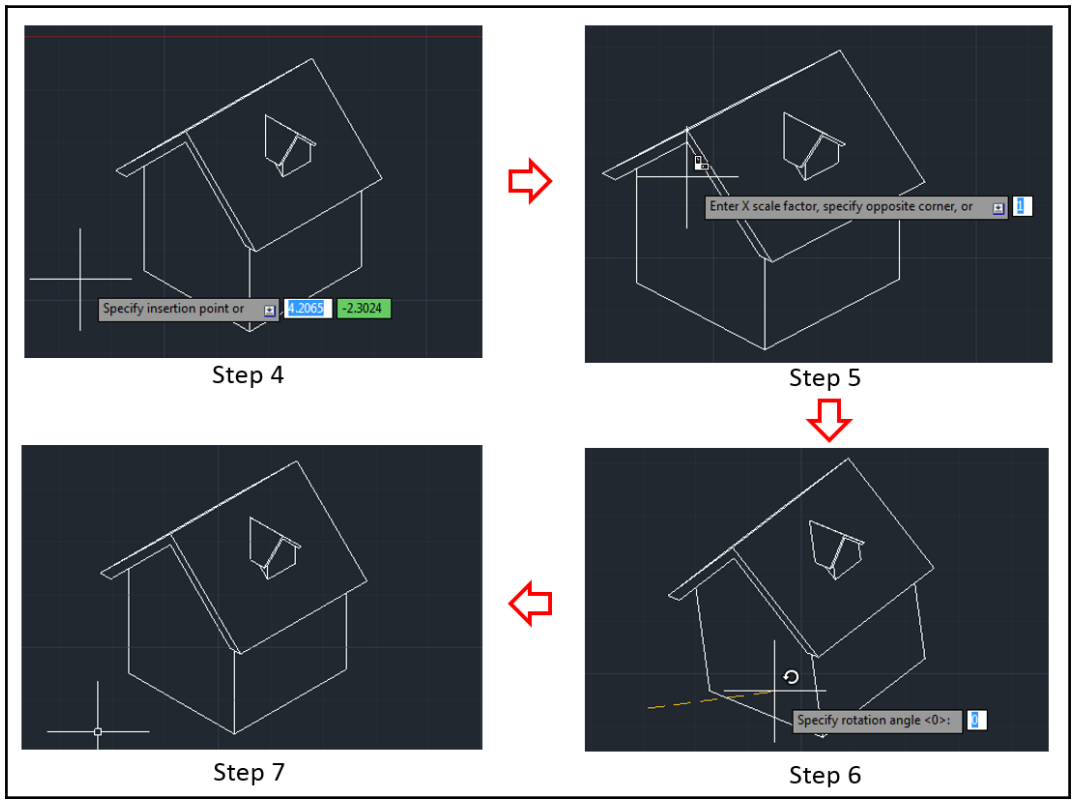


Figure 12.39: Steps for using the Flatshot command (2/2)

The **Flatshot** dialog box is divided into three main parts, as shown in the following annotated figure. The first part is called **Destination**, and in this part, you can choose to insert the 2D shot as a new block or to update an existing block definition. There is another option to export the 2D shot to a file.

The second part is for controlling the color and the line type of foreground or the front-facing lines. The third part is for controlling the obscured or the hidden lines, you can choose to show or to hide them, and you can control the **color** and the **line type** applied to them:

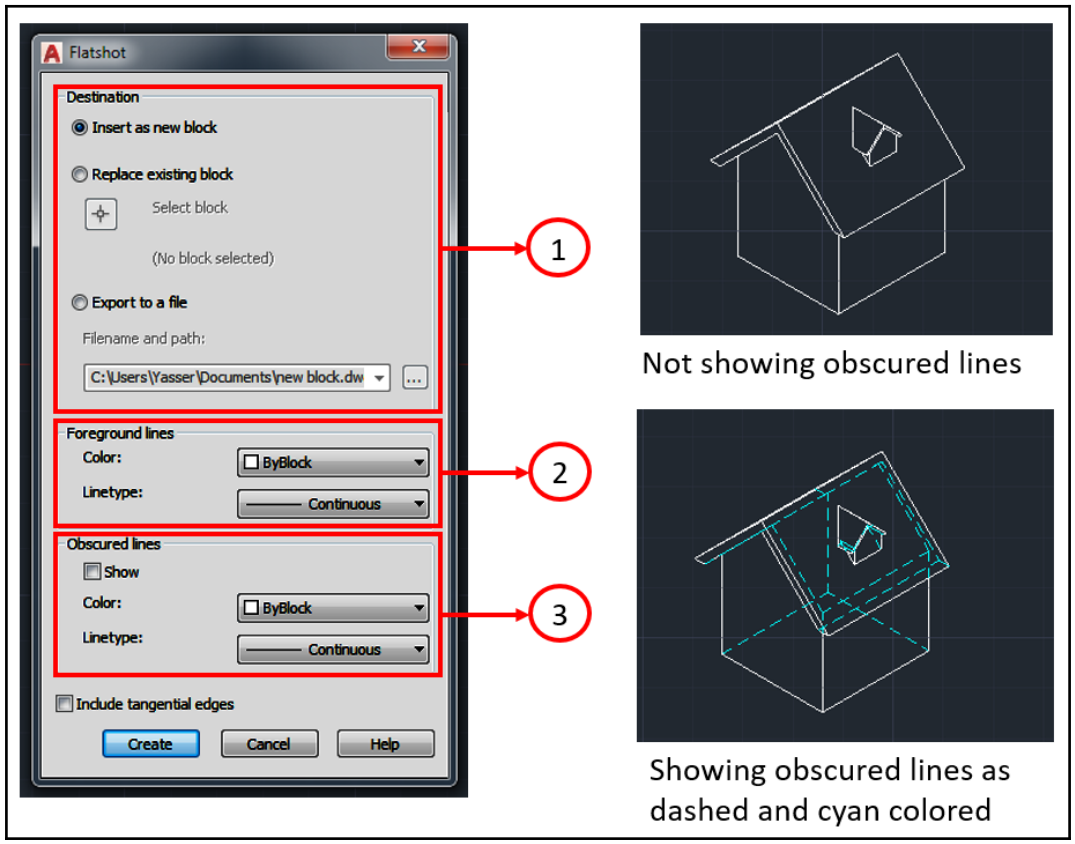


Figure 12.40: Adjusting settings of the Flatshot command

At the bottom of the **Flatshot** dialog box, there is a checkbox for **Including tangential edges**. By checking this option, a line in the created 2D shot will be created wherever a curved surface meets a flats surface, even if they are tangent at the line of intersection. If this is left unchecked, all the tangent intersections between curved and flat surfaces will be ignored.

Using the Section Plane tool

A common scenario is where you have a 3D model and you are required to create a number of sections for that model. The section plane tool is for doing just that, quickly and easily. The **Section Plane** tool button can be found in the **Section** panel, which can be found under the **Home** tab in the **3D modeling** workspace as shown in the following figure:

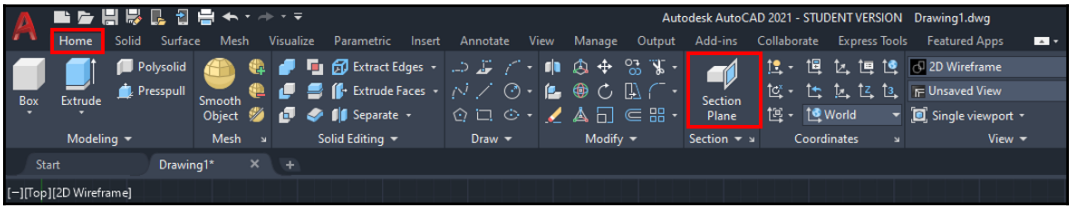


Figure 12.41: Section tool button locations in the 3D modeling workspace

After starting the command, you can create a vertical section plane by determining two points as shown in the following step-by-step example:

1. After starting the `Section Plane` command, specify the location of the first point of the section line. You can do this either with the mouse, or you can type the coordinates using the keyboard and then press *Enter*.
2. Specify the location of the second point.
3. The section plane will created and the command will terminate.

The following figure illustrates the previous steps:

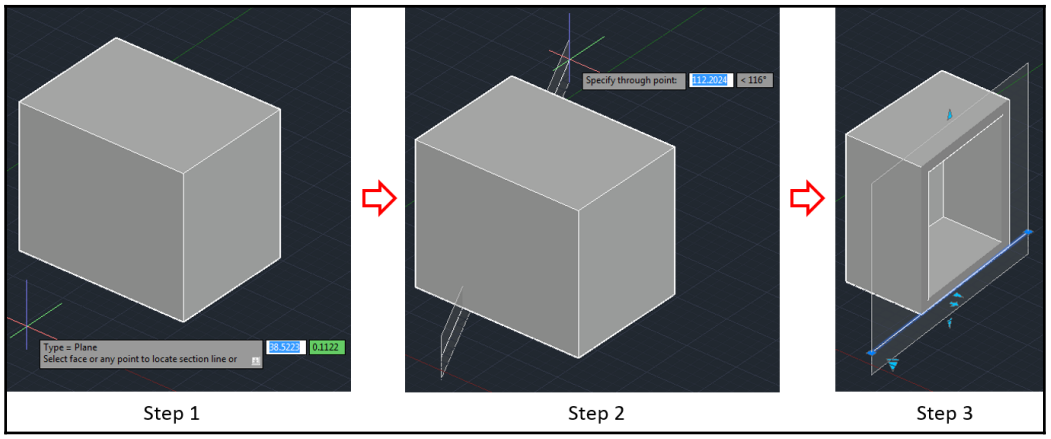


Figure 12.42: Steps for using the Section tool

If you select the section plane, you will find a number of grab points to do quick modifications. The first one is a small square at the starting point of the section plane, which is the base point. It can be used to pick and move the section plane, as shown in the following figure:

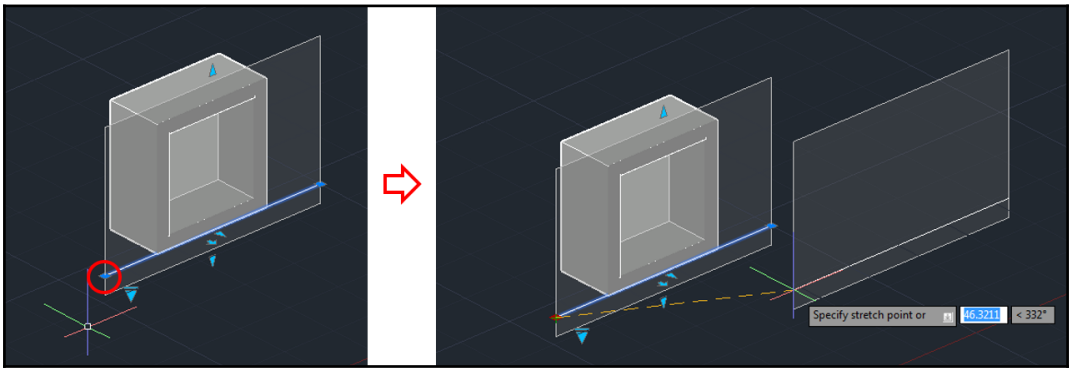


Figure 12.43: Moving the section location using the base point

The other square-shaped grab point (on the end point of the section plane) can be used to change the plane orientation as shown in the following figure:

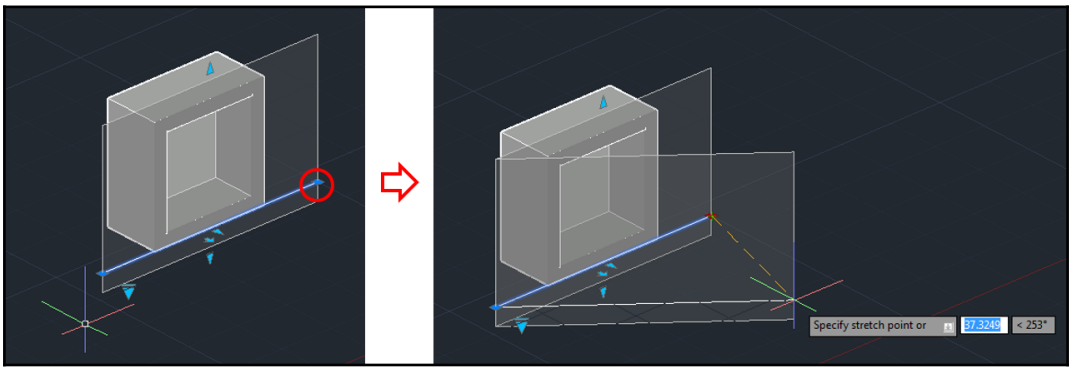


Figure 12.44: Changing the section orientation using the grab point

Near the center point of the section plane, you will find two small arrows. The upper one is pointing outward from the section plane (perpendicular to the section plane). This grab point can be used to change the section plane position along the line normal to the plane orientation as shown in the following figure:

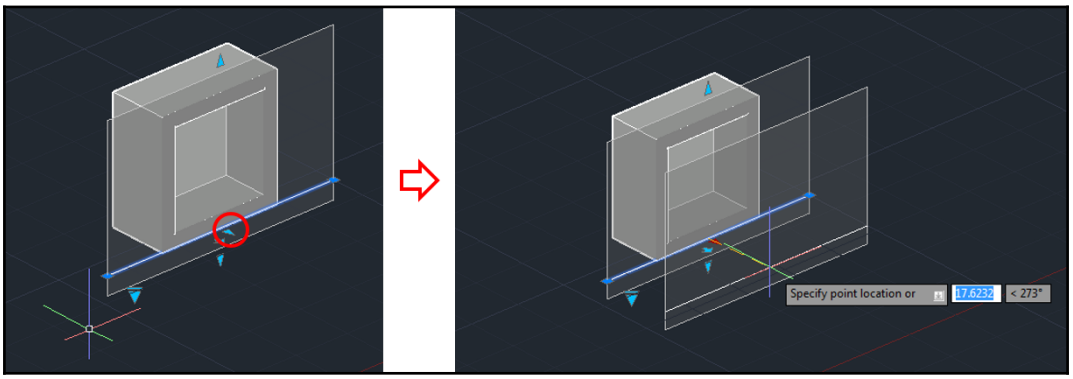


Figure 12.45: Changing the section location along the normal line direction using the grab point

The other arrow is also pointing outward and has a different shape. It is used to change the section plane direction as shown in the following figure:

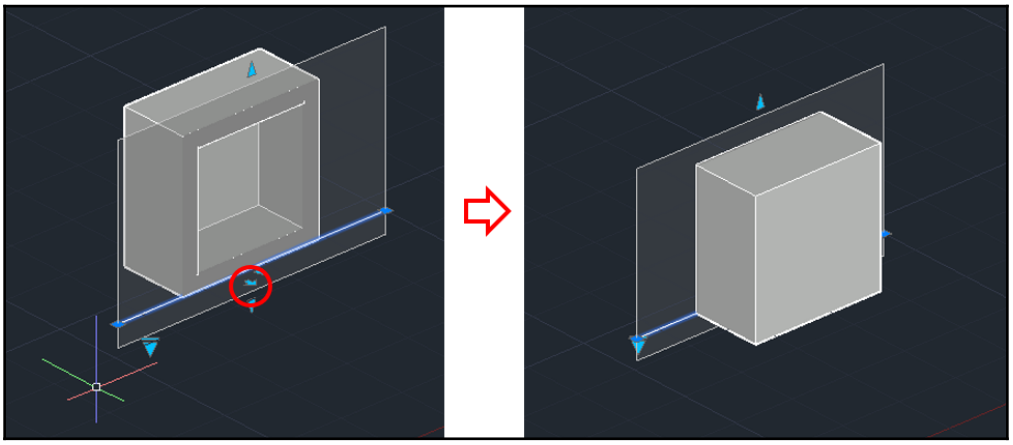


Figure 12.46: Flipping the section using the grab point

Next to the base point you will find a small arrow pointing downward. Click on it to open a list of the different modes of the section plane. As shown in the following figure, there are four modes:

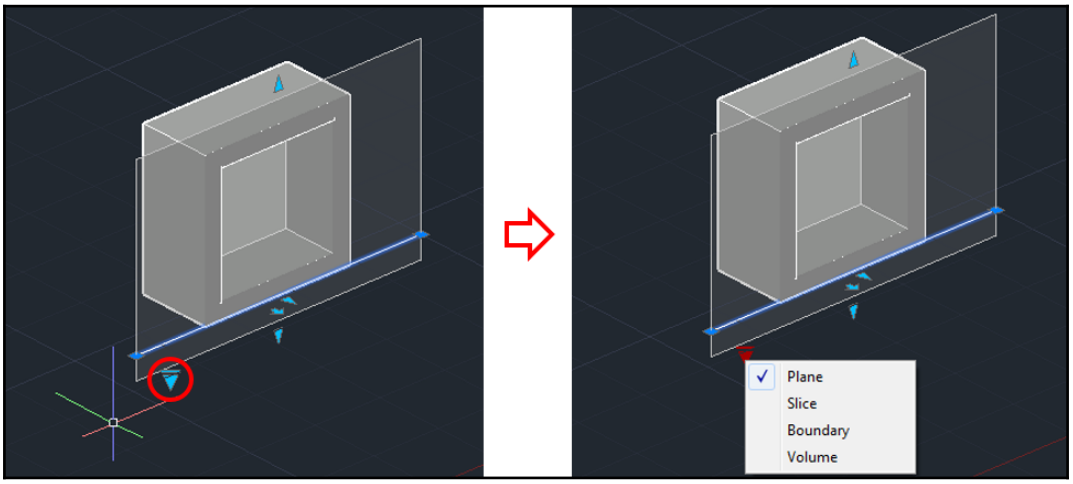


Figure 12.47: Different section modes

- The first mode is called **Plane**, which is the default one. As shown in the previous figures, it cuts the 3D body then shows the cut body in one direction, and hides what is in the opposite direction (which can be switched, as shown previously).

- The next one is called **Slice**. It will create a thin slice of the body around the section plane as shown in the following figure:

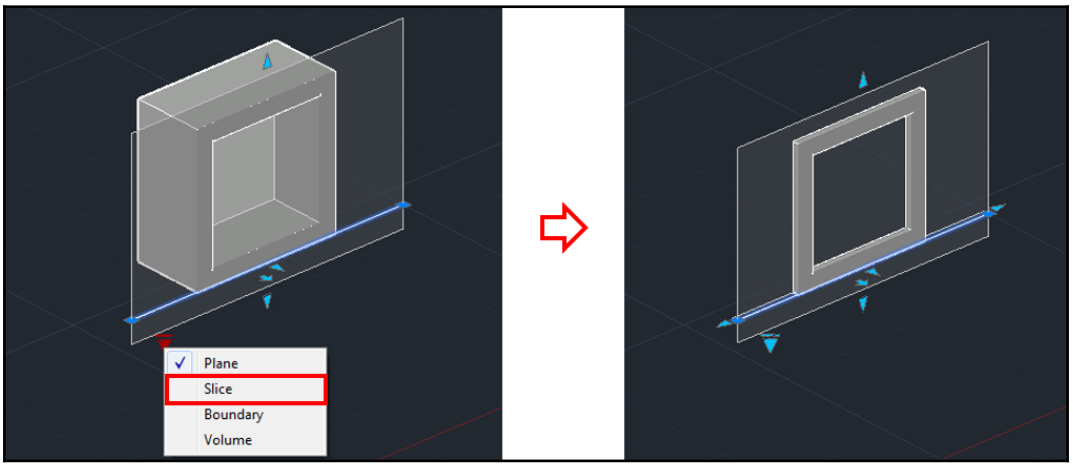


Figure 12.48: Using the Slice section

- The third mode is called **Boundary**. This mode will show the extent of what will be shown of the cut body. As shown in the following figure, it has grab points to control these boundaries:

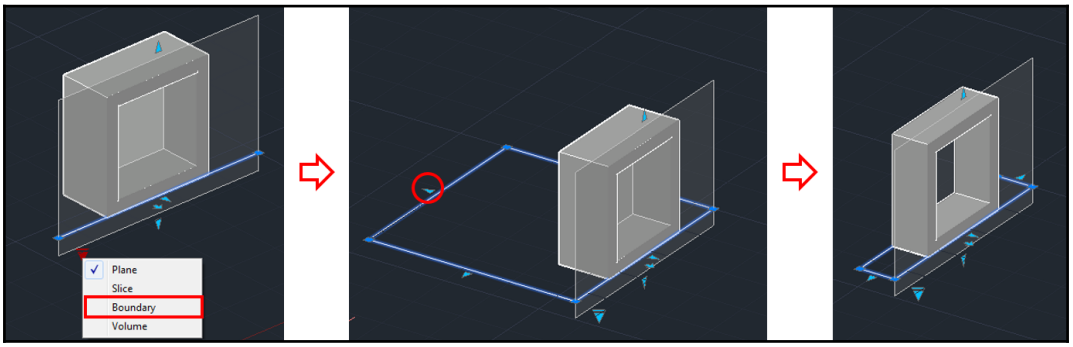


Figure 12.49: Using the Boundary option

- The final option is called **Volume**. It is basically the same as the **Boundary** option, but now you can also control the height limit, as shown in the following figure:

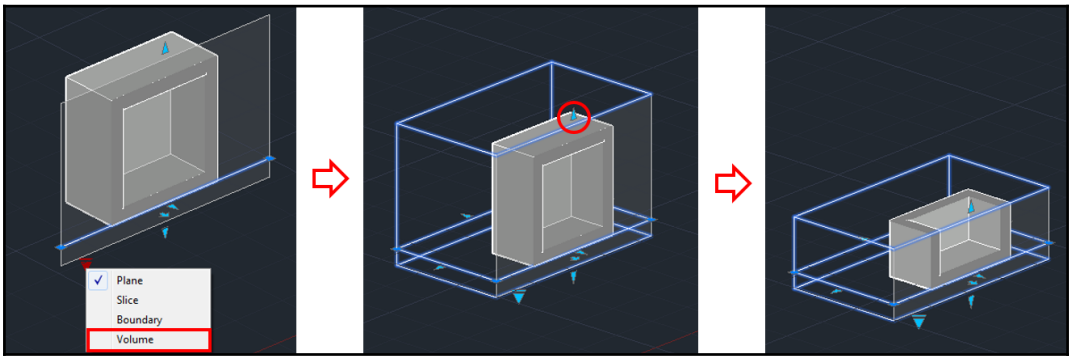


Figure 12.50: Using the Volume option

As shown in the previous examples, the section plane is flat, which may not be appropriate to all the cases, as in some situations you may want the section plane to have a jog at a certain location to show specific details. To add a jog to the section plane, you can use the **Add Jog** tool button located in the **Section** panel under the **Home** tab in the **3D modeling** workspace as shown in the following figure:

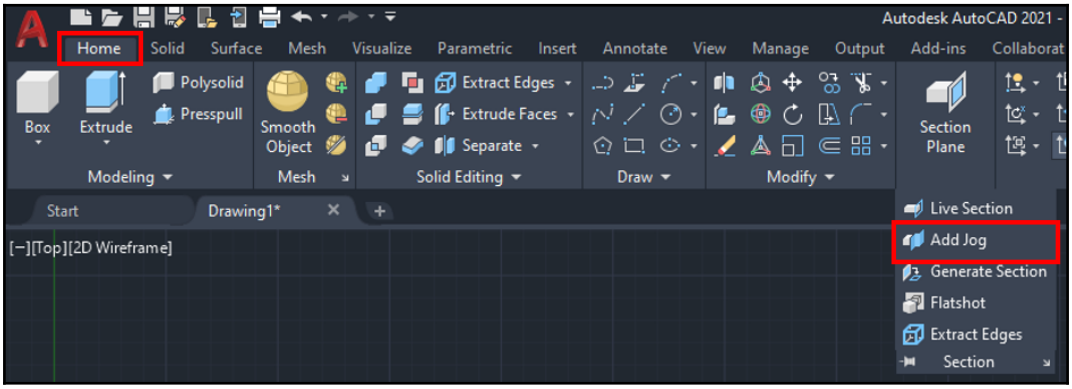


Figure 12.51: Add Jog tool button location in the 3D modeling workspace

The jog can be added to any section plane as shown in the following step-by-step example:

1. Click on the **Add Jog** button then click on the desired section plane. You will be prompted to determine the jog location. Move the mouse cursor to the desired location then click again.

2. The jog will be automatically created with a 90-degree angle.
3. Use the new grab points to modify the section line as shown in the following figure.

The following figure is an illustration of the previous steps:

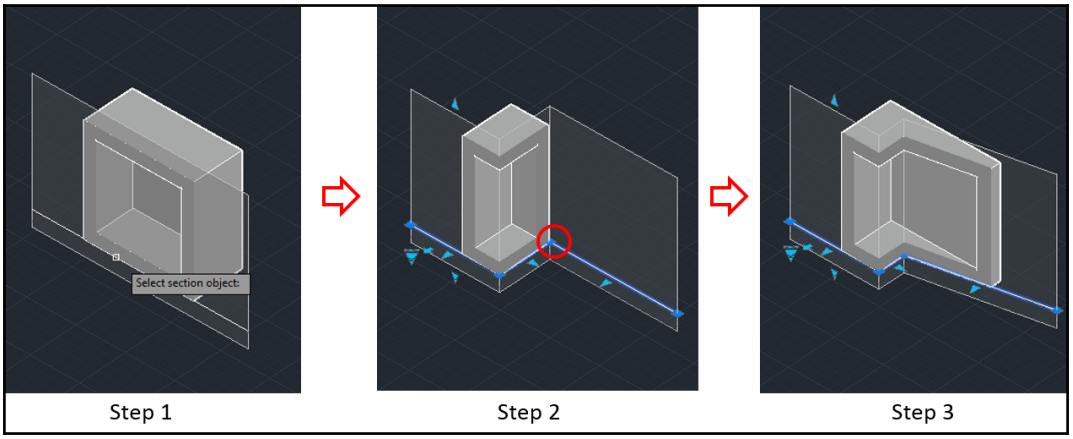


Figure 12.52: Steps for using the Add Jog tool

You can create a 2D block of the section using the **Generate Section to Block** tool. You can find the **Generate Section** tool button in the **Section** panel under the **Home** tab in the **3D modeling** workspace as shown in the following figure:

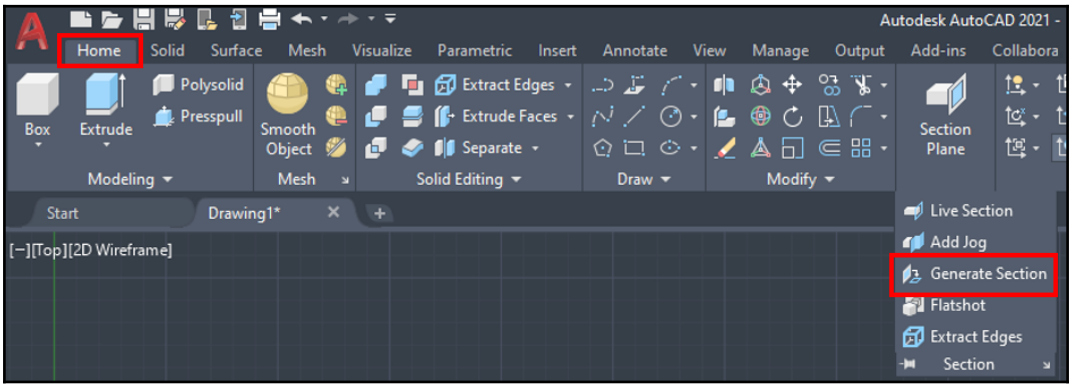


Figure 12.53: Generate Section tool button location in the 3D modeling workspace

How to generate a 2D block of the section is shown in the following step-by-step example:

1. After clicking on the **Generate Section** tool button, the **Generate Section/Elevation** window will appear. Click on **Select Section Plane**.
2. Select the desired section plane.
3. You will return back to the **Generate Section/Elevation** window. Click on **Create**.
4. A 2D block will be automatically generated and inserted in the drawing, and you will be prompted to specify the insertion point, and the scale and the orientation of the block. Notice that the 2D block will be inserted in the **XY** plane even if the section plane was vertical as shown in the example figure.

The following figure shows an illustration of the previous steps:

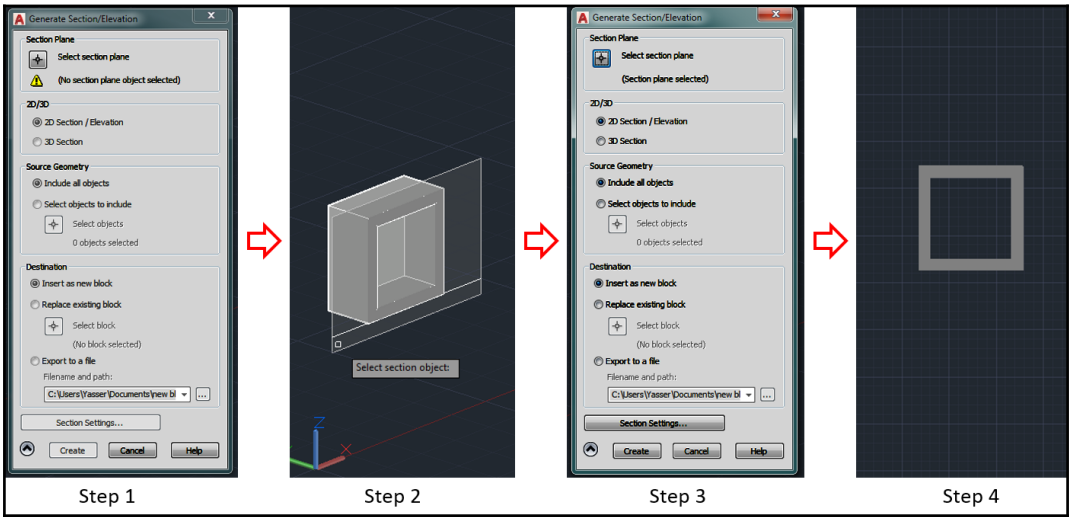


Figure 12.54: Steps for using the Generate Section command

As shown in the previous example, there are different options regarding the inserted block: you can create a new block, replace an existing block definition, or export it to a file. Also, some objects can be excluded from the created section using the **Select Objects to include** option found under the section called **Source Geometry**.

Summary

In this chapter, we learned how to do conversions between 2D and 3D models, starting with the four commands used for 2D-to-3D conversion, which are `Extrude`, `Revolve`, `Sweep`, and the `Loft` command. We learned about the details of each command and the key differences between them. After that, we looked at the opposite scenario, which is creating a 2D shot from a 3D model using the `Flatshot` command.

After covering different methods of creating 3D objects, in the next chapter, the modify commands will be discussed, starting with the conventional 2D modify commands that can also be used with 3D objects (albeit with some limitations), then moving to the specialized 3D modify commands.

13
Modifying 3D Objects

The commands used in the creation of 3D solid objects were discussed in the previous two chapters. In practice, most 3D models cannot be created using a single step of object creation. The reason for this is the fact that practical 3D models can be complicated and rich in detail. Hence, the real-life modeling process involves the creation of simple 3D objects that are close to the desired final shape, and then combining/modifying the simple objects that have been created and adding details to them using the `modify` commands. The set of `modify` commands starts with simple object manipulation, such as moving or rotating the object, the performance of standard Boolean operations, such as combining and subtracting, and a set of other specialized editing commands, as we will see in detail in this chapter.

So, to sum up, by the end of this chapter, we will be able to do the following:

- Use basic `modify` commands
- Use Boolean operations on 3D objects
- Work with `solid editing` commands
- Work with `face editing` commands
- Work with `edge editing` commands

Let's now begin by exploring basic object manipulation for 3D bodies in AutoCAD.

Exploring basic modify commands

In this section, we will start with the basic modify operations for 3D modeling. These basic operations include changing the location of the bodies, changing their orientation, creating copies of the object, and scaling it. These basic operations are executed using the standard modify commands, such as move, copy, rotate, and scale, all of which will be discussed, as well as their 3D modified versions. In addition to this, we will learn how to use the **3D Gizmo** tool, which is a quick and intuitive way to perform the basic modify commands on 3D objects.

Using 2D modify commands with 3D objects

AutoCAD has a full set of modify commands. Details pertaining to these commands and their use with 2D shapes were discussed in detail in `Chapter 3`, *Learning Modify Commands*. In this section, we will revisit some of these commands, which are as follows:

- The `move/copy` commands
- The `mirror` command
- The `rotate` command
- The `scale` command

These commands are usable with 3D objects, but may have some limitations, and may also have a 3D version of the command that eliminates these limitations, as we will see in the following sections.

Move/copy commands

You can find the **Move** and **Copy** command buttons in the **Modify** panel on the **Home** tab in the **3D Basics** workspace, or in the **Modify** panel on the **Home** tab in the **3D Modeling** workspace, as shown in the following screenshot:

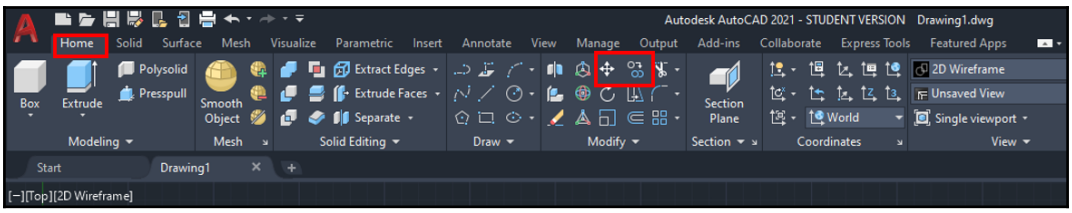


Figure 13. 1: Locations of the move/copy tool buttons in the 3D Modeling workspace

Both the `move` and `copy` commands will work with 3D objects, exactly like they did with the 2D shapes. For more details on using them, please refer to `Chapter 3`, *Learning about Modify Commands*, of this book. The only difference is that now, you have a third dimension that you can move your object to.

When using the 3D perspective view, if **Ortho mode** is not active, object movement will just be restricted to the **XY** plane of the UCS. To access the third dimension (**z-axis**), you have to activate **Ortho mode**. Once activated, you can access the third dimension and it will be displayed near the mouse cursor, irrespective of whether you are moving in the **z-axis** or in the **XY** plane. Another method involves entering the displacement coordinates containing a **z** displacement component. The following diagram shows the `move` command behavior with different **Ortho mode** settings:

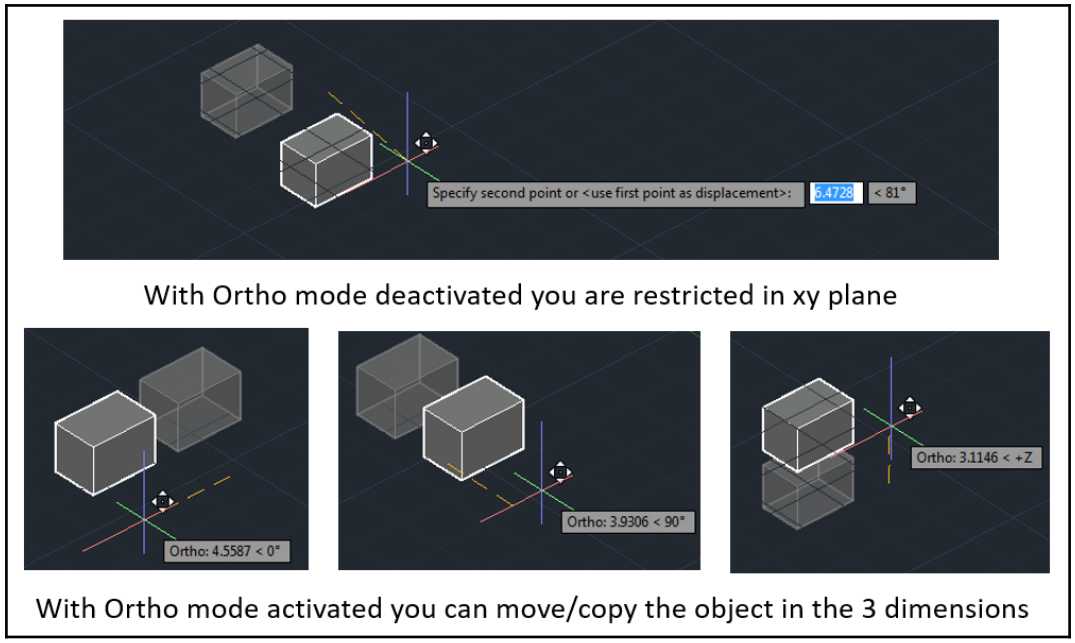


Figure 13. 2: Using move/copy commands in 3D

You can also restrict movement to any of the three main standard planes (**XY**, **XZ**, and **YZ**) by switching the view to one of the six standard views.

Mirror/3D mirror commands

You can find the **Mirror** command button in the **Modify** panel on the **Home** tab in both the **3D Basics** and the **3D Modeling** workspaces, as shown in the following diagram:

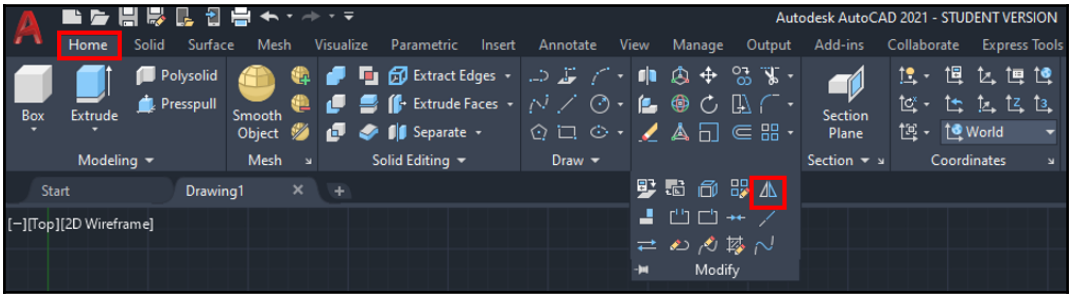


Figure 13. 3: Location of the mirror tool buttons in the 3D Modeling workspace

The `mirror` command will work normally with 3D objects, but with limitations. You select the objects, and then you specify the mirror line as usual, regardless of the object position. The mirror line will be created in the UCS **XY** plane, and the mirror plane will be created by passing through that line and perpendicular to the **XY** plane.

It is impractical to use the `mirror` command in the 3D perspective view. If you are going to use it, set the view to top or bottom standard views.

The 3D version of the `mirror` command is called 3D mirror. The 3D mirror command performs the same function as the standard `mirror` command, but it overcomes the limitations of using the latter with 3D objects. You can find the **3D Mirror** command button in the **Modify** panel on the **Home** tab in both the **3D Basics** and the **3D Modeling** workspaces, as shown in the following screenshot:

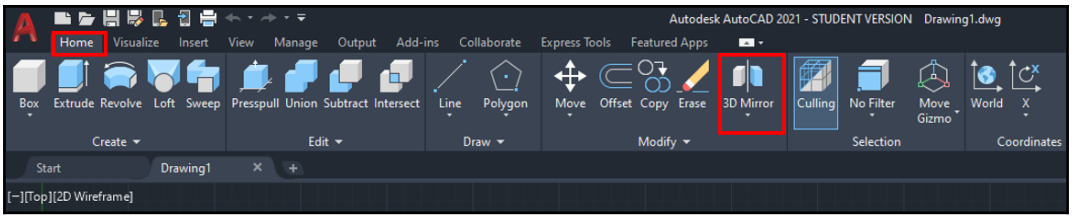


Figure 13. 4: Location of the 3D mirror tool buttons in the 3D Basics workspace

The `3D mirror` command used is shown by means of the following step-by-step example:

1. After you start the command, select the object(s) to be mirrored and then press *Enter* to end the selection phase.
2. Specify the mirror plane. The default method involves determining three points. Click to specify the three point locations, as shown in the diagram.
3. As with the regular `mirror` command, you have the option to either keep or erase the original object.
4. The mirror object will be created and the command terminates.

The following diagram is an illustration of the previous steps:

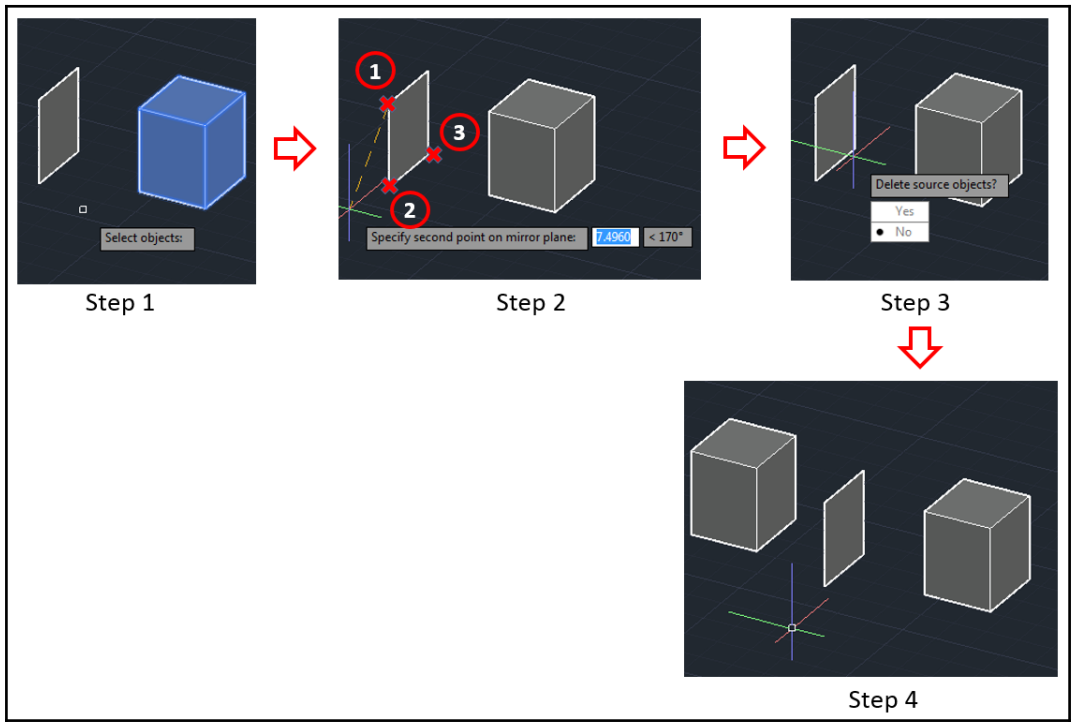


Figure 13. 5: Steps involved in using the 3D mirror command

As shown in the previous example, you can set the mirror plane to be co-planar with any general 3D plane, thereby eliminating the limitation of the regular `mirror` command.

There are alternative methods/options to specify the mirror plane. In the example, the default `3point` method was used. The other methods will appear in the command bar, as shown here:



Figure 13. 6: Additional options in the 3D mirror command

These options are explained as follows:

- **Object**: The first option is called **Object**. To select it, click on the option name in the command bar, or you can just type `O` and press *Enter*. This option allows you to select a planar object, such as a circle, a polyline, or a rectangle, as the mirror plane, as shown in the following diagram:

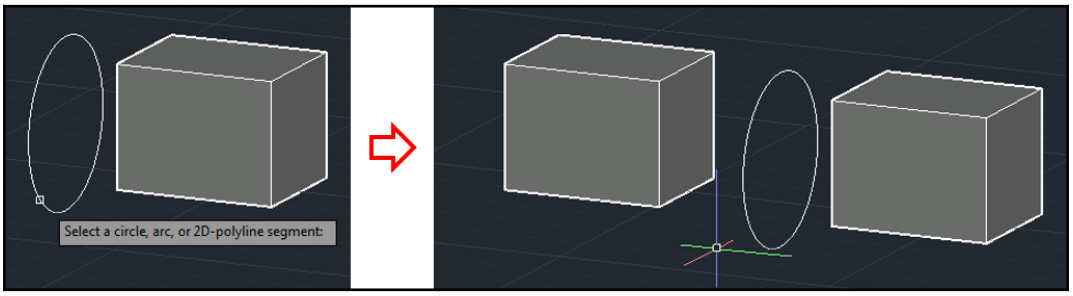


Figure 13. 7: Using an object as a 3D mirror plane

- **Last**: The second option is called **Last**. To select it, click on the option name in the command bar, or you can just type `L` and press *Enter*. This option will use the mirror plane used the last time the command was activated.
- **Z-axis**: The third option is called **Z-axis**. To select it, click on the option name in the command bar, or you can just type `Z` and press *Enter*. The mirror plane is created by determining a point on the mirror plane, and then another point is specified. The first and the second points will form a vector, which is the **Z-axis** of the mirror plane, as shown in the following diagram:

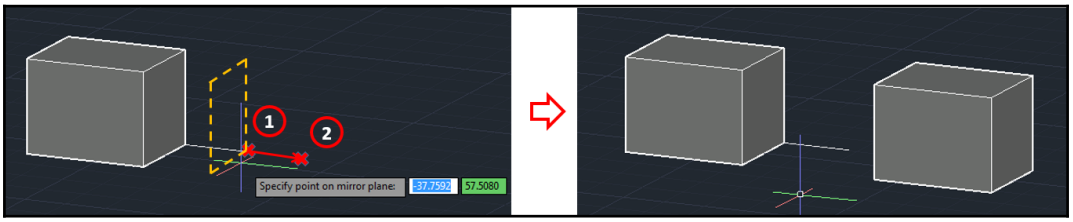


Figure 13. 8: Using the Z-axis option

- **View**: The fourth option is called **View**. To select it, click on the option name in the command bar, or you can type `V` and press *Enter*. This option will use the viewport viewing angle as the direction of the mirror plane, and then you specify a point to determine the location.

The next options are called **XY**, **YZ**, and **ZX** planes. These three options will set the mirror plane to be parallel to the three main UCS planes.

The last option is the default **3points** method, which was demonstrated in the previous example.

Rotate command

You can find the rotate command button in the **Modify** panel on the **Home** tab in the **3D Modeling** workspace, as shown in the following diagram:

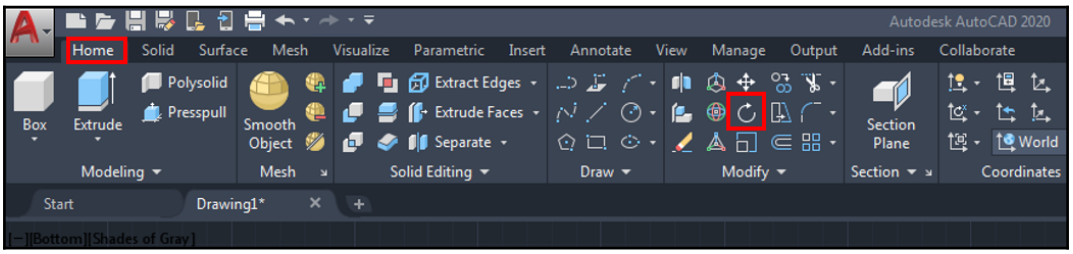


Figure 13. 9: Location of the rotate tool button in the 3D Modeling workspace

Just like the 2D shapes, the rotate command will work normally with 3D objects, but it will have some limitations. In the 2D case, after selecting the object, you will be prompted to set the rotation base point. If you are working in 3D, regardless of the object's location, the base point will be placed in the UCS **XY** plane, and the rotation axis will be set parallel to the **Z-axis**, passing through the rotation base point. The rotate command will work normally, but you will be restricted to rotations around the **Z-axis**.

The 3D version of the rotate command is called 3D Rotate. It can also be found in the **Modify** panel on the **Home** tab in the **3D Modeling** workspace, as shown in the following screenshot:

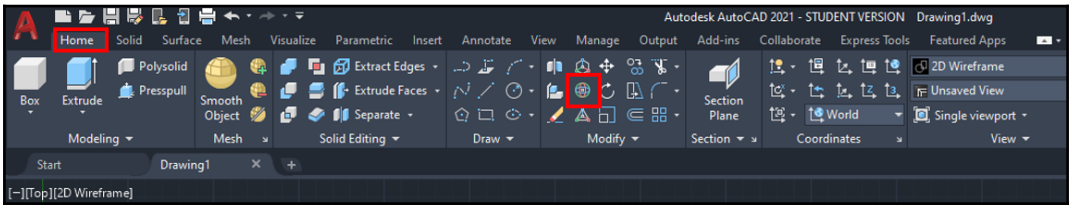


Figure 13. 10: Location of the 3D Rotate tool button in the 3D Modeling workspace

The `3D Rotate` command will activate the **Rotate Gizmo**. The **Rotate Gizmo** is part of the **3D Gizmo** toolset. It is easier and a quicker rotate tool for 3D modeling that doesn't have the limitations of the regular rotate command. The **Rotate Gizmo** will be covered in detail in this chapter on the *3D Gizmo* section.

Scale command

You can find the **Scale** command button in the **Modify** panel on the **Home** tab in the **3D Modeling** workspace, as shown in the following screenshot:

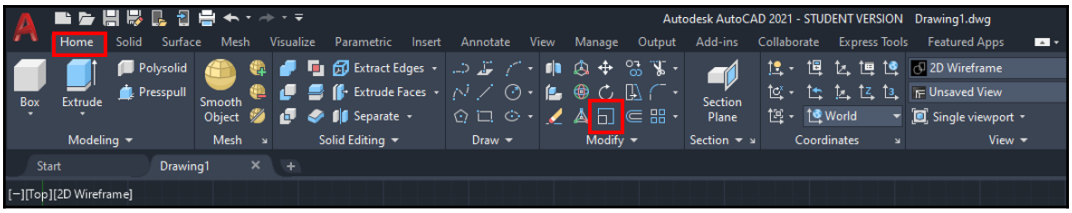


Figure 13. 11: Location of the scale tool button in the 3D Modeling workspace

The `scale` command will work perfectly with 3D objects with no problems or limitations. Nevertheless, there is a 3D version of the `scale` command called `3D scale`. The `3D scale` command will activate the **Scale Gizmo**, which is part of the **3D Gizmo** toolset. It provides an easier and more intuitive way to scale 3D objects. The Scale Gizmo will be covered in detail in this chapter on the *3D Gizmo* section.

Now that we have learned how to use the standard 2D modify tools and their 3D counterparts, let's have a look at the **3D Gizmo** tool.

Using the 3D Gizmo tool

The **3D Gizmo** tool is an alternative method for performing standard moving, copying, rotating, and scaling operations. The difference is that it is more intuitive and saves time compared with using the standard modify commands, given that any of these four operations (`move`, `copy`, `rotate`, and `scale`) is performed by simply clicking and dragging the Gizmo tool icon. The 3D Gizmo tool is located in the **Selection** panel on the **Home** tab in the **3D Basics** workspace. If you are using the 3D Modeling workspace, you can find it in multiple locations. It can be found in the **Selection** panel on the **Home** tab, and is also repeated in both the **Solid** and **Mesh** tabs, as shown in the following screenshot:

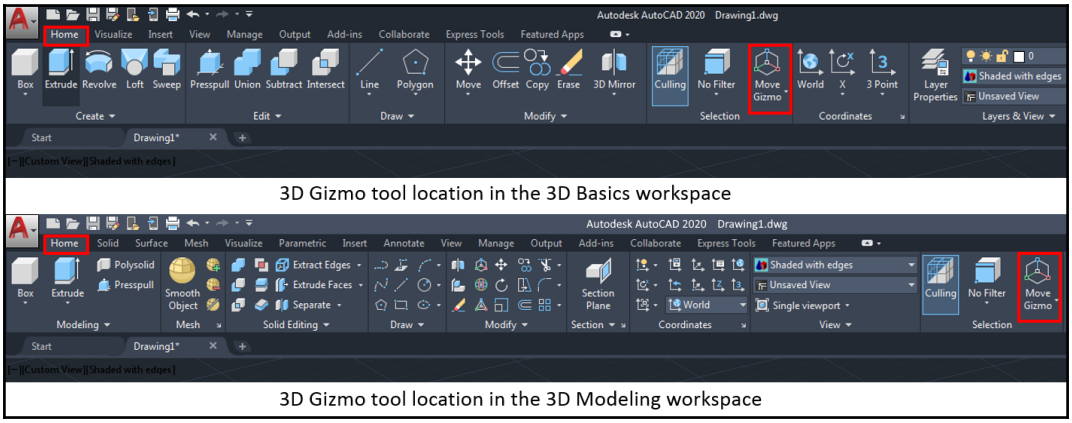


Figure 13. 12: Locations of the 3D Gizmo tool button

If you click on the lower part of the 3D Gizmo tool button, a list containing four entries will expand, showing the four different types for the 3D Gizmo tool, which are as follows:

- **Move Gizmo**
- **Rotate Gizmo**
- **Scale Gizmo**
- **No Gizmo**

The following screenshot indicates the expanded Gizmo tool, showing the four different options as mentioned:

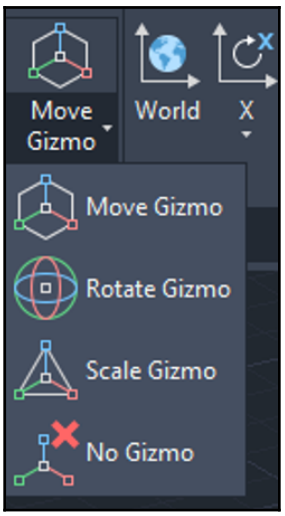


Figure 13. 13: 3D Gizmo tools

Let's now dive deeper into the details of using the first of these four Gizmo tools, the Move Gizmo.

Move Gizmo

To use the **Move Gizmo**, click on the lower part of the Gizmo tool and select **Move Gizmo**, as shown in the previous section. With **Move Gizmo** activated, if you select a 3D object, the **Move Gizmo** icon will appear in the center point of the selected object, as shown in the following diagram. The **Move Gizmo** can be used to quickly move the 3D objects parallel to one of the main UCS axes as follows:

1. Once you select the object, move the mouse cursor near one of the three axes of the **Move Gizmo**. You will notice that an infinite line appears, extending from the axis as shown, indicating that it is ready to move in that direction.
2. Click and move the object to the desired location, and then click again to paste it. You can also enter the distance using the keyboard and press *Enter*, as shown in the following diagram:

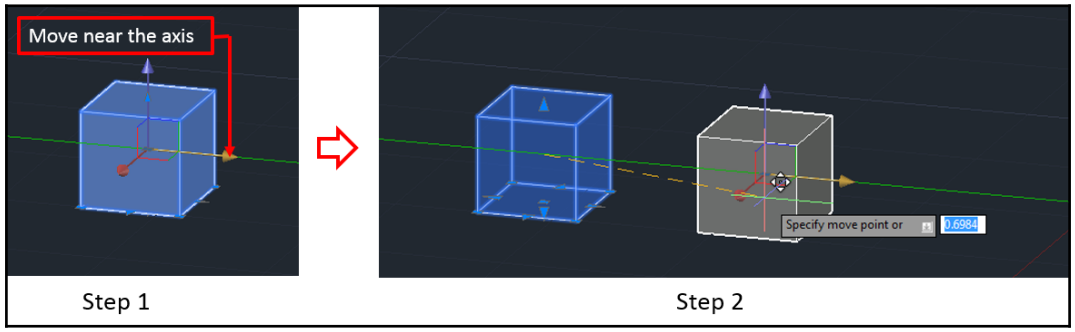


Figure 13. 14: Using the Move Gizmo to move along one of the axes

In addition, you can also restrict movement to a plane instead of just one axis. You can do this by approaching the small plane between any two axes in the **Move Gizmo**, as shown in the following example:

1. Once you select the object, move the mouse cursor near one of the three plane icons of the **Move Gizmo**. You will notice that the plane selected will turn a yellowish color.

2. Click and move the object to the desired location. The object movement will be restricted to the plane selected. Click it again to paste it, as illustrated in the following diagram:

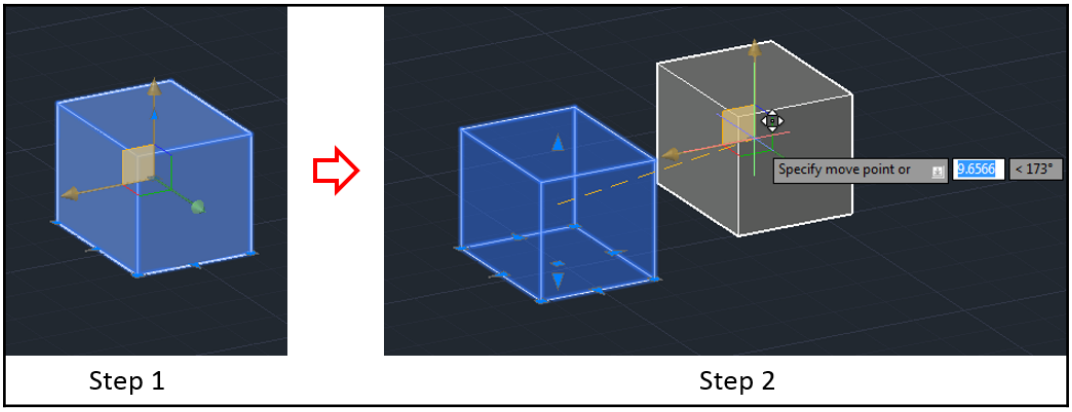


Figure 13. 15: Using the Move Gizmo to move in a specified plane

The **Move Gizmo** has a number of options/alternative methods associated with it. They appear in the command bar right after you click and begin to move, and there are four options, as shown in the following diagram:

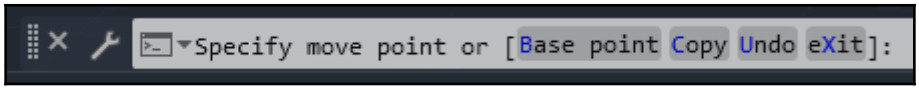


Figure 13. 16: Additional options in the Move Gizmo

These options are explained as follows:

- **Basepoint**: This option is activated by clicking on the option name, or by just typing `B` and then pressing *Enter*. By default, the Gizmo is placed at the center point of the selected object, and that is the base point of the move operations. You can use the **Basepoint** option to change the base point from which you grab and move the object.
- **Copy**: This option is activated by clicking on the option name, or by just typing `C` and then pressing *Enter*. Activate this option to keep the original object and create copies as you paste. You will notice that the **Move Gizmo** will still be active after you paste the first copy, allowing you to paste multiple copies of the object.

- **Undo:** The next option is called **Undo**. Again, you activate it by clicking on the option name, or by typing `U` and then pressing *Enter*. This option is used to undo any changes you have made using the other options. For example, if you activated the copy option and created multiple copies of the object, you can delete the last one created using the undo option, or if you changed the base point, you can return to the default base point location using the undo option.
- **eXit**: The last option is **eXit**. This option can be activated by typing `X` and then pressing *Enter*. This will terminate the `Move Gizmo` command.

Rotate Gizmo

To use the **Rotate Gizmo**, click on the lower part of the Gizmo tool button and select **Rotate Gizmo**, as shown earlier. Now, if you select an object, the **Rotate Gizmo** icon will appear at the center point of the selected object, as shown in the following diagram. The **Rotate Gizmo** can be used to effectively and quickly rotate 3D objects as follows:

1. Once you select the object, hover the mouse cursor over the circle representing the desired rotation direction. Notice that the rotation circle will be highlighted in a yellowish color, and that the rotation axis will be shown infinitely extended.

2. Click and move the mouse to rotate the object and then click again when you arrive at the desired rotation angle. You can also enter the rotation angle by using the keyboard and then pressing *Enter*, as shown in the following diagram:

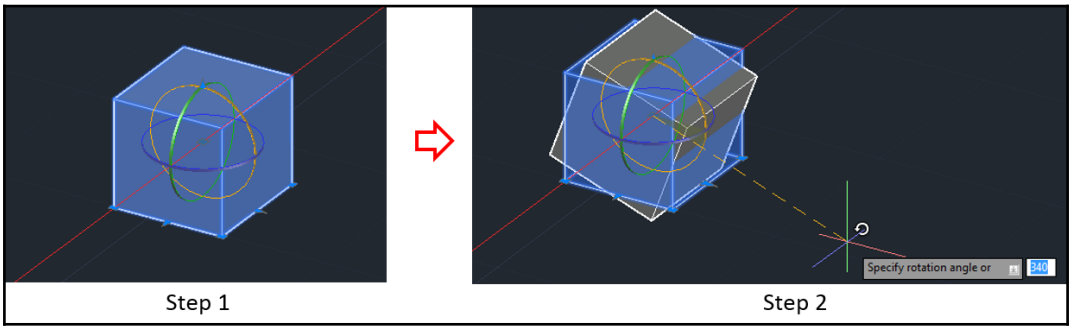


Figure 13. 17: Using the Rotate Gizmo

The **Rotate Gizmo** tool has a number of options/alternative methods that appear in the command bar after you click and start the rotation. There are five options, as shown in the following diagram:

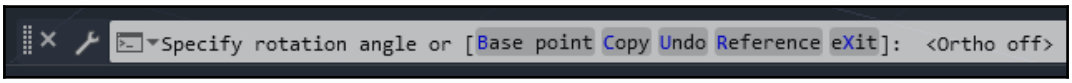


Figure 13. 18: Additional options in the Rotate Gizmo

These options are explained as follows:

- **Base point**: The first option is called **Base point**. It is similar to the base point option in the **Move Gizmo**. To activate it, click on the option name, or just type `B` and then press *Enter*. By default, the **Rotate Gizmo** will be placed at the center of the selected object(s), and the object will be rotated about that point. You can use the **Base point** option to change the rotation base point. Just activate the option and choose the new base point location, as shown in the following diagram:

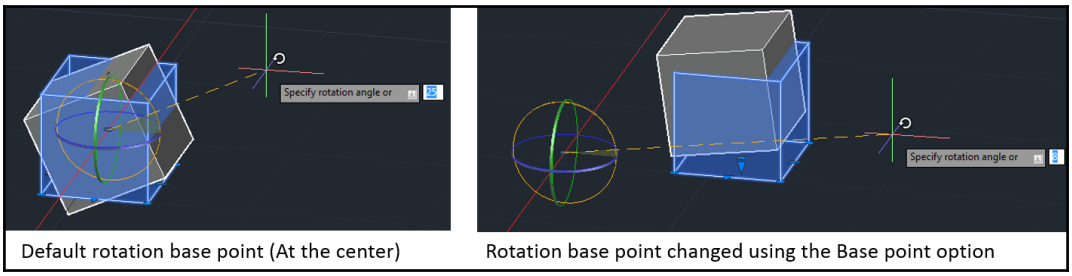


Figure 13. 19: Using the Base point option

- **Copy**: The second option is called **Copy**. To activate it, click on the option name in the command bar, or you can just type `C` and then press *Enter*. This option will keep the original object and create a new rotated copy of the object. Notice that the **Rotate Gizmo** tool will still be active after you create the first copy, thereby allowing you to create multiple rotated copies.

- **Undo**: The next option is called **Undo**. To activate it, click on the option name in the command bar, or you can just type `U` and then press *Enter*. As in the **Move Gizmo**, this option is used to undo any changes you have made using the other options. For example, if you activated the copy option and created multiple copies of the object, you can delete the last one created using the undo option, or if you changed the base point, you can return to the default base point location using the undo option.
- **Reference**: The next option is called **Reference**. To activate it, click on the option name in the command bar, or you can just type `R` and then press *Enter*. When you start rotation, a reference line is generated between the rotation base point and the mouse cursor. By default, this reference line is aligned with the rotated object. The reference option can be used to change the angle of the reference line, as shown in the following diagram:

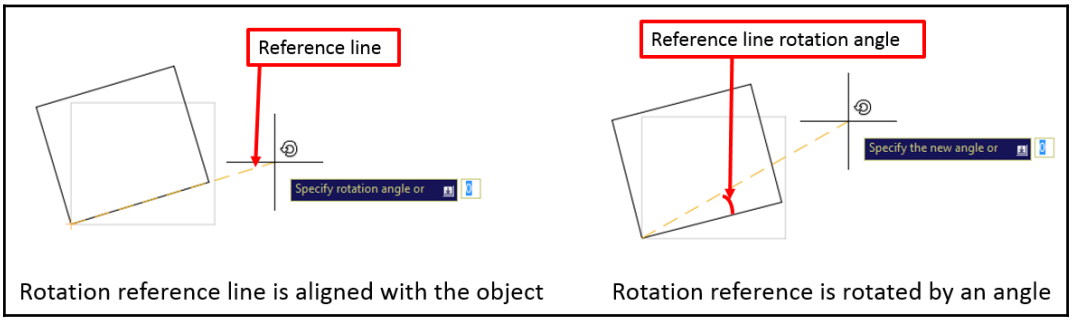


Figure 13. 20: Using the Reference option

- **eXit**: The last option is **eXit**. This option can be activated by typing `X` and then pressing *Enter*. This option will deactivate the **Rotate Gizmo** tool.

Scale Gizmo

To use the **Scale Gizmo**, click on the lower part of the Gizmo tool and select **Scale Gizmo**, as shown at the beginning of this section. Now, with the **Scale Gizmo** activated, if you select a 3D object, the **Scale Gizmo** icon will appear at the center point of the selected object, as shown in the following diagram. The **Scale Gizmo** can be used to quickly scale 3D objects as follows:

1. Once you select the object, hover the mouse cursor over the **Scale Gizmo** icon. Notice that the icon will be highlighted in a yellowish color, indicating that it is ready to be scaled.

2. Click and then move the mouse to scale the object, and then click again when you arrive at the desired scale factor. You can also enter the scale factor using the keyboard and then press *Enter*, as shown here:

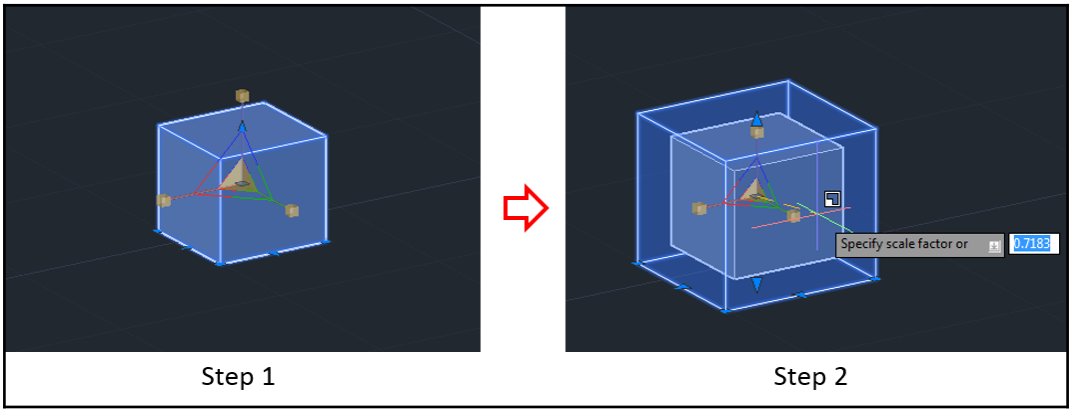


Figure 13. 21: Using the Scale Gizmo

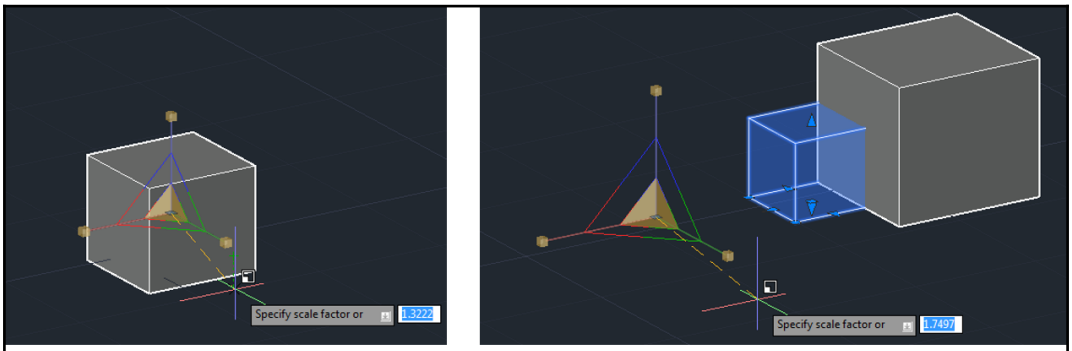
The **Scale Gizmo** tool has a number of options/alternative methods that appear in the command bar after you click and scale the object. There are five options, as shown in the following diagram:



Figure 13. 22: Additional options in the Scale Gizmo

These options are explained as follows:

- The first option is called **Base point**. It is similar to the base point option in the **Move** and **Rotate** Gizmos. To activate it, click on the option name in the command bar, or just type `B` and then press *Enter*. By default, the **Scale Gizmo** icon will be placed at the center of the selected object(s), and the object will be scaled using this center point as the scale base point. You can use the **Base point** option to change it. Just activate the option and click to choose the new base point location, as shown in the following Figure:



Scaling using the default base point (at the center point)

Scaling base point is changed using the Base point option

Figure 13. 23: Using the Base point option

- The second option is called **Copy**. To activate it, click on the option name in the command bar, or you can just type `C` and then press *Enter*. This option will keep the original object and will create a new scaled copy. Notice that the **Scale Gizmo** tool will still be active after you create the first copy, thereby allowing you to create multiple scaled copies.
- The next option is called **Undo**. To activate it, click on the option name in the command bar, or you can just type `U` and then press *Enter*, as with the **Move** and **Rotate** Gizmos. This option is used to undo any changes you have made using the other options. For example, if you activated the copy option and created multiple copies of the object, you can delete the last one created using the undo option, or if you changed the base point, you can return to the default base point location using the undo option.
- The next option is called **Reference**. To activate it, click on the option name in the command bar, or you can just type `R` and then press *Enter*. When you start scaling, a reference line is generated between the scale base point and the mouse cursor. By default, this reference line is aligned with the scaled object. The reference option can be used to change the reference line.
- The last option is **eXit**. This option can be activated by typing `X` and then pressing *Enter*. This option will deactivate the Scale Gizmo tool.

No Gizmo

This is the last entry in the Gizmo tools set. Select this option if you don't want a Gizmo tool to appear when you select any object.

Now that we have learned about different alternative methods to perform the standard modify operations, let's have a look at another type of modify operation, which are the Boolean operations and the commands used to perform them.

Using Boolean operations

Boolean operations are essential in any 3D modeling software. AutoCAD can perform all kinds of Boolean operations on 3D solid bodies and on planar surfaces. The following sections will cover the Boolean operation commands in detail, and these are as follows:

- Using the `union` command
- Using the `subtract` command
- Using the `intersect` command

Let's start by looking at the union command in more detail.

Using the union command

The `union` command is a handy tool when creating complicated 3D models since, in many cases, the 3D object that is required to be created can be broken down into a number of simple 3D shapes. So you can just create the simple 3D shapes using the `primitive` commands and then join them together using the `union` command.

To start the `union` command, you can type the command name, `UNION`, or just type `UN` and then press *Enter*. Alternatively, you can click on the **Union** button found in the **Edit** panel in the **3D Basics** workspace, as shown in the following diagram:

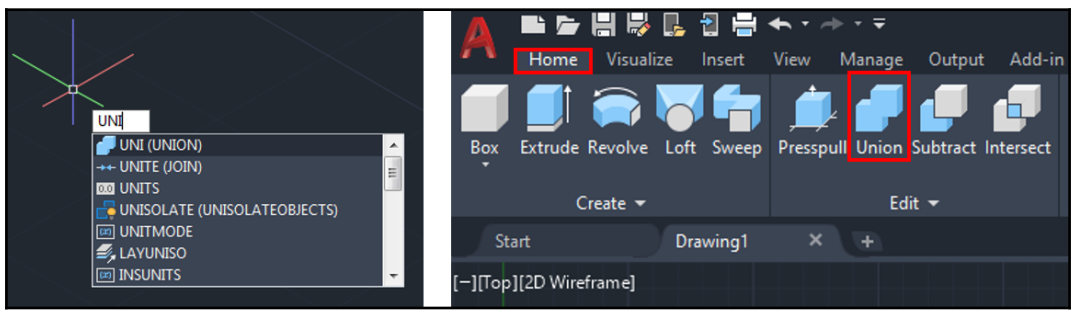


Figure 13. 24: Alternative methods for starting the union command

If you are using the **3D Modeling** workspace, you will find the **Union** button in the **Solid Editing** panel on the **Home** tab, or in the **Boolean** panel on the **Solid** tab.

You can join solid bodies using the `union` command as follows:

1. Start the `union` command.
2. Select the objects to be combined. Notice that the command bar will always display the number of selected objects, and will tell you if one of the objects is excluded from the selection.
3. Once you have selected all the desired objects, press the *Enter* button or right-click. This will combine the selected objects and will end the command.

The following diagram is an illustration of the previous steps:

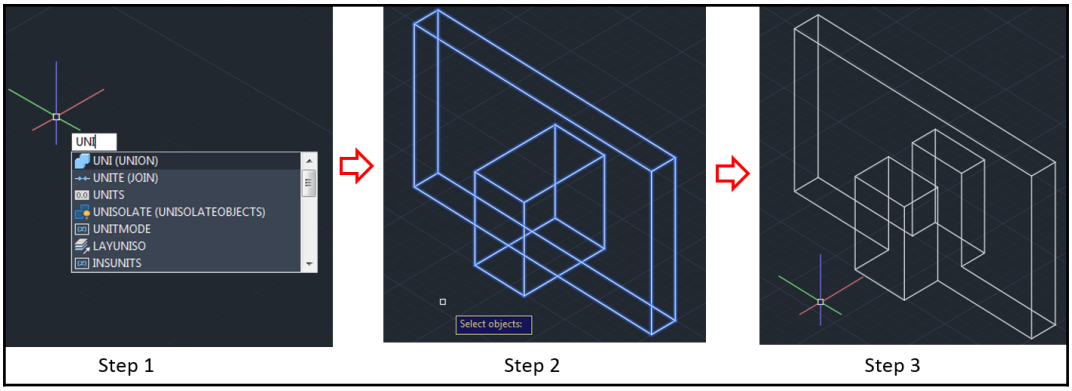


Figure 13. 25: Steps involved in using the union command

The `union` command can also be used with 2D regions or planar surfaces, as shown in the following diagram:

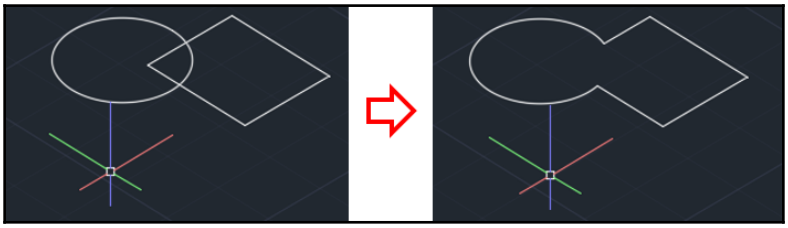


Figure 13. 26: Using the union command with 2D regions

The `union` command can be used to combine either a group of 3D solid objects or a group of 2D surfaces (regions), but it can't create an object by combining a mix of both. The selected objects are not required to be in contact. You can select separated objects and they will be combined in one.

The 3D object created after the union operation will have no grab points, even if the source objects used were primitive solids or had grab points. There will only be a base point to grab and move the object, as shown in the following diagram:

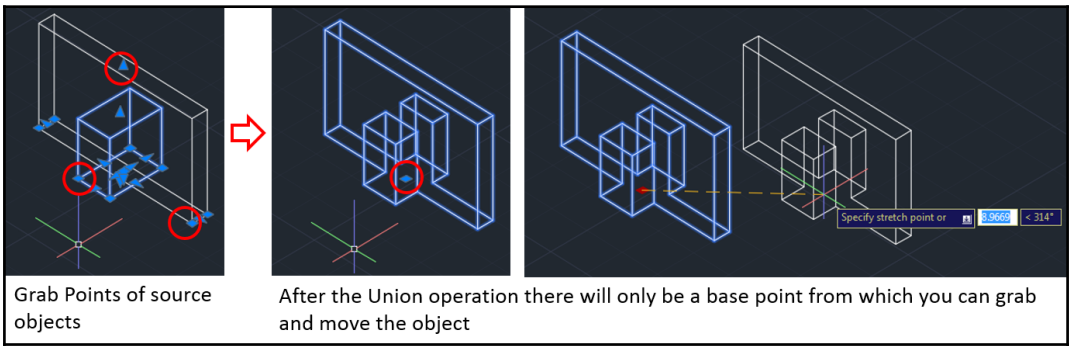


Figure 13. 27: Grab point of objects in union

That's it for the `union` command. Now, let's have a look at the next command in the Boolean group, which is the `subtract` command.

Using the subtract command

Many 3D shapes can be created by subtracting simple 3D objects. For example, if you are drawing a wall that contains windows and doors, it is much simpler to draw a solid wall and then subtract the windows and the doors.

To start the `subtract` command, you can type the command name, `SUBTRACT`, or just type `SU` and then press *Enter*. Alternatively, you can click on the **Subtract** button found in the **Edit** panel in the **3D Basics** workspace, as shown in the following screenshot:

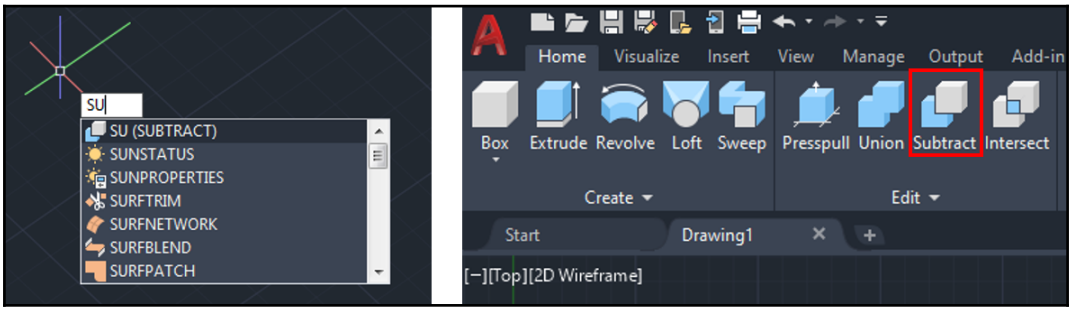


Figure 13. 28: Alternative methods for starting the subtract command

If you are using the **3D Modeling** workspace, you will find the **Subtract** button in the **Solid Editing** panel on the **Home** tab, or in the **Boolean** panel on the **Solid** tab.

Once you start the command, perform the following steps:

1. You will be prompted to select the objects to subtract from. Select the desired objects and then press *Enter*.
2. You will be prompted to select the objects to be subtracted. Select the desired objects and then press *Enter*.
3. The objects will be subtracted and the command terminates.

The following diagram is an illustration of the previous steps:

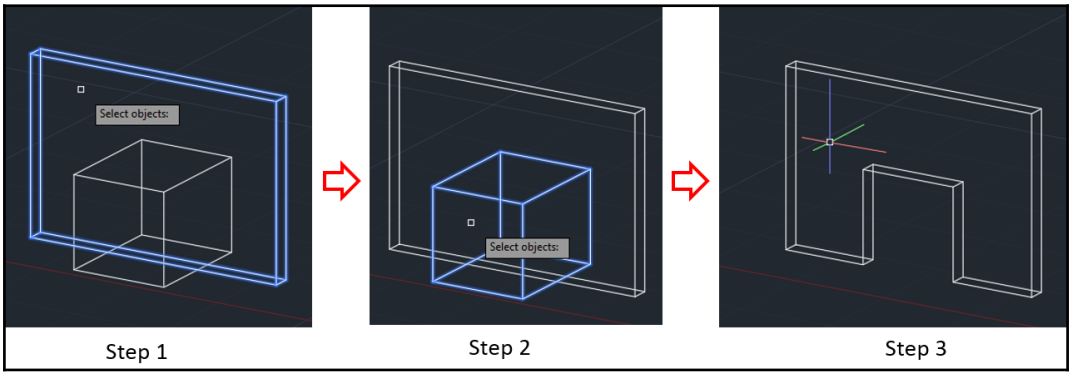


Figure 13. 29: Steps involved in using the subtract command

Similar to the `union` command, the `subtract` command can be used to subtract either a group of 3D solid objects or a group of 2D surfaces (regions), but it can't create an object by subtracting a mix of both. Also, it can't be used with mesh objects. But in case you do select a mesh object, you will be prompted to convert it to a 3D solid or surface.

Also, the created object will have no grab points, even if the objects used in the subtract operation had grab points, as mentioned previously in the *Using the union command* section.

Now, let's look at the final command in the Boolean group, which is the `intersect` command.

Using the intersect command

To start the `intersect` command, you can type the command name, `INTERSECT`, or just type `IN` and then press *Enter*. Alternatively, you can click on the **Intersect** button found in the **Edit** panel in the **3D Basics** workspace, as shown in the following screenshot:

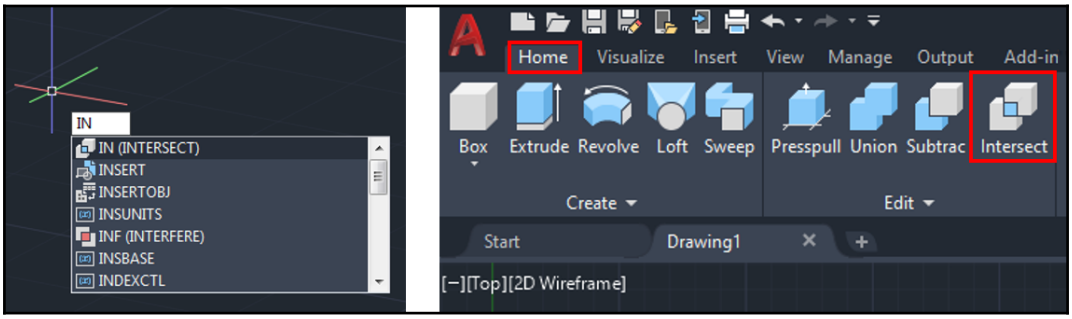


Figure 13. 30: Alternative methods for starting the intersect command

If you are using the **3D Modeling** workspace, you will find the **Intersect** button in the **Solid Editing** panel on the **Home** tab, or in the **Boolean** panel on the **Solid** tab.

The `intersect` command can be used as follows:

1. Start the `intersect` command.
2. Select the intersecting objects.
3. Once you have selected all the desired objects, press the *Enter* button. The intersection will be used to create an object and the command terminates.

The following diagram is an illustration of the previous steps:

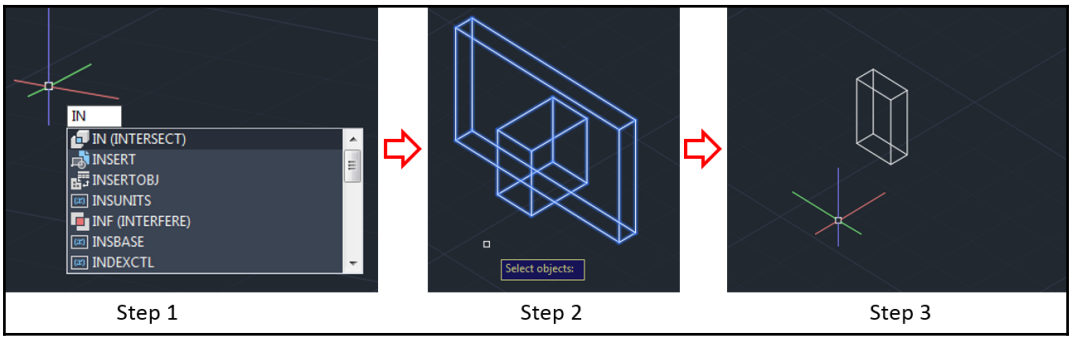


Figure 13. 31: Steps involved in using the intersect command

Similar to the `union` command, the `intersect` command can be used with either a group of 3D solid objects or a group of 2D surfaces (regions), but it can't create an object by using a mix of both. Also, it can't be used with mesh objects. However, in case you do select a mesh object, you will be prompted to convert it to a 3D solid or surface. In addition, the objects used must be intersecting in order for the command to produce a resulting body.

Like other Boolean commands, the created object will have no grab points, even if the objects used in the intersect operation had grab points, as mentioned previously in the *Using the union command* section.

We have now learned how to use different alternatives of the standard modify commands, and how to perform Boolean operations. In the following sections, we will look at three groups of modify commands, each one specific to an element in the 3D object. The first is group is dedicated to solid editing, the second group is dedicated to editing faces of the 3D body, and the final group is dedicated to edge editing commands.

Solid editing commands

In this section, the first of three groups of modify commands will be discussed. As we will see in this section, it adds helpful tools to our solid editing arsenal. You can separate multi-body objects, create a cavity inside a solid body in just one step, slice and object, or create a general curved shape starting from a curved surface. Using these commands, along with their alternative methods and options, will be discussed in detail in this section, Let's now start with the `separate` command.

Using the separate command

Sometimes, you may have a number of separate solid bodies on one solid object definition, and you want to make one of these bodies an individual 3D object. As the name suggests, the `separate` command is used for that specific task. To start the `separate` command, you can type `SOL` and then press *Enter* to open the solid editing option list. Select **Body**, and then select **seParate solids**, as shown in the following screenshot:

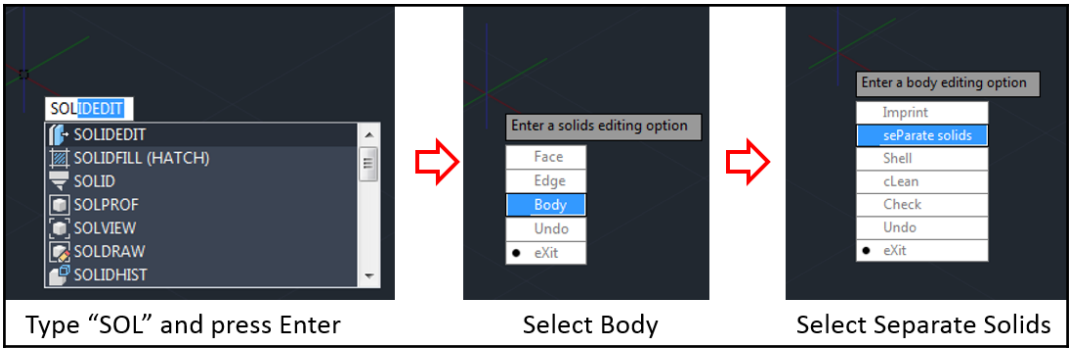


Figure 13. 32: Steps for starting the separate command

If you are using the **3D Modeling** workspace, you will find the **Separate** button in the **Solid Editing** panel on the **Home** tab, or in the **Solid Editing** panel on the **Solid** tab, as shown in the following screenshot:

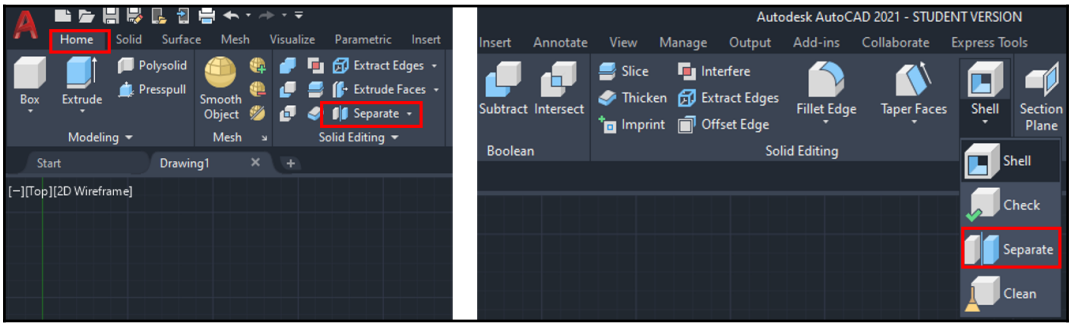


Figure 13. 33: Locations of the Separate tool button

If you have a 3D object that contains multiple non-intersecting bodies, the `separate` command can be used to redefine one or some of them as individual objects as follows:

1. Start the `separate` command.
2. Click on the body to be separated.
3. Exit the body editing list, and then the solid editing list to end the command.

The following diagram is an illustration of the previous steps:

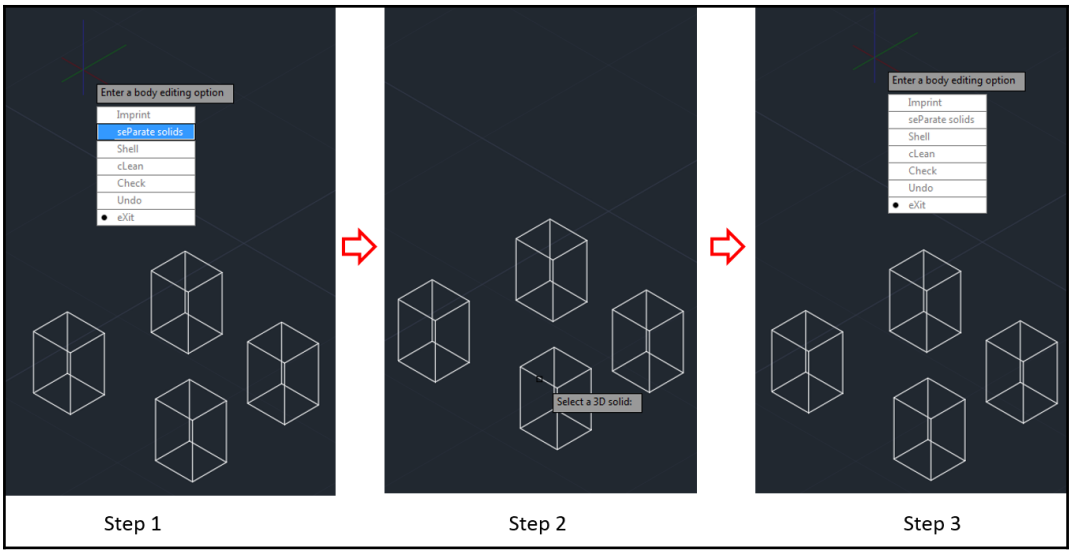


Figure 13. 34: Steps involved in using the separate command

That's it for the `separate` command. Now, let's have a look at the next solid editing tool, which is called the shell command.

Using the shell command

The second solid body editing command in our group is called the `shell` command. It is used to convert a solid body into a hollow shell with a wall of specified thickness, as we will see in this section. To start the `shell` command, you can type `SOL` and then press *Enter* to open the solid editing option list. After that, select **Body**, and then select **Shell**, as shown here:

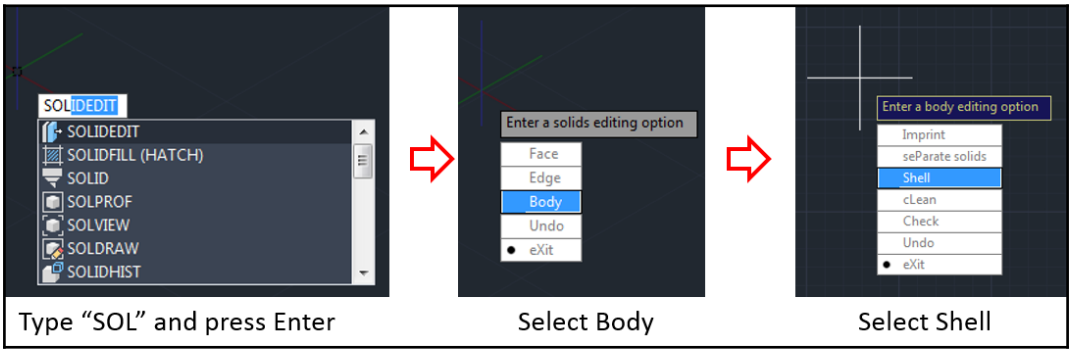


Figure 13. 35: Steps for starting the shell command

If you are using the **3D Modeling** workspace, you will find the **Shell** button in the **Solid Editing** panel on the **Home** tab, or in the **Solid Editing** panel on the **Solid** tab, as shown in the following screenshot:

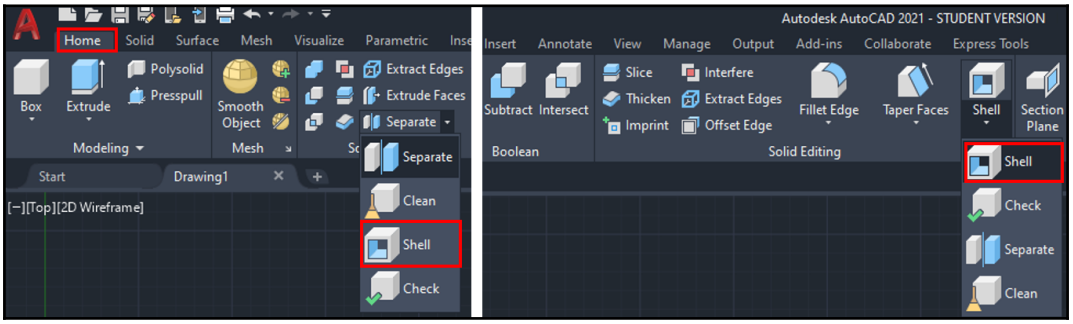


Figure 13. 36: Locations of the Shell tool button in the 3D Modeling workspace

A solid body can be converted into a hollow shape using the shell command, as shown in the following step-by-step example:

1. Start the `shell` command and then, using the mouse cursor, select the desired solid body. You will be prompted to remove faces from the shell operation. We will ignore that for now and just press *Enter*.
2. Specify the shell wall thickness. You can do this either by means of the 2 point method using the mouse cursor or by typing the thickness value and then pressing *Enter*.
3. The hollow shell will be created as shown in the following diagram. Exit the body editing list, and then the solid editing list to end the command.

The following diagram is an illustration of the previous steps:

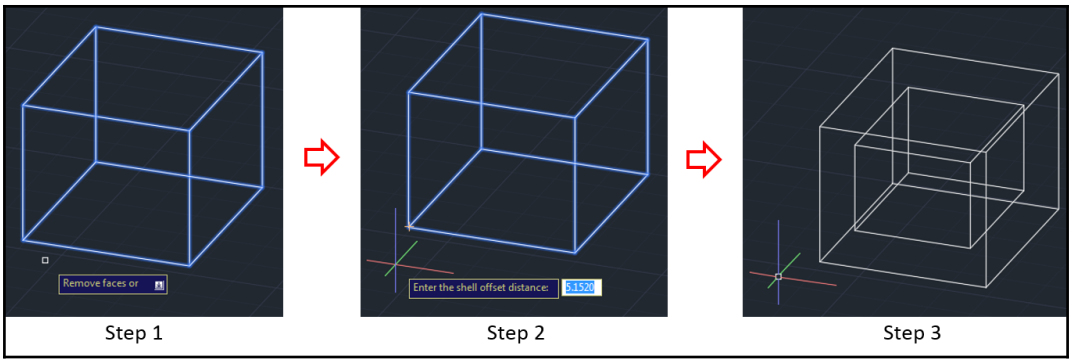


Figure 13. 37: Steps involved in using the shell command

As shown in the previous example, the `shell` command simply offsets the faces of the selected body and then deletes the inner volume bounded by the offset surfaces. The shell command allows you to exclude any of the faces from the offset and the created hollow body will be open from the side of that excluded face, as shown in the following diagram:

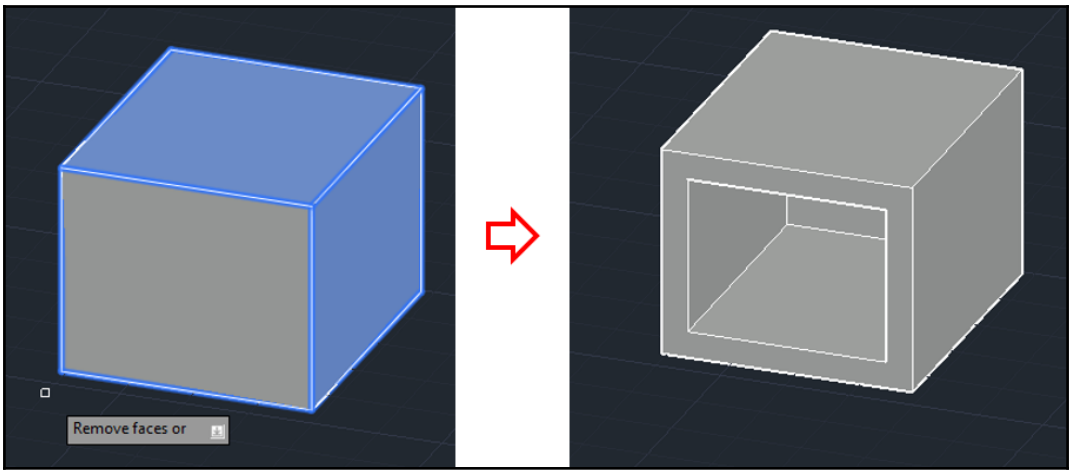


Figure 13. 38: Shell command with one face removed

You can also select multiple faces to be removed and the body will be opened accordingly, as shown in the following diagram, where two surfaces were removed from the shell operation:

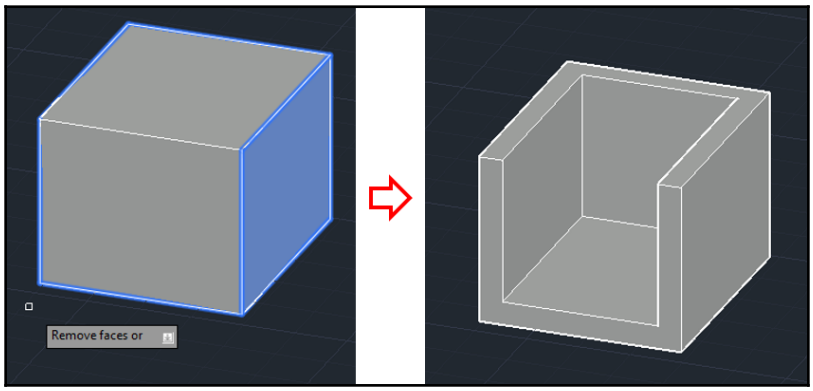


Figure 13. 39: Shell command with two faces removed

We have now learned how to separate a solid body and how to create a hollow shell out it. Now, let's have a look at how to slice it.

Using the slice command

The `slice` command, as the name suggests, can be used to slice a solid body. To start it, you can type the command name, `SLICE` (or just type `SL`), and then press *Enter*. Alternatively, you can click on the **Slice** button found in the **Edit** panel in the **3D Basics** workspace, as shown in the following screenshot:

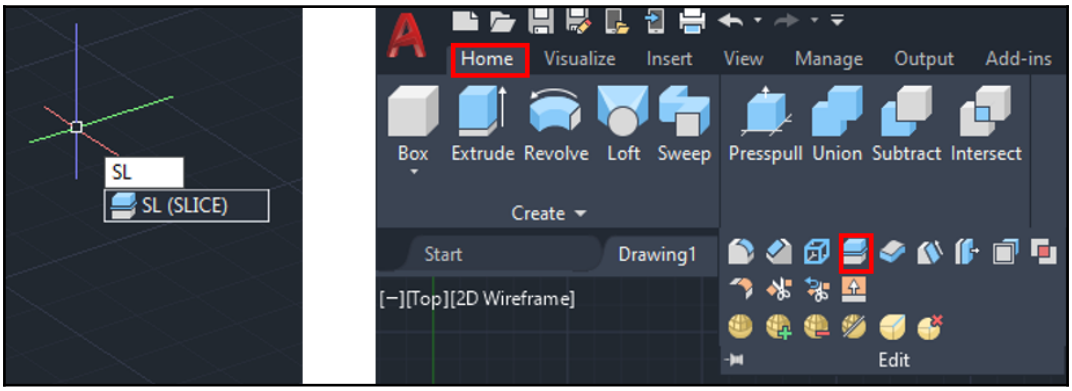


Figure 13. 40: Alternative methods to start the slice command

If you are using the **3D Modeling** workspace, you will find the **Slice** button in the **Solid Editing** panel on the **Home** tab, or in the **Solid Editing** panel on the **Solid** tab.

The `slice` command is used to cut an existing 3D object using a slice plane or surface. In the following example, a plane surface will be used to slice a box as follows:

1. After starting the command, you will be prompted to select the object to be sliced. Click on the box and press *Enter* to end the selection stage.
2. The command bar will display different options for the slicing plane. Type `S` and then press *Enter* to select the surface option.
3. Now, you have to separate the objects. You will be prompted to either keep both of them or to click on the side that you want to keep. Click on the reverse side.
4. The selected part is kept, the other part is deleted, and the command terminated. Now you can delete the slicing surface.

The following diagram is an illustration of the previous steps:

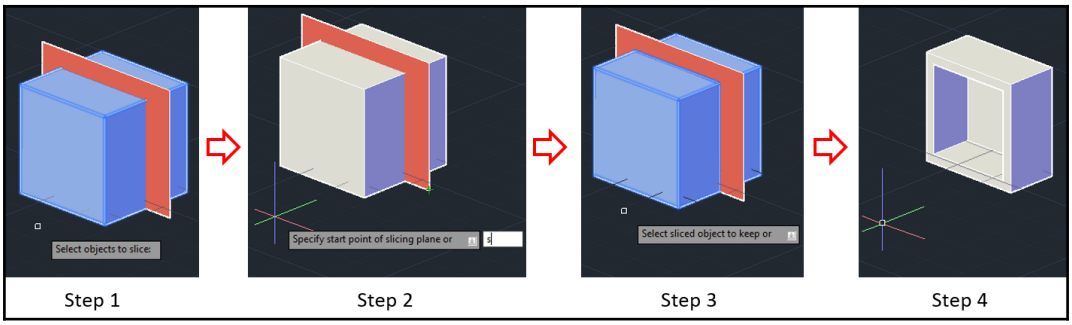


Figure 13. 41: Steps involved in using the slice command

If you select the sliced object, you will find no grab points, and this is the case even if you started with a primitive shape that had grab points before the slice, as shown here:

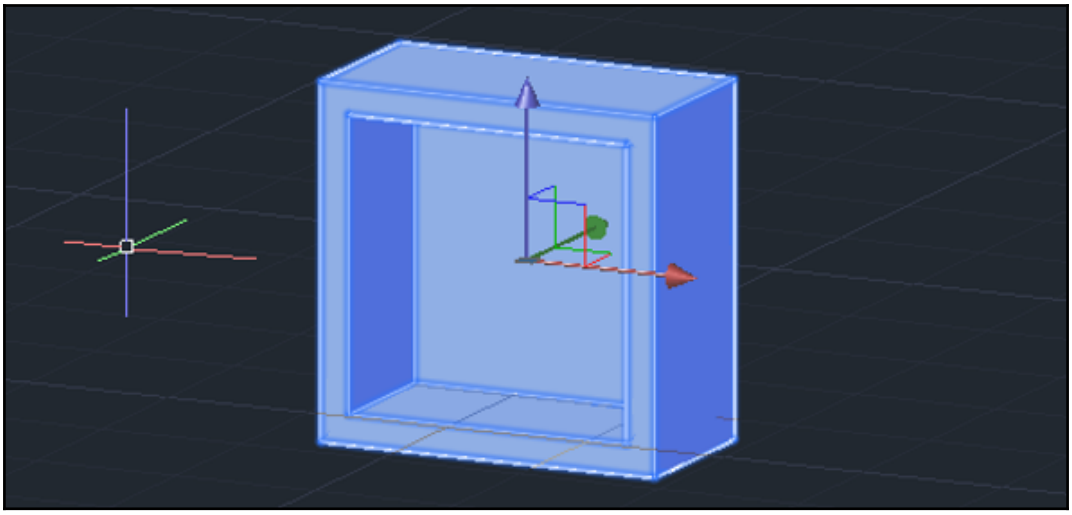


Figure 13. 42: The sliced object has no grab points

As mentioned in the previous example, there are alternative methods/options to specify the cutting plane. In this example, an existing surface is used as a cutting plane. The other methods will appear in the command bar, and these are as follows:



Figure 13. 43: Additional options for the slice command

- **planar Object** can be selected by typing `O` and then pressing *Enter*. This option will align the cutting plane with a planar object you select. The planar object can be a circle, an arc, a rectangle or a polyline.
- **Z-axis** can be selected by typing `Z` and then pressing *Enter*. In this option, you specify a vector by choosing two points. This vector is normal to the cutting plane, with the starting point lying on the cutting plane.
- **View** can be selected by typing `V` and then pressing *Enter*. This option aligns the cutting plane with the view angle. You just specify a point that lies on the plane to specify the plane location.
- **XY**, **YZ**, and **ZX** options can be used to align the cutting plane with the any of the UCS three main planes. You choose the plane direction, and then specify the plane location by choosing a point.
- **3points** can be selected by typing `3` and then pressing *Enter*. This option specifies the cutting plane by determining three points that lie on it.

The following diagram is an illustration of the **planar Object**, **View**, **Z-axis**, and **3Point** methods:

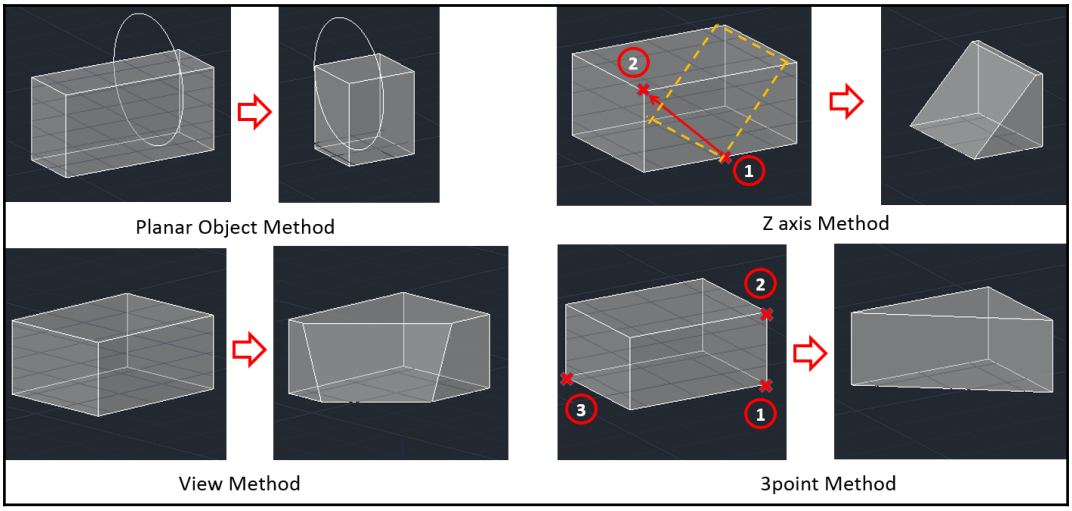


Figure 13. 44: Using additional options with the slice command

That's it for the `slice` command. Now, let's have a look at the next command in our list, which is the `Thicken` command.

Using the Thicken command

The third command in the solid editing group is called *Thicken*. This command is used to add thickness to a generally curved surface. To start the `Thicken` command, you can type the command name, `THICKEN` (or just type `TH`), and then press *Enter*. Alternatively, you can click on the **Thicken** button found in the **Create** panel in the **3D Basics** workspace, as shown in the following screenshot:

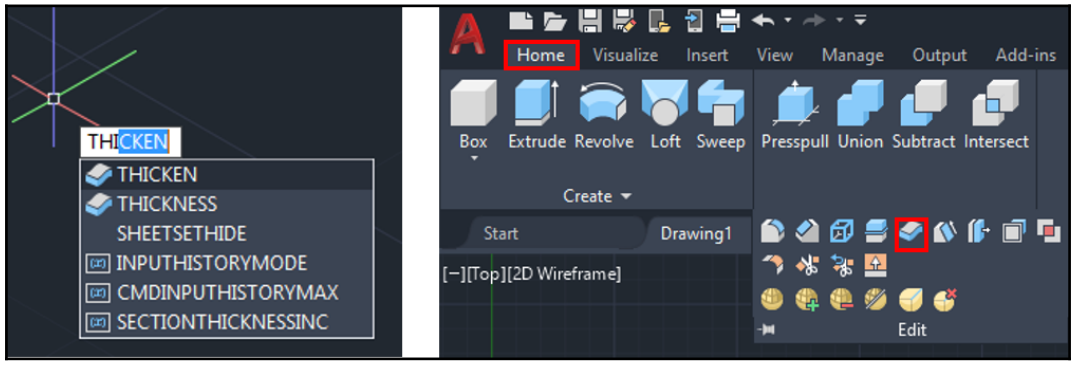


Figure 13. 45: Alternative methods for starting the Thicken command

If you are using the **3D Modeling** workspace, you will find the **Thicken** button in the **Solid Editing** panel on the **Home** tab, or in the **Boolean** panel on the **Solid** tab.

The Thicken command is used to create 3D solid objects by adding thickness to a surface. It can be used as follows:

1. Start the `Thicken` command.
2. Select the surface(s) to be thickened and then press *Enter* to end the selection stage.
3. Type the required thickness value using the keyboard and then press *Enter*.
4. The surface will be thickened and the command terminates.

The following diagram is an illustration of the previous steps:

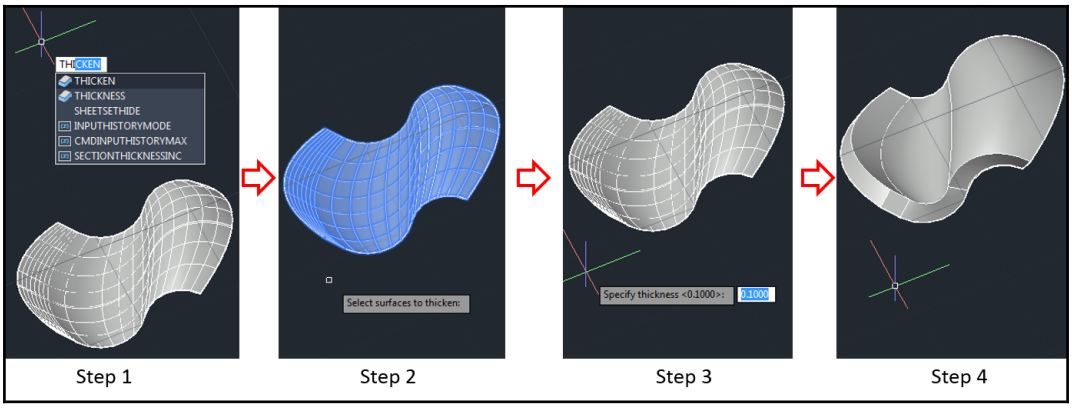


Figure 13. 46: Steps involved in using the Thicken command

If you select the thickened object, you will find a number of grab points. All of these points act like base points, from which you can grab and move the object, as shown here. They cannot be used to modify the thickened object, as shown in the following diagram:

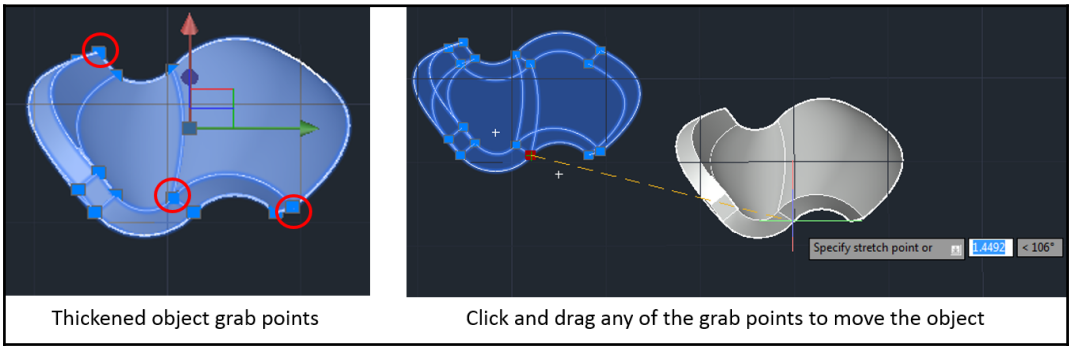


Figure 13. 47: Thickened object grab points

As demonstrated in the previous example, the Thicken command is very similar to the `extrude` command. One difference between these two commands is the fact that the latter cannot be used with curved surfaces, the surface must be planar for the extrude command to work, while the `Thicken` command can generally work with any surface, as shown in the previous example. Another difference is the fact that the `extrude` command works with surfaces, regions, and polylines, and if the polyline is closed, it will automatically generate a 3D solid object, while the `Thicken` command is limited to surface objects. This command won't work with regions or closed polylines. The following table summarizes the differences between the `extrude` and `Thicken` commands, and for more details on the `extrude` command, please refer to `Chapter 12`, *Conversion between 2D and 3D*:

	Thicken command	**Extrude command**
Surfaces (planar)	Works	Works
Surfaces (general/curved)	Works	Does not work
Regions	Does not work	Works
Closed polylines	Does not work	Works

By default, the original surfaces used will be deleted once the Thicken command terminates. You can control whether to delete these objects by using the DELOBJ system variable. To access this, just type `DELOBJ` and then press *Enter*. The current value of the DELOBJ system variable will be displayed, and then you can type the new value and press *Enter*. For more details regarding the DELOBJ system variable, please refer to `Chapter 12`, *Conversion between 2D and 3D*.

Now that we have learned how to use the different solid editing tools in AutoCAD, let's take a look at the next group of modify commands that are specific to face editing

Face editing commands

The second set of modify commands is related to face editing, or to commands that are based on the faces of the 3D solid. As we will see in this section, you can add a taper angle to the solid body face, create an extrusion using the solid body face, or you can offset the faces in a way similar to the 2D curves. In this section, we will take a detailed look at these commands and their alternative methods and options. The first command in our list is called taper faces. Let's now dive deeper into its details.

Using the taper faces command

The first command dedicated to face editing is called taper. This command is used to tilt the face of an existing solid body, as we will see in this section. To start the taper faces command, you can type `SOLIDEDIT` and then press *Enter*, and then click on **Face** followed by **Taper**. Alternatively, you can click on the **Taper Faces** button found in the **Edit** panel in the **3D Basics** workspace, as shown in the following diagram:

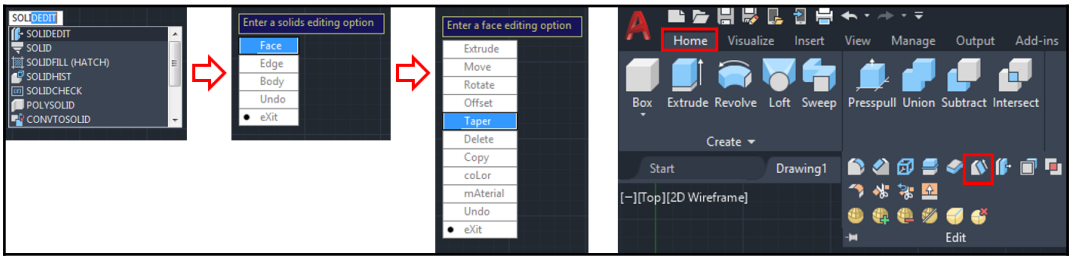


Figure 13. 48: Alternative methods for starting the taper faces command

If you are using the **3D Modeling** workspace, you will find the **Taper Faces** button in the **Solid Editing** panel on the **Solid** tab.

The `taper faces` command can be used to tilt a surface of an existing 3D solid body, as shown in the following example:

1. After you start the `taper faces` command, select the faces to be tapered and then press *Enter* to end the face selection stage.
2. Specify the taper direction by determining the taper axis. This is determined by specifying two points.
3. You will be prompted to enter the taper angle. Type the angle value and then press *Enter* to finish.

The following diagram is an illustration of the previous steps:

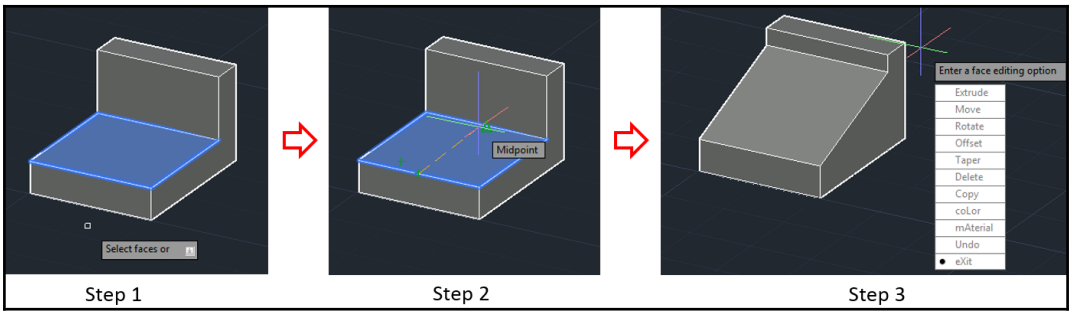


Figure 13. 49: Using the taper faces command

In determining the taper axis, you determine two points. The first point is the base point of the tapering axis, and this side of the face will not be moved; then, you determine the second point, which will be moved to achieve the taper angle, as shown in the following diagram:

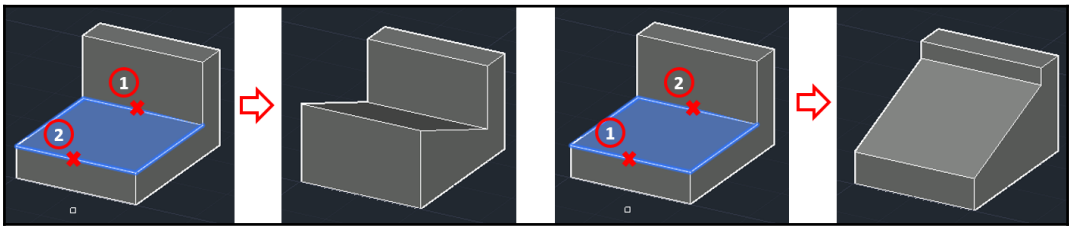


Figure 13. 50: Effect of changing the taper axis direction

The taper angles determine whether the tilting will be inward or outward; negative angles will taper outward, while positive angles will taper inward, as shown in the following diagram:

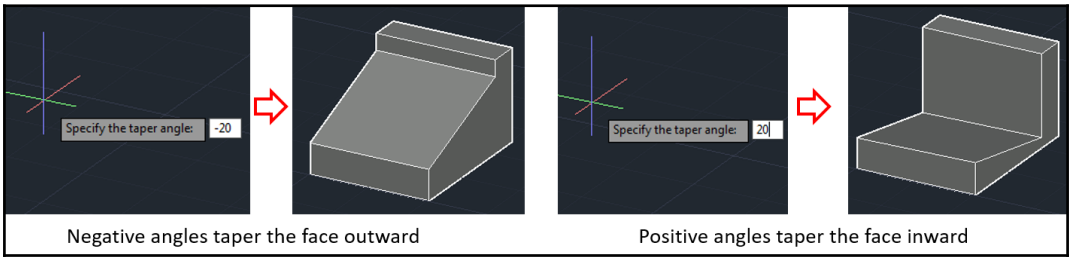


Figure 13. 51: Positive versus negative taper angle

Notice that if you perform taper face operations on a 3D object, the grab point will disappear from the resulting 3D body, even if you started with a primitive 3D object that had grab points.

Now that we have learned how to tilt the face of a solid body, let's have a look at the next command in our group, which is called extrude faces.

Using the extrude faces command

The second command in our face modify group, this can be used to directly extrude a face of an existing 3D solid body. It is very similar to the `PressPull` command, one difference being that the `PressPull` command can also be used to extrude enclosed regions as shown in the previous section, while the extrude faces command is limited to faces of existing solid objects. A second difference is that the extrude faces group has more extrusion options, such as the taper angle and the path, as will be shown in detail in this section. To start the `extrude faces` command, you can type `SOLIDEDIT` and press *Enter*, and then click on **Face** followed by **Extrude**. Alternatively, you can click on the **Extrude Faces** button found in the **Edit** panel in the **3D Basics** workspace, as shown in the following diagram:

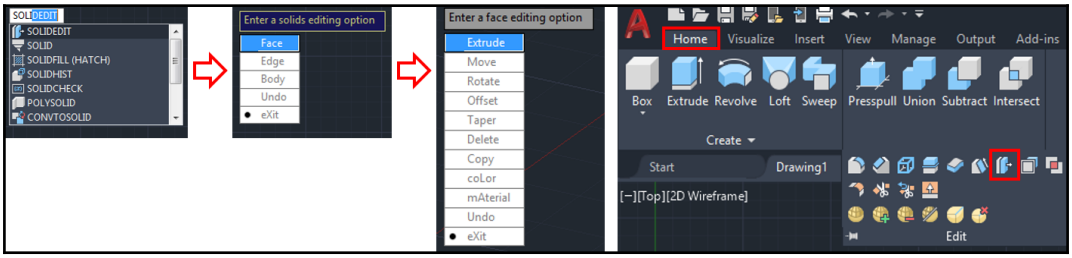


Figure 13. 52: Alternative methods for starting the extrude faces command

If you are using the **3D Modeling** workspace, you will find the **Taper Faces** button in the **Solid Editing** panel on the **Solid** tab.

The `extrude faces` command can be used to quickly extrude a face in an existing 3D solid body, as shown in the following example:

1. After you start the `extrude faces` command, select the faces to be extruded and then press *Enter* to end the face selection stage.
2. Type the extrude distance using the keyboard and press *Enter*.

3. You will be prompted to enter the taper angle, or you can just press *Enter* if you don't want to. The extruded volume will be created, and you will be prompted to select another option from the `SOLIDEDIT` command list. To end the command, choose **Exit** twice.

The following diagram is an illustration of the previous steps:

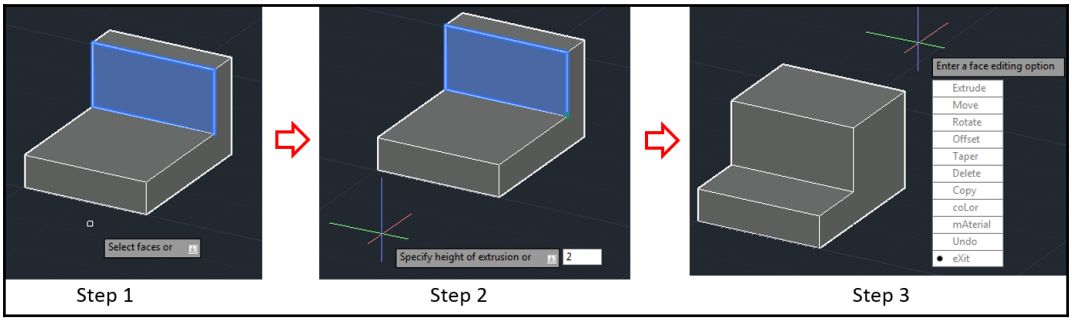


Figure 13. 53: Steps involved in using the extrude faces command

The taper angle shown at the end of the command can be used, as shown in the following diagram:

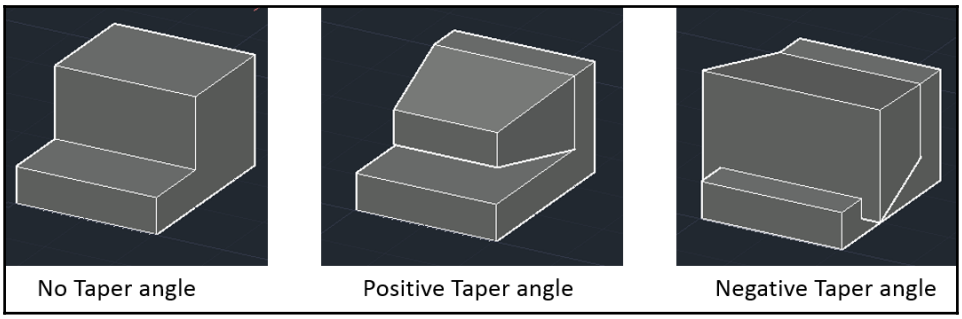


Figure 13. 54: Extrude faces taper angle

The `extrude faces` command has one option called `path`. It appears after you specify the faces to be extruded. To select it, click on the option name in the command bar, or you can just type `P` and then press *Enter*. This option allows you to select a path for the extruded face instead of a straight extrude in the default case. Look at the following example:

1. Start the `extrude faces` command, select the face, and then activate the Path option.
2. Once you click on the path, the face will be extruded accordingly, and then you will be prompted to select another option from the `SOLIDEDIT` command list. To end the command, select **Enter** twice, as shown here:

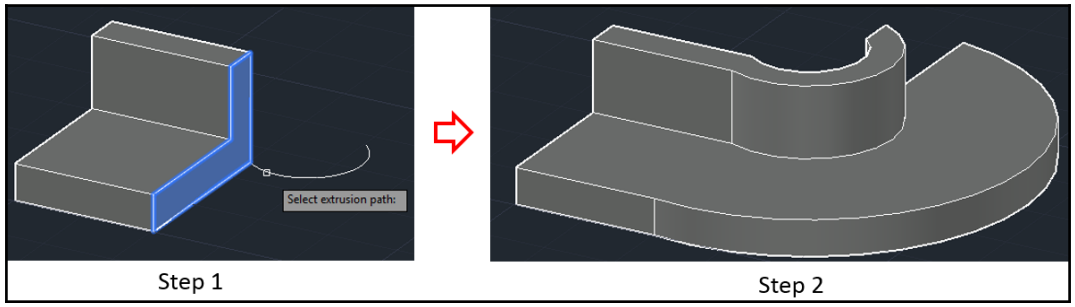


Figure 13. 55: Using the Path option

Notice that if you perform an extrude faces operation on a 3D object, it will no longer have grab points, even if you started with a primitive 3D object that had grab points.

We have learned how to tilt a face, and how to extrude it. Now, let's take a look at the final face modification command in our list, which is the `offset faces` command.

Using the offset faces command

The `offset faces` command is the surface counterpart of the standard offset command used with 2D curves. The same concept is applied to faces of existing solid objects, as will be shown in this section. To start the `offset faces` command, you can type `SOLIDEDIT` and press *Enter* and then click on **Face** followed by **Offset**. Alternatively, you can click on the **Offset Faces** button found in the **Edit** panel in the **3D Basics** workspace, as shown in the following screenshot:



Figure 13. 56: Alternative methods for starting the offset faces command

If you are using the **3D Modeling** workspace, you will find the **Offset Faces** button in the **Solid Editing** panel on the **Solid** tab.

The `offset faces` command can be used to quickly offset a face in an existing 3D solid body, as shown in the following example:

1. After you start the `offset faces` command, select the faces and then press *Enter* to end the face selection stage.
2. Type the offset distance using the keyboard and press *Enter*.
3. The face offset will be executed and the body shape will be modified according to the new face position. After this, you will be prompted to select another option from the `SOLIDEDIT` command list. To terminate the command, select **Exit** twice.

The following diagram is an illustration of the previous steps:

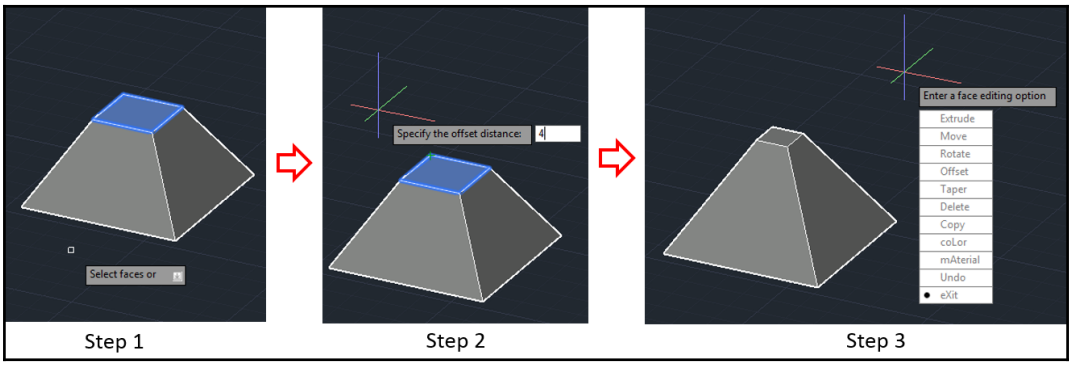


Figure 13. 57: Steps involved in using the offset faces command

Notice that if you perform the offset faces operation on a 3D object, the grab points will disappear from the resulting 3D body, even if you started with a primitive 3D object that had grab points.

Now that we have learned the different face modification commands in AutoCAD, let's have a look at the next specialized group of commands, which are edge editing commands.

Edge editing commands

This section will cover the final group of specialized modify commands. This is a group of commands for editing edges of solid objects. So now let's dive deeper into the details of each command.

Using the FILLETEDGE command

Just like the sharp corners which can be replaced by rounded fillets using the fillet command in 2D shapes, the edges of a 3D solid body can be replaced with a rounded fillet using the FILLETEDGE command. To start the `FILLETEDGE` command, you can type the command name, `FILLETEDGE` (or just type `FILLETE`), and then press *Enter*. Alternatively, you can click on the **Fillet Edge** button found in the **Edit** panel on the **Home** tab in the **3D Basics** workspace, as shown in the following screenshot:

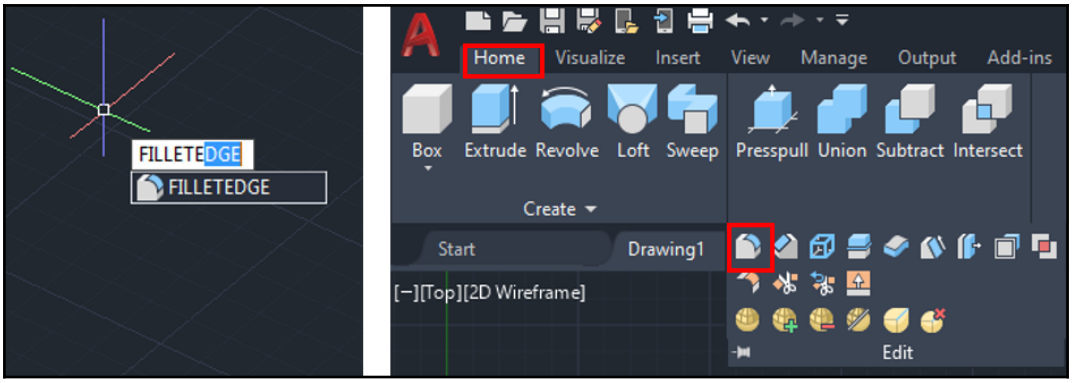


Figure 13. 58: Alternative methods for starting the FILLETEDGE command

If you are using the **3D Modeling** workspace, you will find the **Fillet Edge** button in the **Solid Editing** panel on the **Solid** tab.

To convert a sharp edge of a 3D solid body into a rounded fillet using the `FILLETEDGE` command, perform the following steps:

1. Start the `FILLETEDGE` command.
2. Click on the desired edge(s) on a 3D solid and press *Enter* to end the edge selection stage. Notice that a preview of the fillet will be displayed.
3. You will be prompted to specify the fillet radius. Type the fillet radius value and then press *Enter*, or just press *Enter* if you don't want to change the fillet radius value.
4. Once you type the radius and press *Enter*, the fillet will be created and the command terminates.

The following diagram is an illustration of the previous steps:

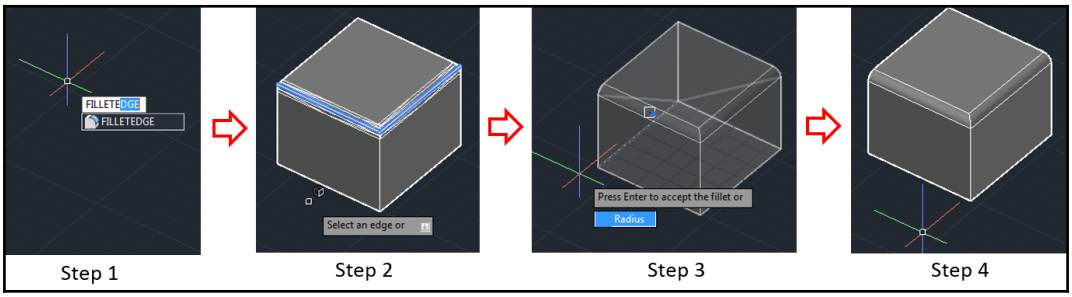


Figure 13. 59: Steps involved in using the FILLETEDGE command

The `FILLETEDGE` command has a number of options that appear in the command bar right after you start the command. There are three options, as shown in the following diagram:

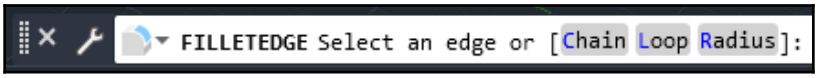


Figure 13. 60: Additional options in the FILLETEDGE command

The first option is called **Chain**. To activate it, click on the option name in the command bar, or you can just type `C` and then press *Enter*. By default, if you are going to select multiple edges, you have to select them one by one, with the chain option activated. If you have a number of edges that are tangent at their connected ends, they will all be selected with one click, as shown in the following diagram:

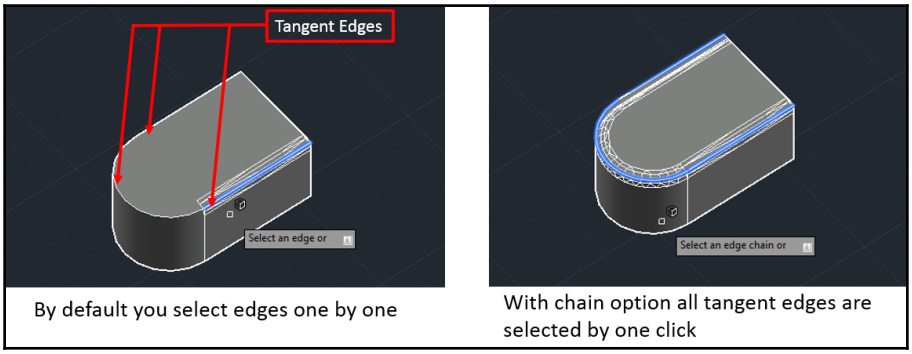


Figure 13. 61: Using the Chain option

The second option is called **Loop**. This can be activated by clicking on the option name in the command bar, or you can just type `L` and then press *Enter*. By default, if you are going to select multiple edges, you must do so one by one. With the loop option activated, you can select the closed loops containing the edge you select with just one click, as shown here:

1. After you start the `FILLETEDGE` command, type `L` and then press *Enter* to activate the loop option.
2. Click on one of the body edges as shown.
3. A loop that contains the selected edge is highlighted as shown in the diagram and you have two options; either to accept the highlighted loop, or click **Next** to view the next loop containing the selected edge.
4. Click **Next** and this will highlight the other loop containing the selected edge.
5. Click **Accept**.
6. You will be prompted to select other edges. Notice that a preview of the loop fillet will be shown.
7. Press *Enter* twice to end the FILLETEDGE command as usual.

The following diagram is an illustration of the previous steps:

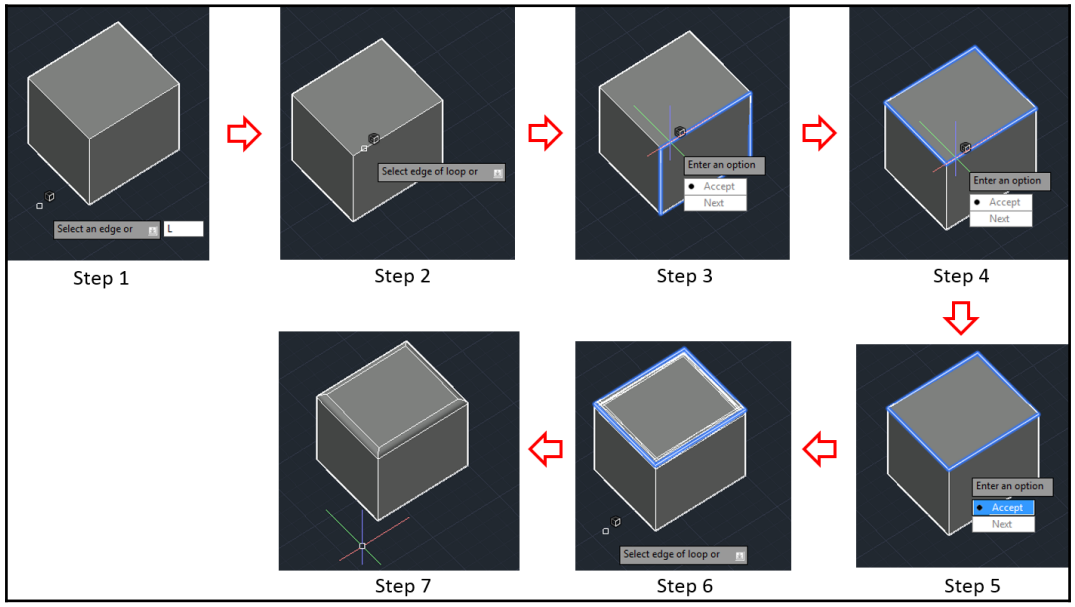


Figure 13. 62: Using the Loop option

The last option is called **Radius**. To activate it, click on the option name in the command bar, or you can just type `R` and then press *Enter*. This option allows you to change the fillet radius.

Using the chamfer edge command

To start the `chamfer edge` command, you can type the command name, `CHAMFEREDGE`, or just type `CHAMFERE` and then press *Enter*. Alternatively, you can click on the **Chamfer Edge** button found in the **Edit** panel on the **Home** tab in the **3D Basics** workspace, as shown in the following screenshot:

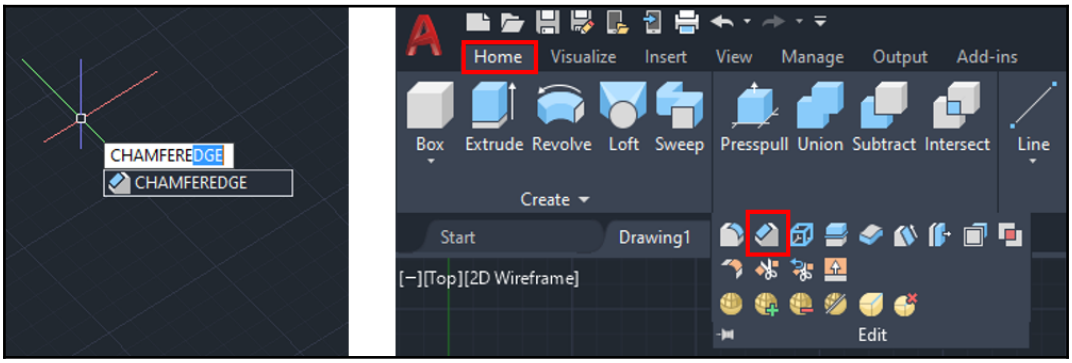


Figure 13. 63: Alternative methods for starting the chamfer edge command

If you are using the **3D Modeling** workspace, you will find the **Chamfer Edge** button in the **Solid Editing** panel on the **Solid** tab.

To chamfer a sharp edge of a 3D solid body using the `chamfer edge` command, perform the following steps:

1. Start the `chamfer edge` command.
2. Click on the desired edge(s) on a 3D solid body and press *Enter* to terminate the edge selection stage. Notice that a preview of the chamfer will be displayed.
3. You will be prompted to specify the chamfer distance. Type the chamfer distance value and press *Enter*, or just press *Enter* if you don't want to change the current chamfer distance.
4. Once you type the distance and press *Enter*, the chamfer will be created and the command terminates.

The following diagram is an illustration of the previous steps:

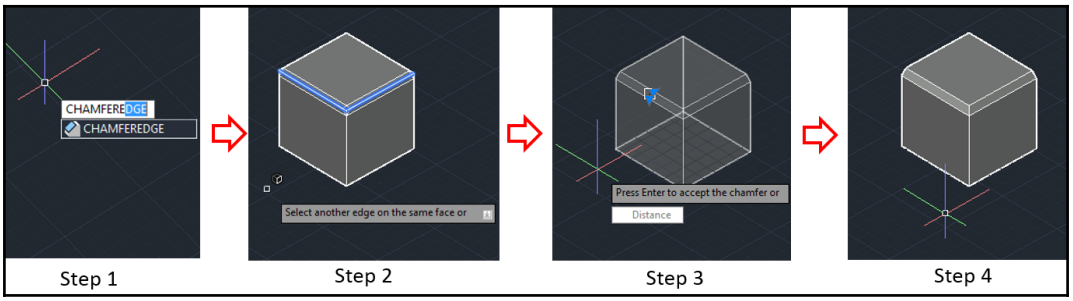


Figure 13. 64: Steps involved in using the chamfer edge command

The `chamfer edge` command has a number of options that appear in the command bar right after you start the command. There are two options, as shown in the following diagram:

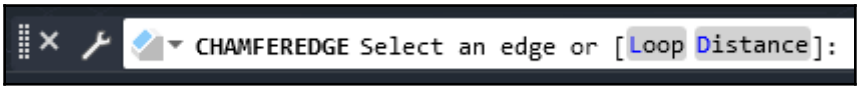


Figure 13. 65: Additional options in the chamfer edge command

The first option is called **Loop**. This can be activated by clicking on the option name in the command bar, or you can just type `L` and press *Enter*. By default, if you are going to select multiple edges, you must do so one by one. With the loop option activated, you can select the closed loops containing the edge you select with just one click.

This option works exactly like the **Loop** option in the FILLETEDGE command. For more details, please refer to the FILLETEDGE section of this chapter

The last option is called **Distance**. To activate it, click on the option name in the command bar, or you can just type `D` and then press *Enter*. This option allows you to change the chamfer distance.

This brings us to the end of this section, where we have learned how to modify the sharp edges of solid bodies using edge chamfer and fillet commands.

Summary

In this chapter, the modify commands used for editing 3D solid objects were discussed in detail, starting with basic commands (including `move`, `copy`, `mirror`, and `scale`) that are used in 2D drafting and can also be used with 3D objects but with limitations. 3D versions of these commands were also introduced that are easier and quicker when used with 3D objects, in addition to removing the limitations of the basic ones.

After this, Boolean operations (`union`, `subtract`, and `intersect`) were discussed in detail, with examples provided for each operation, followed by three groups of editing commands. The first group dealt with solid modifications, including the `slice` and `Thicken` commands, the second group included those commands that specialized in editing faces, and the final group covered edges.

In the previous chapters, `Chapter 11`: Creating Primitive 3D Shapes, `Chapter 12`, *Conversion between 2D and 3D*, and this one, we dealt with solid 3D objects. The next chapter will explore the other types of 3D objects that are available in AutoCAD, which are surface and mesh objects.

14
Surfaces and Mesh Modeling

In the previous chapters, we saw the different commands that are available for creating 3D solid objects. Alongside the `modify` commands, these commands have a countless number of options when it comes to creating 3D shapes, and although using the commands described in the previous chapters (`Chapter 10`, *Introduction to 3D Modeling*, `Chapter 11`, *Creating Primitive 3D Shapes*, and `Chapter 12`, *Conversion between 2D and 3D*) can be used to create almost any 3D model, AutoCAD offers other types of 3D objects that represent different approaches to creating 3D models. These types have different creation methods, different properties, and different modification possibilities.

The first type is surface modeling, where surfaces will be used to represent the 3D model instead of 3D solids. In surface modeling, you can associate the created surfaces with other objects using procedural surfaces. The surfaces and the associated objects can then be manipulated as one group. In addition, you can use **NURBS** (**Non-Uniform Rational B-Splines**) surfaces, which have control vertices instead of their being associated with other objects. These two approaches are totally different from what we have seen in solid modeling in the previous chapters.

The second type is mesh modeling. Unlike solid objects, mesh objects have no mass properties. They are composed of smaller components, which are vertices, edges, and faces. These components can be individually moved, rotated, and scaled. In addition, you can also apply creases and increase the object smoothness, thereby allowing for greater flexibility and providing a different approach to creating the 3D model that is not available in solid modeling.

So, to sum up, in this chapter, we will cover the following topics:

- Introducing the different types of surfaces and learning how to create them
- Learning how to use surface modification tools
- Understanding the mesh modeling concept and learning how to create different mesh objects
- Using the mesh modification capabilities and tools

So let's begin by taking a closer look at surface modeling.

Surface modeling

A 3D model can be defined inside the modeling software either as a solid or as a hollow body (in other words, it is made up of external surfaces). As mentioned in the introduction to this chapter, AutoCAD offers the option to create the 3D model either way. In this section, surface modeling will be discussed, the two surface types available in AutoCAD (planar and network) will be introduced, and the creation and modification commands will be explored in detail.

Creating surfaces

A surface object is a shell with an infinitesimally small thickness. Using surfaces to create the 3D model affords the advantage of using associative modeling or sculpting using the control vertices of NURBS surfaces. In the following sub-sections, we will learn how to create the two main types of surfaces available in AutoCAD, which are as follows:

- Planar surfaces
- Network surfaces

The latter has the option to be associative or to be a NURBS surface, as we will see in this section. After that, we will learn about additional surface creation commands that use the edges of existing surfaces to close gaps and holes in the model, and we will learn about the `surface offset` command.

Let's now dive deeper into the first type of surface – planar surfaces.

Creating planar surfaces

The first type of surface in AutoCAD is a planar surface, which has no curvature. To start the planar surface command, you can type `PLANESURF`, or just use the short version and type `PLA`, and then press *Enter*. Alternatively, the **Planar** surface tool button is located in the **Create** panel on the **Home** tab in the **3D Basics** workspace. If you are using the **3D Modeling** workspace, you can find it in the **Create** panel on the **Surface** tab, as shown in the following screenshot:

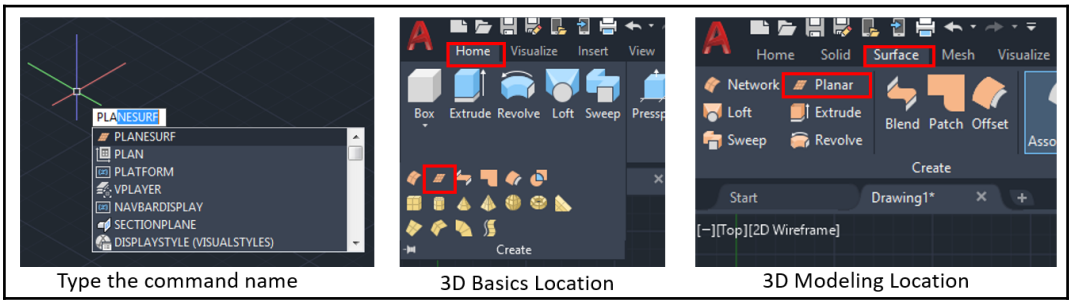


Figure 14. 1: Alternative methods for starting the planar surface command

The `planar surface` command can be used to quickly create planar surfaces using one of two available methods. The first method is limited to creating rectangular-shaped surfaces, where the surface is created by specifying two opposite corners for the rectangle. The second method allows you to create general planar surface shapes by using a closed 2D object. The following is an example of using the first (default) method:

1. Start the planar surface command.
2. Using your mouse, specify the first corner of the rectangular surface. Alternatively, you can type the coordinates using the keyboard and then press *Enter*.
3. Specify the opposite corner, again, either by using the mouse or the keyboard.
4. Once the second point is specified, the surface will be created and the command terminates.

The following diagram is an illustration of the previous steps:

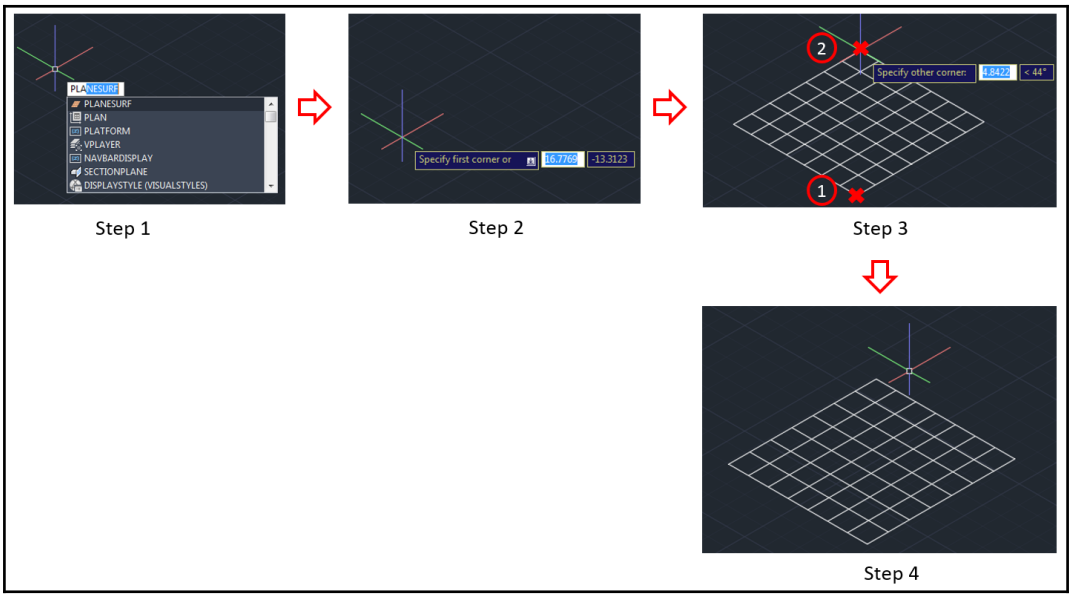


Figure 14. 2: Steps for creating a planar surface

If you select the rectangular planar surface created by the default method, you find four grab points on the corners. These points act just like base points, from which you can grab and move the planar object, as shown in the following diagram:

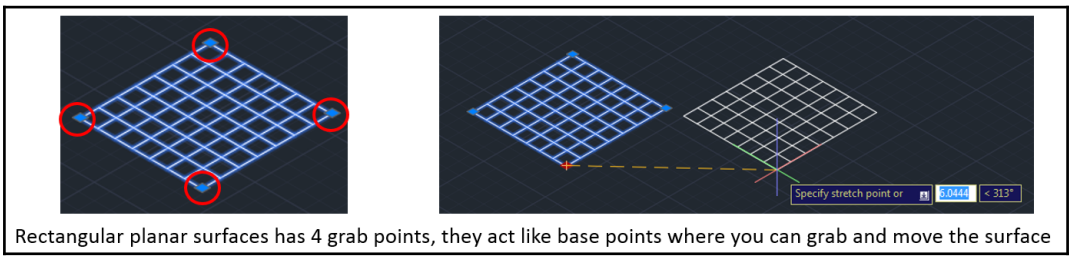


Figure 14. 3: Planar surface grab points

The second method available in the planar surface command can be accessed by choosing the **Object** option that appears in the command bar right after you initiate the command. To select it, click on the option name in the command bar, or you can just type `O` and then press *Enter*, as shown here:

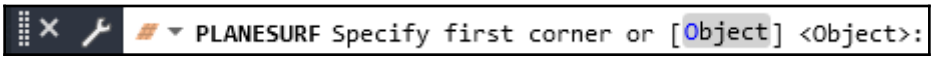


Figure 14. 4: Planar surface command options

This option allows you to create a planar surface, but you are not limited to rectangular shapes. You can select any closed 2D planar shape, such as a closed polyline, a circle, or a set of lines that create a closed shape. The following is an example of using the **Object** option:

1. After starting the `planar surface` command, type `O` and then press *Enter* to select the **Object** option.
2. Click on a closed shape, like the closed polyline shown in the example.
3. Press *Enter* to confirm the selection. The surface will be created and the command terminates.

The following diagram is an illustration of the previous steps:

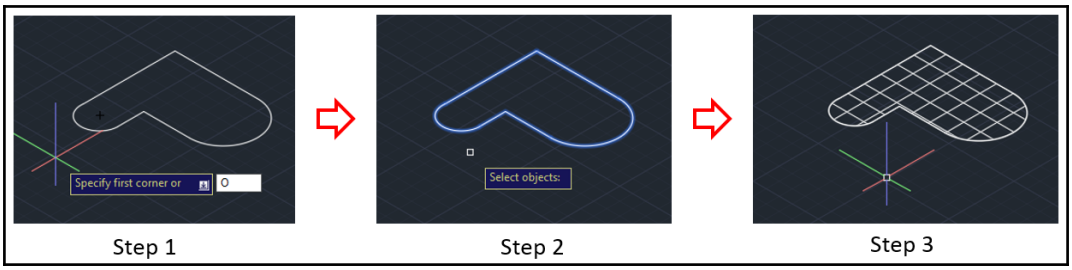


Figure 14. 5: Steps involved in using the Object option

When using the **Object** option, you will have access to the surface associativity option. This option will be activated by default, and it behaves in a similar way to the associative option found in the hatch command. If you have selected the objects that are used to create the surface (the polyline in the previous example), and the surface associativity option is activated, you can use the associated object grab points (the polyline in the previous example) to modify both the polyline and the surface, as shown in the following diagram:

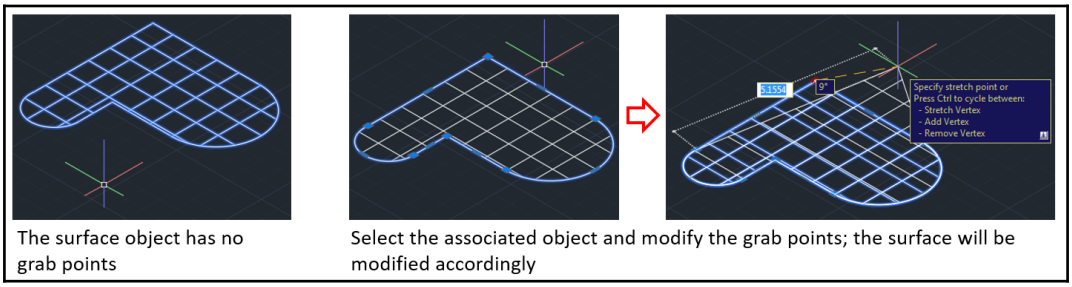


Figure 14. 6: Modifying the surface using the associated object

As mentioned earlier, the surface associativity option is turned on by default. To turn it off, you can click on the **Surface Associativity** tool button located in the **Create** panel on the **Surface** tab in the **3D Modeling** workspace. If you turn it off before creating the surface, the created surface will have grab points just like the default rectangular surface option, but they will only act as base points, as shown here:

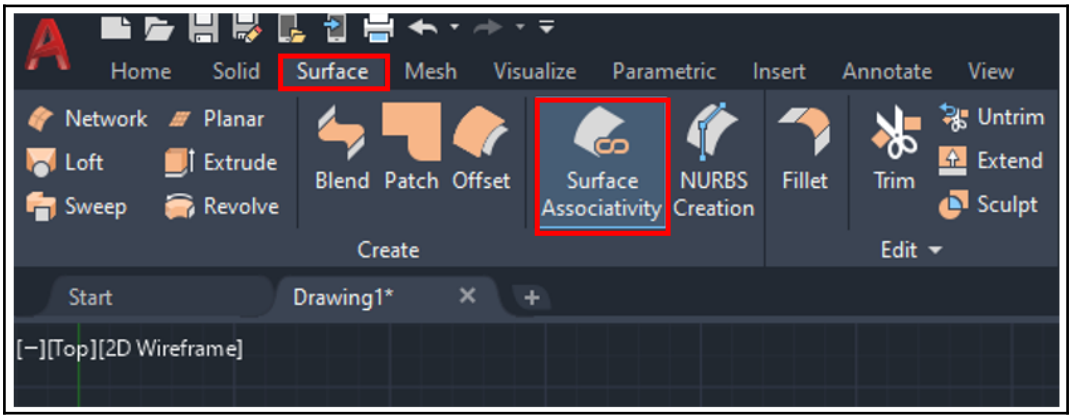


Figure 14. 7: Surface Associativity tool button location

By default, the objects used to create the surface will be deleted once the planar surface command terminates. You can control weather to delete these objects using the DELOBJ system variable. To access this, just type `DELOBJ` and then press *Enter*. The current value of the DELOBJ system variable will be displayed. Then, you can type the new value and press *Enter*. For more details on the DELOBJ system variable, please refer to the *extrude command* section in `Chapter 12`, *Conversion between 2D and 3D*.

Now that we have learned about the first surface type, which is the planar surface, let's have a look at the second type, which is `network surfaces`.

Creating network surfaces

To start the `network surface` command, you can type `SURFNETWORK` (or just use the short version and type `SUR`) and then press *Enter*. Alternatively, the **Network** surface tool button is located in the **Create** panel on the **Home** tab in the **3D Basics** workspace, and if you are using the **3D Modeling** workspace, you can find it in the **Create** panel on the **Surface** tab, as shown in the following screenshot:

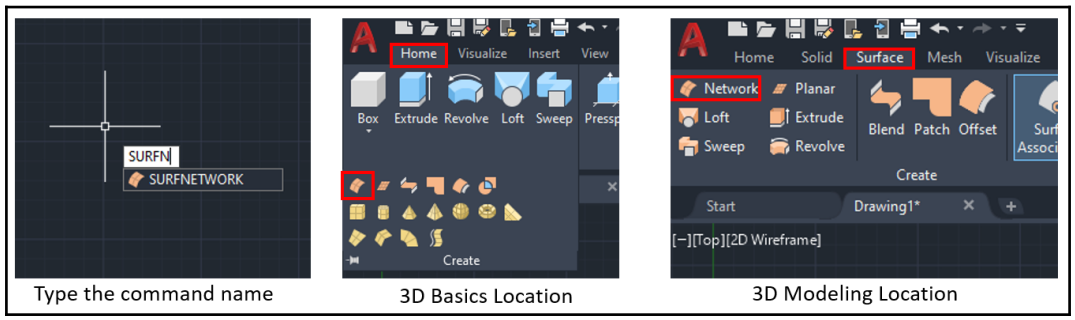


Figure 14. 8: Alternative methods for starting the network surface command

The `network surface` command is the second surface creation method available in AutoCAD. Unlike the `planar surface` command, you are now not limited to just planar surfaces; you can create general curved surfaces using a network of curves, as shown in the following example:

1. After starting the `network surface` command, select the curves of the first direction as shown in the following diagram, and then press *Enter* to end the selection stage.

2. Select the curves representing the second direction.
3. Press *Enter* to end the selection stage. The surface will be created and the command terminates.

The following diagram is an illustration of the previous steps:

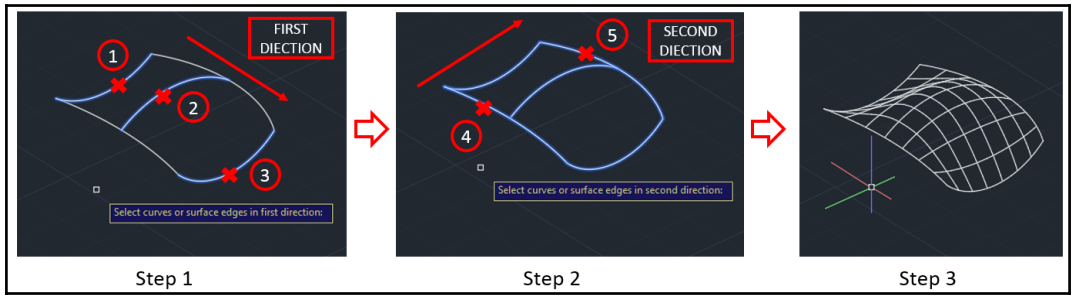


Figure 14. 9: Steps for creating a network surface

If you select the surface, you find four grab points that can be used to modify the surface shape, as shown in the following diagram:

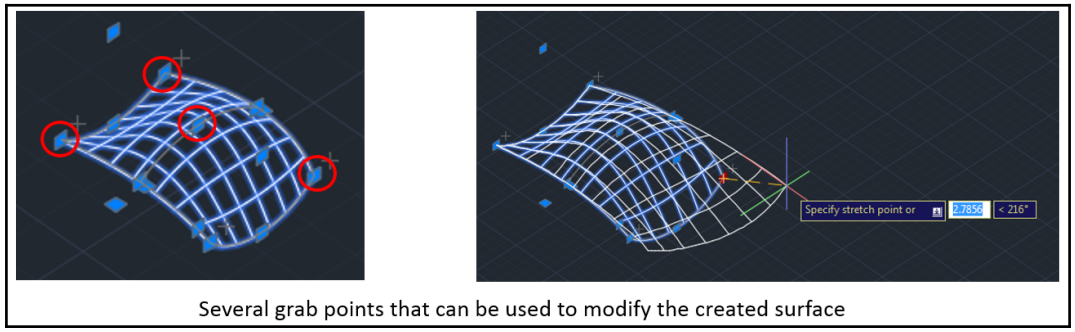


Figure 14. 10: Network surface grab points

We have now learned how to create two major types of surfaces in AutoCAD. Both these commands create a surface from scratch. Let's now look at a group of other surface creation commands that use existing objects in the drawing to create the surface from, starting with the blend surface command.

Blend surface command

A common scenario in 3D modeling using surfaces is to have two separate surfaces and you want to connect them using a smooth curved surface. The **Blend** surface command is just the tool for the job. To start the blend surface command, you can type `SURFBLEND` (or just use the short version and type `SURFB`) and then press *Enter*. Alternatively, the **Blend** surface tool button is located in the **Create** panel on the **Home** tab in the **3D Basics** workspace, as shown in the following diagram, and if you are using the **3D Modeling** workspace, you can find it in the **Create** panel on the **Surface** tab, as shown here:

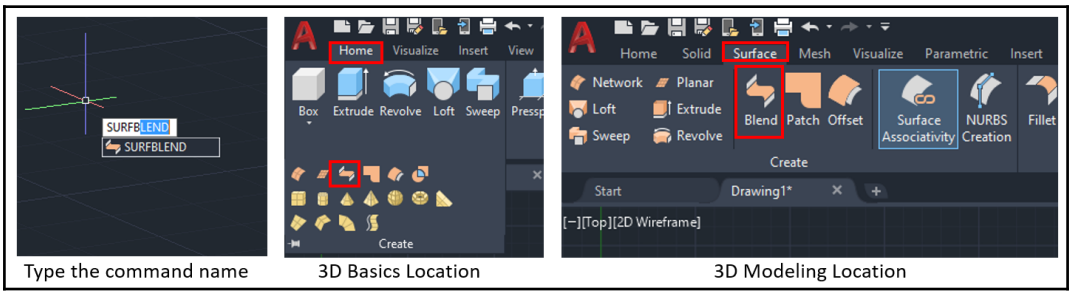


Figure 14. 11: Alternative methods for starting the blend surface command

The `blend surface` command can be used to create a surface connecting the edges of two existing surfaces. These edges can be open or closed and you can control the blending curve shape as shown in the following example:

1. After starting the `blend surface` command, select the edge of the first surface, and then press *Enter* to end the selection stage.
2. Select the edge of the second surface, and then press *Enter* to end the selection stage.
3. You will be prompted to change either the continuity, the bulge magnitude, or you can simply press *Enter* to create the surface and terminate the command.

The following diagram is an illustration of the previous steps:

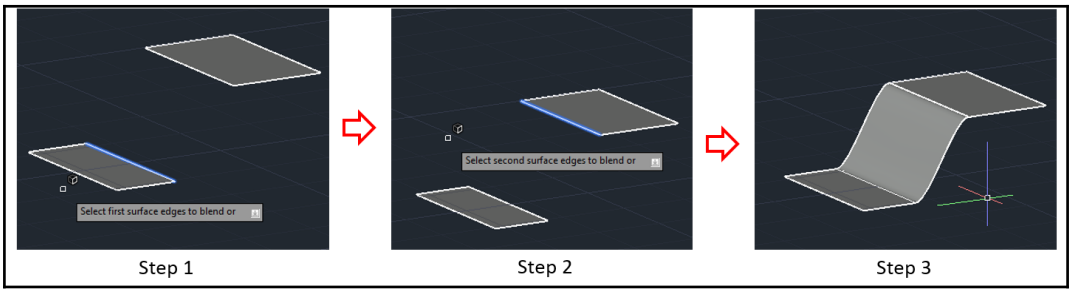


Figure 14. 12: Steps involved in using the blend surface command

The following diagram shows other examples of using the blend curve command:

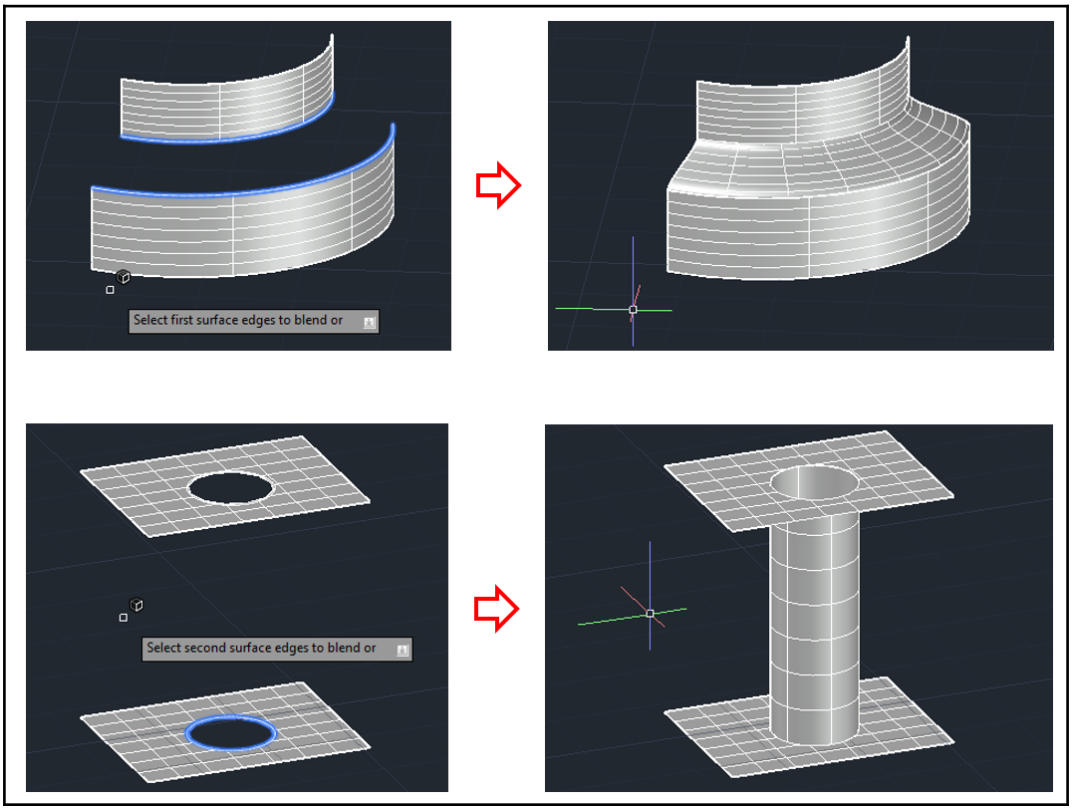


Figure 14. 13: Examples of using the blend surface command

If you select the surface, you will find two small arrows at the two ends of the surface. If you click on one of them, it will open the continuity options for this side of the blending surface. There are three options that control how the surfaces flow to one another, as shown in the following diagram. You can also access all these options during the creation of the blending curve by selecting the continuity option that appears just after the command terminates:

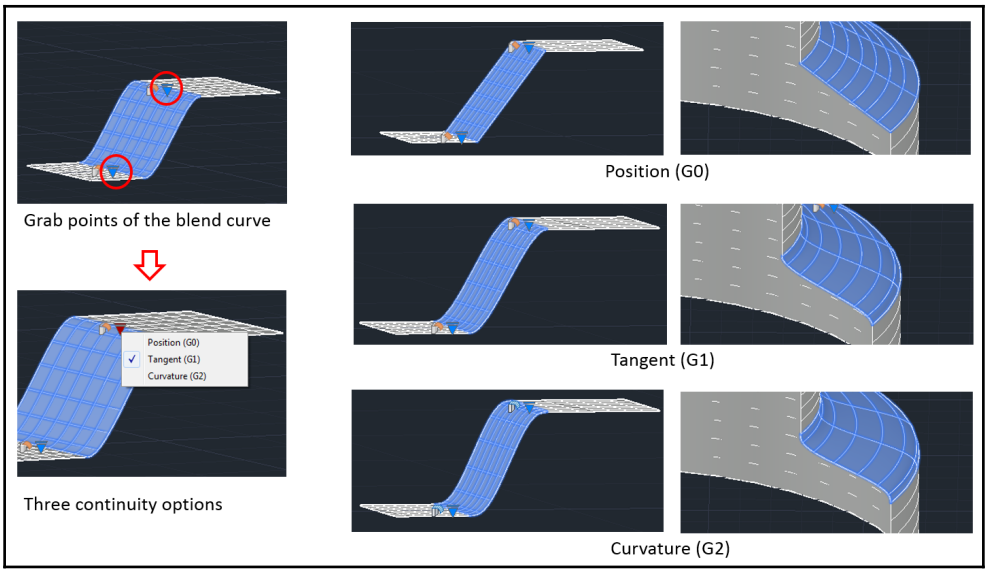


Figure 14. 14: Different blend settings

Another method for controlling the shape of the blending surface at the meeting point with the originating surfaces is to use the **Bulge** option. Once you select it, you enter a value between `0` and `1`, with `0` representing no curvature and `1` maximum curvature:

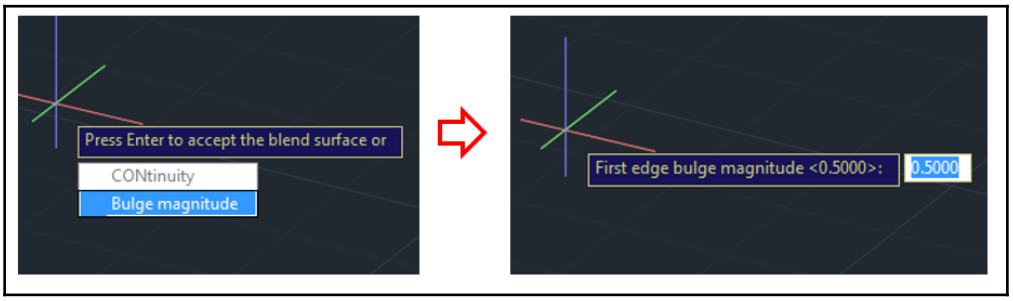


Figure 14. 15: Modifying the bulge magnitude

The `blend surface` command has an option called **Chain** that can be used to quickly select a loop of tangent-connected edges. This option will appear in the command bar right after you initiate the command. To activate it, click on the option name in the command bar or you can type `CH` and then press *Enter*. Now, if you select an edge, all the tangent-connected edges will be automatically selected, as shown in the following diagram:

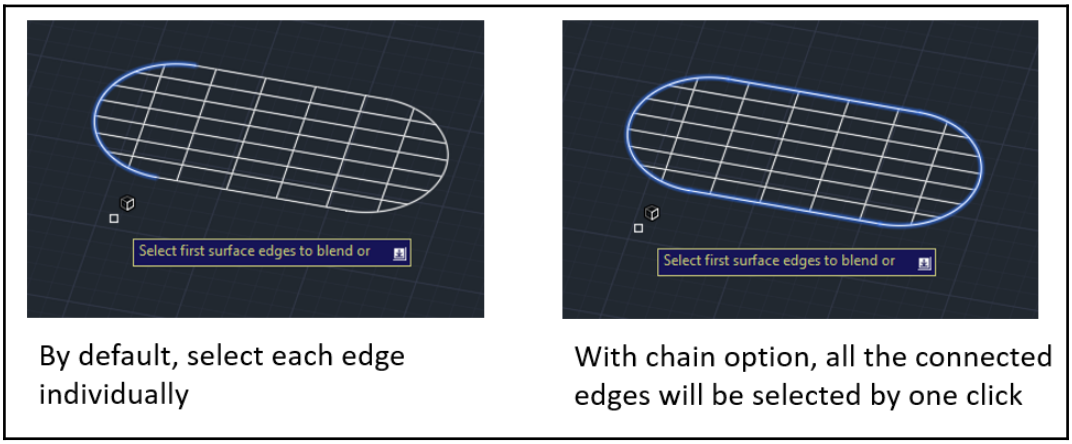


Figure 14. 16: Using the Chain option

We have now learned how to smoothly blend two separate surfaces. Let's now learn how to seal a hole in an existing surface using the `patch` command.

Surface patch command

Having holes in the 3D model is a common problem, especially when using surfaces. AutoCAD offers a solution to this problem by means of the `surface patch` command. To start the surface patch command, you can type `SURFPATCH` (or just use the short version and type `SURFP`) and then press *Enter.* Alternatively, the **Patch** surface tool button is located in the **Create** panel on the **Home** tab in the **3D Basics** workspace, and if you are using the **3D Modeling** workspace, you can find it in the **Create** panel on the **Surface** tab, as shown in the following screenshot:

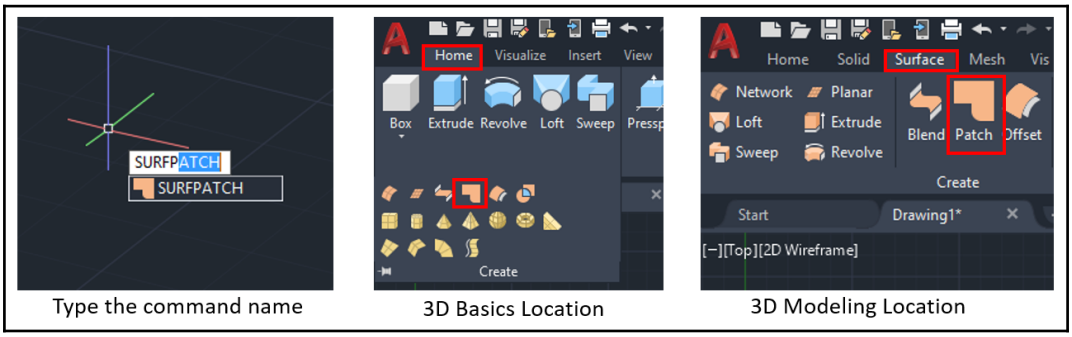


Figure 14. 17: Alternative methods for starting the surface patch command

The `surface patch` command can be used to close a gap or a hole in an existing surface, as shown in the following example:

1. After starting the `surface patch` command, select the edge of the opening or the gap, and then press *Enter* to end the selection stage.
2. You will be prompted to change the settings for **continuity**, **Bulge magnitude**, the **Guides** option, or you can simply press *Enter* to skip.
3. Once you press *Enter*, the patch surface will be created and the command terminates.

The following diagram is an illustration of the previous steps:

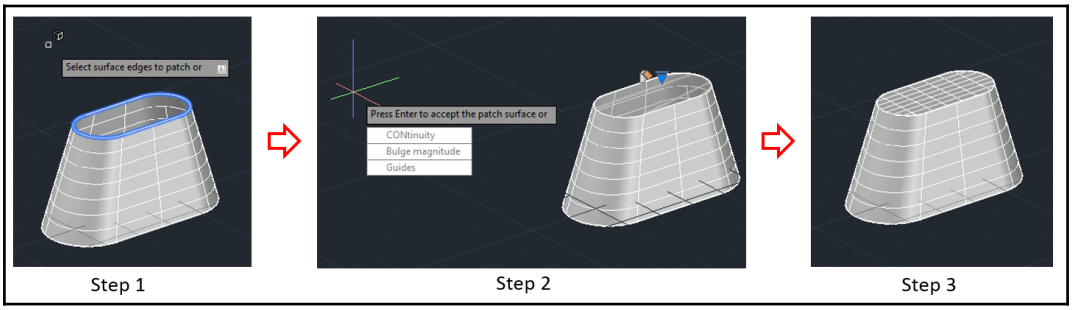


Figure 14. 18: Steps involved in creating a surface patch

The following diagram shows another example of using the `surface patch` command:

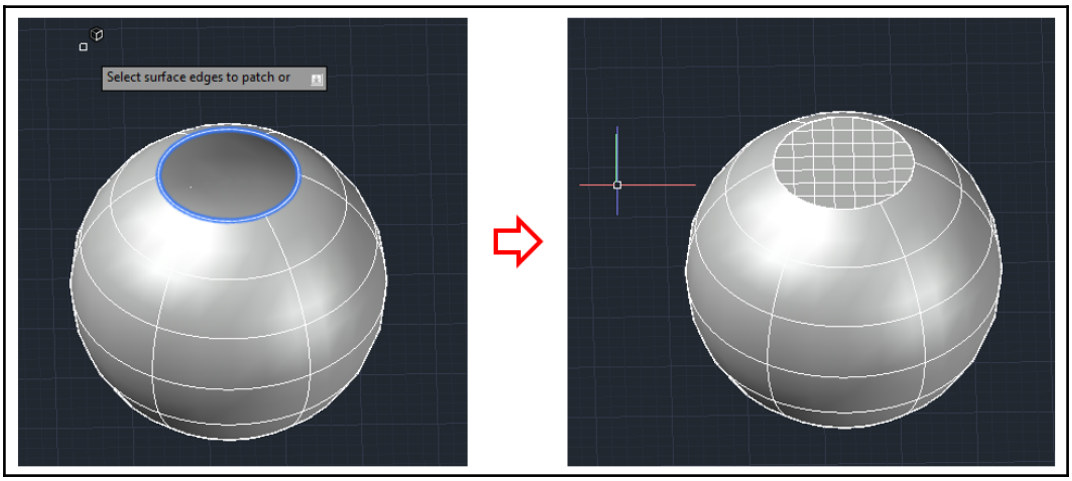


Figure 14. 19: Another example of using the surface patch command

The `surface patch` command has a number of different methods/options to control the shape of the patch surface. The first option is called continuity. You can change the continuity setting before the patch command terminates as shown in the previous example. Alternatively, if you select the patch surface, you will find a small arrow. Click on it to open the continuity options for the patch surface. There are three options that control how the surfaces flow to one another, as shown in the following diagram:

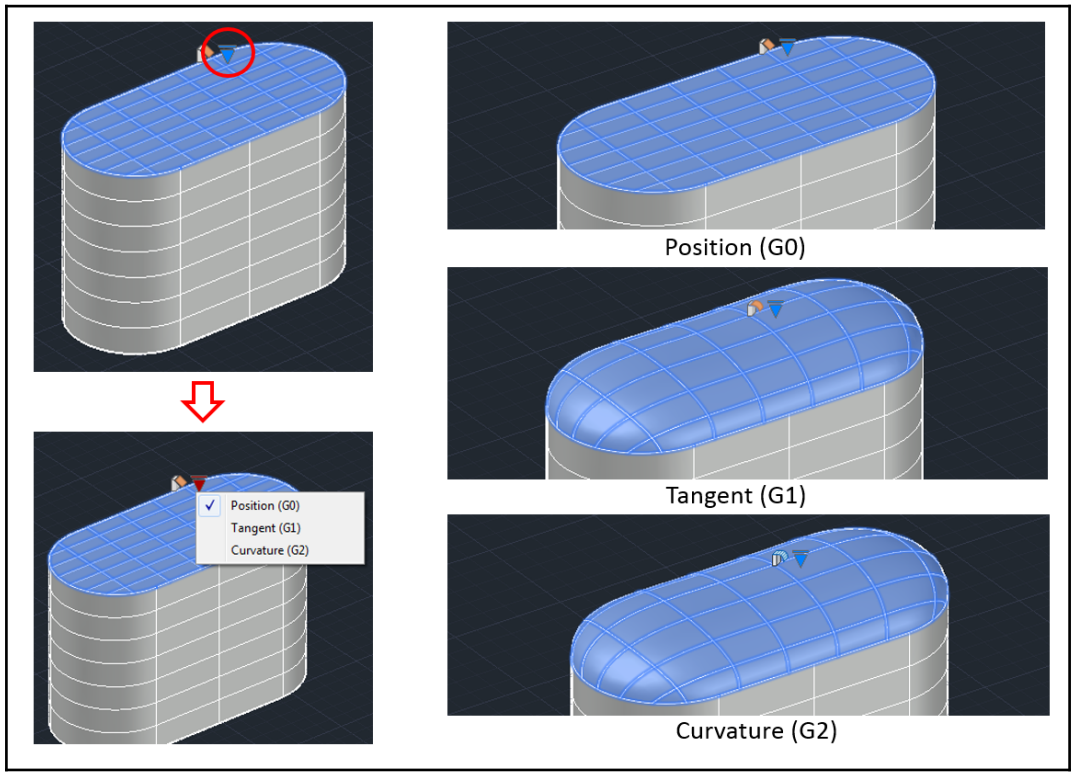


Figure 14. 20: Options for the surface patch shape

Another method for controlling the shape of the patch surface at the meeting point with the originating surfaces involves the use of the **Bulge** option. Once this option is selected, you enter a value between `0` and `1`, with `0` representing no curvature and `1` maximum curvature, as shown here:

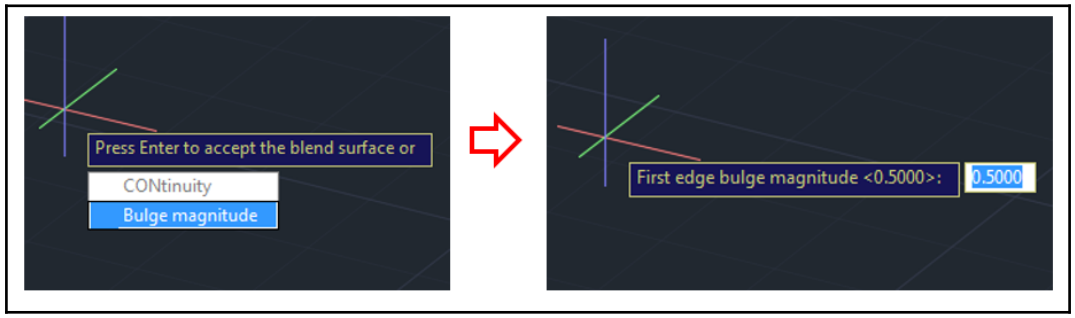


Figure 14. 21: Adjusting the bulge magnitude

The last option to control the shape of the patch surface is called **Guides**. This appears in the list just before the command terminates. It allows you to use an arc, for example, as a guide to the shape of the patch surface, as shown in the following example:

1. Choose the **guide** option before the command ends.
2. Click on the curve(s) to be used as guides and then press *Enter* to end the selection stage.
3. Press *Enter* again to skip the options menu. The command terminates when the surface is created:

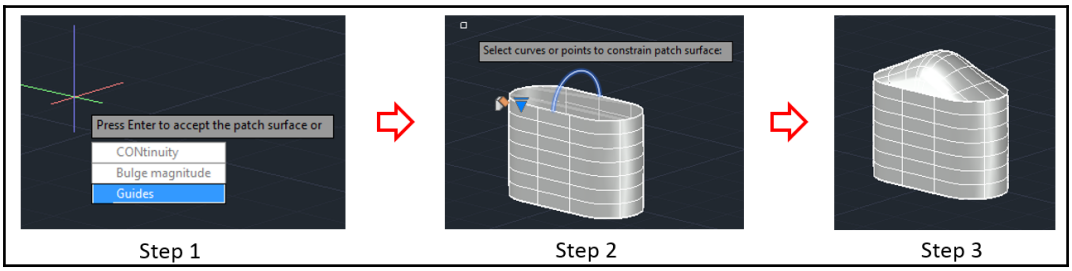


Figure 14. 22: Steps involved in using the Guides option

We have learned how to blend two separate surfaces and how to seal holes in existing ones. Let's now take a look at another type of surface creation command, which is the `surface offset` command.

Surface offset command

The `surface offset` command is basically the surface counterpart of the standard offset command used in 2D drafting. To start the `surface offset` command, you can type `SURFOFFSET` (or just use the short version and type `SURFO`) and then press *Enter*. Alternatively, the **Offset** surface tool button is located in the **Create** panel on the **Home** tab in the **3D Basics** workspace, and if you are using the **3D Modeling** workspace, you can find it in the **Create** panel on the **Surface** tab, as shown in the following screenshot:

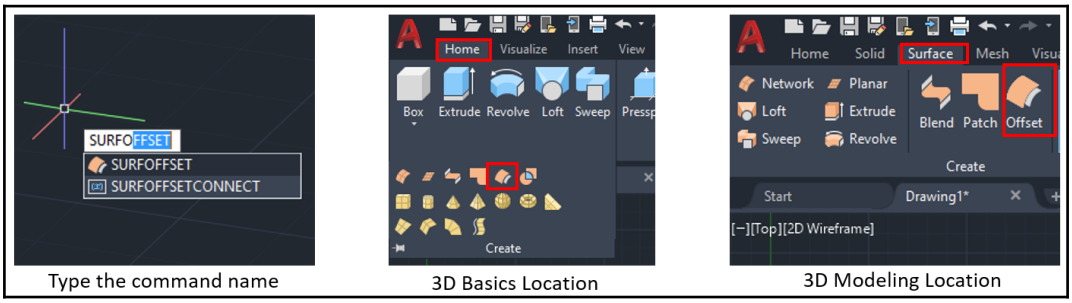


Figure 14. 23: Alternative methods for starting the surface offset command

Just like the `2D offset` command, the `surface offset` command can be used to create an offset surface in a specific direction and with the desired offset distance, as shown in the following example:

1. After starting the `surface offset` command, select the surface(s) you want to offset and then press *Enter* to end the selection stage.
2. Notice that the offset direction will be shown with small arrows on the surface, and you will be prompted to enter the offset distance. Type the offset distance value using the keyboard and then press Enter.
3. Once you press *Enter*, the offset surface will be created and the command terminates.

The following diagram is an illustration of the previous steps:

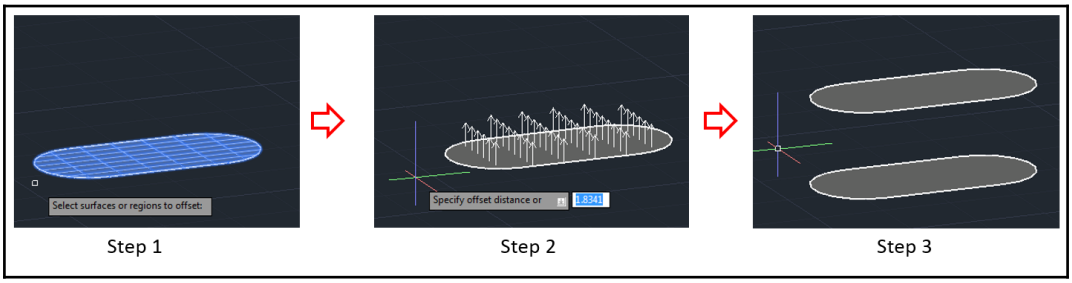


Figure 14. 24: Steps involved in using the surface offset command

In the previous example, a planar surface was used with the `offset` command. The following diagram shows examples of other surfaces that can be used with the `offset` command:

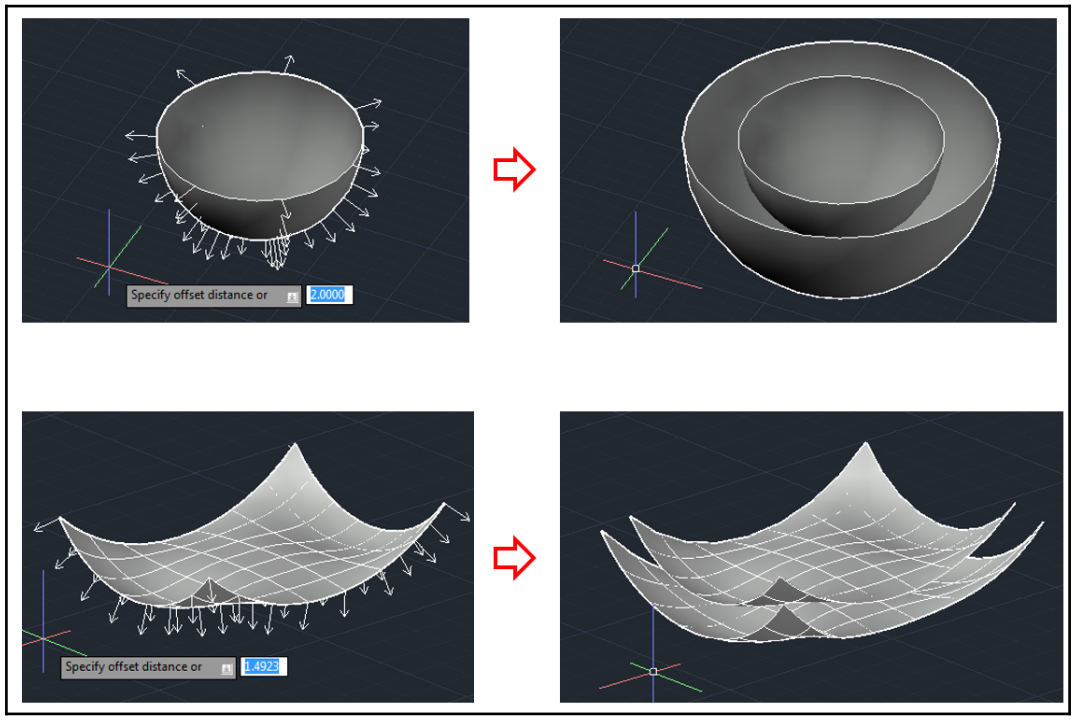


Figure 14. 25: Examples of using the surface offset command

The `surface offset` command has a number of different methods/options that appear in the command bar right after you select the surface(s) to be offset. There are four options, as shown in the following diagram:

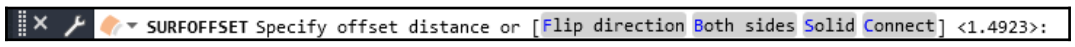


Figure 14. 26: Additional options in the surface offset command

The first option is called **Flip direction**. To select this option, click on the option name in the command bar, or you can type `F` and then press *Enter*. This option allows you to change the offset direction. Just activate the option and you will notice that the direction arrows will flip, as shown in the following diagram:

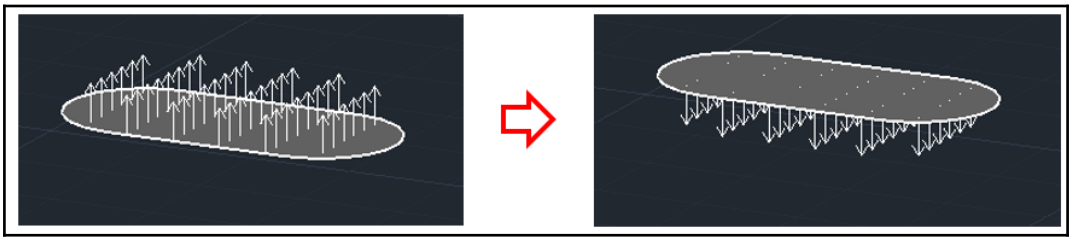


Figure 14. 27: Flipping the offset direction

The second option is called **Both sides**. To activate it, click on the option name in the command bar, or you can just type `B` and then press *Enter*. This option will offset the surface in both sides at the same time.

The third option is called **Solid**. To select it, again click on the option name in the command bar, or you can type `S` and then press *Enter*. This option will create a solid between the original and the offset surface, as shown in the following diagram. This option is very helpful with general curved surfaces, like the one shown in the following example, since the extrude and sweep commands will not produce a solid with curved surfaces:

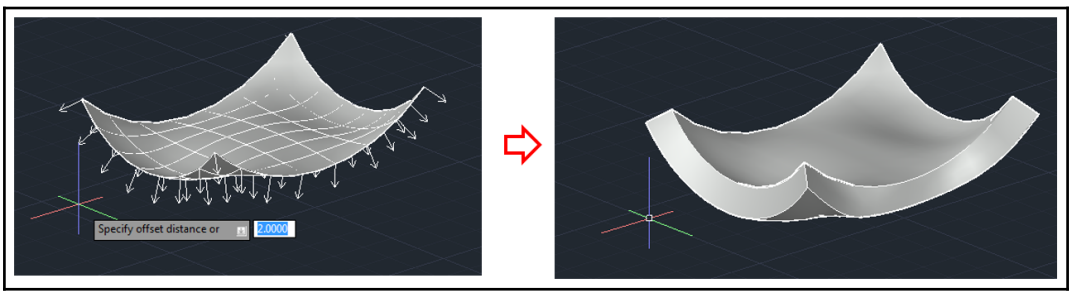


Figure 14. 28:Using the Solid option

The last option is called **Connect**. This option can be activated by clicking on the option name in the command bar or by just typing `C` and then pressing *Enter*. The connect option has two settings – "Yes" or "No", and it can be used in cases where you have multiple connected surfaces. You can select whether the offset of each surface will be independent or connected, as shown in the following example:

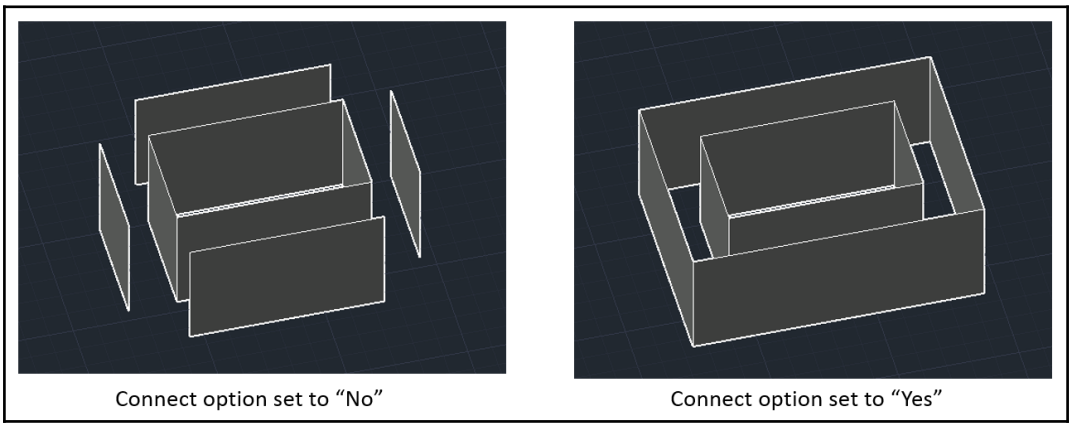


Figure 14. 29:Difference between using and not using the Connect option

Now that we have learned about different methods to create surfaces, let's have a look at the commands dedicated to surface editing in AutoCAD.

Modifying surfaces

As mentioned in `Chapter 10`, *Introduction to 3D Modeling*, although the creation commands of 3D objects have a huge number of possibilities, they can't create every imaginable shape in existence, and the actual 3D modeling process usually starts with the creation of simple shapes. Then, these shapes are edited and combined together to produce the desired 3D model, and surface modeling is no exception to that.

In this section, we will learn about the set of surface editing tools offered by AutoCAD. Let's now look closely at the first command in this set, which is the surface fillet command.

Surface fillet

The surface fillet command is very similar to the standard fillet command used in 2D drafting. If you have two surfaces meeting at a sharp edge, you can replace them with a rounded fillet with a given radius using the surface fillet command, as we will see in this section. To start the surface fillet command, you can type `SURFOFFILLET` (or just use the short version and type `SURFF`) and then press *Enter*. Alternatively, the **Fillet** surface tool button is located in the **Edit** panel on the **Home** tab in the **3D Basics** workspace, as shown in the following screenshot:

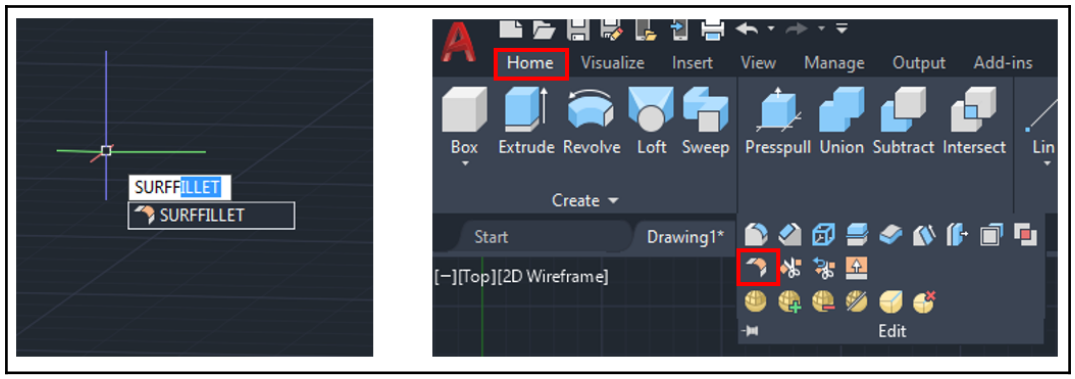


Figure 14. 30: Alternative methods for starting the surface fillet command

If you are using the **3D Modeling** workspace, you can find it in the **Edit** panel on the **Surface** tab.

Just like the `2D fillet` command, the `surface fillet` command can be used to create a round fillet with a specific radius between two surfaces, as shown in the following example:

1. After starting the `surface fillet` command, select the first surface.
2. Click on the second surface.
3. You will be prompted to set the fillet radius or change the trim option setting, or you can just skip and press *Enter*. The fillet will be created and the command terminates.

The following diagram is an illustration of the previous steps:

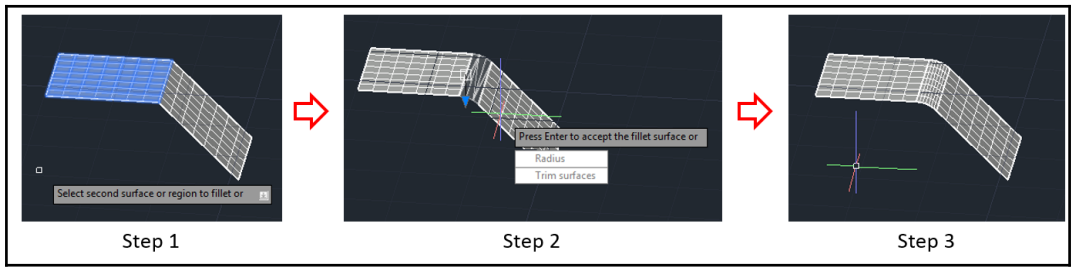


Figure 14. 31: Steps involved in using the surface fillet command

If you select the fillet surface, you will find a small arrow. You can click and drag this arrow to modify the fillet radius or, alternatively, you can enter a new value for the radius using the keyboard as shown here:

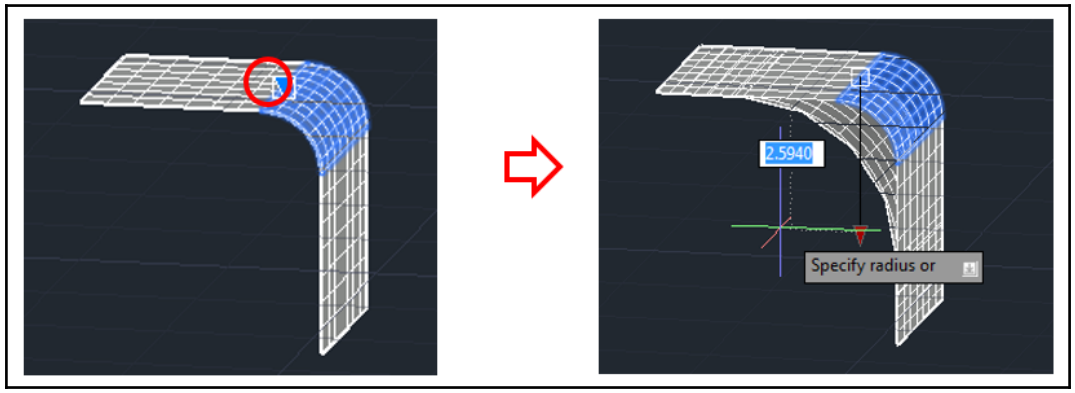


Figure 14. 32: Modifying the fillet radius using the grab point

The `surface fillet` command has a number of different methods/options that appears as a list just before the command terminates. The list contains two options:

- The first option sets the radius of the fillet. To do so, you click on the **Radius** option and then enter the value of the fillet radius and press *Enter*.
- The second option is called **Trim**. This option has two settings – **Trim**, and **No Trim**. If you select **No Trim**, the round fillet will be created, but the surfaces will not be trimmed to the edges of the fillet, as shown in the following diagram:

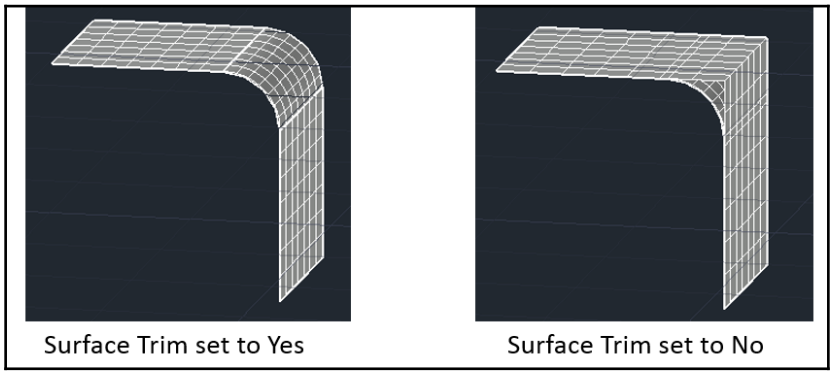


Figure 14. 33: Surface Trim option

Let's now have a look at the next command in the surface editing set, which is the surface trim command.

Surface trim

The `surface trim` command is the 3D counterpart of the standard trim command used in 2D drafting, but instead of trimming lines and curves, you will trim surfaces. To start the `surface trim` command, you can type `SURFTRIM` (or just use the short version and type `SURFT`) and then press *Enter*. Alternatively, the **Trim** surface tool button is located in the **Edit** panel on the **Home** tab in the **3D Basics** workspace:

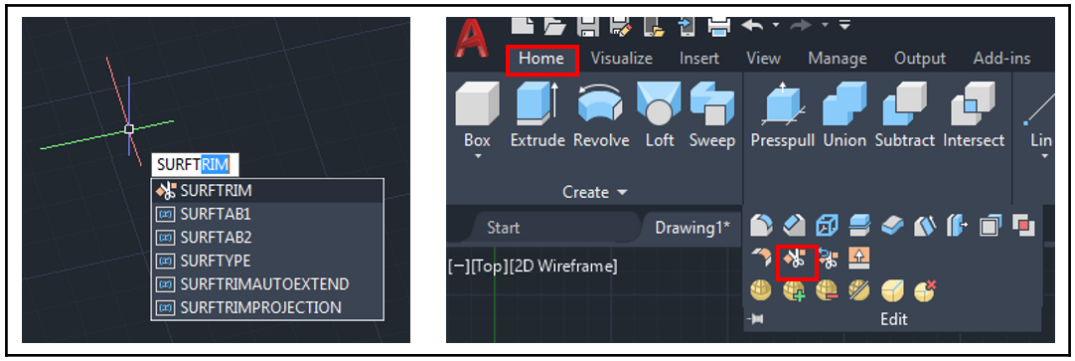


Figure 14. 34: Alternative methods for starting the surface trim command

If you are using the **3D Modeling** workspace, you can find it in the **Edit** panel on the **Surface** tab.

The `surface trim` command can be used to trim surfaces in a similar way to the 2D trim command as shown in the following example:

1. After starting the `surface trim` command, select the surface(s) you want to trim and then press *Enter* to end the selection stage.
2. Select the surface/edges that will be used for cutting and then press *Enter* to end the selection stage.
3. Just like the 2D trim command, click on the side you want to get trimmed and then press *Enter* to terminate the command.

The following diagram is an illustration of the previous steps:

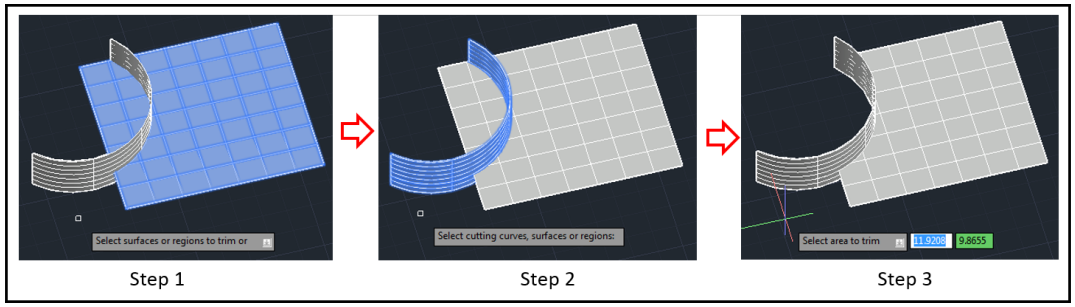


Figure 14. 35: Steps involved in using the surface trim command

In the previous example, a planar surface was used with the `trim` command, and a surface was used for trimming. The `surface trim` command will work fine with curved surfaces and with curves as cutting edges, as shown in the following examples:

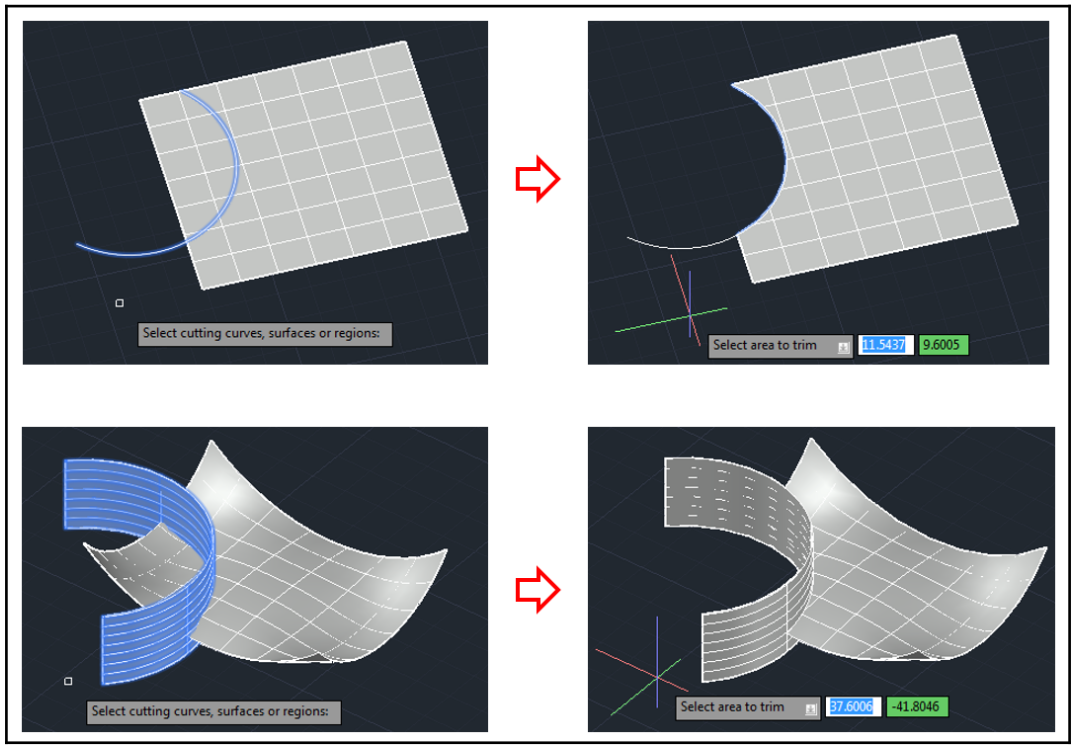


Figure 14. 36: Using a curve to trim a surface, and trimming a curved surface

The trimming edge or surface is not required to be intersecting with the trimmed surface. The `surface trim` command can project the non-intersecting cutting edge/surface on the trimmed surface, and you can even control the manner of projection using the options that appear on the command bar when using the `trim` command:

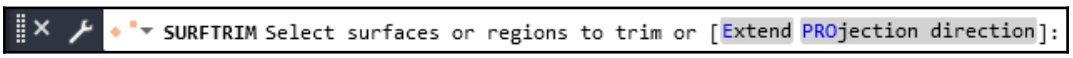


Figure 14. 37: Additional options in the surface trim command

The first option is called **Extend**. This can be selected by clicking on the option name in the command bar, or by just typing `E` and then pressing *Enter*. The **Extend** option has two settings, *Yes* and *No*, and it controls whether to activate the automatic projection of the cutting edge/surface on the trimmed surface.

The second option is called **PROjection direction**. This can be selected by clicking on the option name in the command bar, or you can just type `PRO` and then press *Enter*. This option allows you to choose between different options for the cutting edge projection method on the trimmed surface.

The following diagram is an example of using a non-intersecting cutting surface:

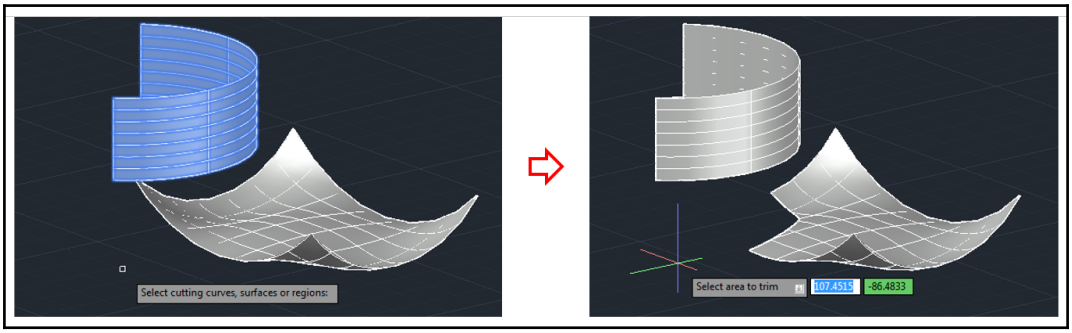


Figure 14. 38: Using the Projection option

We have learned how to trim surfaces using the trim command. However, AutoCAD allows you to undo the trimming operations executed by the surface trim command using a very specific command for this task, which is called the `surface untrim` command.

Surface untrim command

`Surface untrim` is an undo command specific to the operations performed using the surface trim command discussed in the previous section. To initiate the `surface untrim` command, you can type `SURFUNTRIM` (or just use the short version and type `SURFUN`) and then press *Enter*. Alternatively, the **Untrim** surface tool button is located in the **Create** panel on the **Home** tab in the **3D Basics** workspace:

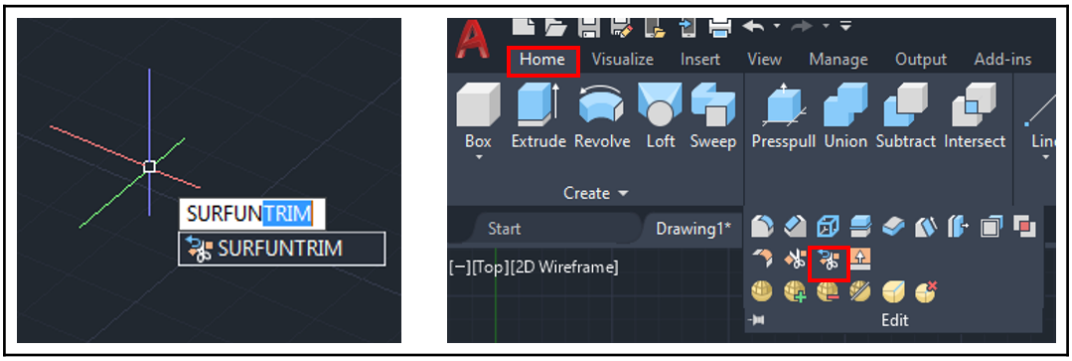


Figure 14. 39: Alternative methods for starting the surface untrim command

If you are using the **3D Modeling** workspace, you can find it in the **Edit** panel on the **Surface** tab.

As the name suggests, if you have a surface that was cut using the surface trim command, you can undo the trimming using the surface untrim command, as shown in the following example:

1. Start the surface untrim command.
2. Select the edges of the trimmed part, as shown in the diagram.
3. Press *Enter* to end the selection stage. The trimmed part will be restored and the command terminates.

The following diagram is an illustration of the previous steps:

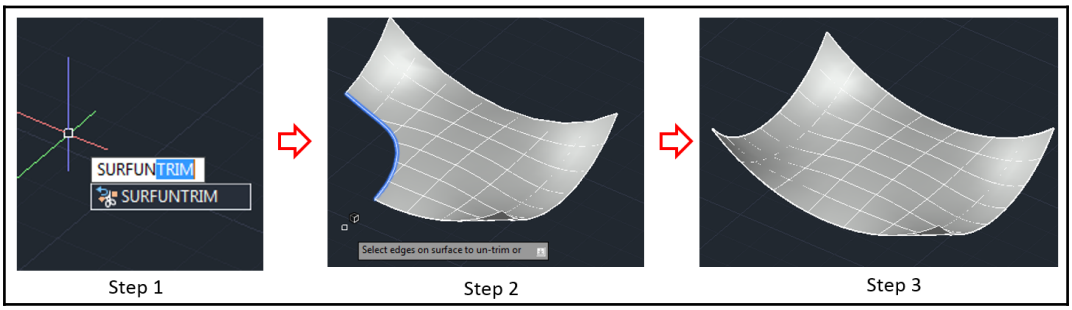


Figure 14. 40: Steps involved in using the surface untrim command

The `surface untrim` command has one option that allows you to select the surface instead of the edge of the trimmed part, as shown in the previous example.

We have now learned how to create a surface fillet, and how to trim and untrim a surface. Now, let's have a look at another surface editing tool called the sculpt command.

Sculpt command

A difficult aspect of creating a 3D model using curved 3D surfaces is making sure that their edges are perfectly matched, and that the model doesn't have any holes. The `sculpt` command allows you to just focus on creating the surfaces without caring much about the edges, and then the `sculpt` command will trim the extra parts, thereby creating perfectly matched edges between the surfaces of the model, as we will see in this section. To start the `sculpt` command, you can type `SURFSCULPT` (or just use the short version and type `SURFS`) and then press *Enter*. Alternatively, the **Sculpt** surface tool button is located in the **Create** panel on the **Home** tab in the **3D Basics** workspace, as shown in the following screenshot:

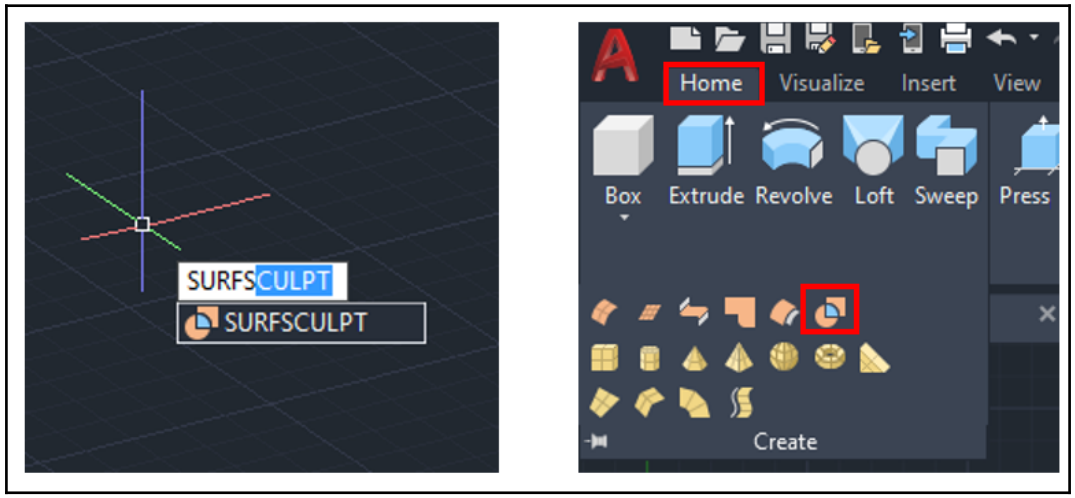


Figure 14. 41: Alternative methods for starting the sculpt command

If you are using the **3D Modeling** workspace, you can find it in the **Edit** panel on the **Surface** tab.

If you have a collection of intersecting surfaces that form an enclosure, the sculpt command can be used to turn that enclosure into a 3D solid object. The surfaces are not required to have clean meeting edges. The `sculpt` command will trim the extending parts of the intersecting surfaces, as shown in the following example:

1. Start the `sculpt` command.
2. Select the surfaces.
3. Press *Enter* to end the selection stage. The solid will be created and the command terminates.

The following diagram is an illustration of the previous steps:

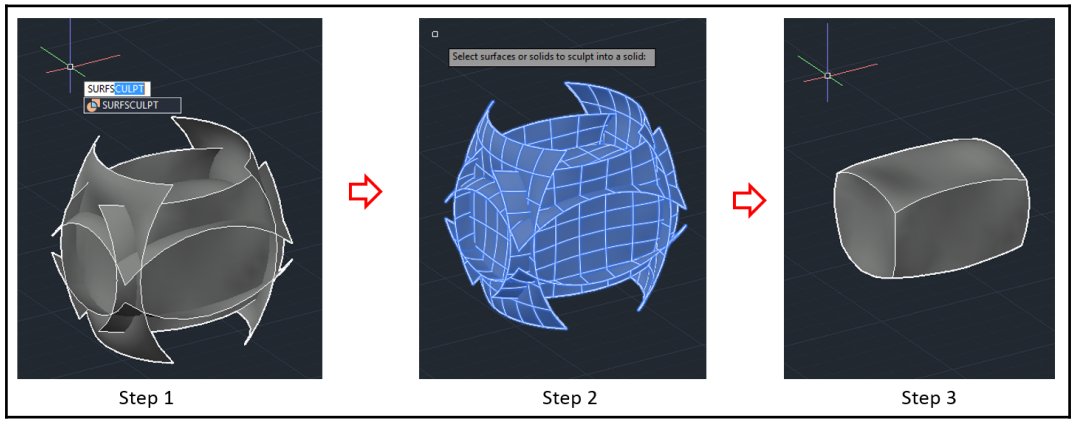


Figure 14. 42: Steps involved in using the sculpt command

There may be some confusion between the `sculpt` and `surface trim` commands since visually, both may produce the same results. The `sculpt` command will trim the extended surfaces as shown in the previous example and will create a solid body using the remaining surfaces, while the trim surface command will only trim them but they will remain surfaces. We have learned about the creation and modification commands used in surface modeling, so let's now take a look at the other major type of 3D objects in AutoCAD, which are mesh objects.

Mesh objects

In addition to 3D solids and surfaces, AutoCAD offers another type of 3D object, called mesh objects. Mesh objects are very similar to solids and surfaces. You can create surface meshes having the same shapes as surfaces, and you can create primitive mesh forms (boxes, pyramids, spheres, and so on) just like solid primitives, although mesh objects don't have the mass property present in the solid objects, but they can be modified in many other ways that cannot be done with regular 3D solid objects. The mesh object is divided into smaller volumes or cells, while the vertices, edges, and faces of these smaller volumes can be individually modified in many ways, as will be shown in this section of the chapter.

Creating mesh objects

To create a mesh object, you have three different options that will be discussed in detail later in this chapter:

1. Create a primitive mesh object from the available list of primitive mesh creation commands.
2. Create a regular 3D solid or surface and then convert it using the `smooth object` command.
3. Use curves to create mesh objects using operations such as `revolve` and `sweep`.

Creating primitive mesh objects

Just like 3D solids, AutoCAD can create the same list of primitive shapes, but this time as mesh objects. The full list of mesh primitive buttons can be found in the **Create** panel on the **Home** tab in the **3D Basics** workspace. If you are using the **3D Modeling** workspace, you can find them on the **Mesh** tab, as shown in the following screenshot:

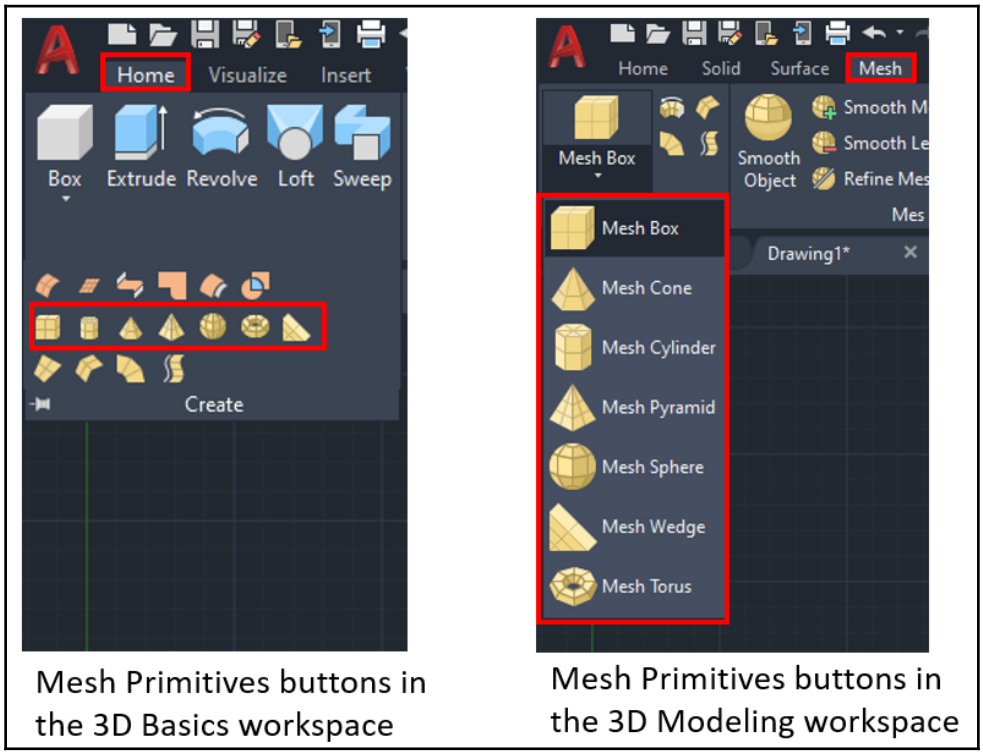


Figure 14. 43: Locations of the tool buttons for creating primitive mesh shapes

The mesh primitives will work exactly like the `solid primitive` commands. For more details about each command, you can refer to `Chapter 11`, *Creating Primitive 3D Shapes*. The following diagram shows a preview of the primitive mesh box and the primitive mesh cone:

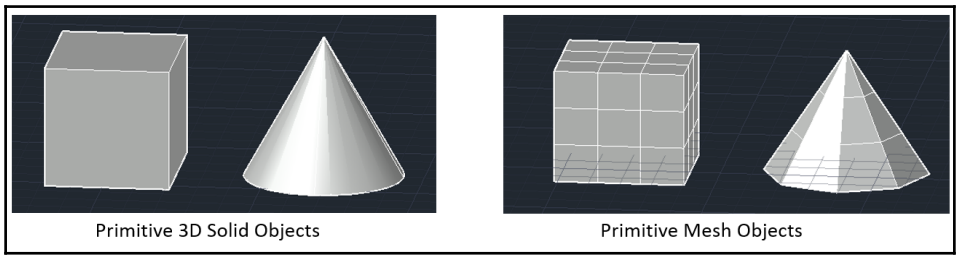


Figure 14. 44: Primitive solids versus primitive mesh shapes

The number of mesh divisions or cells in the mesh object can be controlled using the mesh primitive options. You can access it by clicking on the small arrow in the bottom-right corner of the **Primitives** panel in the **3D Modeling** workspace. This will open the **Mesh Primitive Options** window, as shown in the following diagram. On the left, you have the **Tessellation Divisions** option mentioned earlier to control the number of divisions in the mesh object.

You select a shape from the list of all available primitives, and then type the desired number of divisions for each side. On the right-hand side, you have a small window showing a preview of the mesh, as shown here:

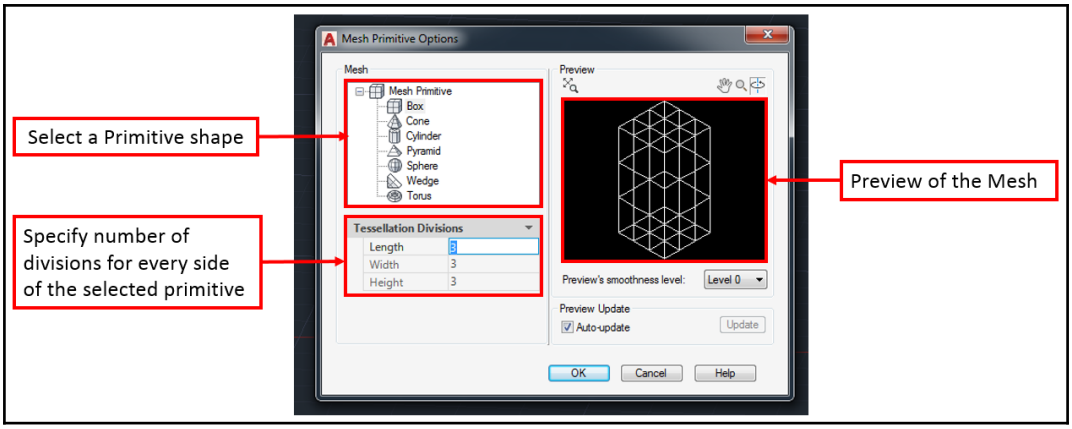


Figure 14. 45: Mesh Primitive Options window

Let's now have a look at the second way of creating a mesh object, which is by using the smooth command.

Using the smooth object command

The second method to create a mesh object is to start by creating a regular 3D solid object, and then convert it to a mesh using the smooth object command. The `smooth object` command can be started by typing the command name `MESHSMOOTH`, or by using the short version and just typing `MES`, and then pressing *Enter*. An alternative way is to click on the **Smooth Object** button located in the **Edit** panel on the **Home** tab in the **3D Basics** workspace, as shown in the following screenshot:

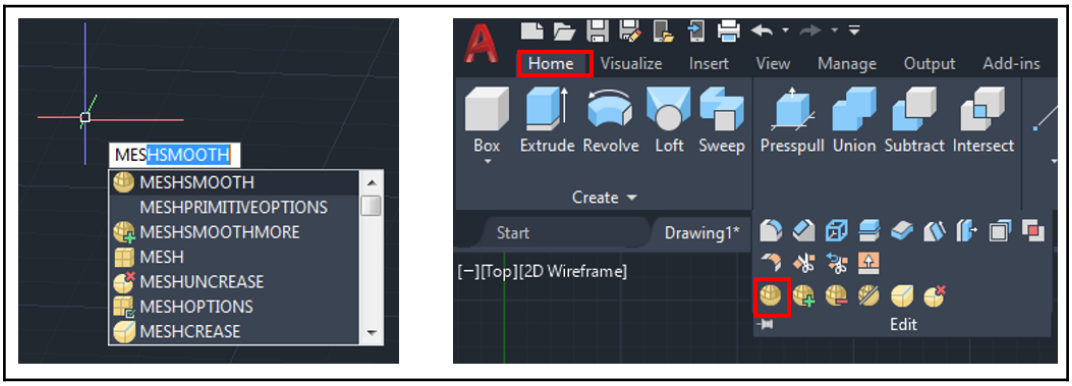


Figure 14. 46: Alternative methods for starting to use MESHSMOOTH

If you are using the **3D Modeling** workspace, you can find it in the **Mesh** panel on the **Mesh** tab.

The `smooth object` command can be used to convert a 3D solid object to a mesh object. It will also apply a controllable smooth operation to the body, as shown in the following steps:

1. Start the `smooth object` command.
2. Click on the desired object.
3. Press *Enter* to end the selection stage. The object will be converted and the command terminates.

The following diagram is an illustration of the previous steps:

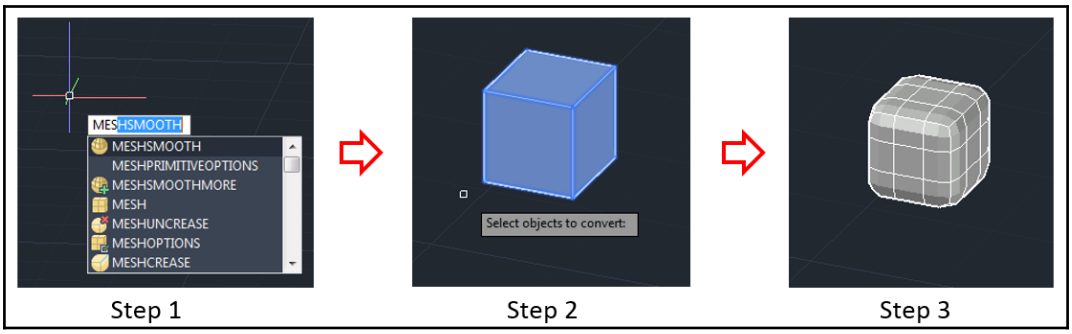


Figure 14. 47: Steps involved in using MESHSMOOTH

As shown in the previous example, in addition to converting the box into a mesh object, the box edges were smoothed out. You can control whether you want this smooth operation to be applied, and you can also control the degree of smoothness using the Mesh Tessellation options.

To start the mesh tessellation options, you can type the command name, `MESHOPTIONS`, and then press *Enter*, or you can click on the small arrow in the bottom-right corner of the mesh panel in the **3D Modeling** workspace. This will open the **Mesh Tessellation** option window, where you can change the smoothness level of the created object, as shown in the following screenshot:

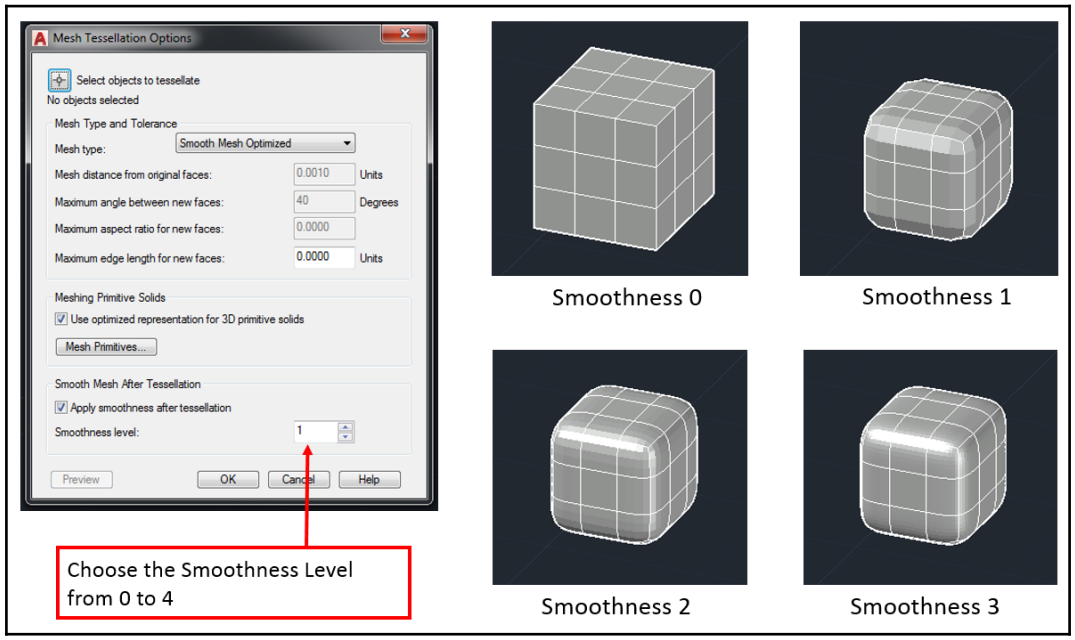


Figure 14. 48: Using mesh tessellation

You can also change the smoothness level of an existing mesh object by using the **Smooth More** and **Smooth Less** commands. You just select the object and then click on the **Smooth More**/**Smooth Less** buttons. You can find them in the **Edit** panel on the **Home** tab in the **3D Basics** workspace, or in the **Mesh** panel on the **Mesh** tab in the **3D Modeling** workspace, as shown in the following screenshot:

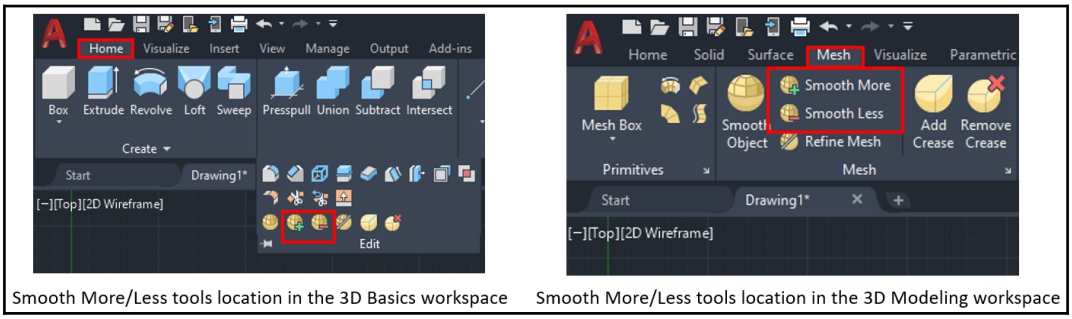


Figure 14. 49: Locations for the Smooth More/Smooth Less tool buttons

We have learned how to create a primitive mesh object and how to convert a 3D solid to a mesh using the smooth command. Let's now look at the commands associated with the third mesh creation method.

Using curves to create mesh objects

Mesh objects are not only limited to solids. It is also possible to create surface mesh objects. For that purpose, there are four mesh creation commands that perform operations on normal 2D curves to create the surface mesh. These commands will be discussed in detail in the following sub-sections.

The REVSURF command

The first command in the surface mesh creation set, this is very similar to the revolve command discussed in `Chapter 9`, *External References and Dynamic Blocks*. A curve is revolved with a given angle and around a specified axis to create the surface mesh, as we will see in this sub-section. To start the `REVSURF` command, you can type the command name and then press *Enter*, or use the short version and just type `REVS` and then press *Enter*. An alternative way is to click on the **REVSURF** tool button located in the **Primitives** panel on the **Mesh** tab in the **3D Modeling** workspace, as shown in the following screenshot:

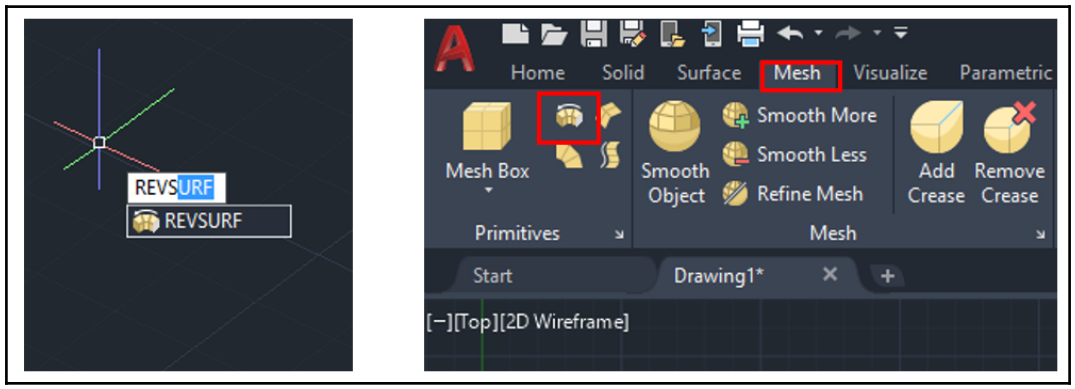


Figure 14. 50: Alternative methods for starting the REVSURF command

The `REVSURF` command is very similar to the revolve command. You specify the curve to be revolved, and then you specify the revolution axis and angle, as shown in the following example:

1. After starting the `REVSURF` command, the revolved object selection stage will end and then you will be prompted to specify the revolution axis.
2. Select an object to be used as the revolution axis (the straight line). The revolution axis selection stage will end and then you will be prompted to specify the starting and ending angles.
3. Type the starting angle and then press *Enter*. After that, type the ending angle and press *Enter*.

4. The revolved mesh surface will be created and the command terminates:

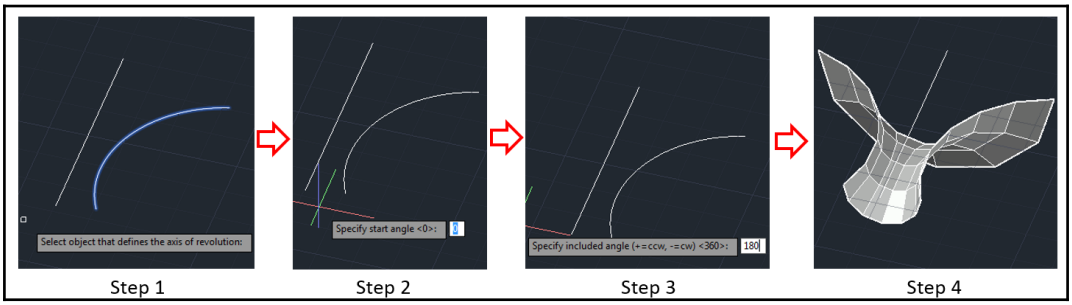


Figure 14. 51: Steps involved in using the REVSURF command

Unlike the revolve command, a surface or region cannot be used with the `REVSURF` command to create a solid mesh object; you can only select a line, an arc, a circle, or a 3D polyline.

The system variables, `SURFTAB1` and `SURFTAB2`, can be used to control the density of the generated mesh. The number of divisions on the revolution direction is determined by using the `SURFTAB1` variable, while `SURFTAB2` controls the number of divisions in the other direction. If the revolved object is a polyline, the number specified by `SURFTAB2` will be created in every straight or arc section the polyline.

Let's now have a look at the next command in the mesh surface creation set.

Ruled surface command

The second command in the surface mesh creation set, this is similar to the Loft command discussed in `Chapter 11`, *Creating Primitive 3D Shapes*. A ruled mesh surface can be created between two 2D curves in one step, as we will see in this sub-section. To start the ruled surface command, you can type the command name, `RULESURF` (or you can use the short version and just type `RU`), and then press *Enter*. Alternatively, you can click on the **Ruled** surface tool button located in the **Create** panel on the **Home** tab in the **3D Basics** workspace, and if you are using the **3D Modeling** workspace, you can find it in the **Primitives** panel on the **Mesh** tab, as shown in the following screenshot:

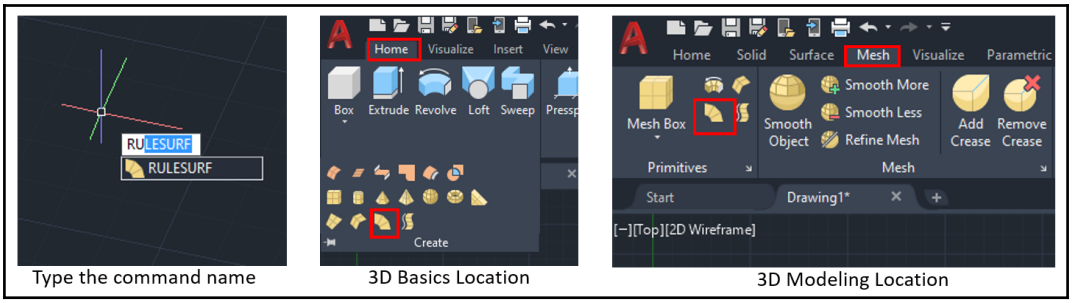


Figure 14. 52: Alternative methods for starting the ruled surface command

The `ruled surface` command can be used to create a ruled mesh surface between two 2D curves, as shown in the following example:

1. Start the `ruled surface` command.
2. Select the first curve. Once you do this, you will be prompted to select the second one.
3. Click on the second curve. The ruled mesh surface will be created and the command terminates.

The following diagram is an illustration of the previous steps:

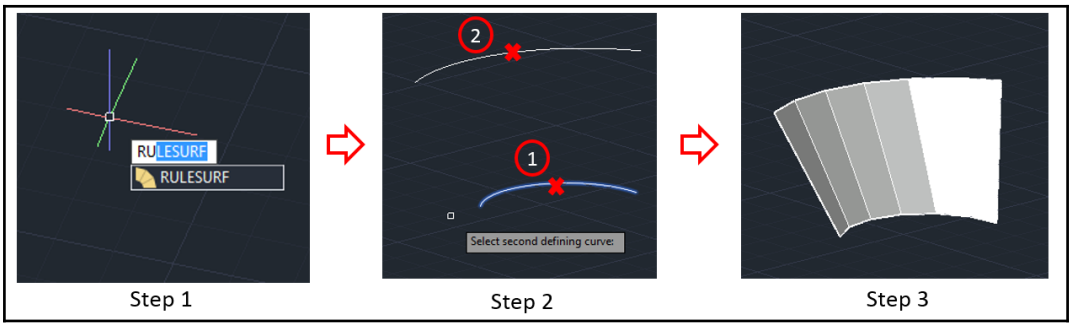


Figure 14. 53: Steps involved in using the ruled surface command

We have covered two surface mesh creation commands. Now, let's have a look at the third one – the edge surface command.

Edge surface command

The third mesh surface creation command in our set, this is an intuitive method for creating a surface mesh by determining the four bounding edges and then a smooth mesh surface is automatically created, as will be shown in this sub-section. To start the edge surface command, you can type the command name, `EDGESURF` (or you can use the short version and just type `EDG`), and then press *Enter*. Alternatively, you can click on the **Edge Surface** tool button located in the **Create** panel on the **Home** tab in the **3D Basics** workspace, and if you are using the **3D Modeling** workspace, you can find it in the **Primitives** panel on the **Mesh** tab, as shown in the following screenshot:

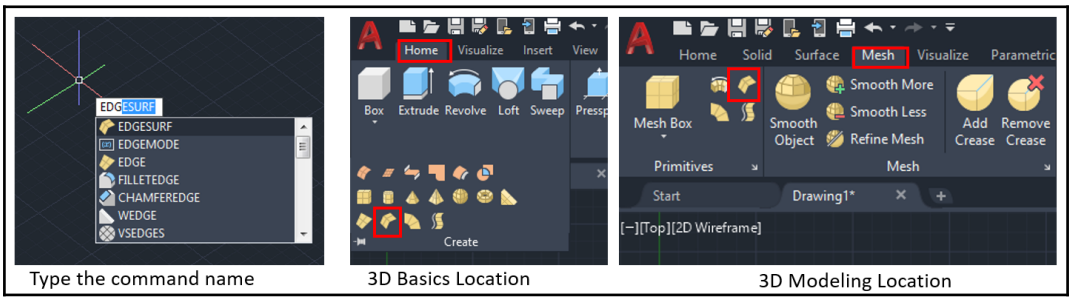


Figure 14. 54: Alternative methods for starting the edge surface command

Using four connected curves, you can create a surface mesh using the `edge surface` command, as shown in the following example:

1. Start the `edge surface` command.
2. Click on the four connected curves, as shown in the following diagram.
3. After selecting the fourth edge, the mesh surface will be created and the command terminates.

The following diagram is an illustration of the previous steps:

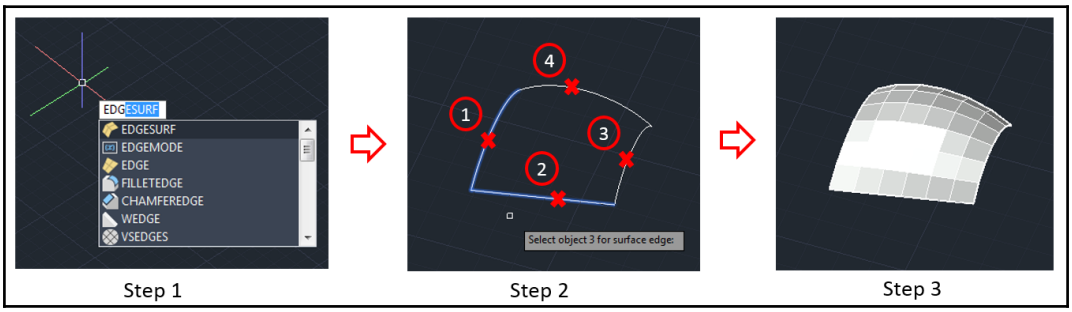


Figure 14. 55: Steps involved in using the edge surface command

Note that the edges can be selected in any direction. The only condition is that they should be connected. And, like the `REVSURF` command, the system variables, `SURFTAB1` and `SURFTAB2`, can be used to control the number of divisions in each direction. The number of divisions set in `SURFTAB1` will be applied to the first edge selected, and `SURFTAB2` will be applied to the other direction.

We have learned about three different surface creation methods. Now, let's take a look at the fourth and final one, which is called the tabulated surface command.

Tabulated surface command

A mesh surface can be regarded as a group of intersecting parallel curves. In the tabulated surface mesh command, you specify the length and the direction for each group of curves and the mesh surface will be automatically created. To start the `tabulated surface` command, you can type the command name, `TABSURF` (or you can use the short version and just type `TABS`), and then press *Enter*. Alternatively, you can click on the **Edge** surface tool button located in the **Create** panel on the **Home** tab in the **3D Basics** workspace, as shown in the following screenshot:

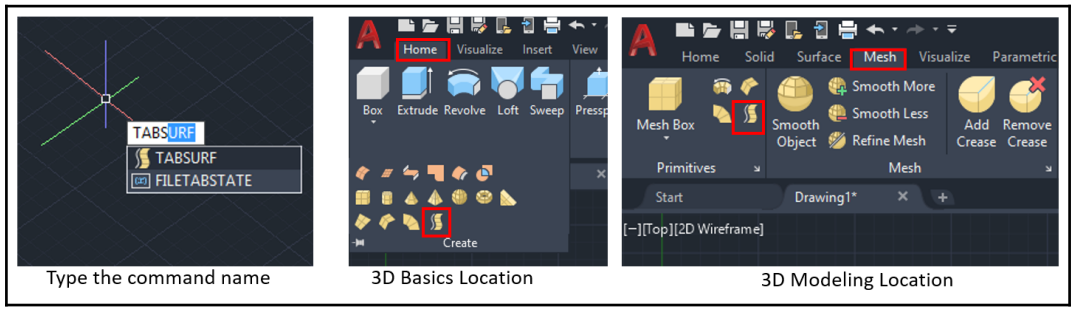


Figure 14. 56: Alternative methods for starting the tabulated surface command

If you are using the **3D Modeling** workspace, you can find it in the **Primitives** panel on the **Mesh** tab, as shown in the following diagram.

The `tabulated surface` command is very similar to the `sweep` command. You select a profile curve or a path curve, and then you select another curve for the sweep direction, as shown in the following example:

1. Start the `edge surface` command.
2. Select the curve to be used as a path or profile.
3. Select a line representing the sweep direction. The mesh surface will be created and the command terminates.

The following diagram is an illustration of the previous steps:

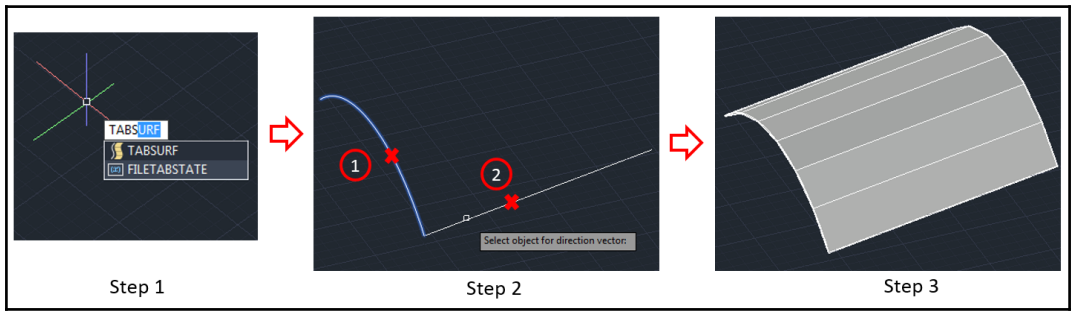


Figure 14. 57: Steps involved in using the tabulated surface command

Unlike the sweep command, the edge surface command can only use a line, an arc, a circle, an ellipse, or a polyline as the path (profile) curve. Sweeping is also limited to only straight paths.

Now that we have learned about the different methods of creating mesh objects, let's have a look at the next logical step, which is the modification toolset dedicated to mesh objects.

Modifying mesh objects

As mentioned earlier, the mesh object is divided into small volumes or cells. Each cell consists of simpler elements, which are faces, edges, and vertices. It is possible to mess around with the elements (vertices, edges, and faces) of each cell to modify its shape, and the other connected cells will dynamically change accordingly, thereby opening a wide range of modification and modeling possibilities, as we will see in this section.

Working with selection filters

To start working with the cell elements (vertices, edges, and faces), we need to adjust the selection filter properly to be able to select these elements. By default, when you try to make a selection in AutoCAD, the whole object will be selected, so to be able to select a vertex of a cell in a mesh object, for example, you need first to set the selection filter to only select vertices. The selection filter tool can be found in the **Selection** panel on the **Home** tab in the **3D Basics** workspace, as shown in the following screenshot:

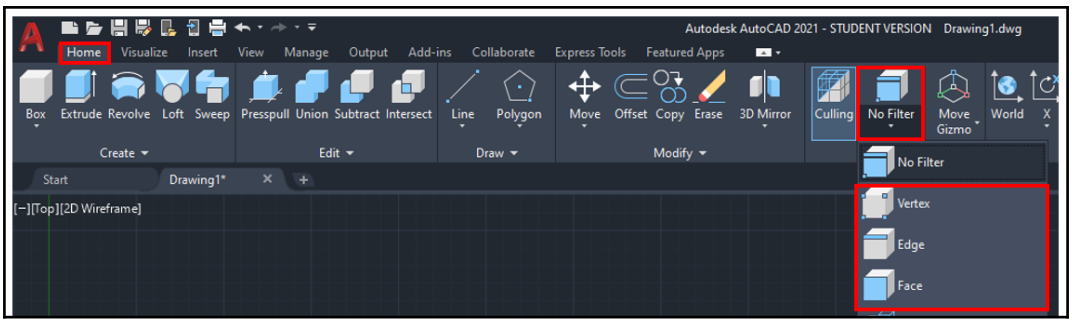


Figure 14. 58: Location of selection filters in the 3D Basics workspace

The default setting is **No Filter** and, as mentioned, it will select the whole object. When you click on the selection filter tool button, this will expand to show three other selection filter options; one for selecting vertices (**Vertex**), one for the edges (**Edge**), and the last one for selecting faces (**Face**).

Modifying the mesh vertices

As mentioned in the previous section, the first step is to set the selection filter to *Vertex*. After that, select any vertex (or vertices) in a mesh object, and then you can use the Move Gizmo (or the move command) to change the location of that vertex in any direction. The following diagram shows two examples of moving a mesh vertex:

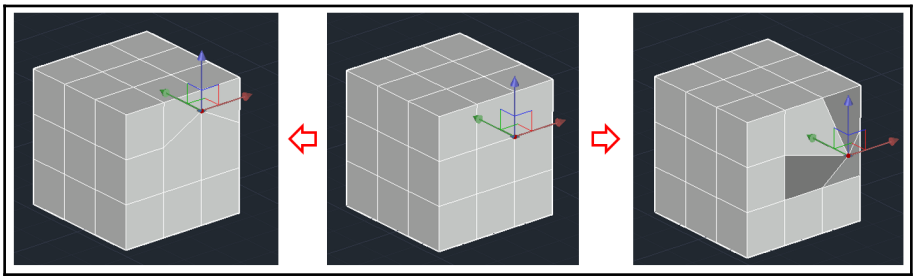


Figure 14. 59: Moving a vertex

Now, let's have a look at modifying the edges of mesh objects.

Modifying the mesh edges

The process of modifying mesh edges is similar to vertices. Start by setting the selection filter to *Edges*. After that, select the desired edge(s) to be modified. then you can change the location, the orientation, or the size of the selected mesh edges. Each of these operations can be done using the respective **Gizmo** tool as shown in the following diagram:

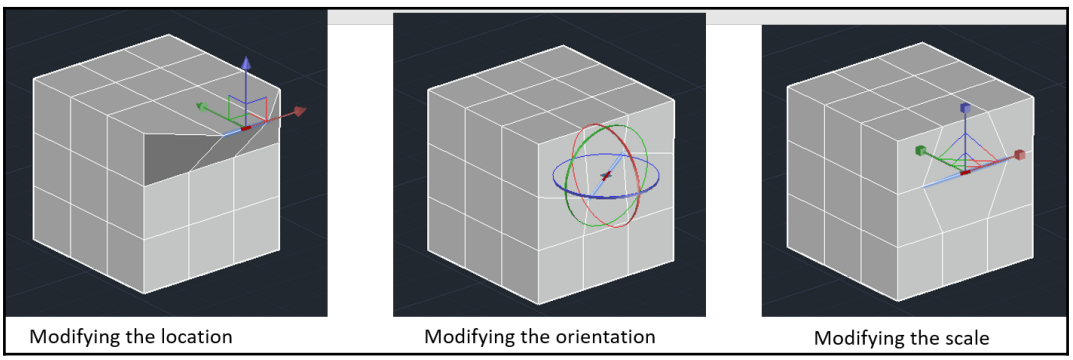


Figure 14. 60: Modifying the location of the mesh edge

Notice that in all cases, the connected cells are dynamically modified with the changes you make.

Now, let's have a look at the modification of the final element of the mesh cells, which are the faces.

Modifying the mesh faces

In a similar manner to the modification of vertices and edges, mesh faces can also be moved, rotated, or scaled using the respective **Gizmo** tool. You start by setting the selection filter to *Face*, select the desired face(s), and then you can modify the mesh face as shown in the following diagram:

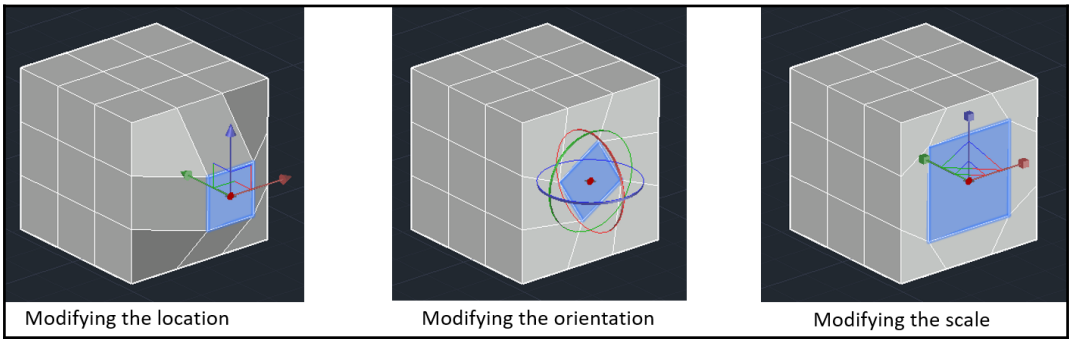


Figure 14. 61: Using the Gizmo tools to Modify the mesh face

Now that we have learned how to perform basic modify operations on the elements of individual mesh cells, let's have a look at the other mesh modification options available in AutoCAD.

Using the extrude face command

The extrude face command is used to directly extrude the face(s) of individual mesh cells. This operation opens up the possibility of adding details to simple objects in a very flexible and intuitive manner. The **Extrude Face** tool button can be found in the **Mesh Edit** panel on the **Mesh** tab in the **3D Modeling** workspace, as shown in the following screenshot:

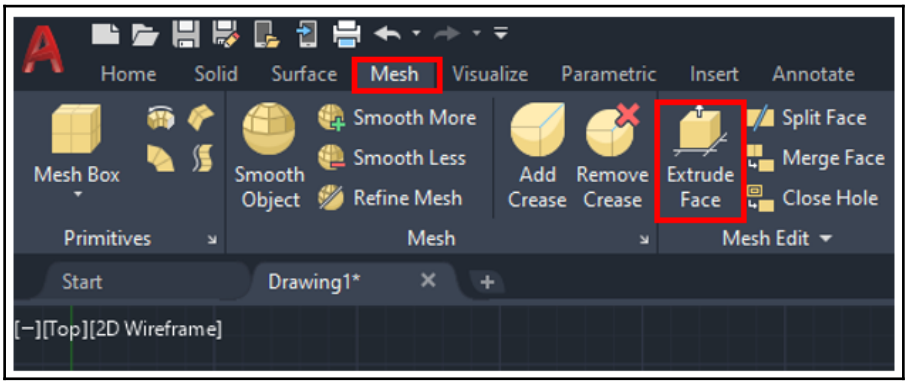


Figure 14. 62: Location of the Extrude Face tool button in the 3D Modeling workspace

The mesh cell face can be extruded directly using this command, as shown in the following step-by-step example:

1. Start the `extrude face` command, select the mesh face(s) to be extruded, and then press *Enter*.
2. Move the mouse cursor to specify the extrusion height. Alternatively, you can type the height value using the keyboard and then press *Enter*.
3. The extrusion is created and the command terminates:

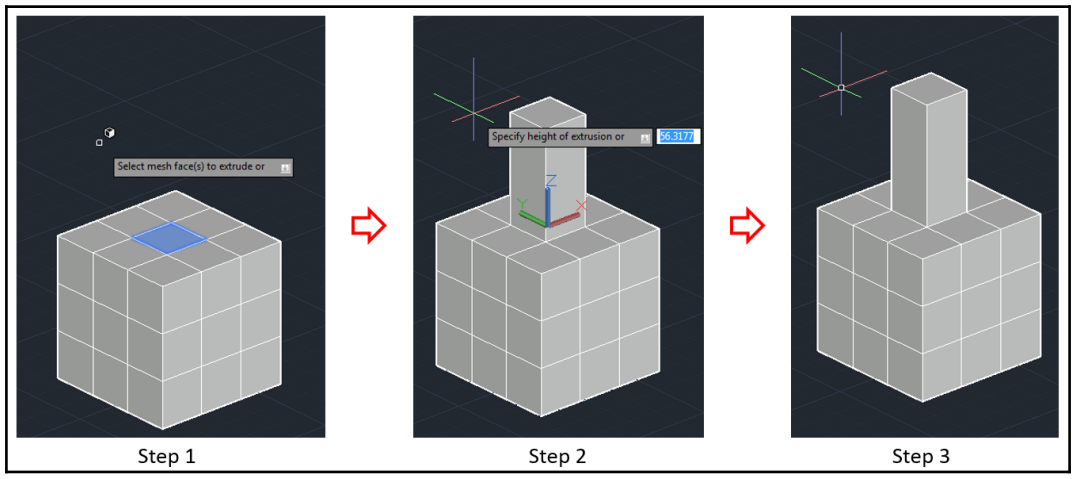


Figure 14. 63: Steps involved in using the extrude face command

Notice that in the previous example, if you move the mouse cursor downward (or entered a negative value for the extrusion height), a hole will be created in the mesh object, as shown in the following screenshot:

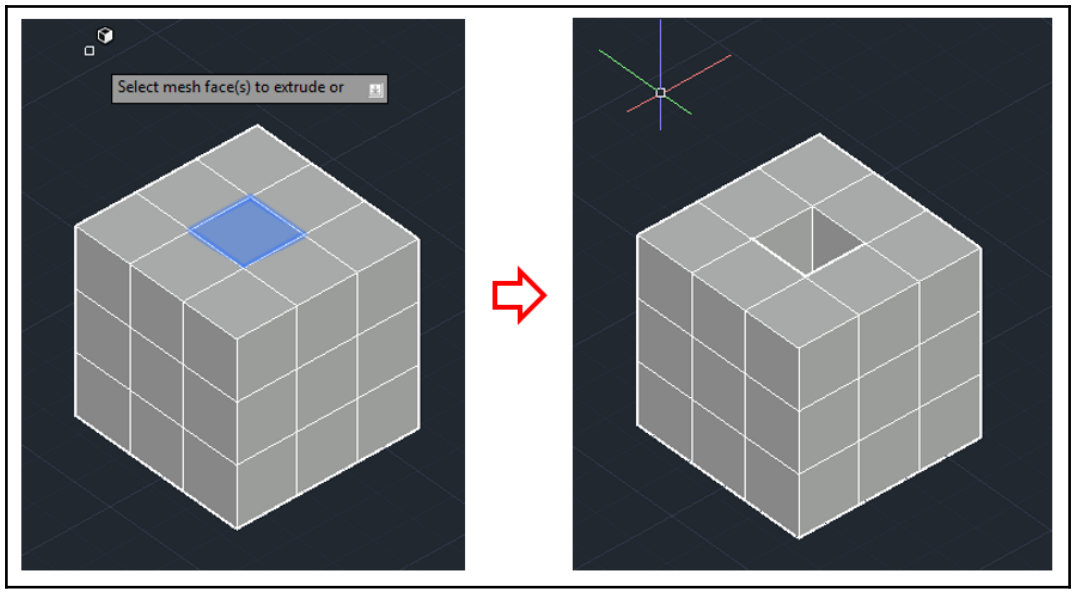


Figure 14. 64: Using the extrude face command to create holes

The `extrude face` command has two sets of options; the first set appears right after the command starts. This set has only one option, called **Setting**. To select this option, click on the option name in the command bar or, alternatively, type `S` and then press *Enter*. In the case of extruding multiple adjacent faces, this option controls whether the adjacent faces will be connected to produce a single extruded body, or whether each face will be extruded separately. The difference between these two settings may not be apparent in the case of mesh objects that are not sufficiently smooth. The following diagram shows the difference between the two settings using a smoothed mesh object:

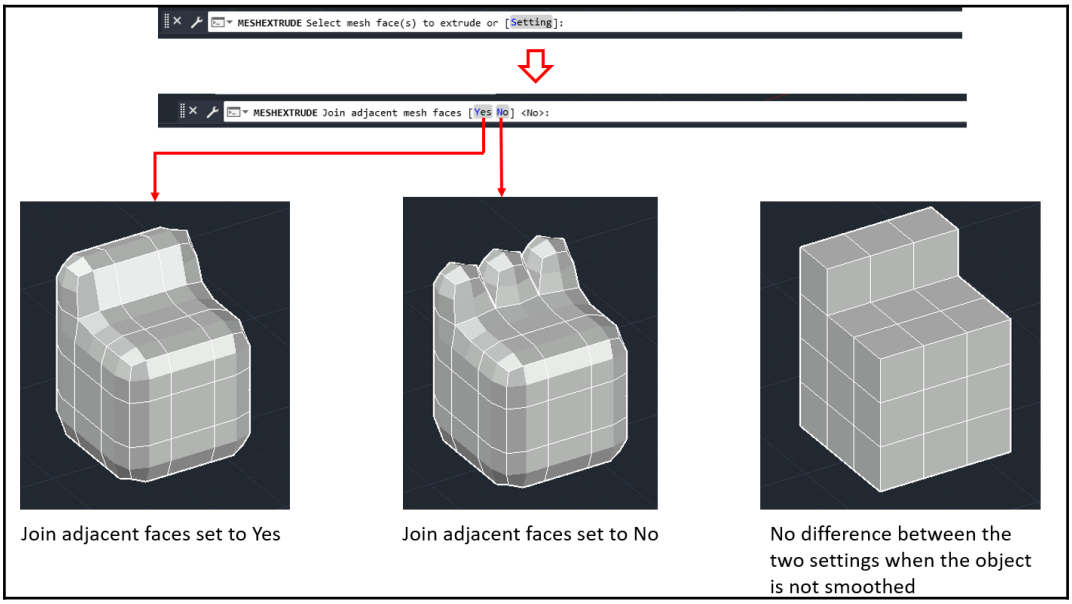


Figure 14. 65: Using the Joint adjacent mesh option

The second set of options in the extrude face command appears after selecting the faces to be extruded. This set has three options, as shown in the following diagram:

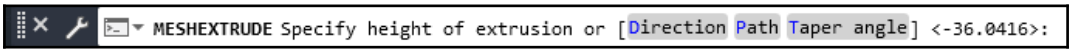


Figure 14. 66: Additional options in the extrude face command

The first option is called **Direction**. This option can be selected by clicking on the option name in the command line, or by typing `D` and then pressing *Enter*. By default, the extrusion will be created perpendicular to the mesh face. This option allows you to change the extrusion direction as desired by specifying two points representing the extrusion displacement vector. The following diagram shows an example of using the Direction option. The line end points are used to specify the direction vector:

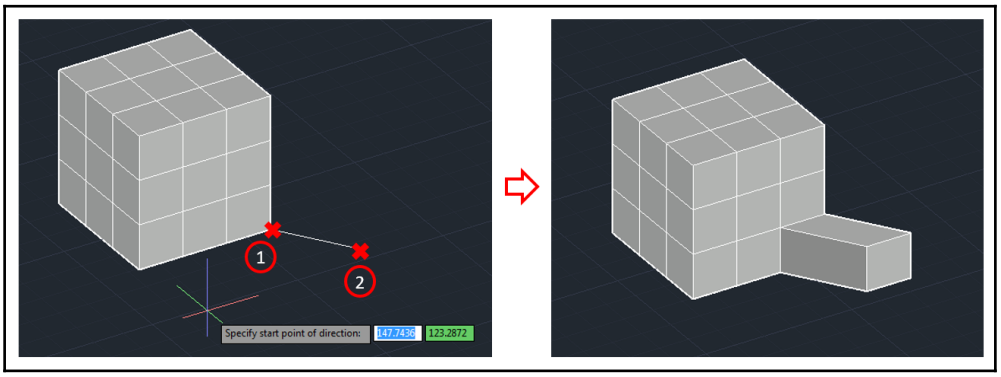


Figure 14. 67: Using the Direction option

The second option is called **Path**. To select it, click on the option name in the command bar or, alternatively, type `P` and then press *Enter*. This option allows you to set a custom path for the extrusion, as shown in the following diagram, where a polyline is used as an extrusion path:

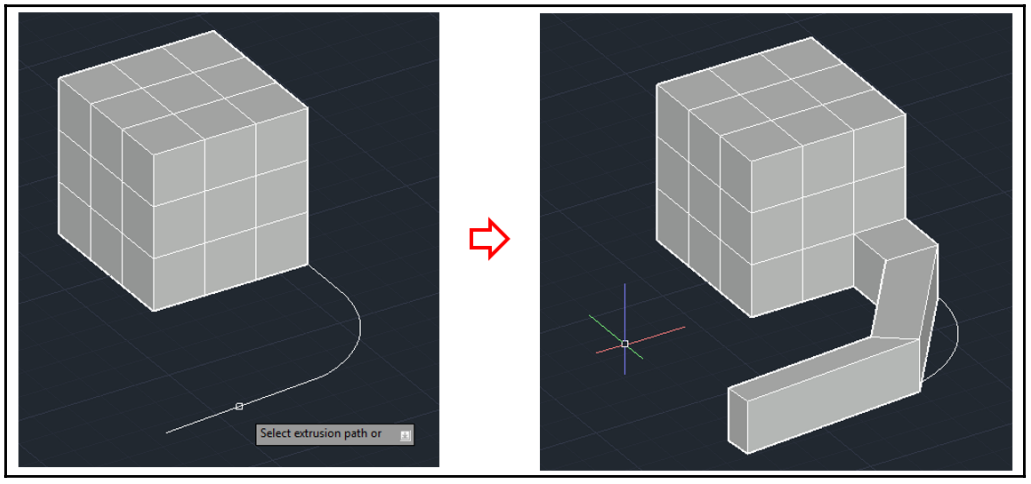


Figure 14. 68: Using the Path option

The last option is called **Taper angle**. This option can be selected by clicking on the option name in the command line or, alternatively, by typing `T` then and pressing *Enter*. As the name suggests, this option adds a taper angle to the extrusion; a positive angle will taper the extrusion inward, while a negative angle will taper the extrusion outward, as shown in the following diagram:

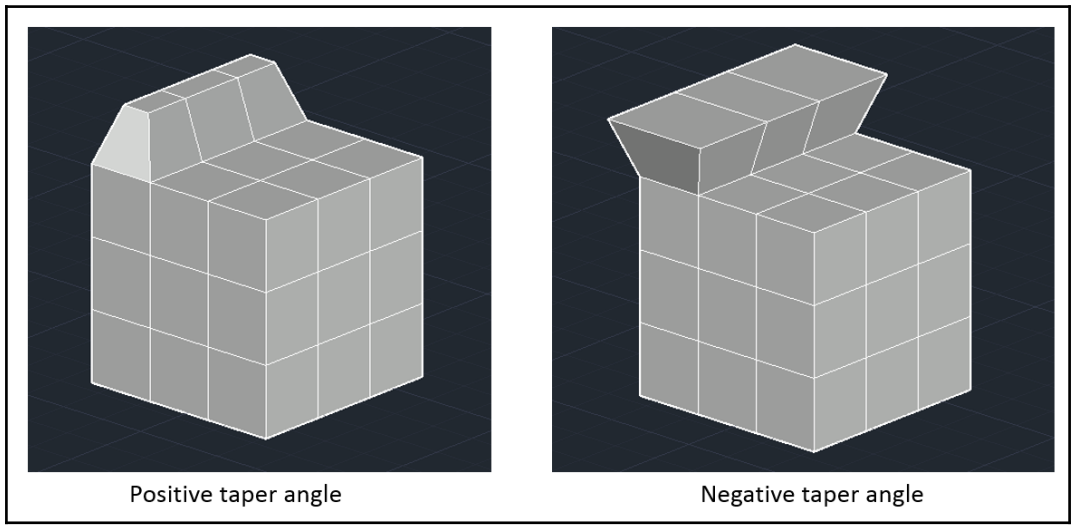


Figure 14. 69: Positive versus negative taper angle

Now that we have learned how to extrude the mesh faces, let's have a look at the next command, which is used to split them.

Using the split face command

Another modification option that is added to the mesh object editing tools is the ability to split and join the mesh faces. In this section, we will start by learning how to split an existing mesh face using the `split face` command. The **Split Face** tool button is located in the **Mesh Edit** panel on the **Mesh** tab in the **3D Modeling** workspace, as shown in the following screenshot:

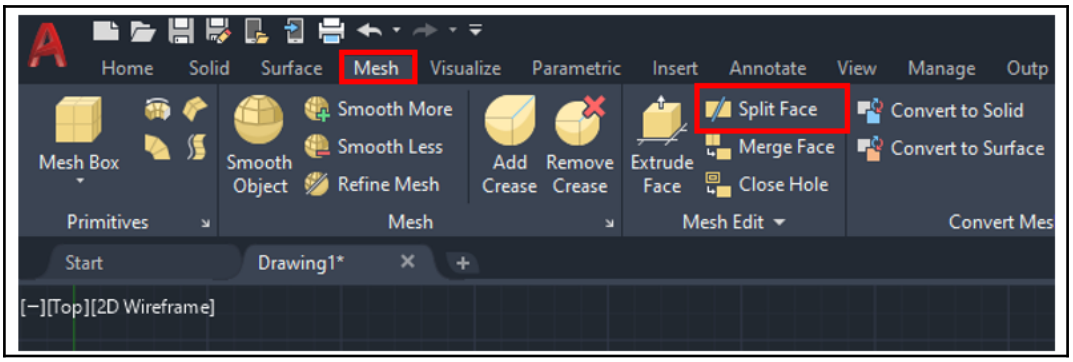


Figure 14. 70: Split Face tool button location in the 3D Modeling workspace

A mesh face can easily be split using the `split face` command, as shown in the following step-by-step example:

1. After starting the `split face` command, select the desired face to be split.
2. Specify the splitting line by determining the location of two points using the mouse cursor, as shown in the following diagram.
3. After the splitting line is specified, the split will be created and the command terminates.

The following diagram is an illustration of the previous steps:

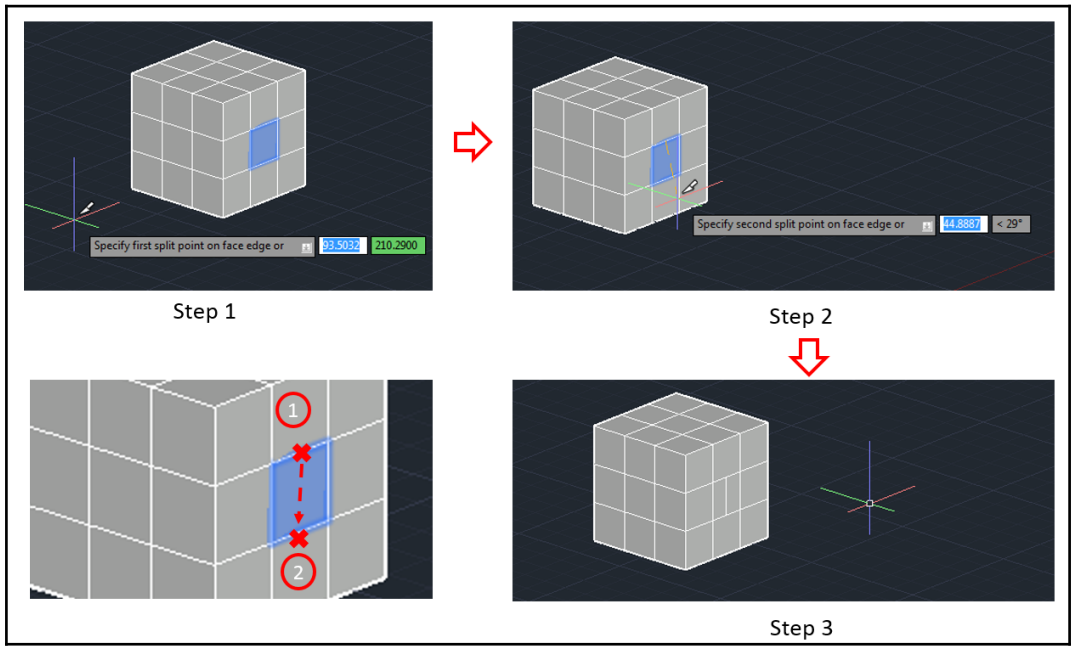


Figure 14. 71: Steps involved in using the split face command

We have learned how to split the mesh's faces, so let's now look at how to merge them.

Using the merge face command.

The **Merge Face** tool button is located in the **Mesh Edit** panel on the **Mesh** tab in the **3D Modeling** workspace, as shown in the following screenshot:

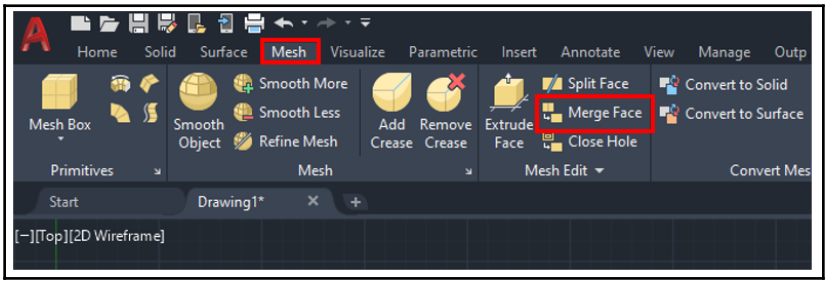


Figure 14. 72: Merge Face tool button location in the 3D Modeling workspace

Multiple mesh faces can easily be joined together using the `merge face` command, as shown in the following step by step example.

1. After starting the `merge face` command, select the desired faces to be joined and then press *Enter*.
2. As shown in the following diagram, the faces will be merged and the command terminated, as shown here:

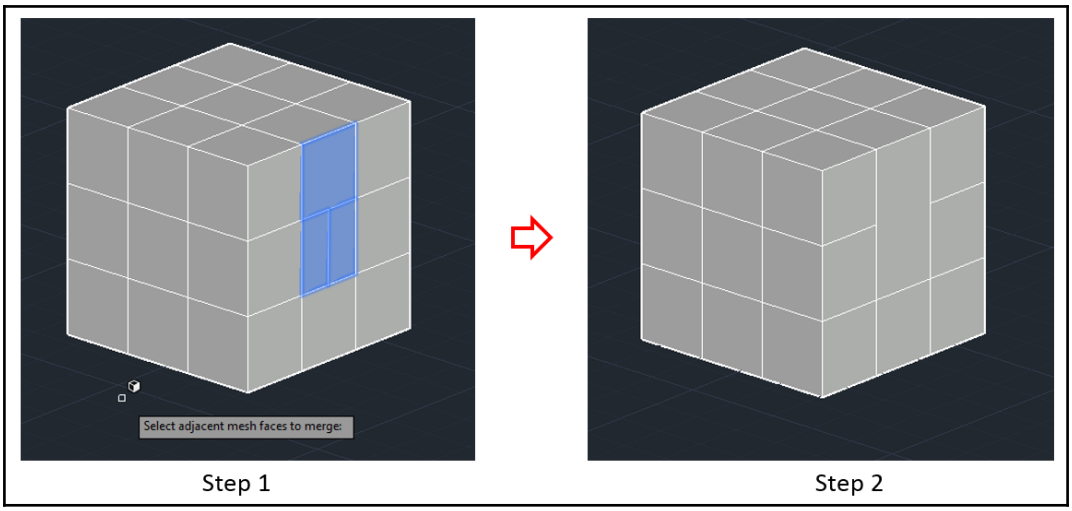


Figure 14. 73: Using the merge face command

Now that we have learned how to merge or split a mesh face, let's have a look at the next mesh modification command, which is called the close hole command.

Using the close hole command

The close hole command can be used to close gaps in mesh objects, you simply select the mesh edges surrounding that hole and the command will automatically create a surface to seal that gap. The **Close Hole** command button is located in the **Mesh Edit** panel on the **Mesh** tab in the **3D Modeling** workspace, as shown in the following screenshot:

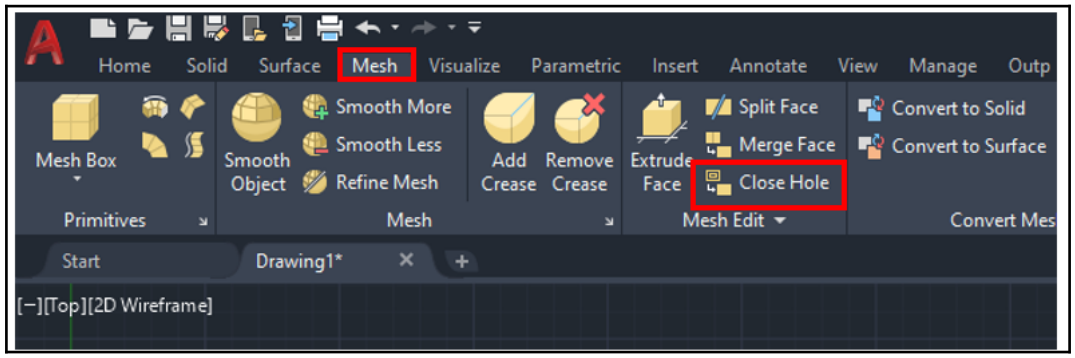


Figure 14. 74: Close Hole tool button location in the 3D Modeling workspace

A gap in the mesh object can be closed, as shown in the following step-by-step example:

1. Start the close hole command, select the edges surrounding the hole (this can be an open loop as shown in the following diagram), and then press *Enter*.
2. A surface will be created that closes the gap and the command terminates, as shown here:

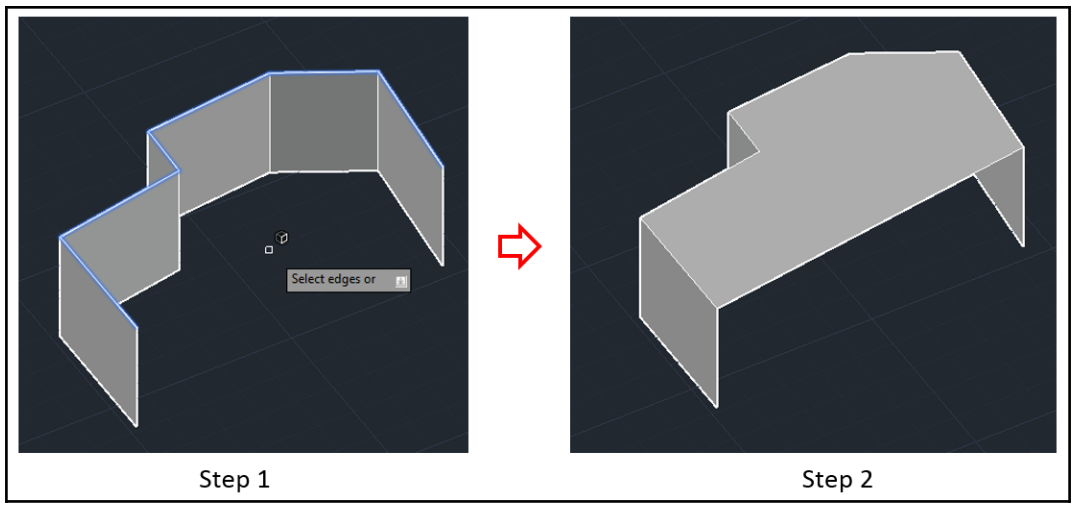


Figure 14. 75: Using the close hole command

The close hole command contains one option that appears in the command line just after the command is started. This option is called **Chain**, and it can be selected by clicking on the option name in the command line or by typing `CH` and then pressing *Enter*. When this option is activated, the group of connected edges can be selected with one click instead of having to select the edges one by one.

Now, let's have a look at the next command, which is called the Collapse Face or Edge command.

Using the Collapse Face or Edge command

The Collapse Face or Edge command can be used to edit the mesh maps on mesh objects. As the name suggests, it collapses a face (or an edge) to a single point and the neighboring faces will be modified accordingly. The **Collapse Face or Edge** tool button can be found in the **Mesh** Edit panel on the **Mesh** tab in the **3D Modeling** workspace, as shown in the following screenshot:

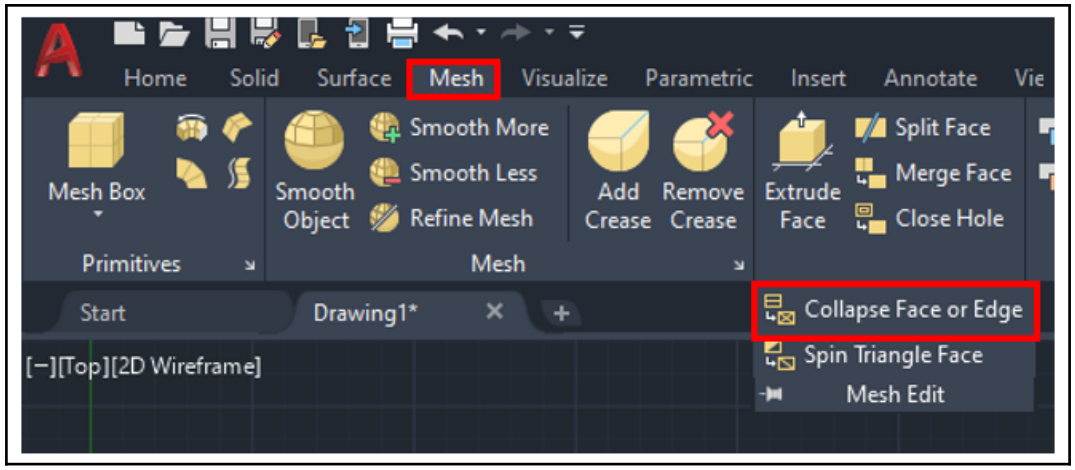


Figure 14. 76: Collapse Face or Edge tool button location in the 3D Modeling workspace

The following is a step-by-step example of collapsing a face in a box mesh object:

1. Start the Collapse Face or Edge command and then click on the desired mesh face to be collapsed.
2. As shown in the following diagram, the middle mesh face is collapsed to a single point and the command terminates, as shown in the following diagram:

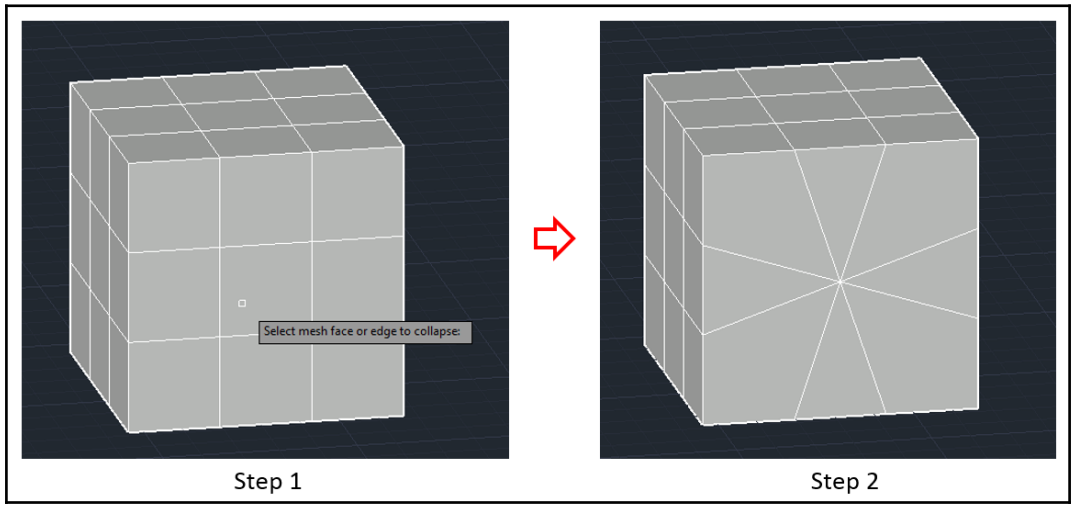


Figure 14. 77: Using the Collapse Face or Edge command

As shown in the previous example, the face is collapsed to a single point, and the other neighboring faces were automatically triangulated to accommodate the new change.

Now that we have learned how to manipulate the faces of the mesh object and how to close holes in the mesh object, let's have a look at how to control the mesh smoothing process by adding creases and using the refine command.

Adding creases to smoothed objects

As shown in *Using the smooth object command* section, the mesh object can be created with a certain degree of smoothness, or the smoothness of an existing mesh body can be increased and decreased. You may arrive at a situation where you don't want the whole body to be smoothed, or you want some of the edges and the faces of the body to remain sharp and flat. This can be done using the **Add Crease** tool. The **Add Crease** tool button can be found in the **Edit** panel on the **Home** tab in the **3D Basics** workspace, or in the **Mesh** panel on the **Mesh** tab in the **3D Modeling** workspace, as shown in the following screenshot:

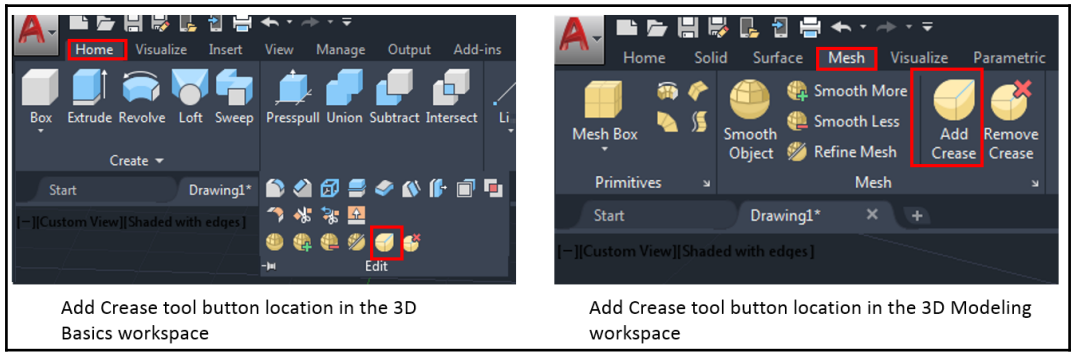


Figure 14. 78: Adding Crease tool button locations

The following step-by-step example shows the exclusion of a set of mesh faces from the smoothing operation of a box mesh object:

1. After starting the add crease command, select the desired mesh faces to be excluded from the smoothing operation and then press *Enter*. The selected faces will be highlighted, as shown in the following diagram.
2. You will be prompted to select the crease mode. Select **Always**.
3. Now, increase the smoothness of the object. As shown in the diagram, the creased faces will remain flat with no curvature following the smoothing operation.

The following diagram is an illustration of the previous steps:

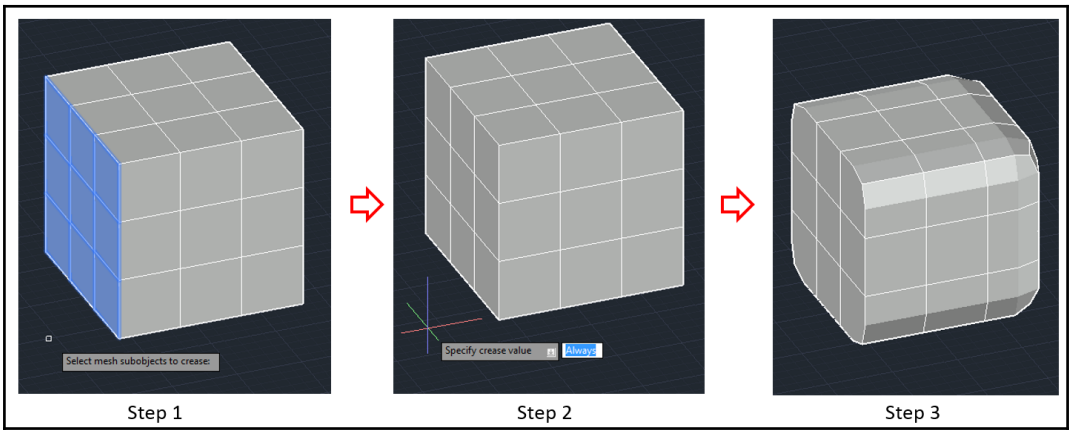


Figure 14. 79: Steps involved in using the add crease command

The crease can be undone using the **Remove Crease** tool button, which is located next to the **Add Crease** button, as shown in the following screenshot:

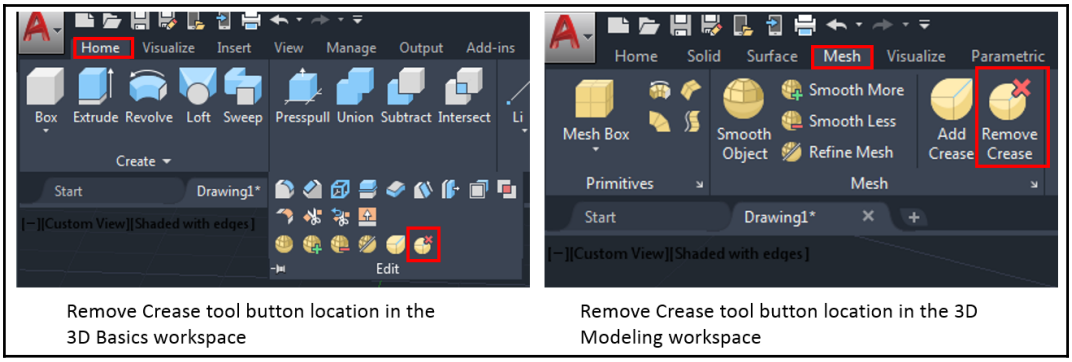


Figure 14. 80: Remove Crease tool button locations

So, if you have a smoothed object that has a crease applied to some of its elements, the crease can be undone as demonstrated in the following step-by-step example:

1. Start the remove crease command, select the creased elements (the mesh faces in the case of the example diagram), and then press *Enter*.

2. The crease will be removed and the command terminated, as shown in the following diagram:

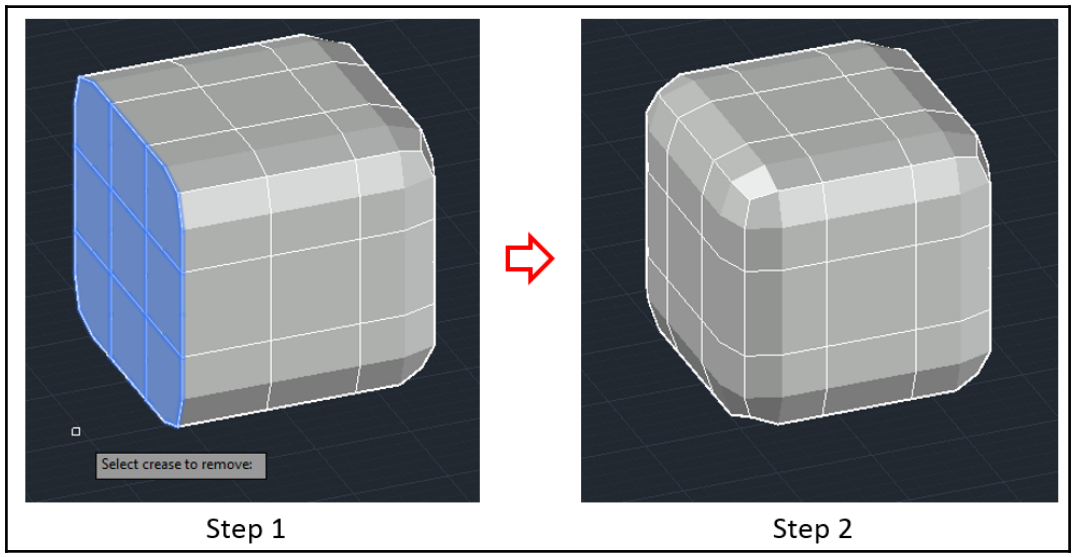


Figure 14. 81: Steps involved in using the remove crease command

We have learned how to control the degree of smoothness on every part of mesh objects. Now, let's have a look at another tool used in mesh modeling, which is mesh refinement.

Refining the fesh

Increasing the smoothness of a mesh object does not increase the mesh cell count, as shown in the following diagram. The number of cells is the same, but the cell edges and surfaces are more curved with the increased smoothness:

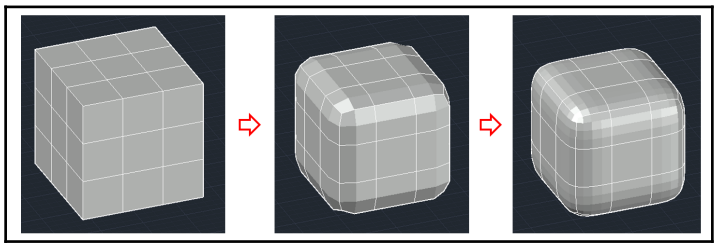


Figure 14. 82: Increasing the mesh body smoothness

If a sufficient number of cells is present, you can have the same smooth body, but this time represented by cells with straight edges and flat surfaces. This operation is called refining the mesh object. The **Refine Mesh** tool button can be found in the **Edit** panel on the **Home** tab in the **3D Basics** workspace, or in the **Mesh** panel on the **Mesh** tab in the **3D Modeling** workspace, as shown in the following screenshot:

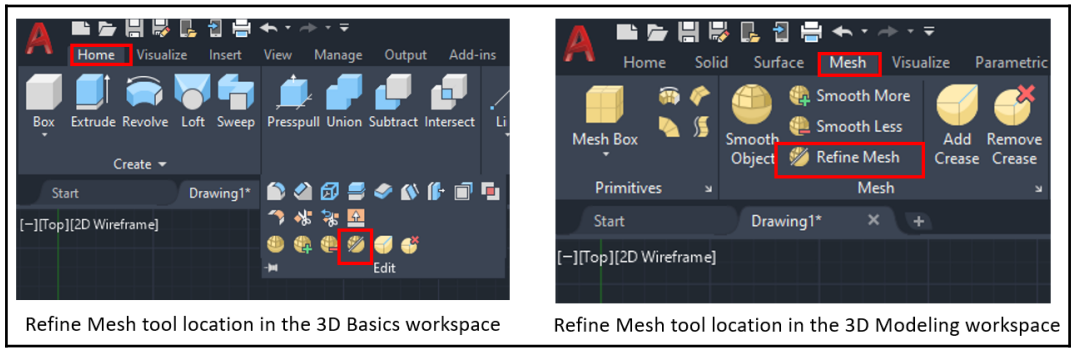


Figure 14. 83: Refine Mesh tool button locations

The refine mesh command works only with smoothed objects. If the object already has straight edges and flat cell faces, then there is no point adding more cells. To refine a mesh object, simply click on the **Refine Mesh** button and then click on the desired mesh object to be refined. The following diagram shows an example of an object that is created with no smoothness, then with the smoothness increased once, and then with the mesh refined:

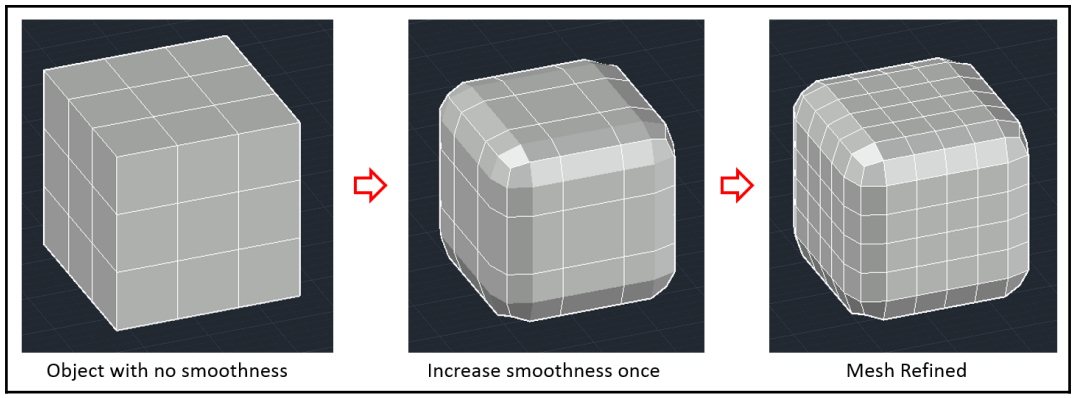


Figure 14. 89:Result of refining followed by increasing smoothness

The previous diagram shows an example of an object being smoothed, and then, in the next refining step, more divisions were added to accommodate the smoothness added in the previous step.

Summary

In this chapter, we have learned two additional approaches for creating 3D models in AutoCAD. Starting with surface modeling, we learned how to create planar and network surfaces, which are the two main surface types in AutoCAD, and we also explored the differences between using procedural and NURBS surfaces, and the possibilities afforded by using them, which doesn't have an equivalent in solid modeling. Then, we looked at a group of other surface creation commands, which are mainly used for patching, closing holes, and blending two separate surfaces. By way of completing the modeling toolset, we also discussed in detail the modification and editing tools that are dedicated to surface modeling.

Then, we moved on to the second additional approach in 3D modeling – mesh modeling. We started by exploring the three different ways to create a mesh object; we looked at mesh primitives, how to convert a solid body to mesh, and finally, a group of mesh creation commands by using curves. After that, we explored the different modification possibilities of the mesh elements, and we discussed the different editing tools that are dedicated to mesh objects. We then concluded by applying what we had learned by means of two small practice projects where we created a simple chair model and a mug using the mesh modeling tools.

In the next chapter, we will move from modeling to the presentation or the output stage. We will learn the concept of paper space, how to properly set up your drawing for printing, and actual printing, including related settings and options.

15
Paper Space Layouts and Printing

To create a high-quality professional print of your project, a number of settings should be adjusted properly, such as the scale of the print, the title block, and the printed line color and width of each line in the drawing. In this chapter, we will discuss, in detail, the topics related to the preparation and printing of drawings in AutoCAD. They are as follows:

- Understanding the paper space
- Creating viewports and title blocks
- Setting pen assignments and printing

By the end of this chapter, we will be able to properly set the paper space tab, create a title block, and print with the desired scale, line colors, and line weights. Let's now take our first step, which is exploring the paper space.

Understanding the paper space

An AutoCAD working environment is divided into two main parts: the model space and the paper space. The model space is where you do all the drawing or modeling, while the paper space is where you prepare your drawings for printing. The presence of these two separate working spaces allows for easier and better focus during the model's creation; for example, the user can use the convenient **1:1** scale in the model space during the model's creation, even if they are planning to print to another scale, which will be adjusted in the paper space (as we will see later). You can even create multiple layouts and viewports to print the same model in different scales. The paper space also houses the title block and notes for the drawing, so you don't need to worry about that interfering with the modeling process.

In this section, we will learn how to switch to or create a paper space tab in the drawing file and how to adjust the paper space tab settings properly.

Introduction to paper space

You can easily switch between the paper space and the model space using the tabs located at the bottom-left corner of the drawing area; there will be one tab for the model space called **Model**, and two (or maybe more) tabs for the paper space, called **Layout1** and **Layout2**, as shown:

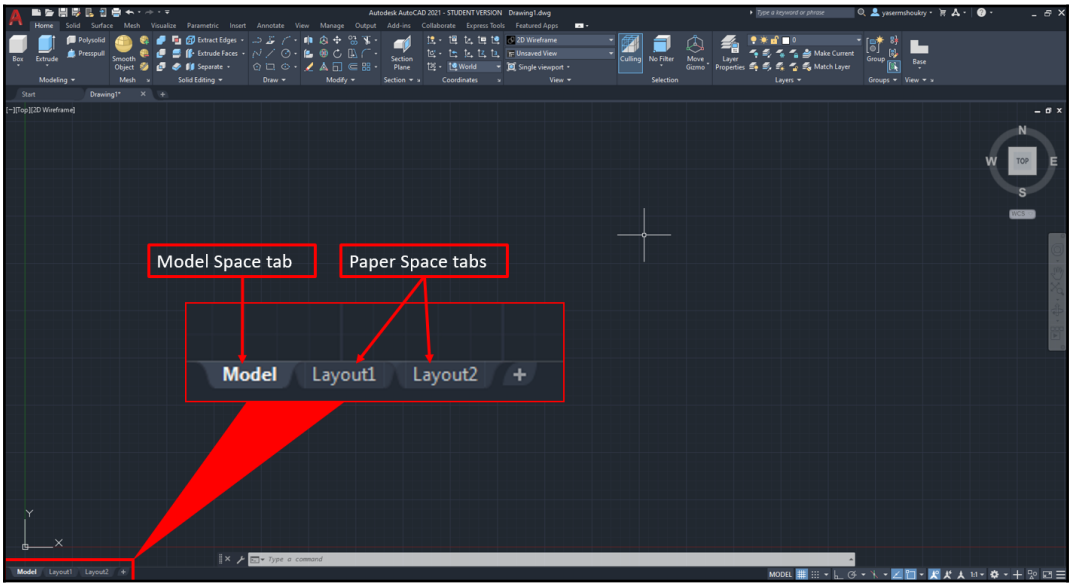


Figure 15.1: The model space and paper space tabs

Hovering over the inactive tabs will show a small preview of the contents of this tab. This will work with either the paper space tabs or the **Model** tab, as shown here:

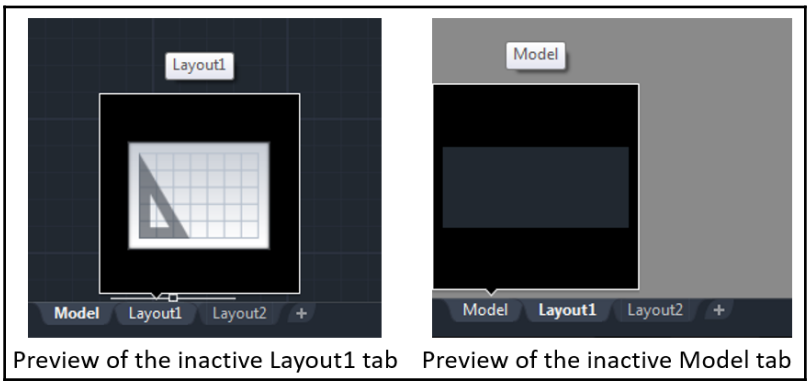


Figure 15.2: The tabs' preview

You can only have one **Model** tab in the drawing file; on the other hand, you can have multiple paper space tabs. You can also create new paper space tabs and delete existing ones. To create a paper space tab, click on the small plus sign on the right side of the last paper space tab and a new **Layout** tab will be created, as shown here:

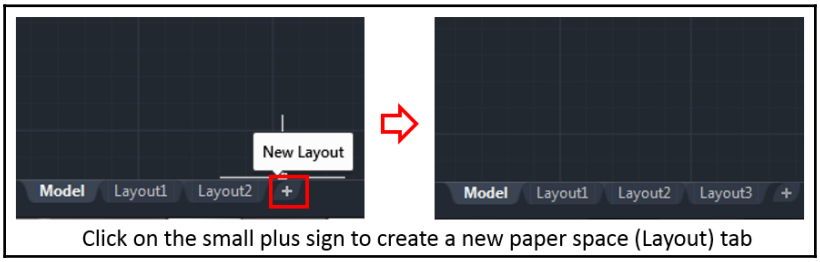


Figure 15.3: Creating a new paper space tab

If you right-click on any of the existing paper space tabs, you will find a number of options to manage them, as shown in the following screenshot. You can delete an existing tab or rename it; you also have the **Move or Copy...** option, which allows you to create an exact copy of an existing paper space tab, or just change its position. Choose the new location of the tab (or the copy) then press **OK**. Alternatively, to change the tab's position, you can simply click and drag the tab and move it to the desired position:

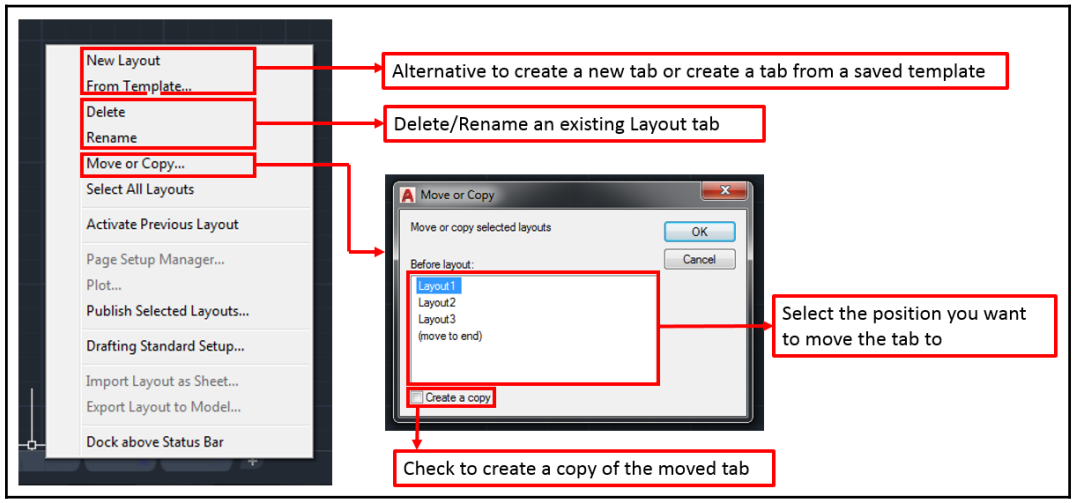


Figure 15.4: Managing layout tabs

As mentioned earlier, the purpose of the paper space tabs is to prepare your model/drawings for printing. You can manage the printing paper area by adding different viewports showing different parts of your drawing/model, with each viewport having a different printing scale. In addition, you can also add a title block for your drawing, showing all the relevant information.

Exploring the layout tab

Let's have a quick tour of the paper space tab. When you switch to a new or empty paper space tab, you will find a preview of the printing paper, showing the borders of the printable area. This border will depend on the size or type of paper used and the selected printer. Inside the printable area, you will find a viewport showing, by default, all the contents of the model space, as shown:

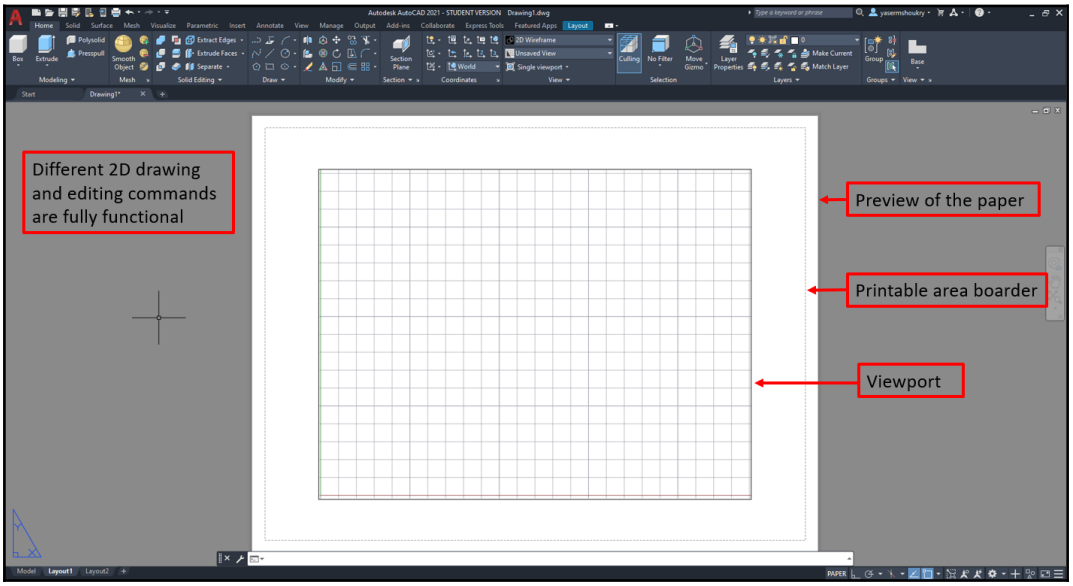


Figure 15.5: A preview of the paper space tab

You can carry out all the two-dimensional navigation maneuvers, such as zooming and panning, but not three-dimensional rotation or use of the ViewCube. In addition, all the two-dimensional drawing and drafting tools are available and work normally to create the title block of the drawing, which we will show later in this chapter. Even the commands for creating three-dimensional shapes will work, but the created objects will only be viewed from the top.

We have explored the different contents of the layout tab. Let's now learn how to adjust its settings properly.

Setting up the paper space tabs

The first step for using the paper space tab properly is to adjust the settings. To access the settings, right-click on the desired layout tab and choose **Page Setup Manager...**. This will open the **Page Setup Manager** window, where you will find a list of all the layout tabs available in the drawing file, as in the following figure. You will find four options on the right, which are as follows:

- **Set Current**: This is used to change the current/active layout. You choose the desired layout tab from the list then click on **Set Current**.
- **New...**: This is used to create a new layout tab.
- **Modify...**: This is used to adjust or modify the settings for the current/active layout tab.
- **Import...**: This is used to import a saved file with previously saved layout tab settings.

The following figure shows the details of the **Page Setup Manager** window:

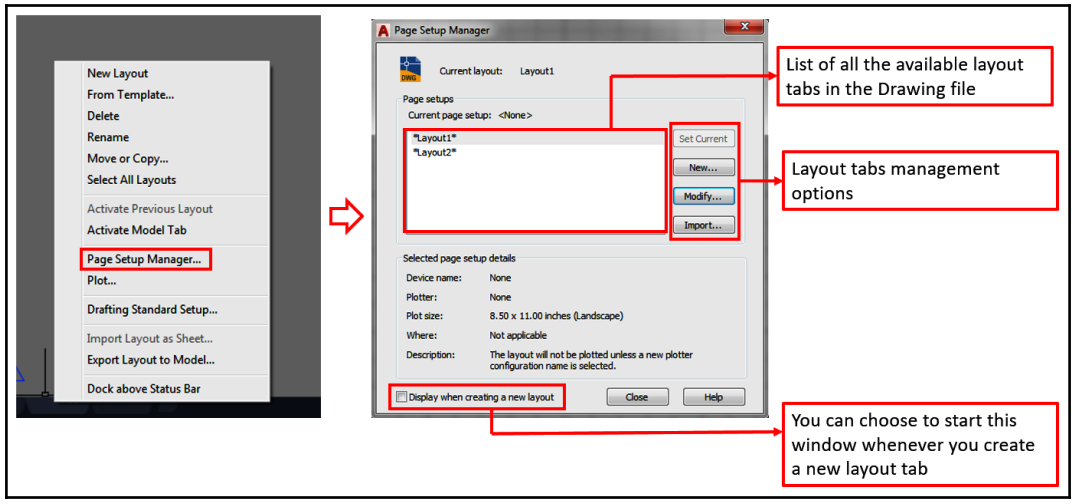


Figure 15.6: The Page Setup Manager window

So, to adjust the paper space tab settings, click on **Modify**, which opens the **Page Setup** window. The **Page Setup** window is divided into a number of sections. Starting from the top left, you will find the section called **Page setup**, where you can quickly select a previous or a saved setup to use in your current layout tab. If there are no saved setups and this is the first tab that you modify, the settings will then display **<None>**, as in the following figure.

The next section is called **Printer/plotter** and this where you choose the printer or plotter you are planning to use for printing your drawing. You simply click on the button next to **Name** and it will open all the available printers or plotters that are connected directly to your PC, or through a network connection, to choose from. Notice that besides the printers and plotters, you will find other options, such as printing to a PDF file or printing to an image file.

After selecting the plotter or printer, go to the next section, where you select the paper size you are planning to use. Click on the button on the **Paper size** section and it will open all the available paper sizes that are compatible with the selected printer or plotter. The locations of the **Printer/plotter** selector and the **Paper size** section are shown here:

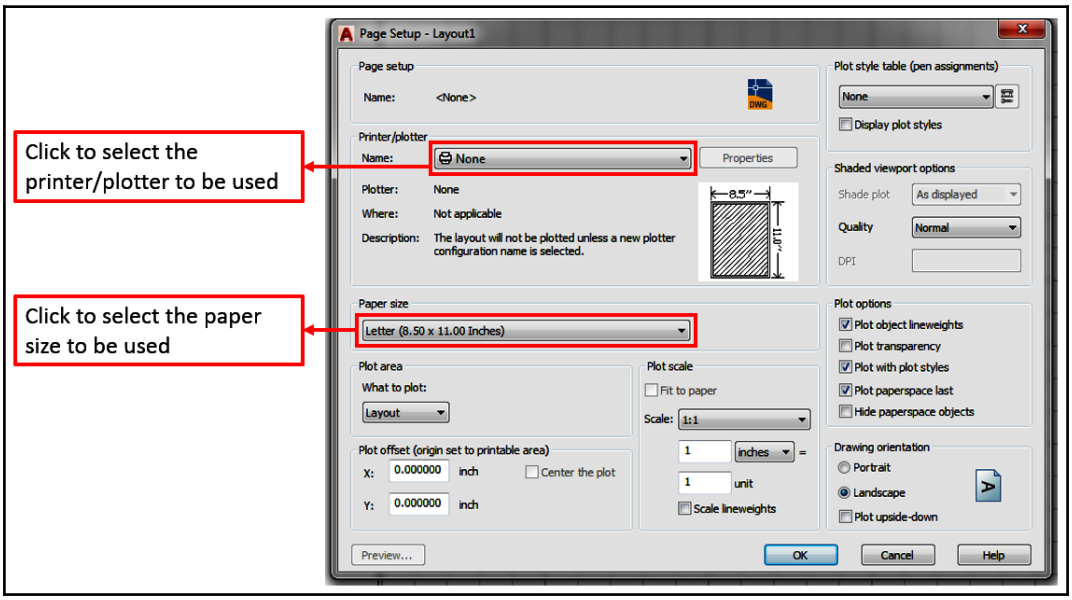


Figure 15.7: Adjusting the paper size for the layout

We have now set the printer and the paper size—the next step is to adjust the scaling factor. At the bottom, in the middle, you will find the **Plot scale** section, where you can set the units used in the layout tab and add a global scaling factor. For the units, you have two different options: **inches** or **mm**. Then, after that, you can set a global scaling factor for the layout. The lower number represents the units displayed in the layout tab and the upper number is for the objects inside the model, as shown here:

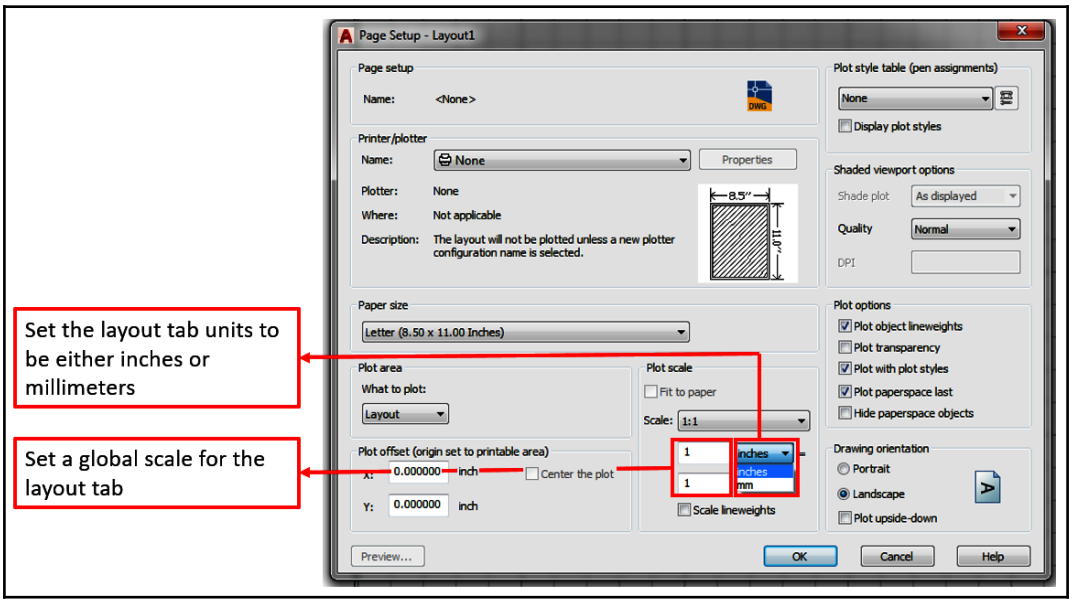


Figure 15.8: Adjusting the scale for the layout

As mentioned earlier, the scaling factor is global for the **Layout** tab, so if you have created viewports with a given scale factor, the final scale on the printed paper will be a multiplication of these two scaling factors, so you should be careful when dealing with this scaling factor. A good practice is to set this scale factor to **1:1** and just rely on the scaling factor of the viewports.

Now that we have a properly adjusted layout tab, let's create the viewports that will display our model.

Creating viewports and title blocks

A viewport is like a window that you can view a part, or all, of the drawing in the model space from. You can create just one or multiple viewports in the same layout. Also in case you want to focus on specific parts of the drawing, it is possible to create viewports with different custom shapes to capture exactly the desired region of the drawing (even if the desired region has an irregular shape). Moreover the custom shaped viewports grants the flexibility of placing multiple viewports, in addition to the ability of adding different styles of title blocks and notes. Also, each viewport can have a separate scaling factor. In this section, we will learn how to create viewports with different shapes and how to properly adjust their related options and settings.

Creating rectangular viewports

To create a new rectangular viewport, you can click on the button named **Rectangular**, found in the **Layout Viewports** panel on the **Layout** tab (the **Layout** tab appears when you switch to any of the paper space tabs). An alternative method is to type the command name, `MVIEW`, (or just type `MV`), then press *Enter* to start the command, as shown here:

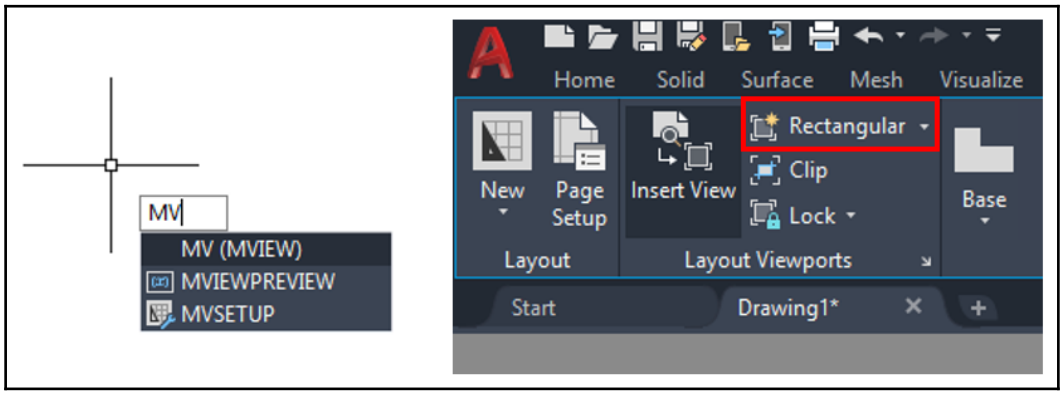


Figure 15.9: Creating a new viewport

After starting the command, you can create a rectangular viewport by simply specifying two opposite corners, as shown in the following step-by-step example:

1. Start the `Rectangular` viewport command.
2. Using the mouse cursor, specify the first corner location of the viewport. Alternatively, you can type the coordinates using your keyboard and then press *Enter*.
3. Click to specify the second corner location. Once you do that, the command terminates and the viewport is created, as shown:

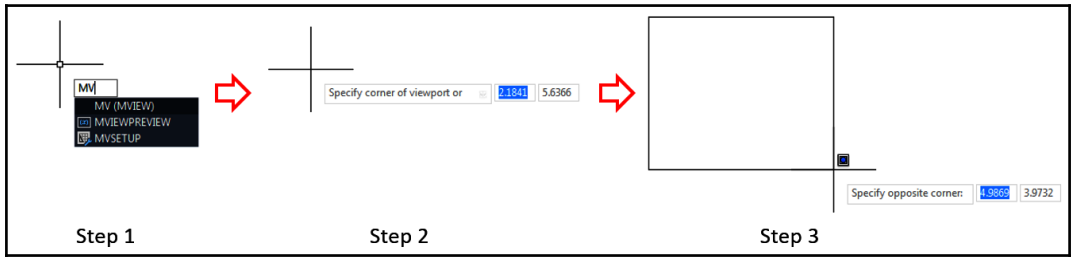


Figure 15.10: Steps for creating a rectangular viewport

We have created the viewport; now, let's explore how to modify it and adjust its settings.

Adjusting the viewport settings

By default, the viewport will display a zoom option, which extends the view of the model space (that is, it will be zoomed out to display all the objects in the model space). You can control what is viewed inside the viewport either by resizing the viewport or changing the zoom/scale factor. This can be done by trying the different methods explained here.

Method 1

In this method, we double-click inside the viewport. This will switch the effect of all the commands you use (navigation and even drawing/editing) to the model space, but through the viewport (so, you are controlling the model space but through the viewport). You can change the zoom or pan or rotate the view, and even draw some new objects or edit existing ones. After you have finished, you just double-click outside the viewport and the control is switched back to the active layout tab. Notice that if you switch to the model space tab, you will find all the changes or additions you made through the viewport present.

In addition, if you select the viewport, you will find a number of grab points that can be used for quick modifications. Starting with the center of the viewport, you will find a basepoint that can be used to grab and move the viewport. Besides that, the four corner grab points can be used to change the size of the viewport, as in the following figure. Notice that you can't change the viewport shape; it will keep a rectangular shape when you move the corner point:

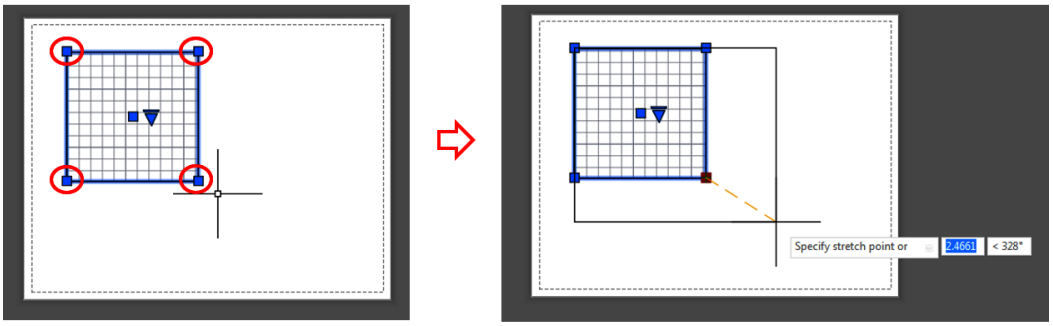


Figure 15.11: Resizing the viewport using corner grab points

Finally, near the basepoint, there is a small arrow. Click on this arrow to open a list of all the available preset scale factors for the viewport. You can quickly change the scale factor by clicking on the desired one. Notice that at the top of the list, you will find the current scale factor for the viewport. In the following figure, you will see that **Custom** is selected, which means it is not using one of the preset standard scale factors from the list:

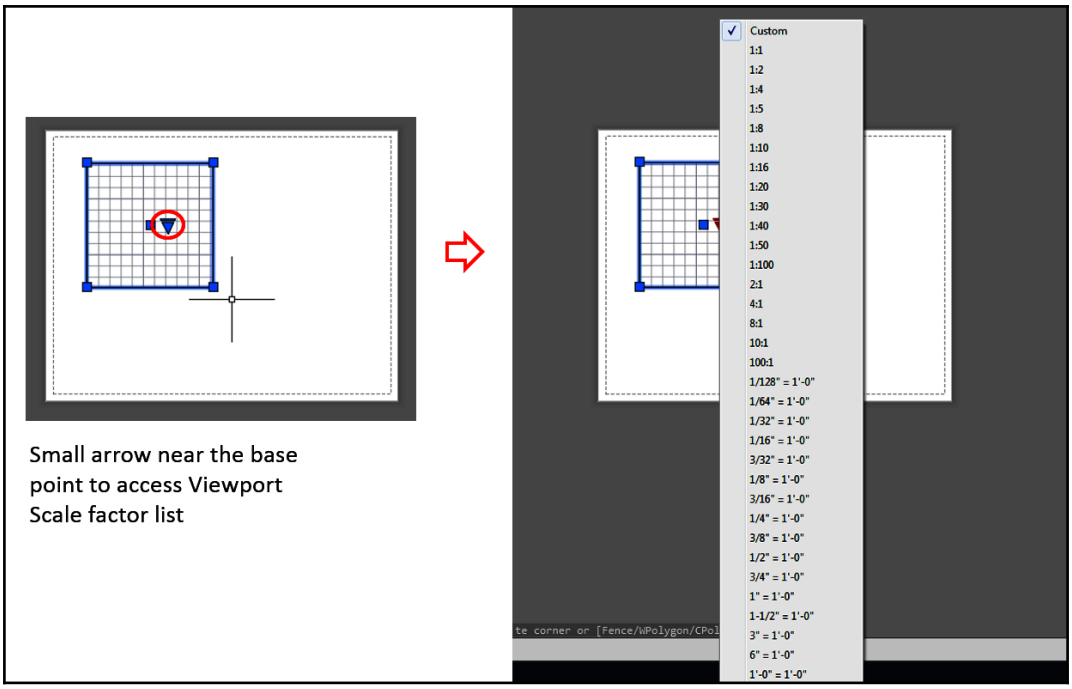


Figure 15.12: Adjusting the viewport scale

Method 2

An additional method to control and customize the viewport is using the properties palette. The properties palette for the viewport displays the usual properties of most of the objects in AutoCAD, such as the color, layer, the coordinates of the viewport's center, the length, and the height.

In the **Misc** section at the bottom, you will see an option called **On**, which can be used to turn the viewport on or off. Also, you have an option called **Standard scale**, which opens a list of the previously shown preset standard scale factors to choose from. If you want to use a scale factor outside of this list, you can use the **Custom scale** option, which can be used to set the scaling factor to any desired value. The following screenshot shows the location of the scaling options for the viewport in the **PROPERTIES** palette:

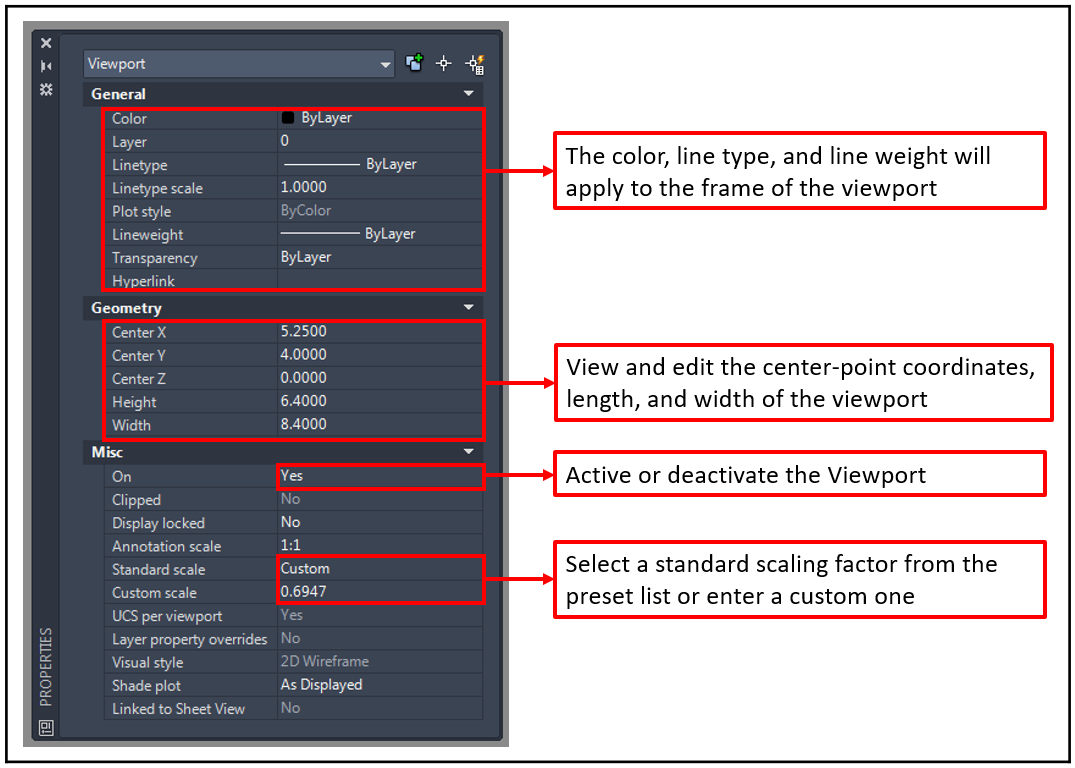


Figure 15.13: The viewport's properties palette

Let's now have a look at a special case, where we want to create multiple views for a three-dimensional model.

Creating orthographic and isometric views for a three-dimensional model

In the case of a three-dimensional model, it is common for the printed drawing to have three main orthographic views and an isometric view for the model. To enable this, you can create multiple viewports and adjust the viewing angle using the first method discussed in the previous section. Double-click inside the viewport then, using the ViewCube, you can change the view to either one of the standard views (which may be the same as the orthographic projections of your model) or a general viewing angle for the isometric view, as shown:

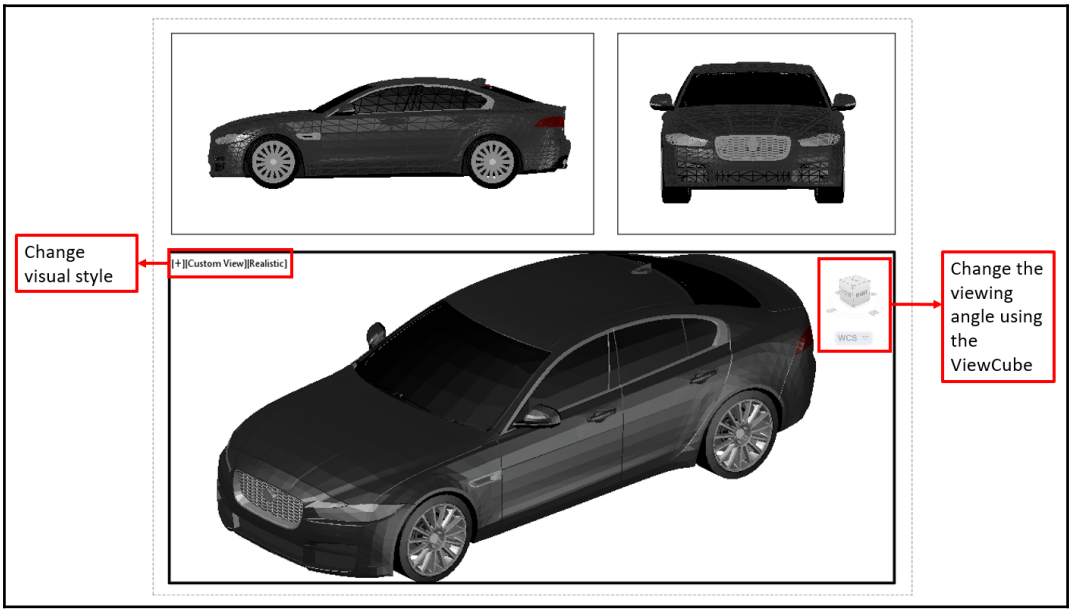


Figure 15.14: The viewports for a three-dimensional model

In addition, the visual style can also be changed for every viewport, as desired, using the controls in the top-left corner, as seen in the previous figure.

We have now created a viewport and adjusted the settings properly. Let's look at another scenario where we may want to create a custom-shaped viewport.

Creating viewports with general shapes

Viewport shapes in AutoCAD are not just limited to rectangular; you can create a viewport with almost any shape using the polygonal viewport command.

To start the `polygonal` viewport command, click on the button named **Polygonal**, found in the **Layout** viewports panel, on the **Layout** tab. Alternatively, you can select the **Polygonal** option that shows up in the command bar when you start the rectangular viewport command, as shown:

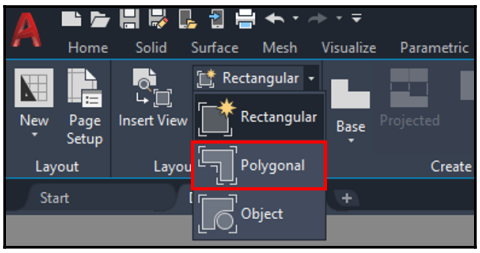


Figure 15.15: The Polygonal viewport tool button's location

The creation of the polygonal viewport is identical to the good old standard polyline, except it will always auto-close the shape. The perimeter of the shape created is divided into connected sections. Each section can be a straight line or an arc section. You specify the starting and ending point of each section and whether it is straight or curved, just like with the polyline command. For more details on the polyline command, please refer to `Chapter 2`, *Basic Drawing Tools*. The following figure shows some examples of the shapes that can be created using the `Polygonal` viewport command:

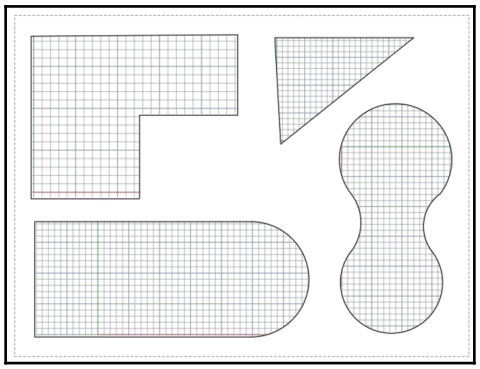


Figure 15.16: Examples of viewport shapes that can be created using Polygonal viewport tool

An alternative method to create a general-shaped viewport is to use the **Object** viewport tool. The **Object** viewport tool button is located in the **Layout** viewports panel on the **Layout** tab. Alternatively, you can select the **Object** option that appears in the command bar when you start the rectangular viewport command, as shown:

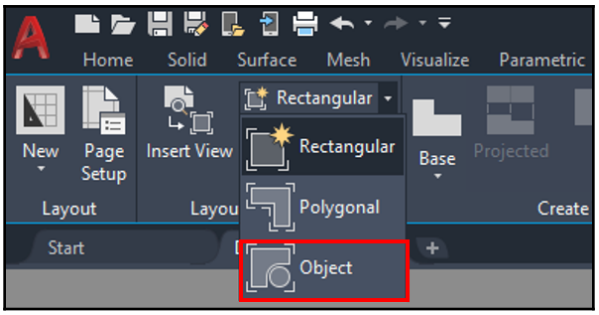


Figure 15.17: The Object viewport tool's button location

After starting the command, you simply click on any closed object in the layout tab (such as a circle, an ellipse, a rectangle, or a closed polyline) and it will be converted to a viewport.

The next step in the drawing preparation process is to draw a title block. In the next section, we will look at how to use fields to create a smart title block.

Creating a title block

As mentioned earlier, all the drawing and editing commands will work normally in the paper space tabs; you can use them to create a frame and a title block for your drawing. The dashed lines showing the printable area will help you set the limits of your border frame. A good practice is to use fields with the text of the title block. Adding that to your template will automatically update the text if it undergoes any changes during the life cycle of the drawing's creation.

An example of this is the date of the drawing. If you have multiple drawings—and they have several versions—it will be a tedious task to always update the date on the drawing and the repeated process of updating often produces mistakes. You can simply add the text of the drawing date as a field, which will update automatically. To do so, type the `Field` command name, then press *Enter*. Alternatively, you can click on the **Field** tool button located in the **Data** panel on the **Insert** tab, as shown:

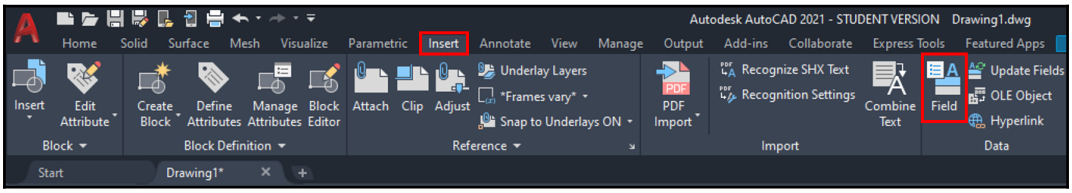


Figure 15.18: The Field tool button's location in the three-dimensional modeling workspace

This will open the **Field** window. Here, you have all the types of field text that can be inserted in the drawing. To get the date field, select **Other** in **Field Category**, select **SystemVariable** on **Field names**, then select **date** from the **System variable** list in the middle, and finally, pick the desired date format and press **OK**, as shown:

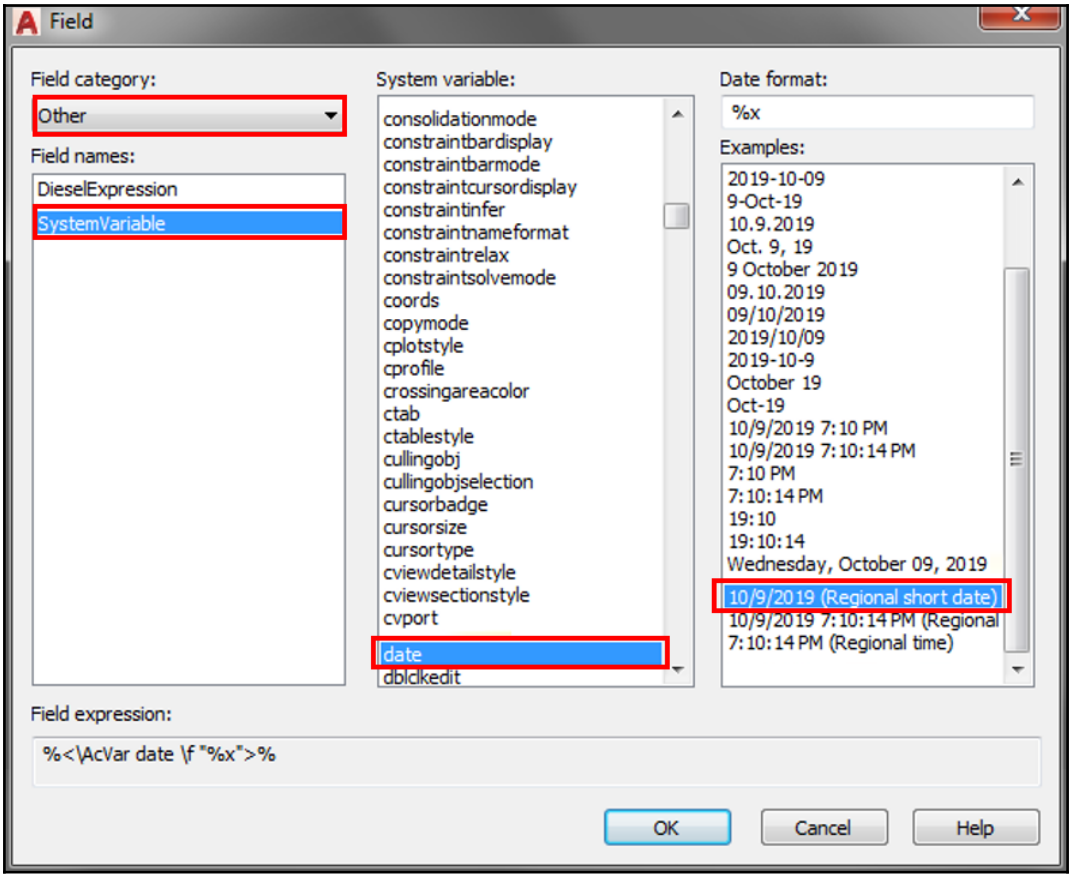


Figure 15.19: The Field window

The **Field** window will close and then you will be prompted to paste the date field text at the desired location. Another example of using fields in the title block is with the drawing name, project name, and drawing author. All this data can be added to the drawing properties by opening the application menu and selecting **Drawing Properties** from **Drawing Utilities**, as shown:

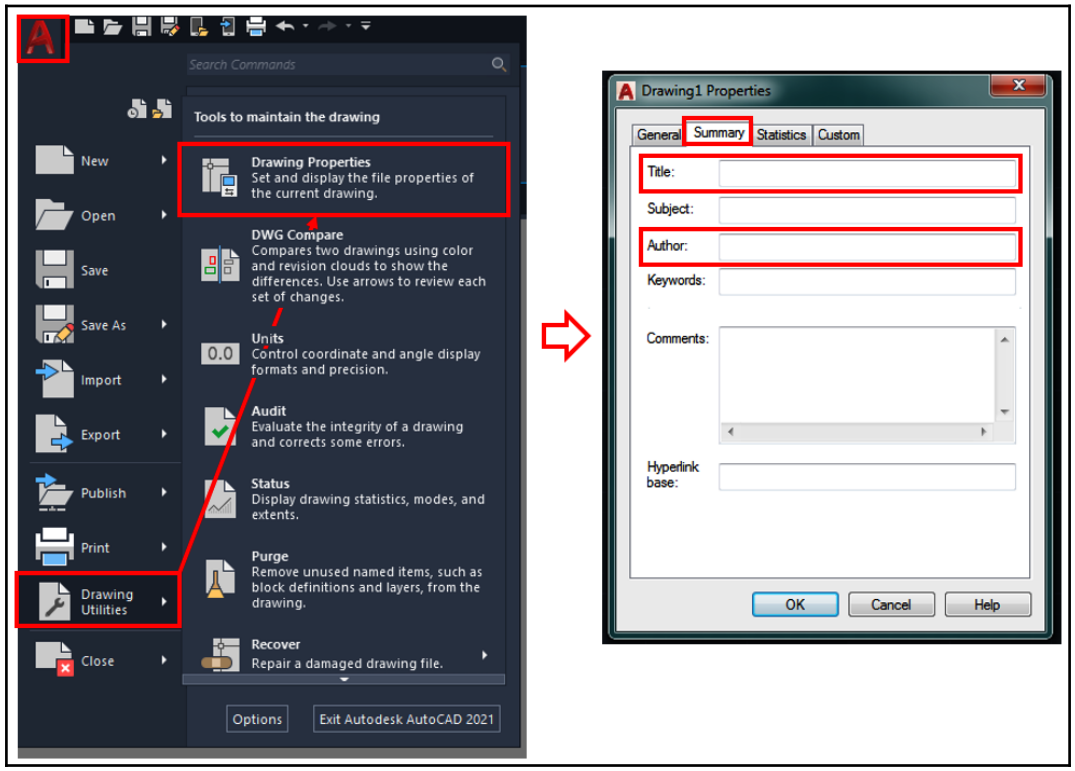


Figure 15.20: Adding a drawing title and an Author name

In case you want to add a property that is not present in the **Summary** tab, such as the project name, you can click on the **Custom** tab, click on the **Add...** button, and type the new property name and the value for it, as shown:

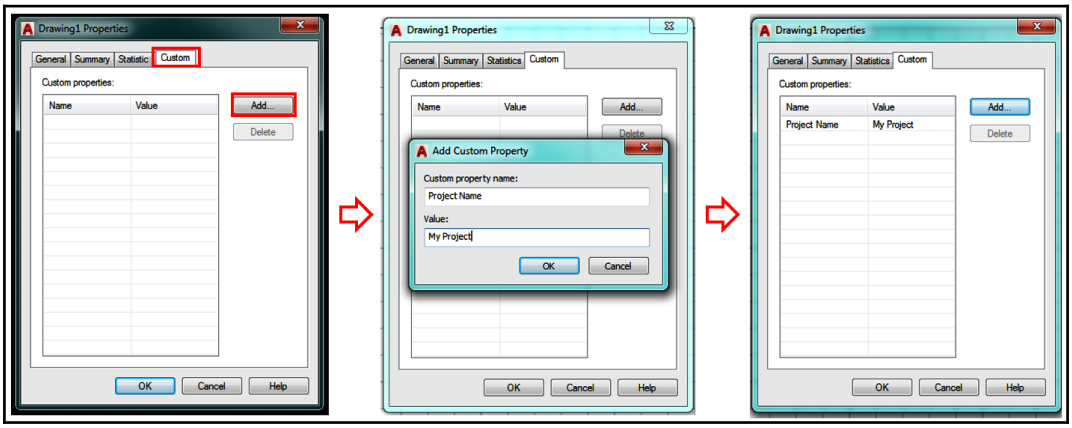


Figure 15.21: Adding a new custom property

After the data is added to the file properties, if you open the **Field** window on **Document** in **Field category**, you will find all the file properties that were present in the **Summary** tab, and the custom properties you have added, such as the project name, as shown:

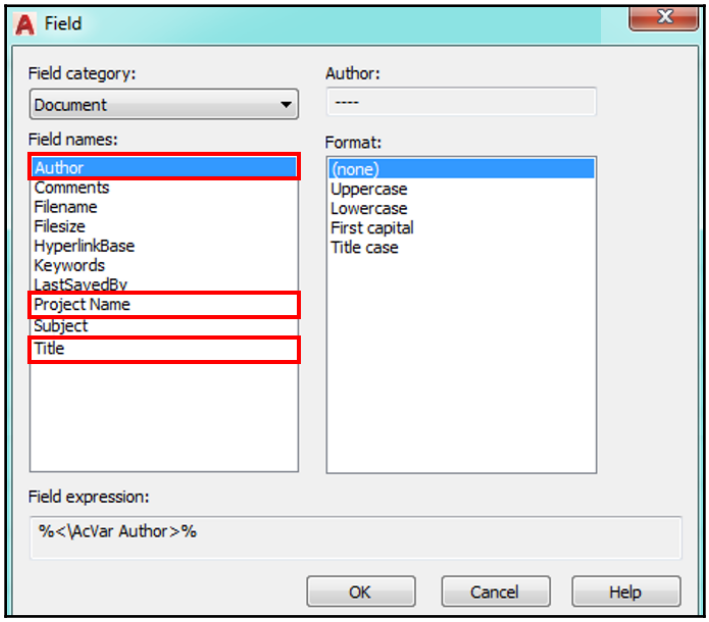


Figure 15.22: Adding fields

You can select the property and set the formatting as desired. Now, if you want to make an adjustment to the author name, for example, it will automatically update in all the layouts. Simply click on the **Update Fields** button.

We have learned about the tools used to create a viewport with any desired shape, how to adjust the viewport settings properly, and how to create a smart title block for our drawing. Let's now look at detailed practical examples for creating and setting viewports for two different scenarios.

Examples for creating and setting layout tabs and viewports

In this section, two practical examples for the process of adjusting the settings of the layout tab, drawing a title block, and creating a viewport with an appropriate scale are shown in detail. The first example is a hammer drawing, while the other is a plan of a small apartment.

Example 1

In this example, we have a drawing for different views of a hammer. The unit for length used in the drawing is millimeters. The general dimensions of the hammer are shown in white in the following figure:

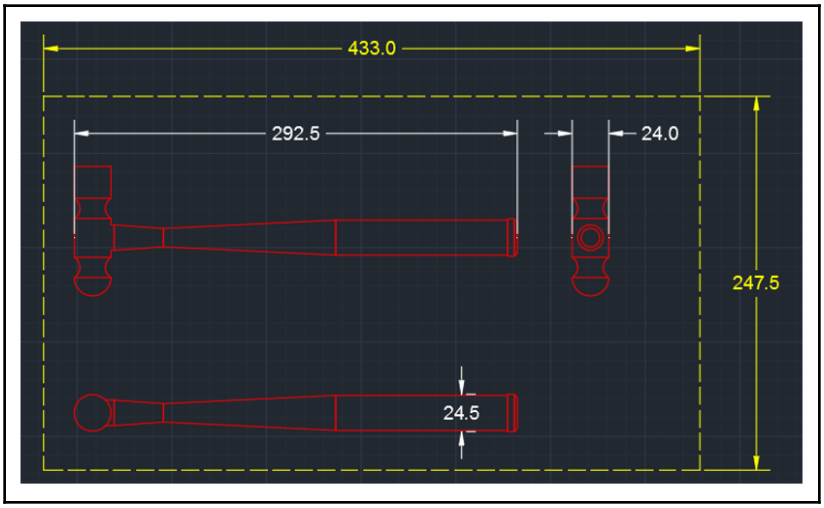


Figure 15.23: Different views and dimensions of the hammer

The first step is to draw an enclosing rectangle, illustrated in yellow in the preceding figure (you can delete it later), and measure it to get the dimensions of the area that will be printed. In the preceding figure, the dimensions are **433** mm x **247.5** mm. An A4-sized paper in landscape orientation has dimensions of 297 mm x 210 mm, so by using a scale of **1:2**, the drawing will fit on an A4 paper's printable area.

The following is a step-by-step tutorial for setting up the layout tab and creating and setting the viewport, starting with the steps for setting up the layout tab:

1. Click on the **Layout1** tab to activate it, then right-click on the tab name and choose **Page Setup Manager**.
2. The **Page Setup Manager** window will show up. Make sure **Layout1** is selected, then click on **Modify** to open the **Page Setup** window.
3. In the **Page Setup** window, set the printer to any of the **AutoCAD PDF** options (normally, you would choose the printer you will use but for now, we will just choose **AutoCAD PDF**). Choose **ISO A4 (297 x 210 MM)** for the paper size, make sure that **mm** is chosen for the units, and set the scaling factor to **1:1**, as shown:

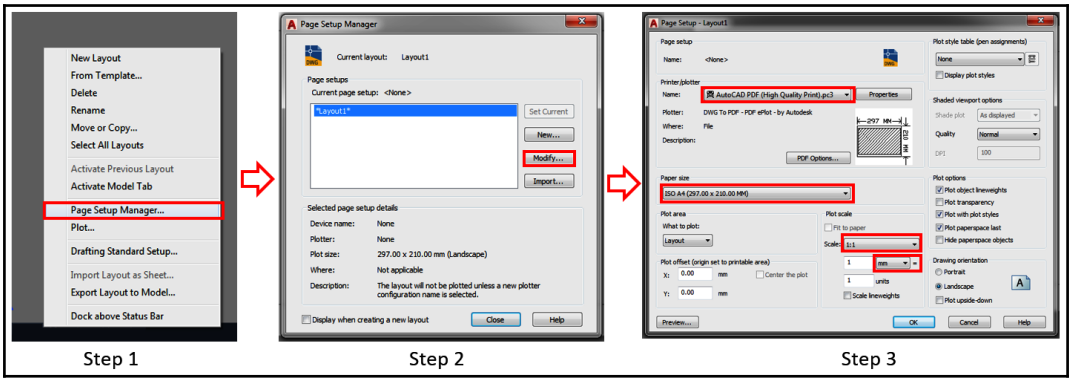


Figure 15.24: Steps for adjusting the layout settings

After setting up the layout tab, you can draw a border frame inside the printable area and add a title block, then create and set the viewport, as in the following steps:

1. Start the `Polygonal` viewport command and draw a viewport, avoiding the area occupied by the title block, as in the the following figure.
2. Select the viewport, click on the small arrow to open the preset standard scale factor list, and select the **1:2** scale.

3. Delete the yellow dashed line and dimensions from the model:

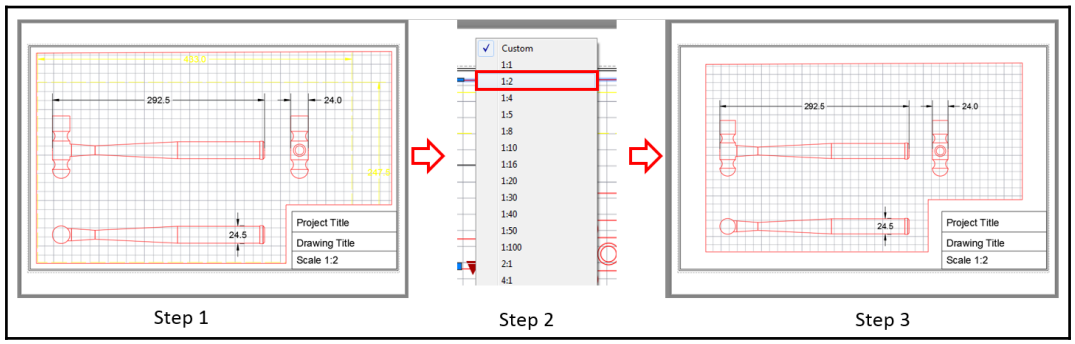


Figure 15.25: Creating and adjusting the viewport

That's it for the first example. Let's now look at the next example, which is a drawing for a small apartment.

Example 2

In this example, we have a drawing for a small apartment. The unit for length used in the drawing is meters and the general dimensions of the apartment are shown in yellow in the following figure:

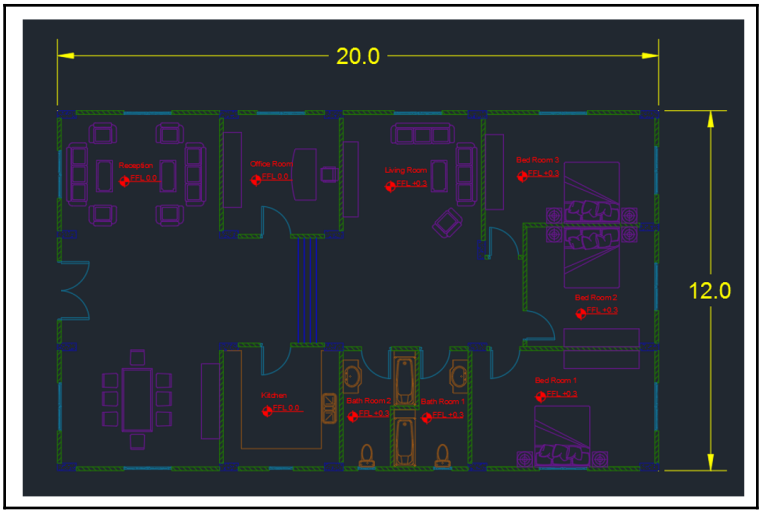


Figure 15.26: The apartment's general dimensions

Start by measuring the dimensions of the apartment, as in the preceding figure. The apartment is **20** m x **12** m. An A1-sized paper in landscape orientation will have dimensions of 841 mm x 594 mm; so, for example, if we use a scale of **1:30**, we will get the following:

The length will be 20 / 30 = 0.666 m

= 666.66 mm

The width will be 12 / 30 = 0.4 m

= 400 mm

This should nicely fit on an A1 paper's printable area.

The following is a step-by-step tutorial for setting up the layout and creating and setting the viewport, starting with the steps for setting up the layout tab:

1. Click on the **Layout1** tab to activate it, then right-click on the tab name and choose **Page Setup Manager**.
2. The **Page Setup Manager** window will show up. Make sure that **Layout1** is selected, then click on **Modify** to open the **Page Setup** window.
3. In the **Page Setup** window, set the printer to any of the **AutoCAD PDF** options. (normally, you would choose the printer you will use but for now, we will just choose **AutoCAD PDF**.) Choose **ISO A1 (841 x 594 MM)** for the paper size, make sure that **mm** is chosen for the units, and set the **1000:1** scale factor.

Notice that the layout scaling factor is set to **1000:1** because the drawing in the model space is in meters and the units in the layout tab are in millimeters, as shown:

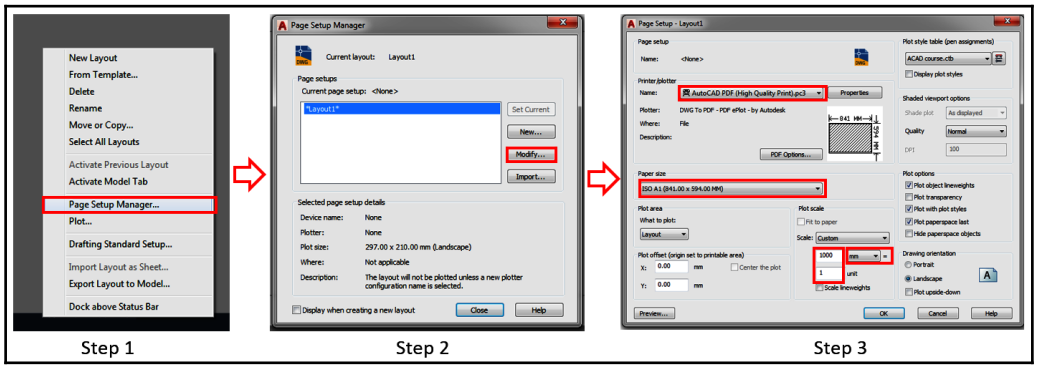


Figure 15.27: The steps for adjusting the layout settings

After adjusting the settings for the layout tab, you can draw a border frame inside the printable area and a title block with a width of 0.12 m, as shown:

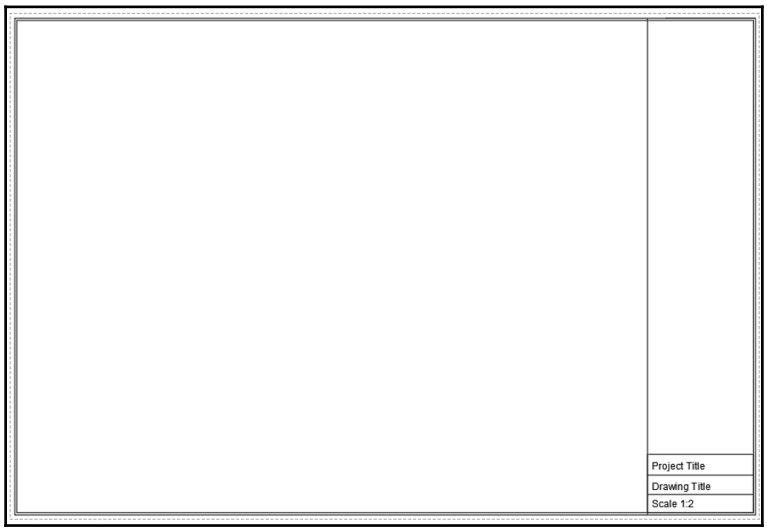


Figure 15.28: An example of a simple title block

The next step is to create and set the viewport, as shown:

1. Start the `Rectangular` viewport command and draw a viewport, avoiding the area occupied by the title block, as in following figure.
2. Select the viewport, click on the small arrow to open the preset standard scale factor list, and select the **1:30** scale.
3. Delete the yellow dimensions from the model:

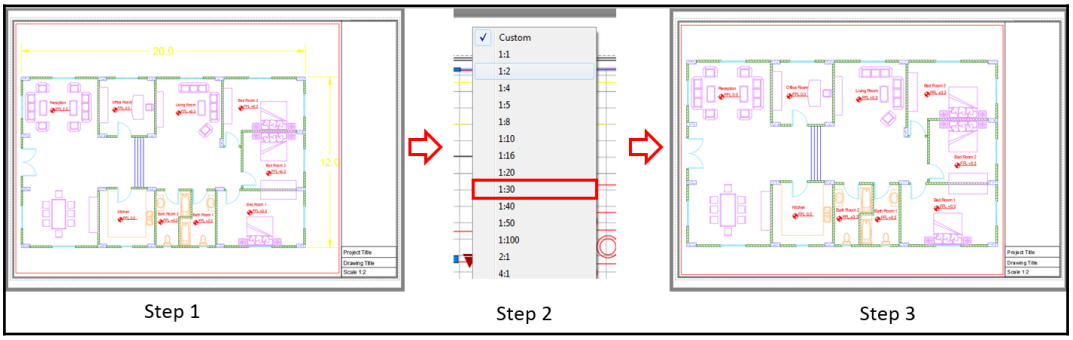


Figure 15.29: Creating and adjusting the viewport

Now that we have completed the preparations needed before printing the drawing, let's take a detailed look at the actual printing process.

Understanding the printing process

After creating and adjusting the settings for the paper space tabs, drawing the title block, and creating the viewports with different shapes as desired, as well as tweaking their settings properly, the drawing is now ready to be printed. The details for using the different settings of the print command are discussed in the following sections.

Starting the print command

To print, you can click on the application menu and choose the **Plot** option on the **Print** submenu, or you can press *Ctrl + P* to initialize the print command, as shown:

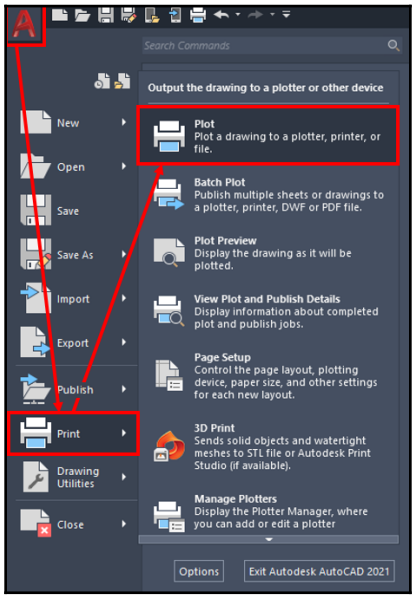


Figure 15.30: The location of the Plot command button

This will open the **Plot** window, which will be discussed in the next section.

Choosing what to print

As seen in the following figure, the **Plot** window is very similar to the **Page Setup** window of the layout tab. In the following screenshot, we will be using the same file used in example 2 of the previous section. Notice that the settings for the printer name, the paper size, and the scale factor are all the same as that of the layout and they can be changed if needed.

On the left side, below the **Paper size** section, you will find the **Plot area** section. This section controls what to plot. If you click on the **What to plot** button, you will find four different options, as shown:

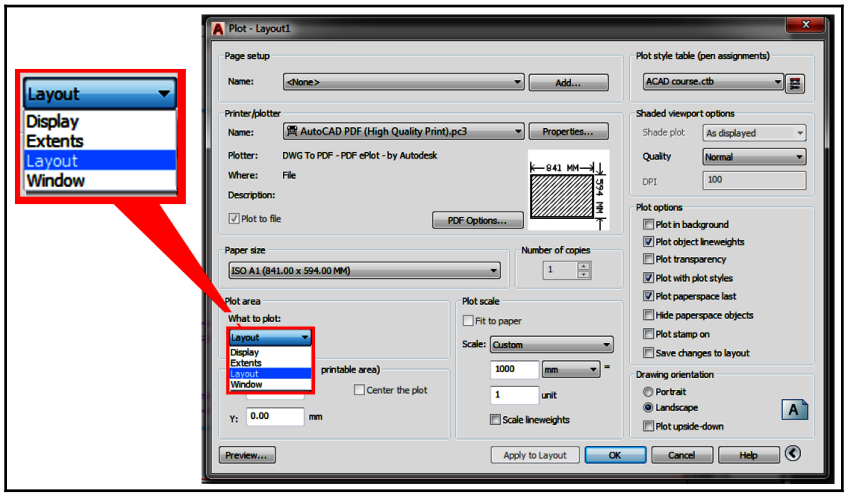


Figure 15.31: Different options for the plotted area

Let's take a more detailed look at these four options:

- **Display**: Choosing this option will limit the printed content to what is displayed on the screen and ignores anything else. For this option to work properly, you need to select the Fit to paper option on the Plot scale section to make sure it puts everything on the screen in the printable area, despite the zoom or the scaling factor.
- **Extents**: This option is mostly used when you are printing from the model space tab. (Nonetheless, it will work fine with the layout tabs.) It will put everything in the active tab (whether this is a paper or model space tab) in the printable area. Again, for this option to work properly, you need to make sure that the **Fit to paper** option on the **Plot scale** section is checked.

- **Layout**: This option will print what is inside the printable area present in the paper preview in the **Layout** tab. This is the option to use when you want to print with a specified scale factor.
- **Window**: When you choose this option, you will be prompted to specify a rectangular area by specifying two opposite corners and what is inside that specified rectangle will be printed. This option can be used for both the model space and paper space tabs, and it is preferred to use it with the **Fit to paper** option to keep the scale factor.

In the bottom-left corner, you can find the **Plot offset** section, where you can adjust the origin coordinates of the printable area, or you can simply set it to be centered by checking the **Center the plot** option. Below the **Plot offset** section, you will find the **Preview...** button, which is used to preview the plot before sending it to the printer or plotter. The location of the **Preview...** button and the **Plot offset** settings is shown here:

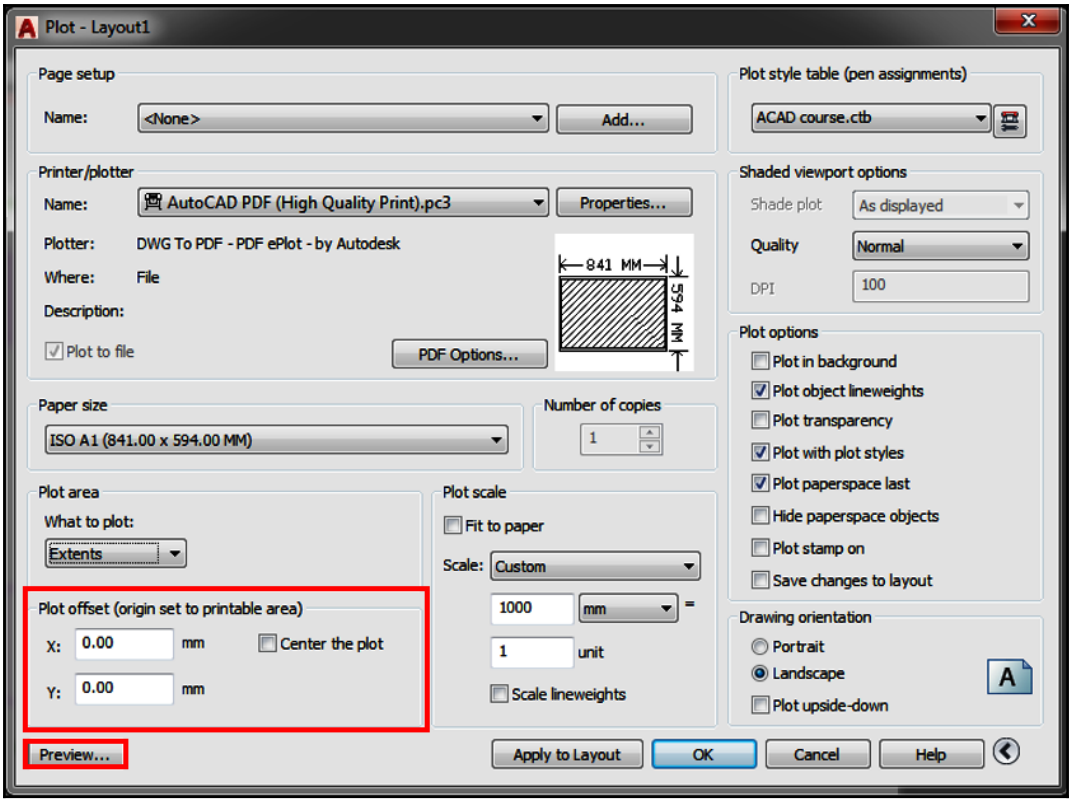


Figure 15.32: The Plot offset setting and Preview buttons' locations in the Plot window

We have chosen what to print and adjusted its position on the printed paper. Let's now learn how to control the printed line weights and colors for every line in our drawing using plot styles.

Using plot styles

In the model space, each line or curve will have a set of properties, such as the color and the line weight. You can print the lines and curves using the same colors and line weights already available in the model space, or you can customize how each line appears in the printed drawing. In the top-right corner of the **Plot** window, you will find the **Plot style table (pen assignments)** section. This section is used to control the pen assignments of different lines in the model.

A common example is the use of different colors for different objects in the model inside AutoCAD (to improve visibility while working on the model) while planning to only use the black color (or greyscale) in the printed drawings. That is very common in many of the architectural and MEP drawings.

By default, AutoCAD will use the same object color and line weight in the printed drawings. This setting is called the **acad** plot style. There are other preset plot styles, such as the following:

- **Monochrome**: This one will use black for everything in the printed drawings.
- **Grayscale**: This will use grayscale to represent different colors in the drawing.

The following screenshot shows the **Plot style table** option's location:

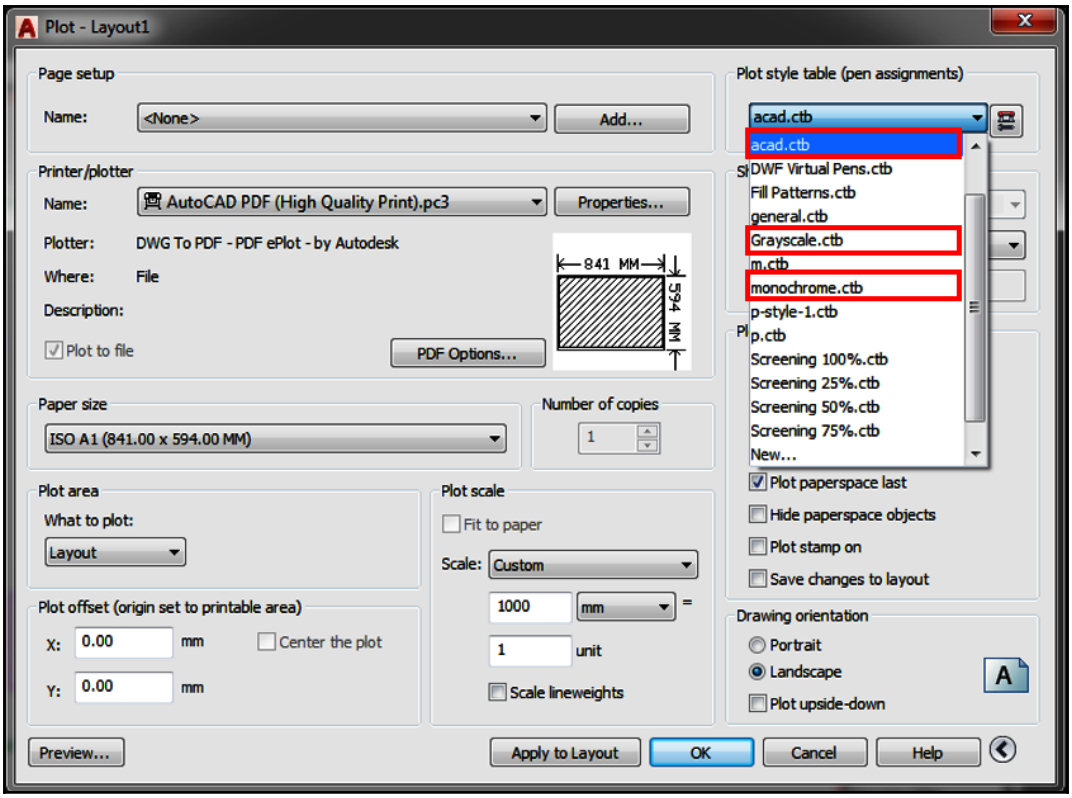


Figure 15.33: Preset plot styles

In addition to the preset plot styles, you can create a new one or modify an existing one. To create a new plot style, follow the steps given here:

1. Open **Plot styles list** and click on **New...**.
2. This will start the four steps: **Add Color**, **Dependent**, **Plot Style**, and **Table**. Starting with a window called **Begin**, you can start a new plot style from scratch or you can browse to an existing CFG, PCP, or PC2 file. For this example, choose to start from scratch and press **Next**:

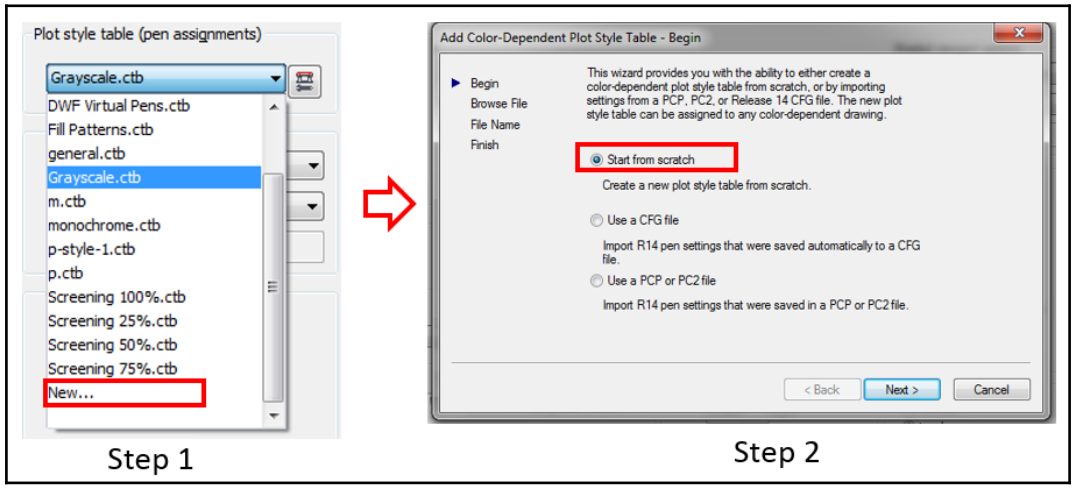


Figure 15.34: The steps for creating a new plot style (1/4)

3. The second step, **Browse File**, will be skipped (but not if an existing file in the previous step was selected). The **File name** option will appear. You can type the desired plot style name, then press **Next**.

4. Now, the plot style is created. By default, it will be set as the `acad` preset plot style. To customize how different colors are printed, click on **Plot Style Table Editor...**:

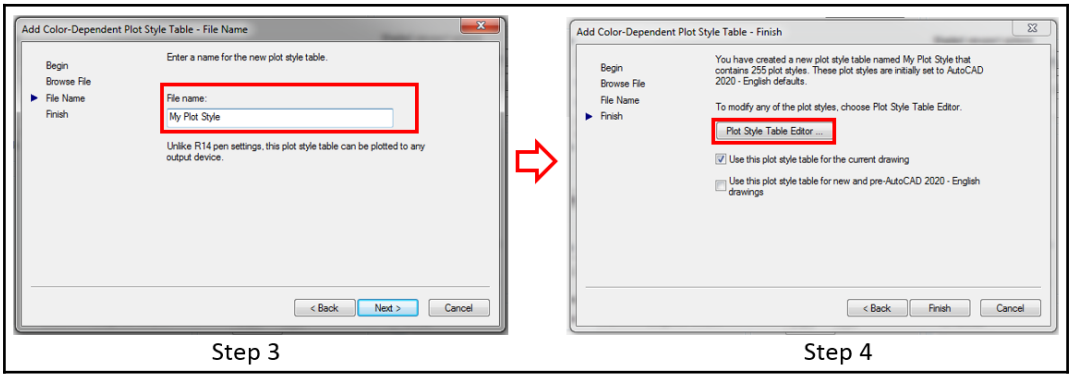


Figure 15.35: The steps for creating a new plot style (2/4)

4. This will open the **Plot Style Table Editor** window. On the left, you will find a list of all the available object colors in AutoCAD. For each color in the list, you can customize the properties of how this color will be printed; for example, you can use the same object color or you can use a different one, or you can also control line weight and style:

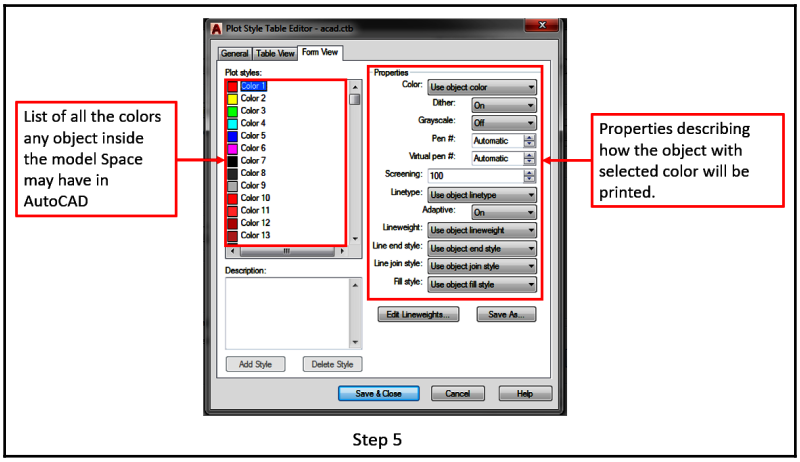


Figure 15.36: The steps for creating a new plot style (3/4)

5. After adjusting these settings, for each color click on **Save & Close**, then click **Finish**, as in the following figure. The plot style will be saved and you can use it for printing in the project in hand or for future projects:

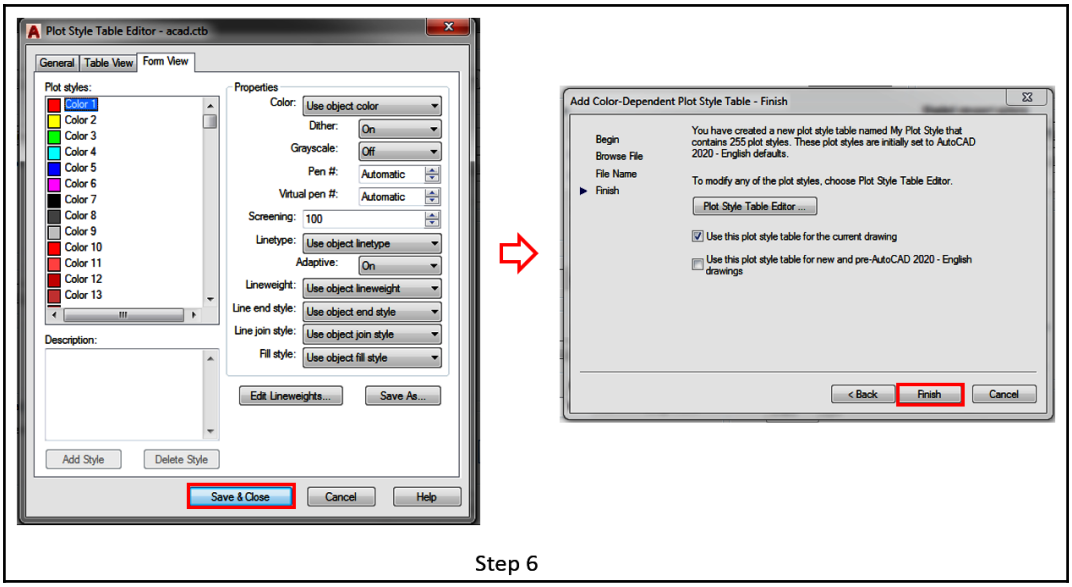


Figure 15.37: The steps for creating a new plot style (4/4)

Let's now have a look at two simple, practical examples for using the plot style table.

Examples for using Plot Style Table Editor

In this section, we will create a plot style for two simple practical examples. The first one is example 1 from the *Examples for creating and setting layout tabs and viewports* section. The second one is for a small classroom.

Example 1

In this example, we will set the plot style for printing the drawing from the first example in the *Examples for creating and setting layout tabs and viewports* section. The following figure shows three views for a drawing of a hammer. The hammer is drawn in red, while the dimension lines are set to the 255 color:

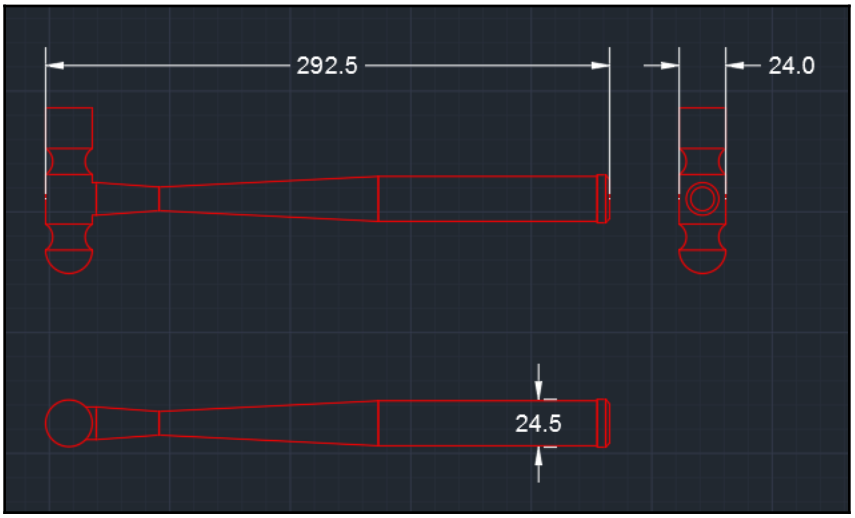


Figure 15.38: Dimensions of the hammer

Let's say that we want to print the hammer in black and the dimension lines in red. So, the first step is to open the **Plot** window, then click on the **Plot styles** list, and select **New**. Name the new plot style `Hammer`, as shown:

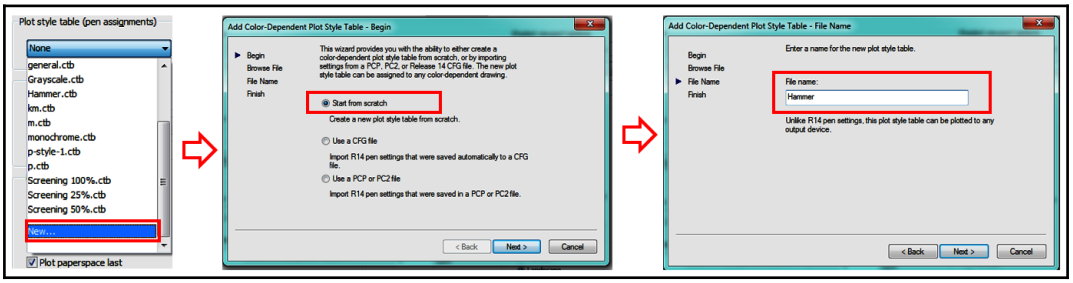


Figure 15.39: Creating a new plot style

Click **Next**, then open **Plot Style Table Editor**. In the **Plot Style Table Editor** window, select the color red from the **Plot styles** list and set the color on **Properties** to **Black**. After that, scroll down to the bottom of the **Plot styles** list and select **Color 255**, then set the color on **Properties** to **Red**, as shown:

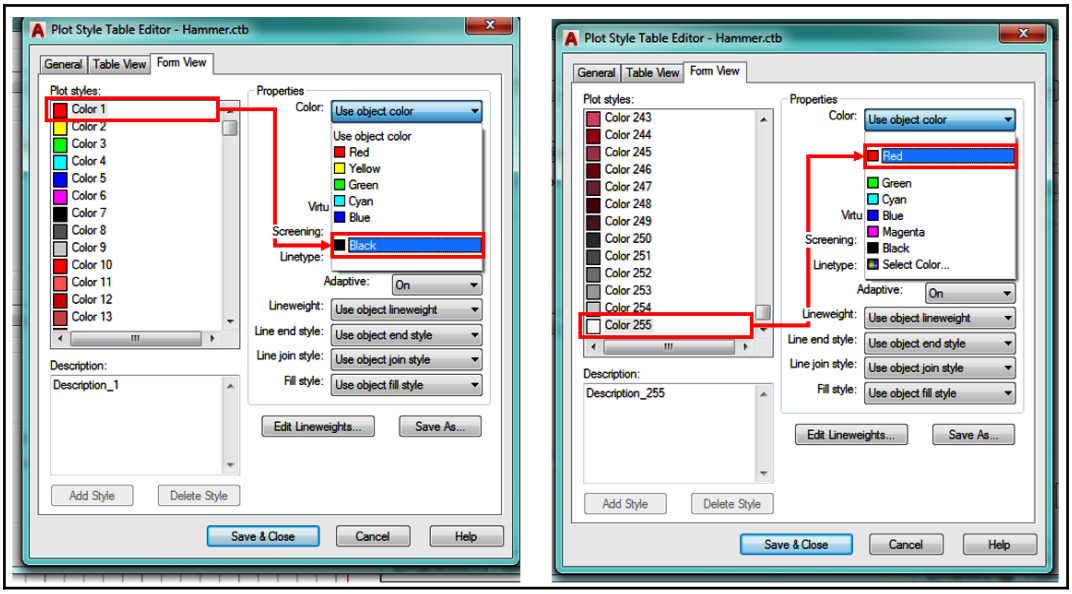


Figure 15.40: Adjusting the line colors

Click **Save & Close** to close **Plot Style Table Editor**, then click Finish to end the plot style creation wizard. Now, if you print or preview the drawing, it should look like this:

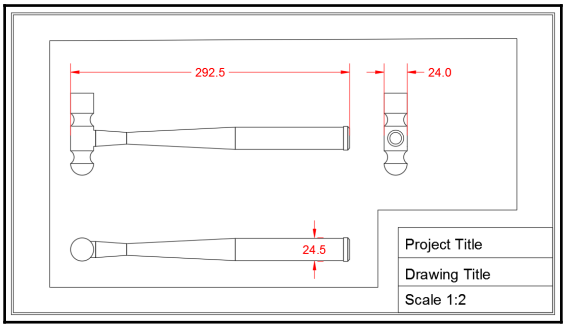


Figure 15.41: A preview of the created plot style

And that's it for the first example; let's now look at the second example.

Example 2

The following is an example of setting the plot style for a classroom plan that is supposed to be printed using only the black color. As you can see in the following figure, the drawing includes the furniture (the chairs and the teacher's desk) in purple, the floor tiles in yellow, and a small stage for the teacher in red:

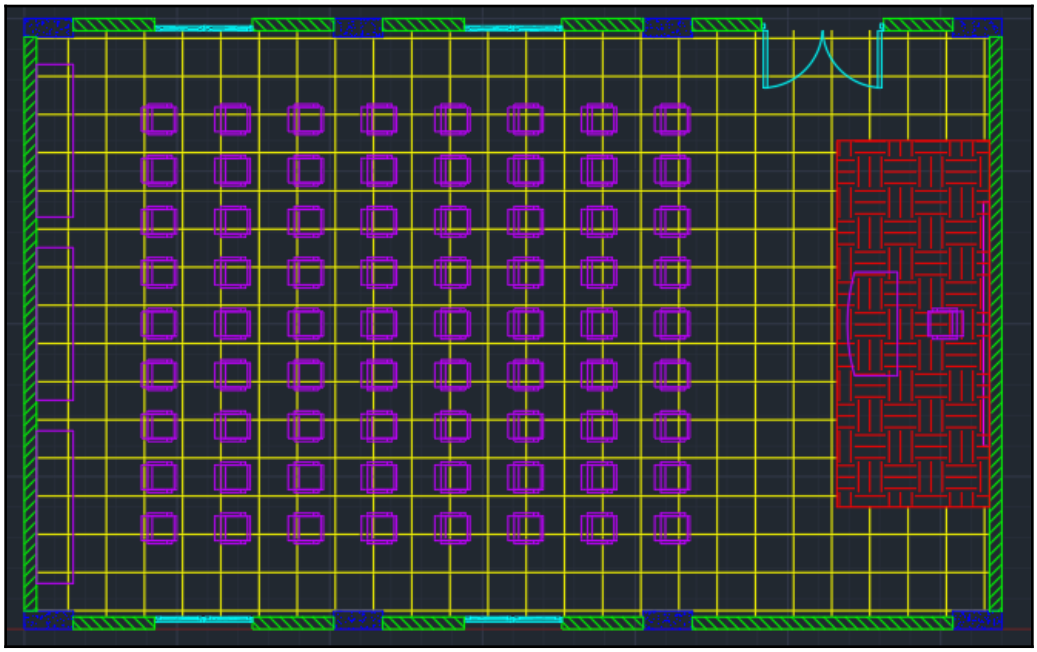


Figure 15.42: A plan view of a classroom

As we said, we want to print it using only black, so we can simply set all the colors in the plot style to print in black (or just use the monochrome plot style, for example). But if all the drawing components are printed using the same line thickness, as in the following figure, the intersections will be very distracting and some of the details, such as the teacher's table, may become barely visible:

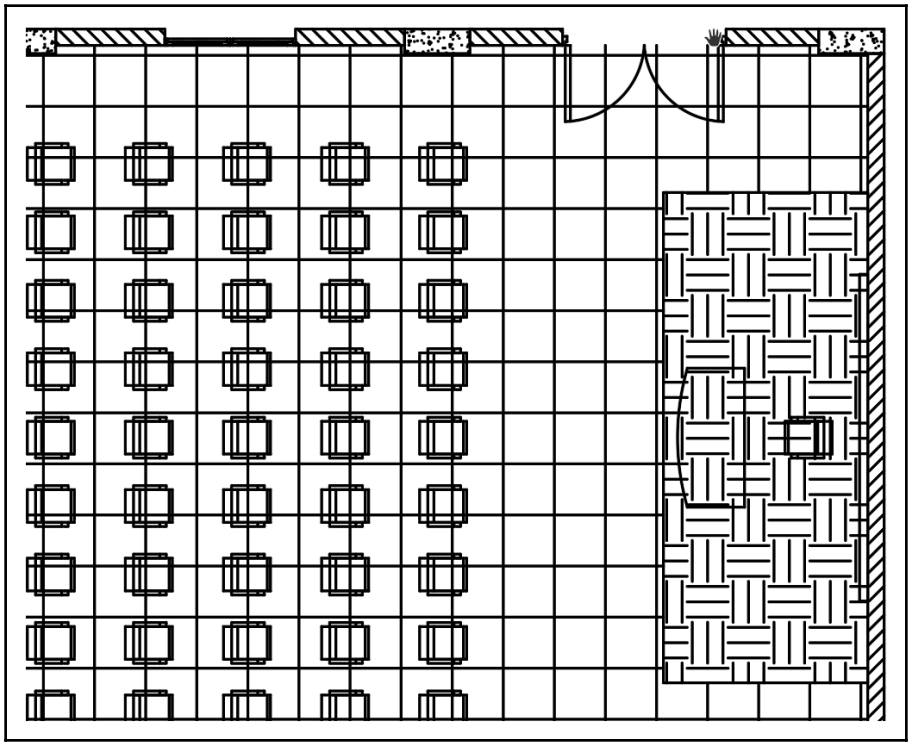


Figure 15.43: A preview of the monochrome plot style

To fix that, we will set the plot style to print the tiles using a lighter line weight than that used for the furniture and the walls. To do so, open the plot style list and select **New...**. Give a name to the plot style (for example, `Classroom`) and click **Next**, as shown:

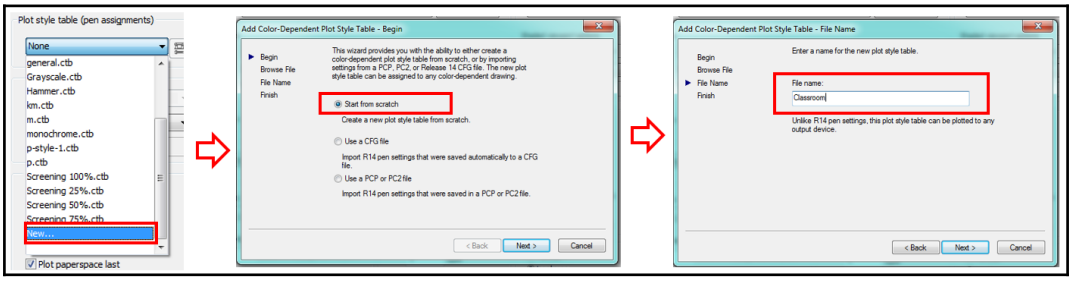


Figure 15.44: Creating a new plot style

Click on **Plot Style Table Editor** and the first step now is to set all the colors in the model to print in black. To do so, select all the colors in the **Plot styles** list (to quickly do this, select the first color then press *Shift* + *End*), then set the printing color on **Properties** to **Black** and the line weight to **0.2500 mm**. Now, the next step is to make the tiles have a thinner line weight. So, select yellow and red and set the line weight to **0.0500 mm**, as shown:

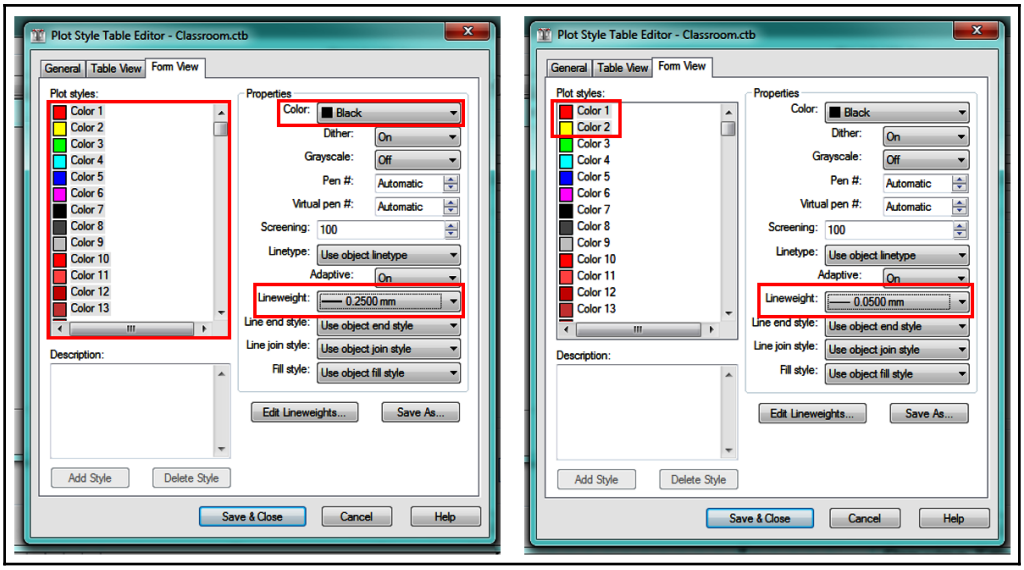


Figure 15.45: Adjusting line weights and colors

Click on **Save & Close** and click **Finish** to end the plot style creation. Now, the output will look as in the following figure. Notice the lighter line weight for the tiles and the improved visibility of the furniture, as shown:

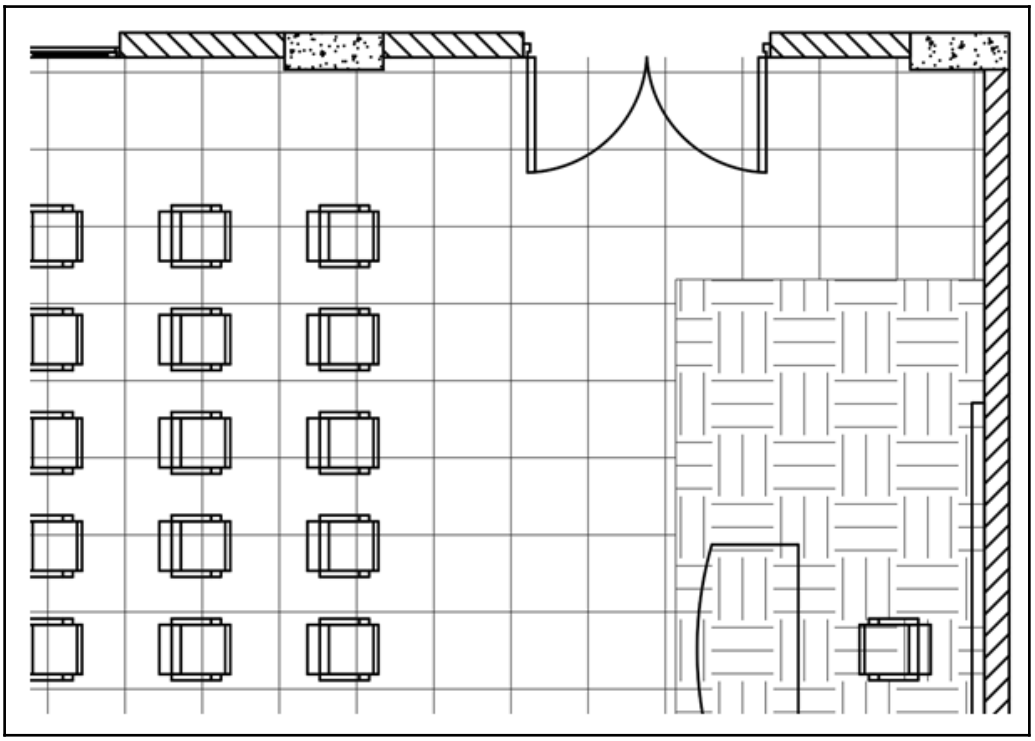


Figure 15.46: A preview of the created plot style

That was a simple example of using the different available tweaks in the plot style table. You can also use multiple line weights instead of only two, as in the previous example, and you can also assign different line styles to different objects in the model. This brings us to the end of the second example.

Summary

The necessary preparations that have to be done before printing a drawing were covered in this chapter, starting with an introduction to paper space, how to create and adjust the settings properly for layout tabs, how to draw a title block, and how to create and set viewports properly. We then looked at two practical examples for creating and setting the scale and shape of a viewport.

After that, we moved on to the actual printing command. We learned about different modes for what to print and how to set the pen assignments to control how every line in the drawing will be printed, with examples of how to set the plot style table for two scenarios.

In the next chapter, we will learn about another type of output in AutoCAD and more about three-dimensional modeling, which is preparation and production of rendered images.

16
Rendering and Presentation

High-quality rendered images are the main ingredient of a good 3D modeling project presentation, and to produce good quality renders, there are a number of conditions and settings that need to be adjusted properly. It is like setting the scene before taking the picture. In this chapter, the following scene-setting aspects will be covered, each in a dedicated section:

- Adjusting lighting conditions
- Displaying shadows
- Applying materials to the components of the 3D model
- Creating cameras

After the scene is set, we will cover the details of creating a rendered image, and after that, we will take a quick look at exchanging other file formats for 3D models.

Setting the lighting of the scene

An essential part of creating a scene is to set the lighting properly, because different lighting conditions can produce totally different results. In AutoCAD, there are different types of light sources, and you can add one or multiple light sources of these different types to the model to set the lighting as desired. In this section, we will start by learning how to set the lighting intensity units in AutoCAD, and then we will explore the different light sources that are available in AutoCAD. Each type will be covered in a dedicated sub-section. We will learn how to create and properly adjust their settings, and we will look at the differences between them. Let's now begin with adjusting the lighting intensity units.

Lighting intensity units

Before working with light sources, the first step is to properly adjust the lighting intensity units so that the light source will behave as expected. AutoCAD has two light source intensity units. The first is based on the system international unit, and the other is based on the American system.

To change the lighting system intensity, click on the **Application Menu** button, and in the **Drawing Utilities** sub-menu click on **Units**. This will open the **Drawing Units** window, where you can find the lighting intensity units in the section called **Lighting**, as shown in the following screenshot:

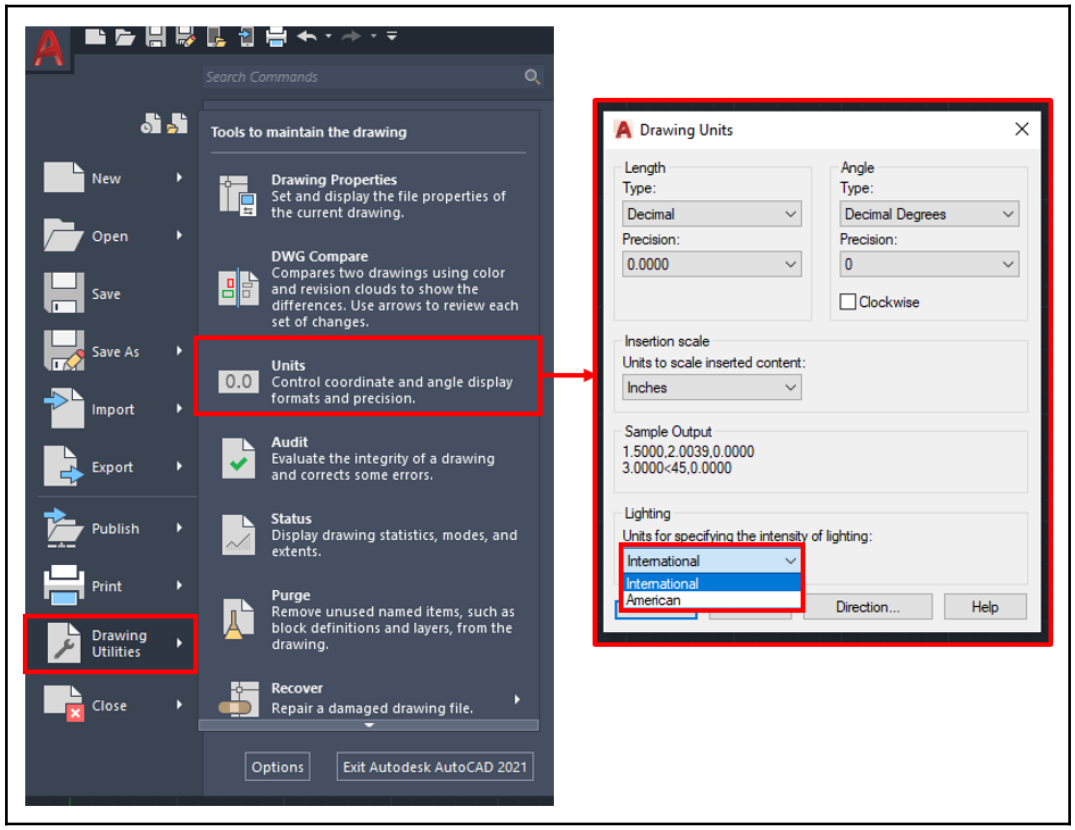


Figure 16.1: Setting the lighting intensity units

Now that we have set the lighting intensity units, we are ready to begin creating light sources in our model, and we will start with the first light source type, which is the Point light source.

Creating Point light sources

The first type is called the Point light source. This light source is basically a point in the model space that emits light in all directions. You can use this type for general lighting conditions.

The **Point** light tool button can be found in the **Lights** panel under the **Visualize** tab in the **3D Basics** workspace, as shown in the following screenshot. An alternative method is to type the command name `POINTLIGHT` (or just type `POI`) then press *Enter*:

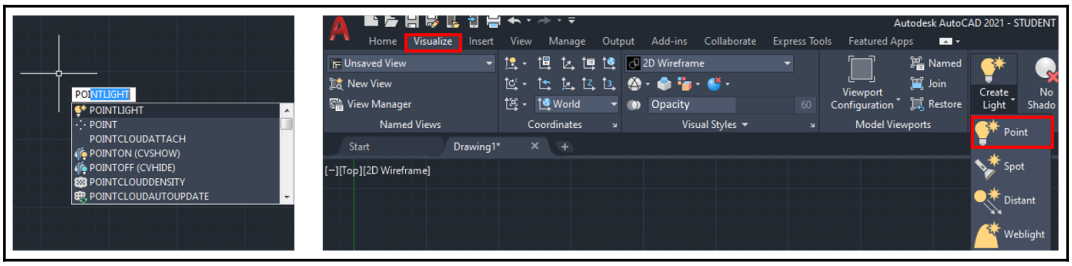


Figure 16.2: Alternative methods to start the Point light source command

If you are using the **3D Modeling** workspace, you can find the **Point** light tool button in the **Lights** panel on the **Visualize** panel.

You can create a Point light source as follows:

1. Start the `Point light` source command.
2. Use the mouse cursor to specify the point source location. Alternatively, you can type the coordinates of the location using the keyboard and then press *Enter*. The upper-end point of the vertical line is selected, as shown in the following screenshot:

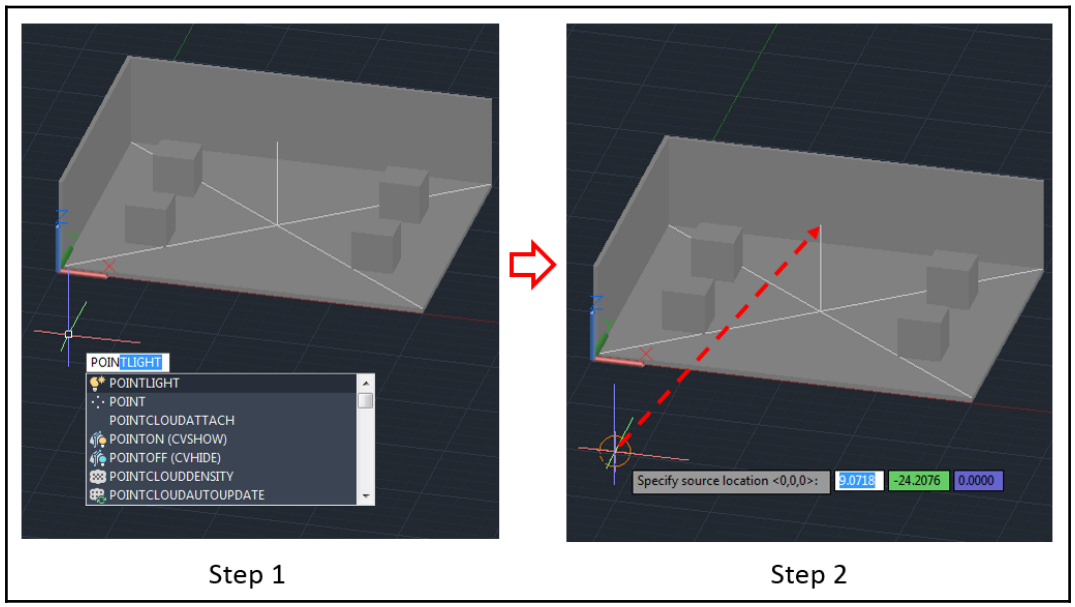


Figure 16.3: Steps for creating a Point light source (1/2)

3. After specifying the location, a settings menu will appear. These different settings will be discussed after this example. To create the light source using the default settings and end the command, click on **eXit**.

   The command terminates and the light source is created. You may have noticed that light is cast evenly in all directions, as shown in the following screenshot:

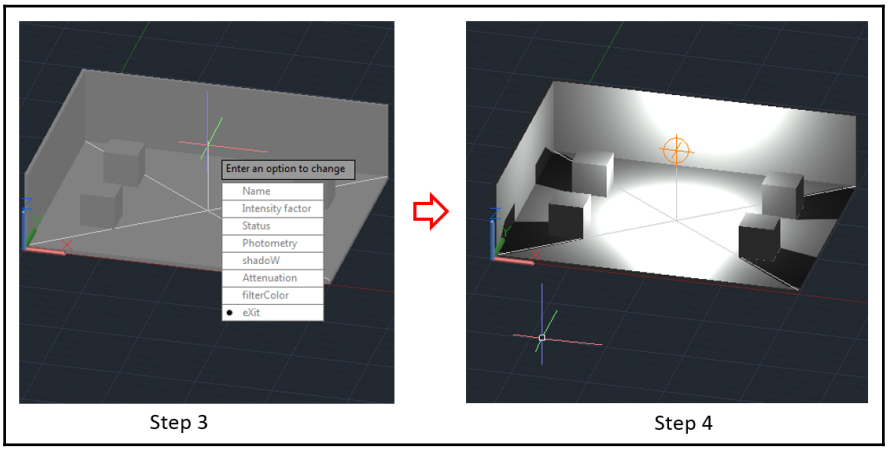


Figure 16.4: Steps for creating a Point light source (2/2)

As shown in the previous example, an option menu containing eight options for the light source appears in the third step just before the command ends, and they are as follows:

- **Name**: Click on this option to specify a name for the created light source; otherwise, AutoCAD will give default names to the light sources created, such as **Pointlight1** and **Spotlight2**.
- **Intensity factor**: This option sets a factor that ranges from **0** to **1** regarding the intensity of the light source.
- **Status**: This option determines whether to turn the light source on or off after it is created.
- **Photometry**: This option adjusts the settings for the intensity and the color of the light source. There are three different options for intensity. You can either set the intensity value in **Cd** (Candelas), use the perceived power with a **luminous flux** value (Lm), or specify the intensity using an **illuminance** value (Lx|Fc). As for adjusting the light source color, there are also three options. You can either choose one of the preset lamps, type a color name, or specify the color temperature in Kelvins.
- **shadoW**: This option specifies whether or not the light source will cast shadows. It also has three options for the shadow softness.
- **filterColor**: This option adds a color filter to the light source. The resulting color will be a combination of the source color and the filter color.
- **eXit**: This option ends the command.

The intensity and the color of the created light source can be modified by selecting the light source and right-clicking and opening the **PROPERTIES** palette (alternatively, you can select the light source, type `PR`, and then press *Enter*). In the **Photometric properties** section of the **PROPERTIES** palette, the **Lamp intensity** option can be used to view and modify the light source intensity. To modify it, just click on the intensity value, type the new desired value, and press *Enter*, as shown in the following screenshot:

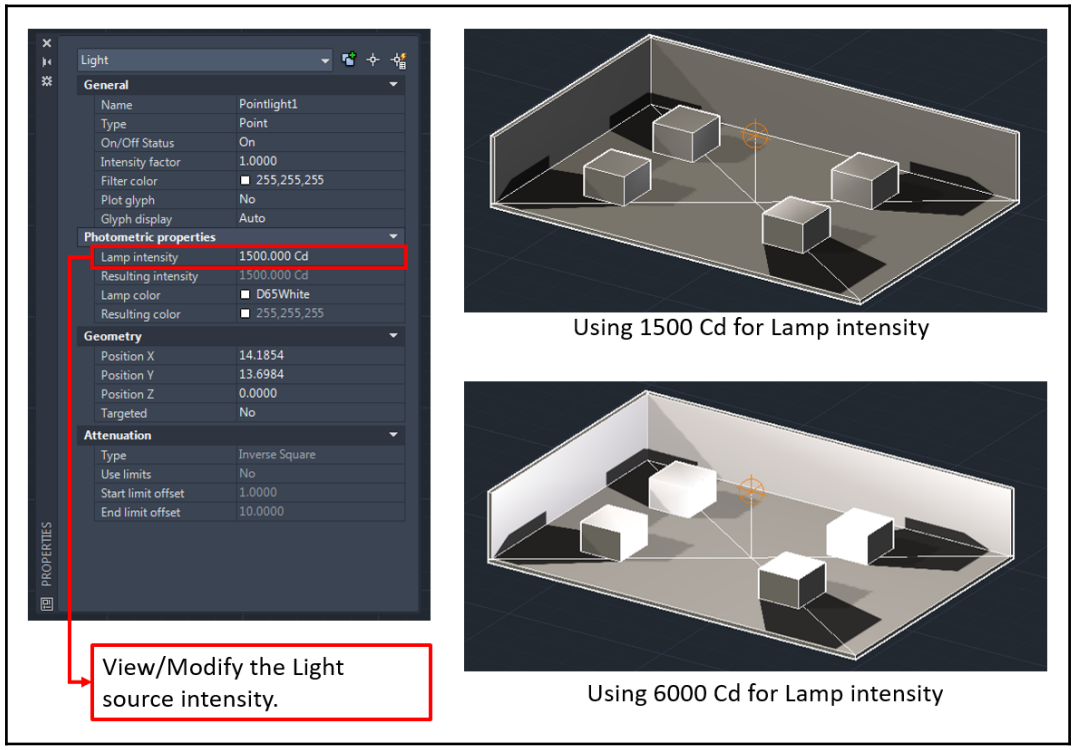


Figure 16.5: Adjusting the intensity of a Point light source

The light source color can also be viewed/modified using the **Lamp color** option in the **PROPERTIES** palette. Click on the color and the **Lamp Color** window will open, where the light source color can be selected from a list of available standard lamps. In addition, a filter color can be added to the light source, and a preview of the final color is shown in the following screenshot:

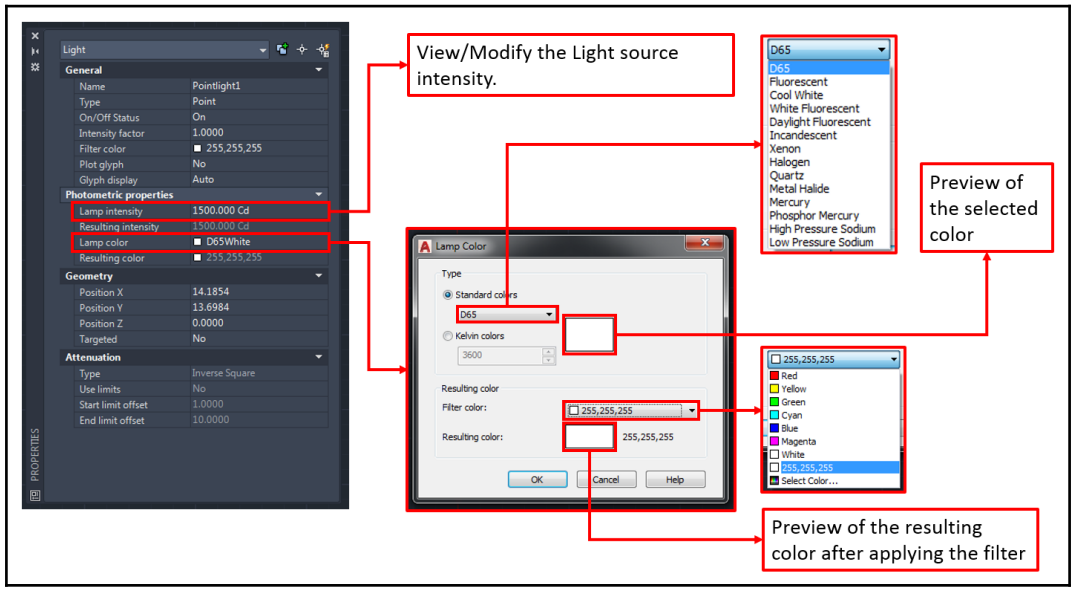


Figure 16.6: Adjusting the color of a Point light source

Let's now have a look at the second light source type, which is the Spotlight.

Creating a Spotlight source

The next light source is called the Spotlight. As the name suggests, it is used to create a light source that emits a cone of light in a specific direction, similar to a follow spot used in theaters. The Spotlight tool button can be found on the **Lights** panel within the **Visualize** panel in the **3D Basics** workspace, as shown in the following screenshot. An alternative method is to type the command name, `SPOTLIGHT`, (or just type `SPO`) and then press *Enter*, as shown in the following screenshot:

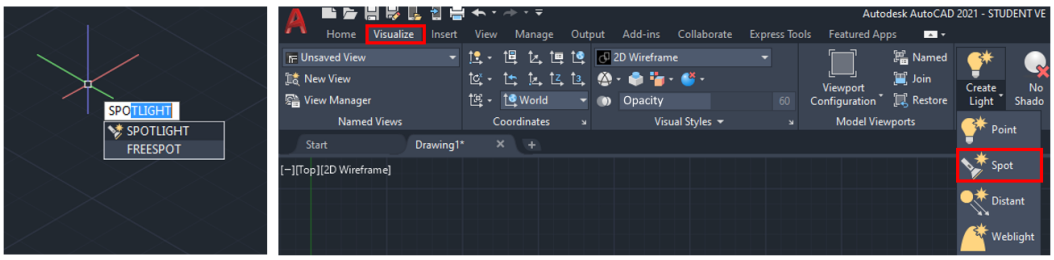


Figure 16.7: Alternative ways to start the Spotlight command

If you are using the 3**D Modeling** workspace, you can find the Spotlight tool button in the **Lights** panel within the **Visualize** panel.

A Spotlight source can be created as follows:

1. After starting the Spotlight source command, specify the location of the Spotlight using the mouse cursor. Alternatively, you can type the coordinates of the location using the keyboard and then press *Enter*. In the example in the screenshot, the upper-end point of the vertical line is selected.
2. Use the mouse cursor to specify the direction of the Spotlight. In the following screenshot, the corner point will be selected so that the Spotlight is facing that corner:

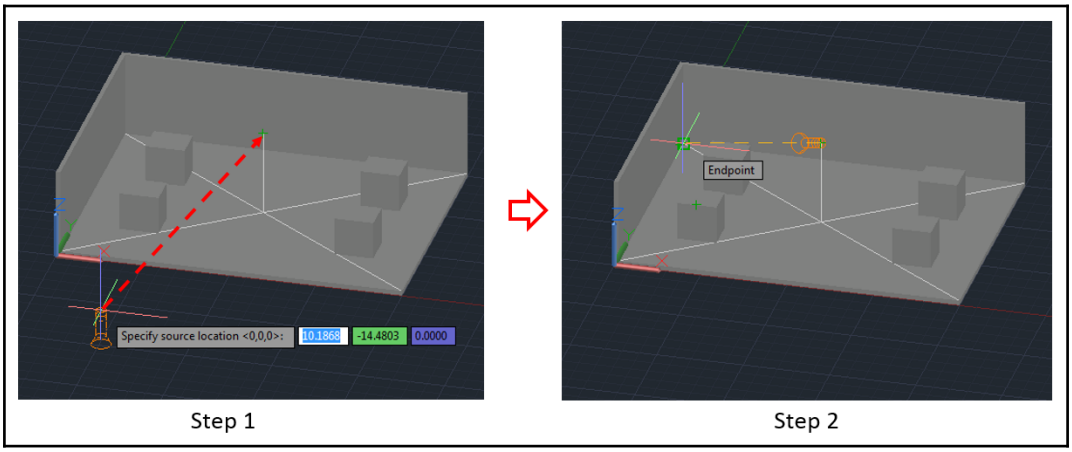


Figure 16.8: Steps for creating a Spotlight source (1/2)

3. After specifying the location, a settings menu will appear. To create the light source using the default settings and end the command, click on **eXit**.

The command terminates and the light source is created. Light is cast as a cone originating from the Spotlight location, as shown here:

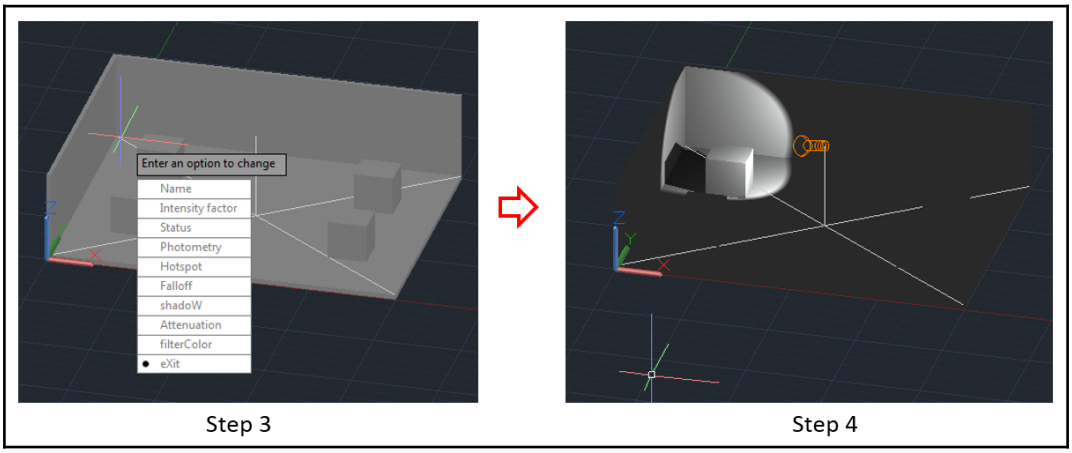


Figure 16.9: Steps for creating a Spotlight source (2/2)

As shown in the previous example, an options menu containing ten options for the light source appears in the third step just before the command ends. The options are the same as those discussed in the previous section, with the addition of two more that are specific to Spotlights, which are **Hotspot** and **Falloff**. These two options are for adjusting the Hotspot and Falloff angles of the Spotlight.

If you select the Spotlight source, you will find a number of grab (control) points that can be categorized into four types:

- Along the Spotlight direction line, you will find two points. These are **Position** and **Target** grab points, which can be used to modify the location and direction of the Spotlight.
- On the outer side of the Spotlight direction line, you will find two outside pointing arrows. These can also be used to change the position and the target of the Spotlight, but change is limited to along the Spotlight direction line.
- On the light cone's outer circle, you will find four small arrows called the **Falloff** grab points. They can be used to modify the outer light cone angle.

- On the light cone's inner circle, you will find another four small arrows called the **Hotspot** grab points. These can be used to change the light cone Hotspot angle. The four grab points are highlighted in the following screenshot:

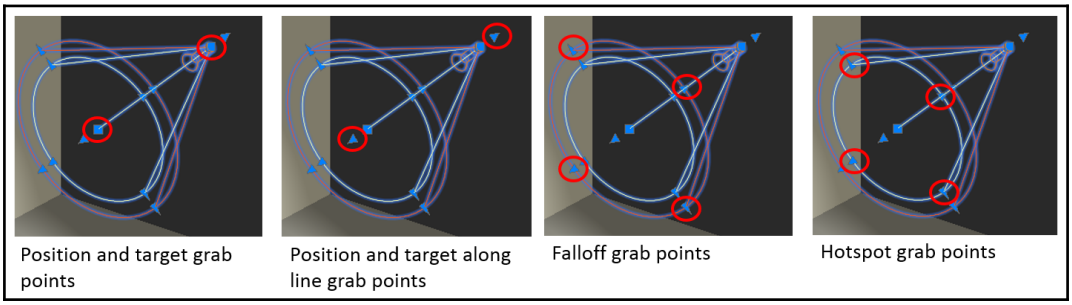


Figure 16.10: Grab points of a Spotlight source

As previously mentioned, **Position** and **Target** grab points can be used to quickly modify the location of the Spotlight and the location of the target point, as shown in the following screenshot:

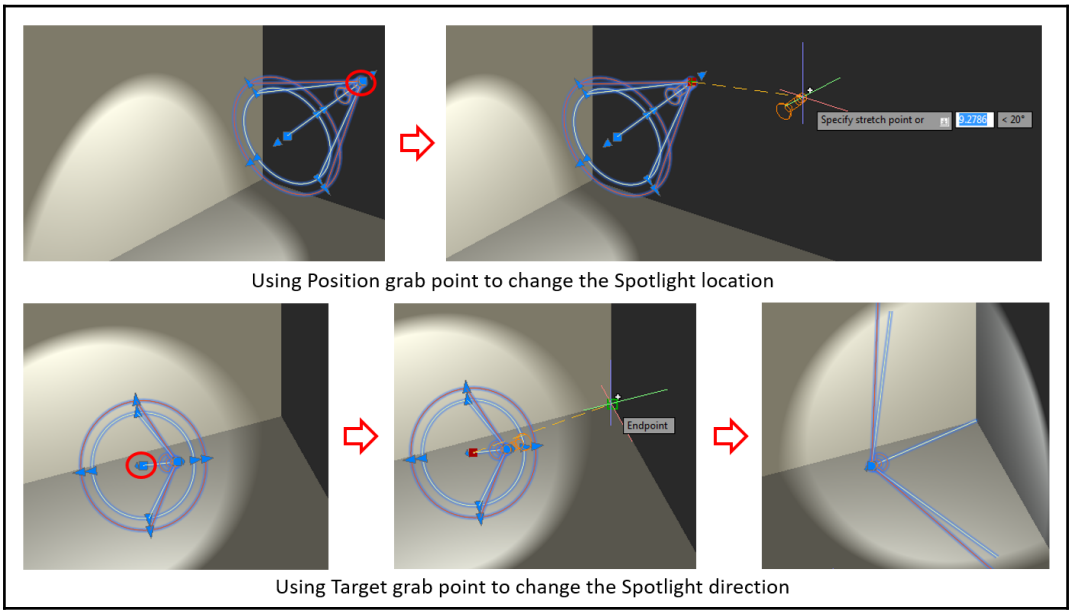


Figure 16.11: Changing the direction and target of a Spotlight source

Position and target line grab points can also be used to change the position of the Spotlight source and target, as shown in the following step-by-step example:

1. Click on **Position** along the line grab point.
2. Move the mouse to change the Spotlight location. Notice that the Spotlight movement is limited to the light cone direction.
3. When you arrive at the desired destination, click again to finish. In the following screenshot, we can see that the light intensity has decreased because the Spotlight has moved away from the wall:

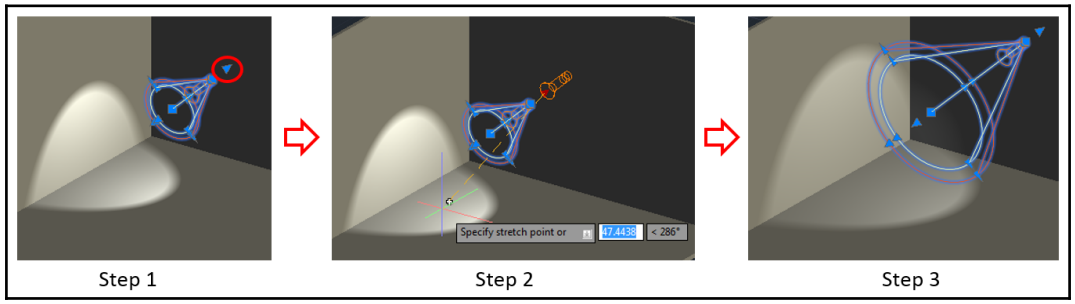


Figure 16.12: Steps for changing Spotlight position along the direction line

Falloff grab points can be used to control the light cone outer angle, as shown in the following screenshot. Notice that only the outer angle has changed; the Hotspot angle did not change:

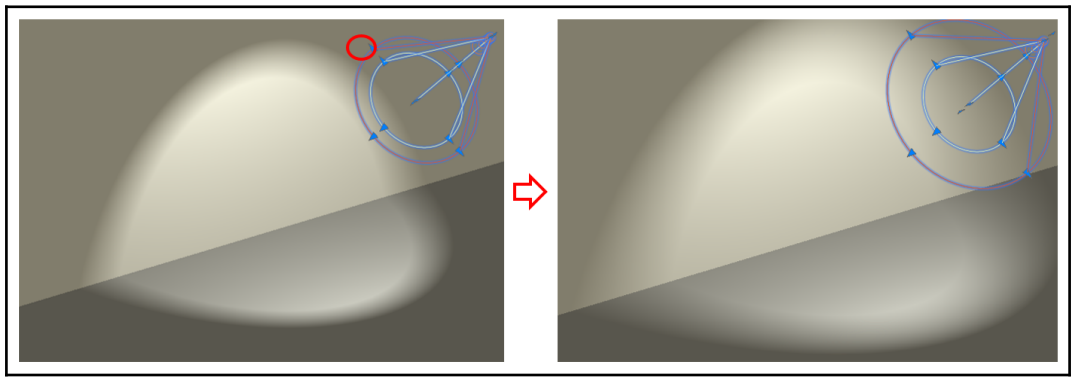


Figure 16.13: Using the Falloff grab point to change the light cone angle

The light cone has an inner angle called the Hotspot angle. This angle forms the limit of the maximum brightness (or the beam angle, as it is called by lighting designers). The brightness will gradually decrease to reach the minimum at the edge of the full cone (at the Falloff angle). The Hotspot grab points can be used to control the Hotspot angle of the light cone, as shown in the following screenshot:

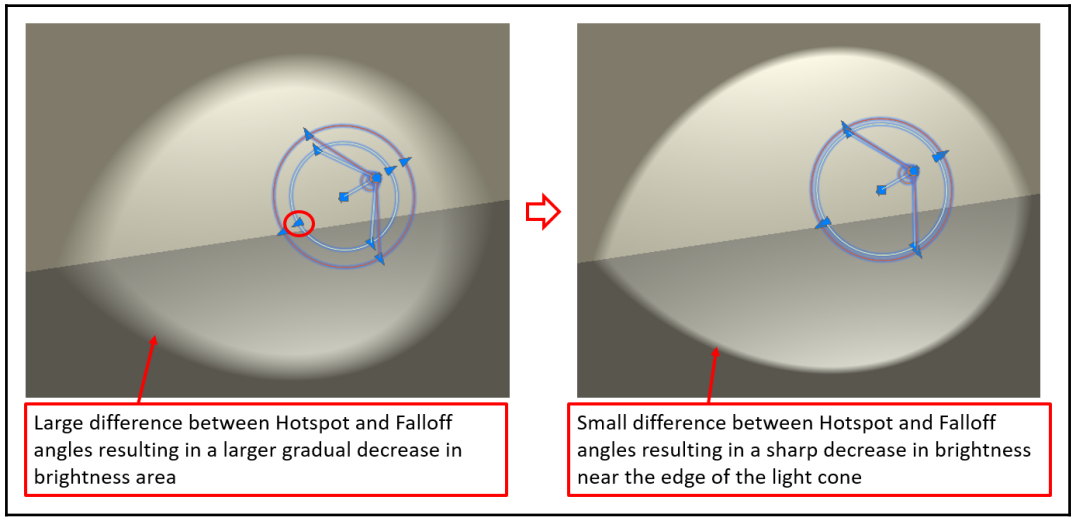


Figure 16.14: Using the Hotspot grab point to change the light cone angle

The intensity and the color of the created Spotlight source can be modified by using the properties palette. For more details, please refer to the *Creating a Point light source* section of this chapter.

Creating a Distant light source

The next light source is called Distant. This type is used to resemble light coming from a very Distant source so that the light rays are all parallel. The **Distant** light tool button can be found in the **Lights** panel within the **Visualize** panel in the **3D Basics** workspace, as shown in the following screenshot. An alternative method is to type the command name, `DISTANTLIGHT`, (or just type `DISTA`) and then press *Enter*:

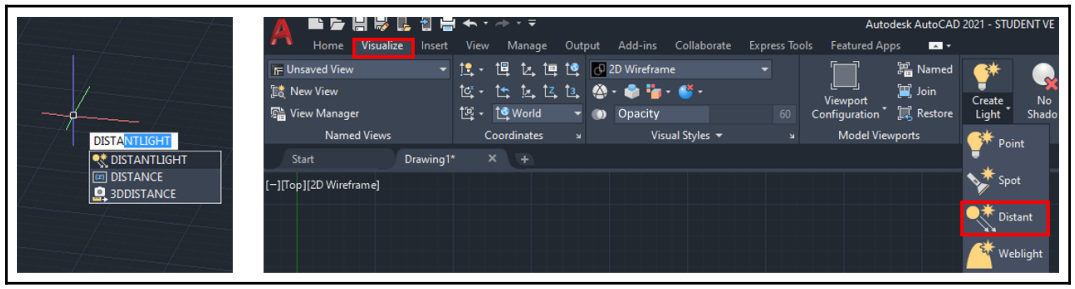


Figure 16.15: Alternative ways to start the Distant light source command

If you are using the **3D Modeling** workspace, you can find the **Distant** light tool button in the **Lights** panel within the **Visualize** panel.

The Distant light tool can be used to create a light that is cast with the same angle all over the model (that is, the light rays are parallel, like light coming from the sun), as shown in the following step-by-step example:

1. After starting the `Distant light source` command, you will be prompted to specify the direction of the light rays by specifying the locations of two points representing the direction vector. Click to specify the location of the first point. In the following screenshot, the inclined line endpoints will be used as the light rays vector.
2. Use the mouse cursor to specify the second point location, as shown in the following screenshot:

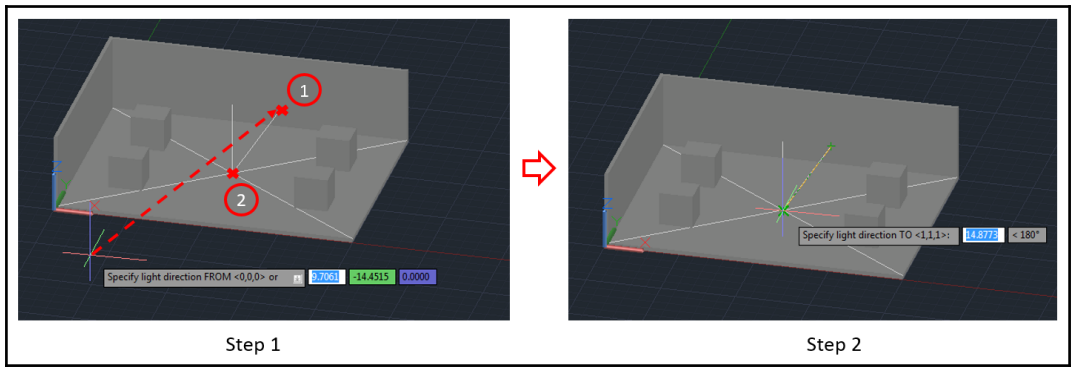


Figure 16.16: Steps for creating a Distant light source (1/2)

3. After specifying the location, a settings menu will appear. To create the light source using the default settings and end the command, click on **eXit**.

The command terminates and the light source is created. The light is cast on all the objects with the same angle, as shown here:

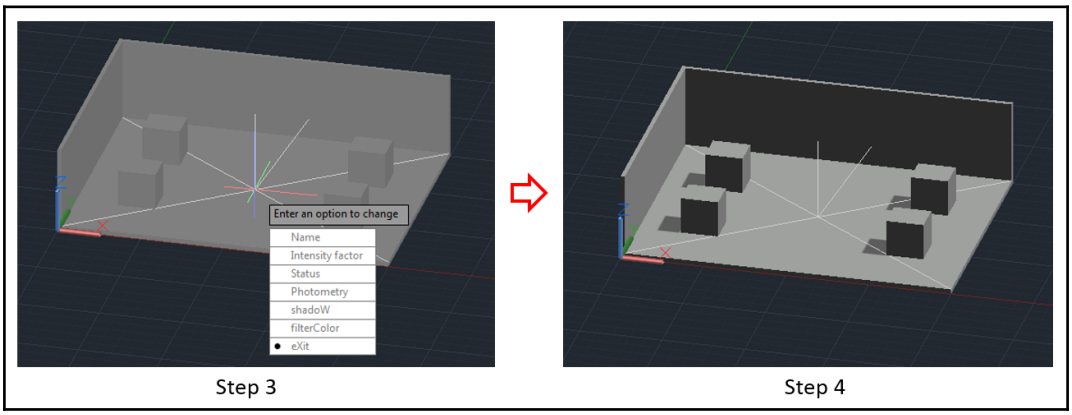


Figure 16.17: Steps for creating a Distant light source (2/2)

As shown in the previous example, an options menu containing seven options for the light source appears in the third step just before the command ends. They are similar to the options that appear in the `Point light source` command. For more details, please refer to the *Creating a Point light source* section of this chapter.

The intensity and the color of the created Distant light source can be modified by using the properties palette. The Distant light source does not have a glyph (an icon in the model showing the location of the light source) in the model area, so to open the Properties palette for a Distant light source, click on the small arrow in the bottom-right corner of the **Visualize** tab, as shown in the following screenshot.

This will open the **Lights in Model** palette, which displays a list of the light sources created in the model. Double-click on the desired Distant light source to open the **Properties** palette, as shown in the following screenshot:

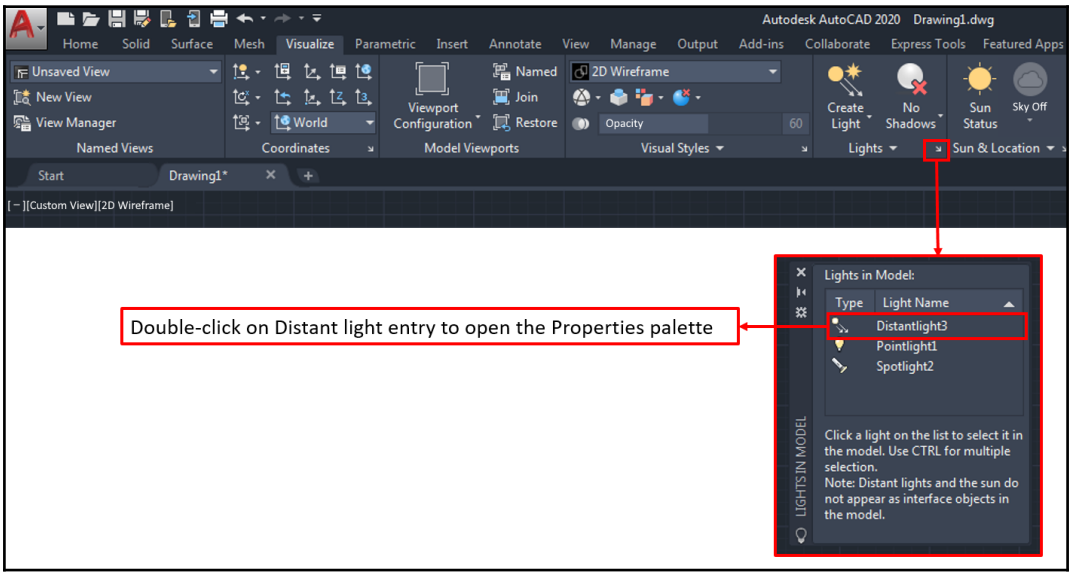


Figure 16.18: The Lights in the Model palette

After that, modify the intensity and the color, as shown in the *Creating Point light sources* section of this chapter.

That's it for the Distant light source. Let's now have a look at the fourth light source type, which is called the Weblight source.

Weblight source

The final light source is called the Weblight source. It is similar to the Point light source. The difference is that it adds the ability to precisely determine the intensity distribution of the light source. The **Weblight** tool button can be found in the **Lights** panel within the **Visualize** panel in the **3D Basics** workspace, as shown in the following screenshot. An alternative method is to type the command name, `WEBLIGHT`, (or just type `WEB`) and then press *Enter*, as shown in the following screenshot:

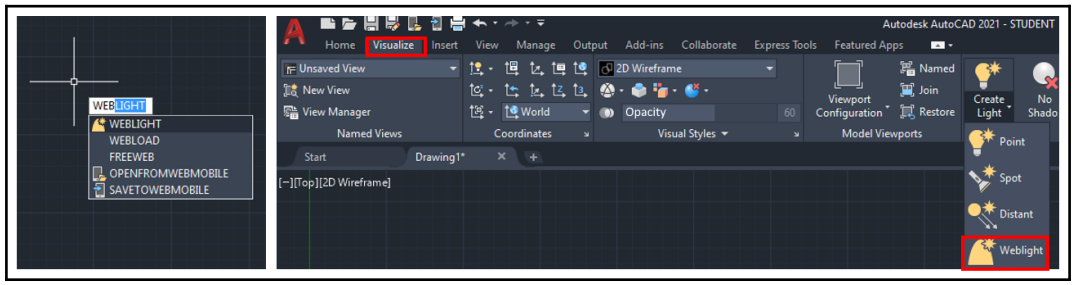


Figure 16.19: Alternative methods to start the Weblight source command

If you are using the **3D Modeling** workspace, you can find the **Weblight** tool button in the **Lights** panel within the **Visualize** panel.

You can add a Weblight source to the model, as shown in the following step-by-step example:

1. After starting the `Weblilght source` command, use the mouse cursor to specify the source location. In the following screenshot, the top end of the vertical line is used as the source location.
2. Use the mouse cursor to specify the target location. In the following screenshot, the bottom end of the vertical line is used as the target location:

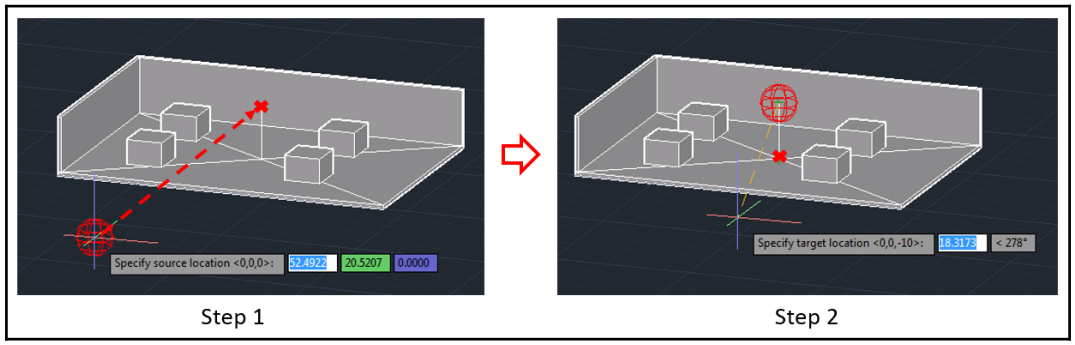


Figure 16.20: Steps for creating a Weblight source (1/2)

3. After specifying the location, a settings menu will appear. For details on these options, please refer to the *Creating a Point light source* section of this chapter (except for the **weB** option, which will be discussed later in the chapter). To create the light source using the default settings and end the command, click on **exit**.
4. The command terminates and the light source is created:

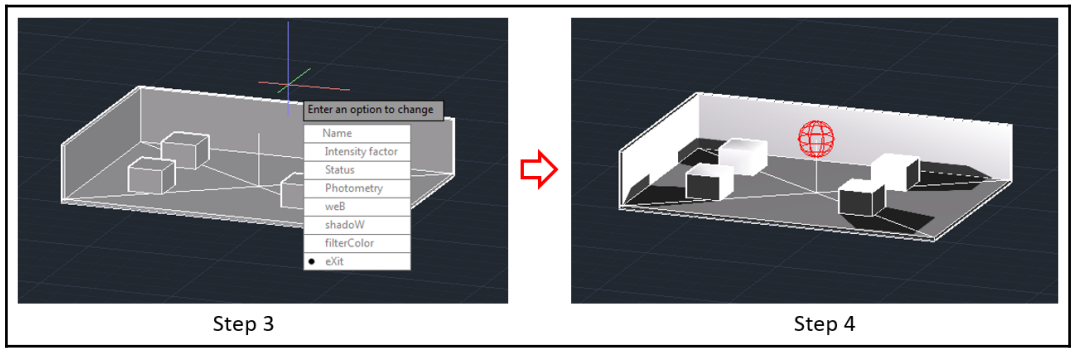


Figure 16.21: Steps for creating a Weblight source (2/2)

In the previous example, the Weblight source was created with default settings, that is, no custom intensity distribution of the light source was set, and that's why it is just emitting the same light in all directions like a point source. To add a map for the intensity distribution, you need to associate the Weblight source with a web file. To do that, select the **Weblight source**, right-click anywhere on the screen, and then select **Properties**.

In the **Photometric Web** section of the **PROPERTIES** palette, click on the small browse icon next to **Web file**. This will open the **Select Web File** window. Browse to the location of the desired web file and open it. You can also click once on the filename to see a preview of the intensity distribution, as shown in the following screenshot. AutoCAD may come with a number of preset web files. In addition to that, light fixture manufacturers may also provide web files for their products that you can download and use:

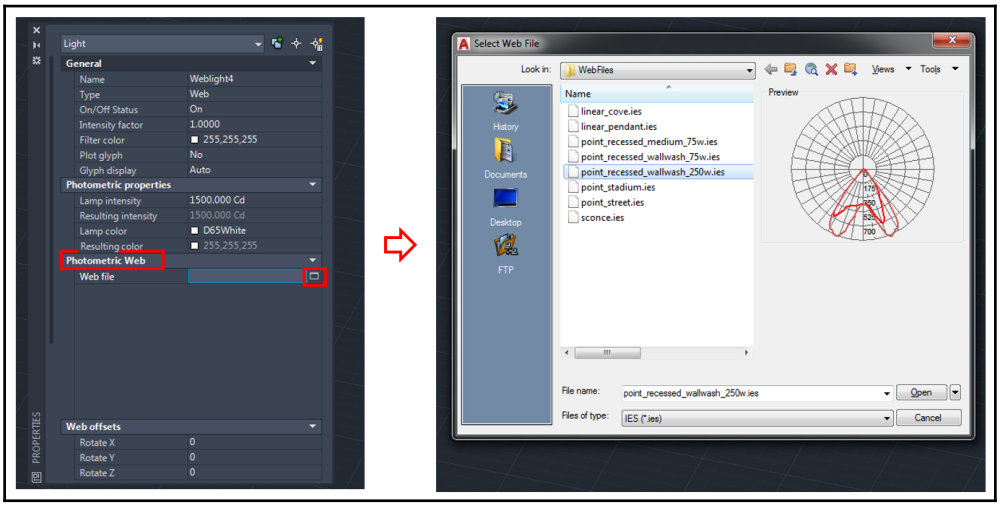


Figure 16.22: Associating a web file with the Weblight source

An alternative method of adding the web file is during the Weblight source creation. Select the option called **weB** from the option list that appears in *step 3* in the previous *Weblight creation* example. Then, type the name of the desired web file.

It is also possible to rotate a given intensity distribution profile around any of the three main axes. This is done using the three rotation options found in the *Web offsets* section in the **PROPERTIES** palette of the Weblight source. Simply type the rotation angle for each axis and then close the **PROPERTIES** palette.

Sun lighting

In addition to the four different light sources discussed in the previous sections, AutoCAD can also add lighting to the model that resembles sunlight. Different aspects that determine sunlight can be controlled, such as the location on Earth and the specific date and time. To enable sun lighting, you simply click on the button called **Sun Status** found in the **Sun & Location** panel on the **Visualize** tab in the **3D Basics** workspace:

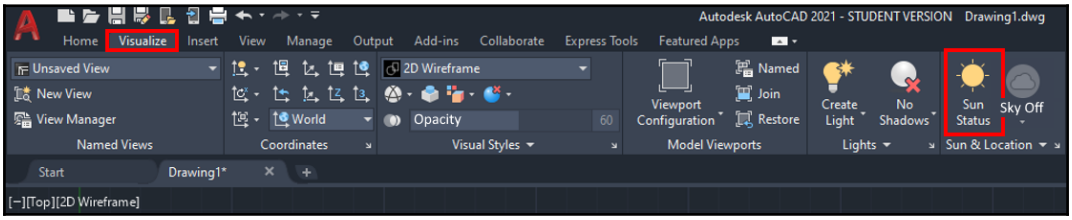


Figure 16.23: Location of the Sun Status tool button in the 3D Basics workspace

If you are using the **3D Modeling** workspace, you can find it in the **Sun & Location** panel, which is also on the **Visualize** tab. When the button has a blue frame, this indicates that sun lighting is activated.

To control the sunlight direction, you can control the date, time, and location. This is done by expanding the **Sun & Location panel**. You will find two sliders to control the date and the time. For the location, click on the **Set the Location** button. The button will expand, showing two different options. The first one is called **From Map**, where you can search for an address on the map similar to a GPS navigation software, or simply enter the longitude, latitude, and elevation of the desired location. The second option is called from a file, where you can use the location stored in a GIS file.

We have now learned how to create and adjust the settings of different types of light sources in AutoCAD. The next step is to choose how we want shadows to be displayed in our model.

Displaying shadows

The presence and the style of shadows in the 3D model can be controlled using a command called `VSSHADOWS`. When activating this command (by typing the command name and then pressing *Enter*), you will be prompted to choose between three different values for the `VSSHADOWS` system variable that correspond to three different settings for the shadows in the model, and these values are as follows:

- Setting the value to `0` means there will be no shadows in the model.
- Setting the value to `1` will create shadows on the model ground plane. The ground plane location is usually the **XY** plane at elevation **Z** = `0`. This can be changed using a system variable called `SHADOWPLANELOCATION`.
- Setting the value to `2` corresponds to full shadows cast on all the objects in the model.

The three options have tool buttons to allow us to quickly switch between them. The buttons can be found in the **Lights** panel on the **Visualize** tab in the **3D Modeling** workspace, as shown in the following screenshot:

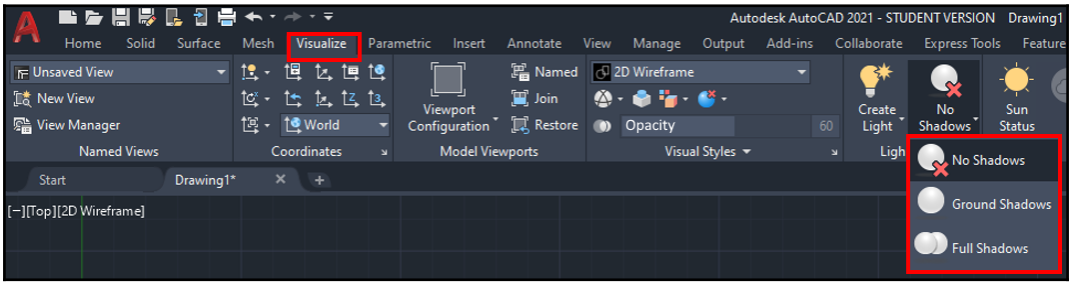


Figure 16.24: Location of the Shadow tools in the 3D Modeling workspace

The following screenshot demonstrates the differences between the three shadow settings. The model in the Figure has sun light activated, as well as a Spotlight:

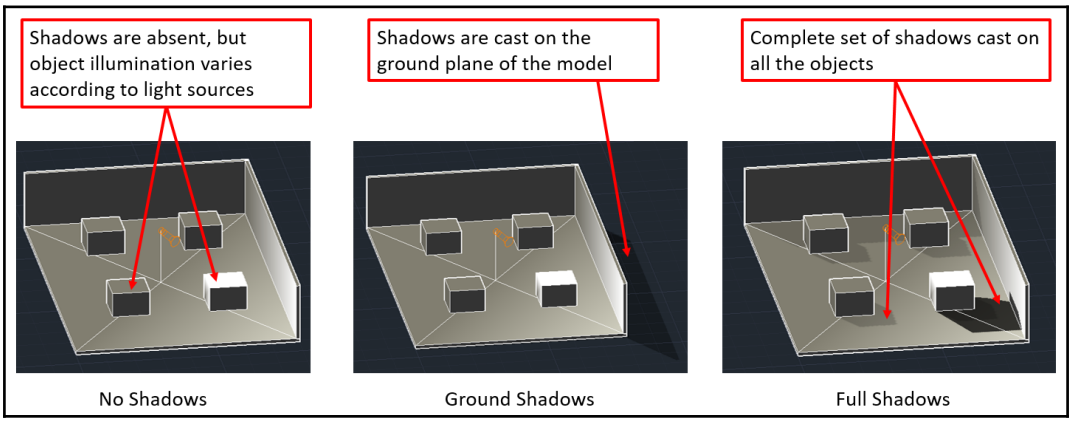


Figure 16.25: Different shadow modes

Now that we have learned how to create and set light sources, and how to control the shadows displayed in our model, the next step is to apply materials to the bodies in the model. Let's take a look at that in detail.

Applying and viewing materials

After creating the 3D model, the final step to polish your model and bring it to life is to assign materials to each part. In this section, the application and settings of the different materials in AutoCAD and the creation of new materials and material libraries will be discussed in detail.

Setting the visual style to display materials

The first step to start working with materials is to adjust the visual style settings to properly display materials. As discussed in `Chapter 8`, *Customization Tools*, AutoCAD offers different visual styles, which vary the degree of detail and impact on the machine's performance. Some of these will not display materials or textures, such as 2D Wireframe and Conceptual. Some will only display the material properties (reflectiveness and transparency), such as the visual style called Shaded. And the Realistic visual style will display both the material properties and the textures.

In addition to the existing visual styles, it is possible to change the materials/textures display setting using the `VSMATERIALMODE` system variable. This has three possible values, which are **0**, **1**, or **2**, and they correspond to the following settings:

- The value **0** corresponds to no materials displayed.
- The value **1** will only display materials.
- The value **2** will display both materials and textures.

You can also switch between these three settings using the tool button found in the **Materials** panel on the **Visualize** tab in both the **3D Modeling** and **3D Basics** workspaces. The location of the tool button in the **3D Modeling** workspace is shown in the following screenshot:

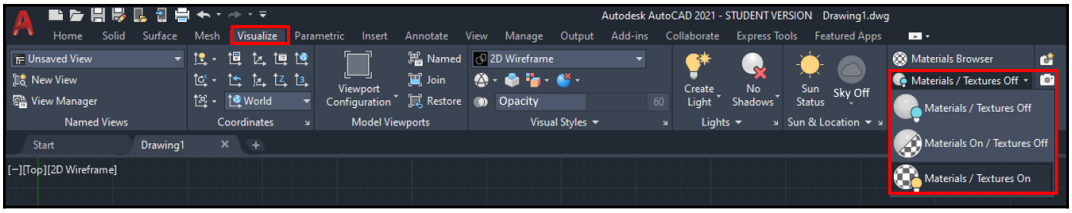


Figure 16.26: Materials and textures display button location in the 3D Modeling workspace

The following screenshot shows a dining table and some chairs with three glass cups and a small box on the table. The table has a wood material assigned to it, the chairs have a fabric material, the cups have glass materials, and the small box has a metallic material, as you can see in the bottom-left corner of the screenshot, where neither the materials nor the textures are displayed. There is no difference between the different objects, although they have different materials assigned to them. In the upper-right corner, the second option, **Materials On/ Textures Off**, is selected. Now, the properties of the material, such as the transparency of the glass cups, are displayed. In the bottom-right corner, both materials and textures are displayed, which gives a fully realistic view:

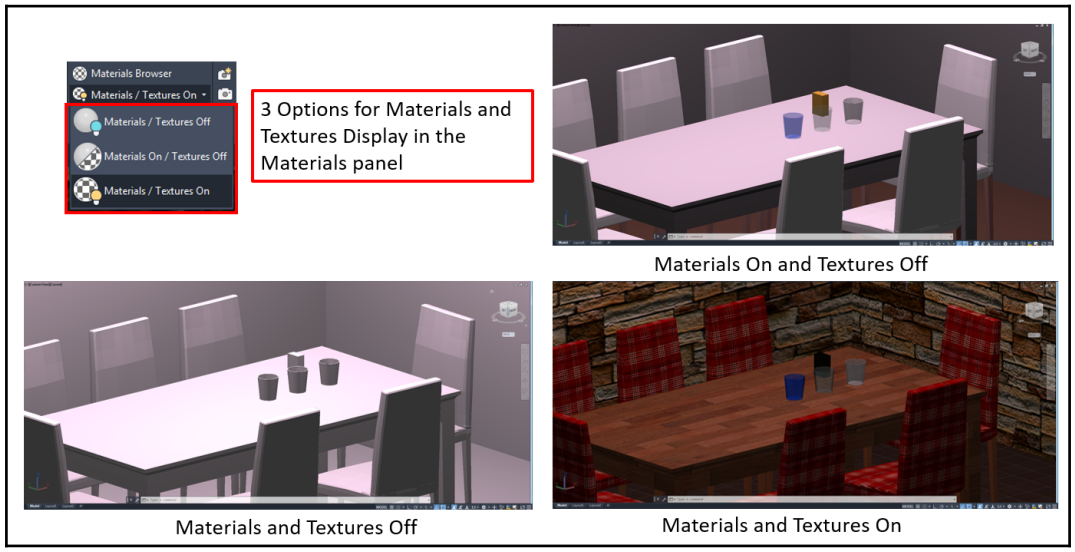


Figure 16.27: Difference between showing material properties and material textures

Now that we know the differences between different visual style settings and how materials are displayed in each one, let's now explore the different materials available in AutoCAD.

Exploring the Materials Browser

AutoCAD has a library that contains a large collection of different materials. To access the material library, click on the **Materials Browser** button found in the **Materials** panel on the **Visualize** tab in both the **3D Modeling** and **3D Basics** workspaces. The following screenshot shows the **Material Browser** button located in the **3D Modeling** workspace:

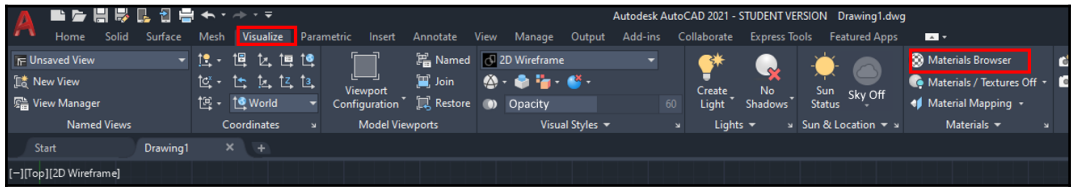


Figure 16.28: The Materials Browser button location in the 3D Modeling workspace

Clicking on the **Materials Browser** button will open the **Materials Browser** palette, as shown in the following screenshot. At the top, there is a search bar that you can use to quickly get a material by typing the name. Below that there are two big sections. The one on the bottom displays all the materials that are present in the current material library. The library may be divided into categories for easier navigation. To access these subcategories, click on the button named **Autodesk Library** (the library name) to select the category to be displayed, as shown in the following screenshot:

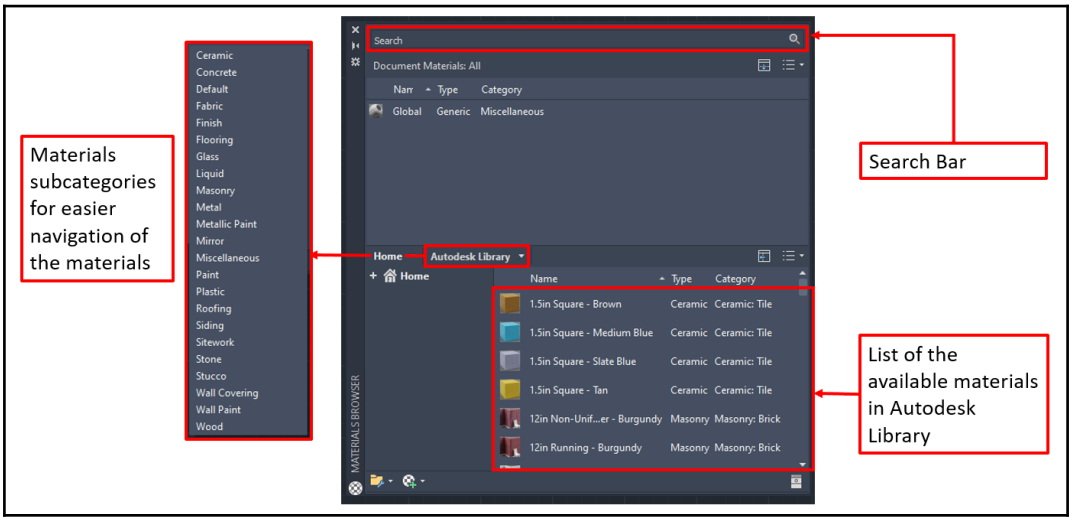


Figure 16.29: Materials Browser palette overview

The upper large section is for displaying materials, and you can apply filters, such as only showing the materials applied to the model or the materials of the selected objects. The different filters are shown in the following screenshot:

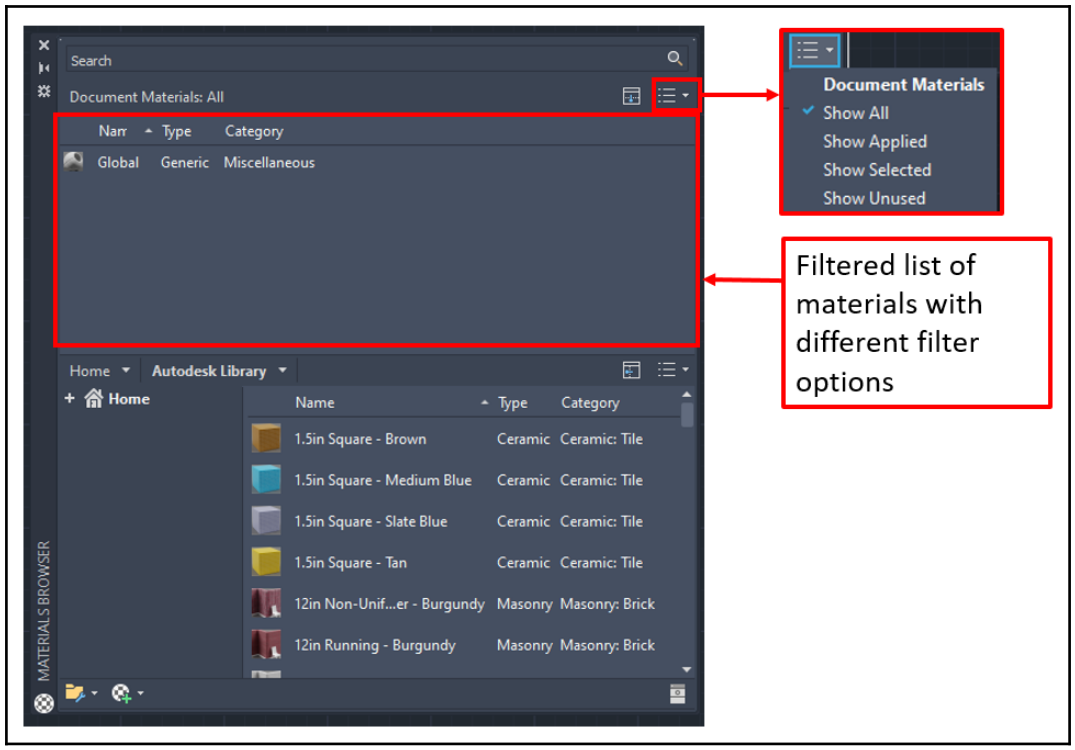


Figure 16.30: Filtered list in the Materials Browser palette

The materials list has different display options. They can be accessed by clicking on the small button in the top-right corner of the list, as shown in the following screenshot. The default one is called **List View**. It displays a small thumbnail beside the description of the material. Another option is called **Text View**, which only displays the names of the materials:

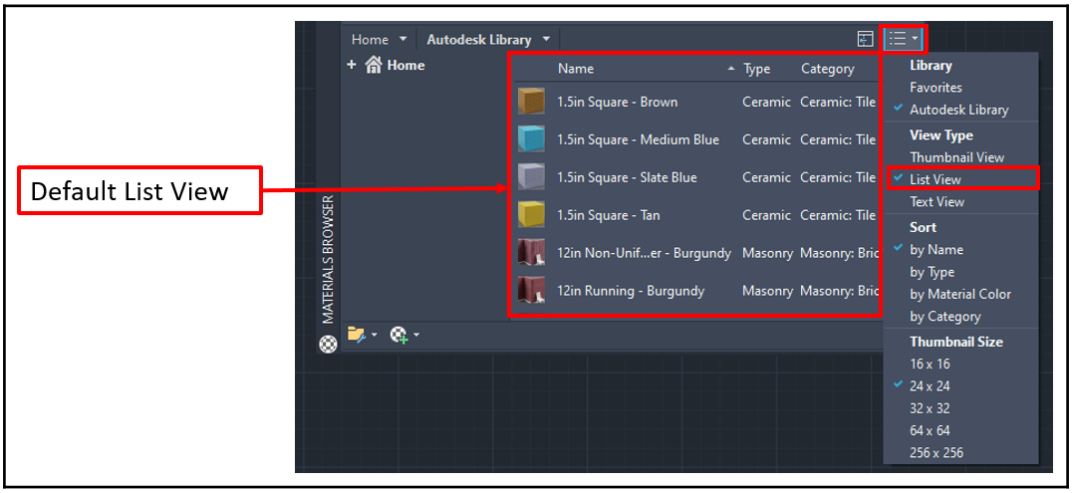


Figure 16.31: List View in the Materials Browser palette

The final display option is called **Thumbnail View**. This displays big thumbnails showing previews of the different materials. It is also possible to control the thumbnail size using the different thumbnail size options.

We have seen the material libraries in AutoCAD and learned to navigate them. Let's now apply the materials to the objects in the model.

Applying materials to objects

Applying a material to an object in the model is a simple drag and drop operation, as shown in the following step-by-step example:

1. Open the **Materials Browser**, click on the material name, and drag it to the desired object.
2. Release the mouse button to apply the material:

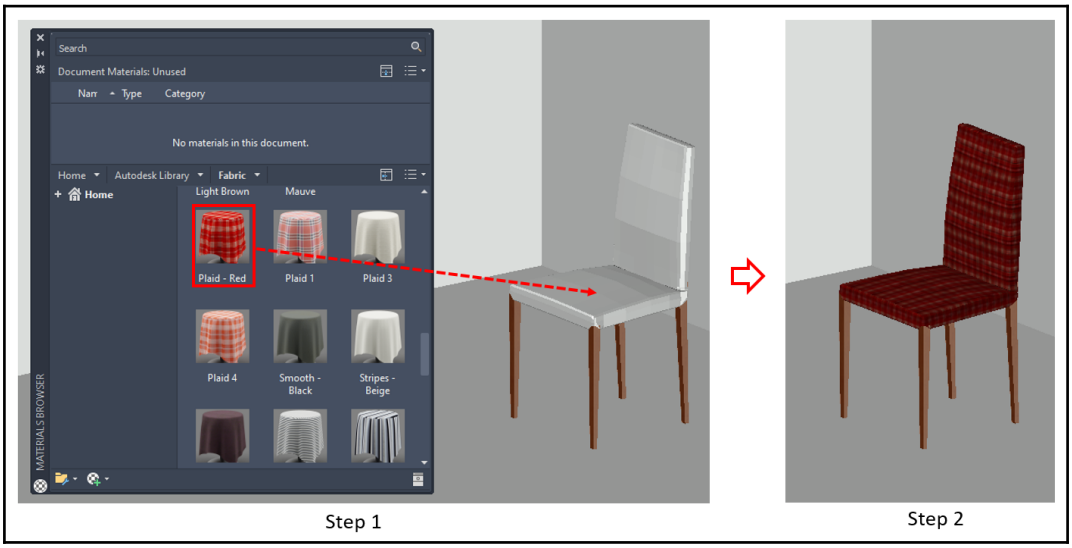


Figure 16.32: Applying material to an object

An alternative method to apply materials is to associate the materials with layers. This will automatically apply the material to all objects in that layer that have their material property set to By Layer.

To assign a material to a layer, click on the **Attach By Layer** button found in the Materials panel on the **Visualize** tab in both the **3D Modeling** and the **3D Basics** workspaces. The location of the **Attach By Layer** button in the **3D Modeling** workspace is shown in the following screenshot:

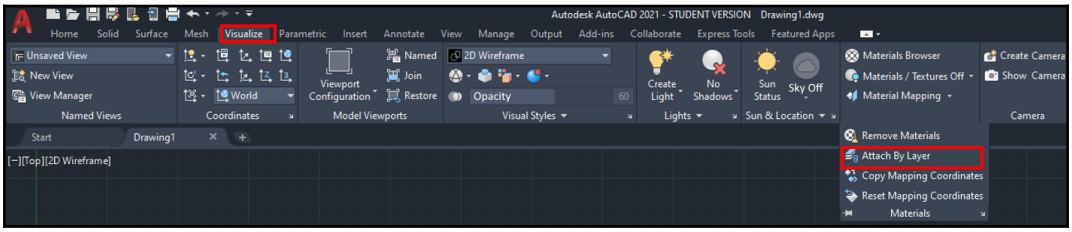


Figure 16.33: Location of the Attach By Layer button in the 3D Modeling workspace

Clicking on the **Attach by Layer** button will open the **Materials Attachment Options** window. This window is divided vertically. On the left, you have a list of the materials that were used in the model, and on the right, you have a list of all the layers present in the model. To assign a material to a layer, simply click on the material name, drag it, and then drop it on the desired layer name. If you have a material attached to a layer and you want to return to the default **Global** material, click on the big **X** sign that will appear near the layers that have material attachments, as shown in the following screenshot:

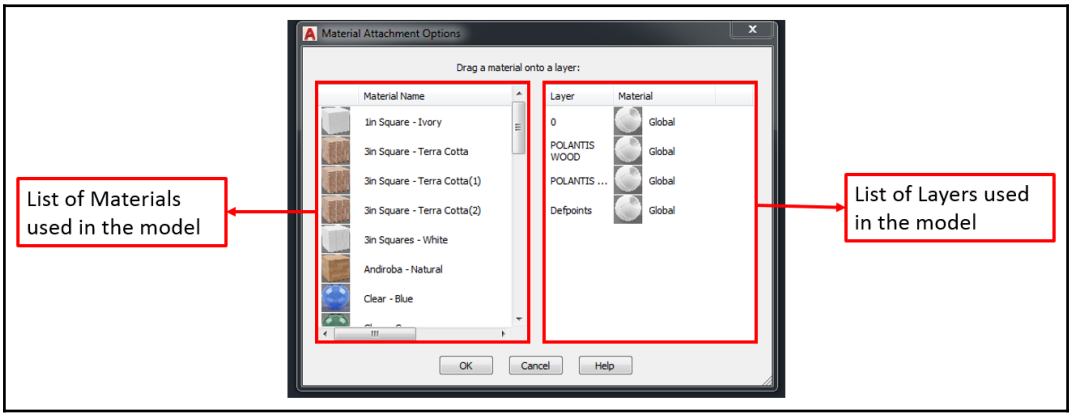


Figure 16.34: Material Attachment Options window

Now that we have selected the material and applied it to the object, the next thing to look at is how the material's texture map will be wrapped around the 3D object. This can be controlled by using the **Material Mapping** options. The details of using the **Material Mapping** options are shown in the next section.

Material Mapping

Material textures can be mapped on object surfaces in different ways. AutoCAD allows you to choose the method by which the material textures will be mapped by using the `MATERIALMAP` command. To use it, type `MATERIALMAP` (or just type `MATE`) and then press *Enter*. This command has four different options for material texture mapping, which are as follows:

- Box
- Planar

- Sphere
- Cylindrical

An alternative method for accessing these four options is by clicking on the small arrow near the **Material Mapping** tool button found in the **Materials** panel on the **Visualize** tab in both the **3D Modeling** and **3D Basics** workspaces. The following screenshot shows the location of the **Material Mapping** tool button in the **3D Modeling** workspace:

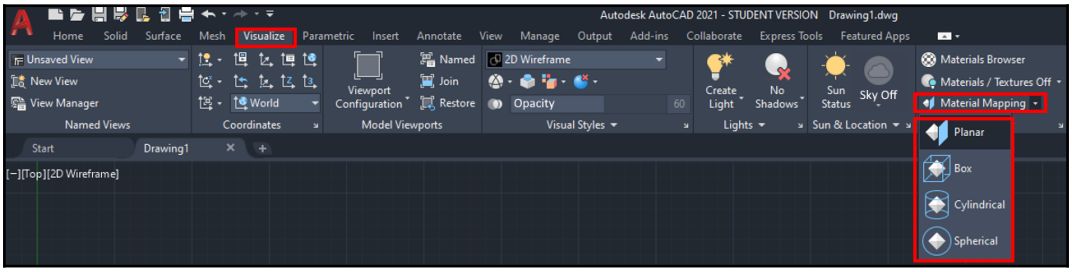


Figure 16.35: Location of the Material Mapping tool button in the 3D Modeling workspace

To apply any of these four mapping options, just click on the desired mapping options, and then click on the object in the model. A wireframe will be shown after that to adjust the alignment of the texture map. After finishing alignment, press *Enter* to accept the mapping. The following screenshot shows the result of applying the four different texture mapping options to a sphere object:

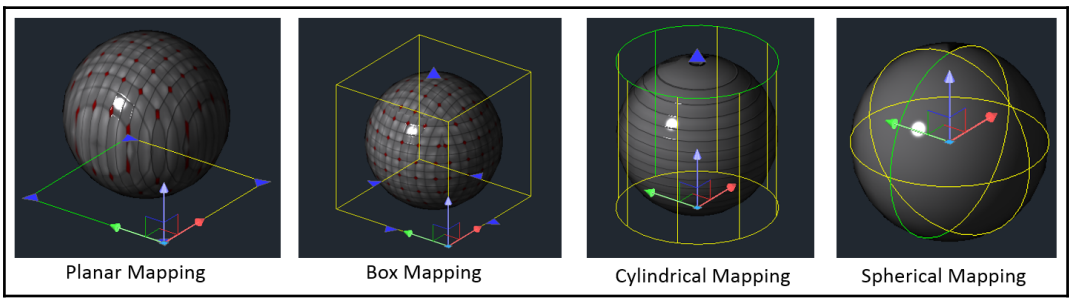


Figure 16.36 -Different material mapping methods

As mentioned earlier, **Material Mapping** can be used to adjust the texture map alignment with the object. The following is an example of that. In the following screenshot, there is a box with a 4-inch square with inset diamond material applied, and the tile pattern is repeated five times along the height of the box. This can be changed to three, for example, by using **Material Mapping** as follows:

1. Click on the **Box Material Mapping** button, then click on the **Box object**, and then press *Enter* to confirm selection.
2. The **Mapping control** wireframe will appear. Click on the upper blue arrow.
3. Move the mouse cursor upward until the tile pattern is repeated three times, and then click again. After that, press *Enter* to end the command:

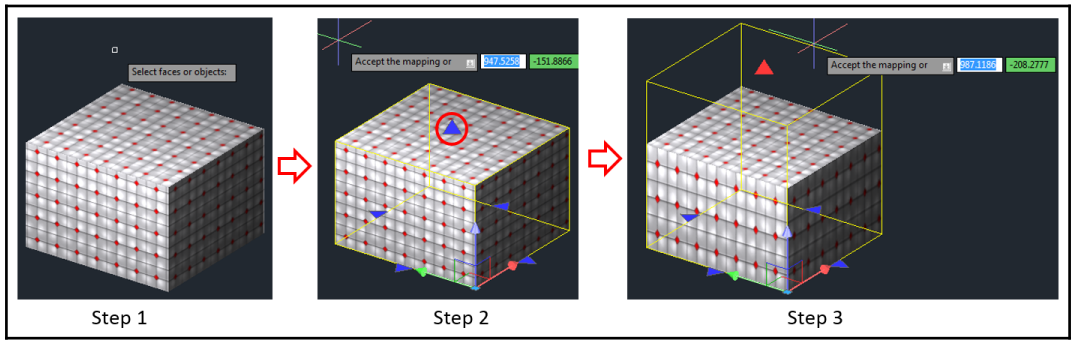


Figure 16.37: Adjusting the material map

We have now learned how to navigate the materials libraries, how to apply materials to objects, and how to control the texture mapping. In the next section, we will learn how to add a new material that may have a custom texture map to the materials library.

Using custom materials/texture maps

In addition to the materials available in the Autodesk library, it is possible to add a new material with a custom texture map. To add a new material, click on **Create New Material** at the bottom of the **Materials Browser** palette. This will open a list of different types (templates) to start creating the material from. So, for example, if you have a texture image file for wood, you can use the **Wood** material, where the material properties for wood will be set. A texture image will be loaded that can be modified, or you can load an external texture image by clicking on the text under the texture image preview (as shown in the following screenshot), browse to the desired image file, and press **OK**.

Below the material types list is an entry called **New Generic Material**. This will also open the **Materials Editor** palette, but this time nothing is preset and all the material properties will be available to tweak as desired. The texture image file will be blank. You can click on the text under the texture image preview to select the texture image file, as shown in the following screenshot:

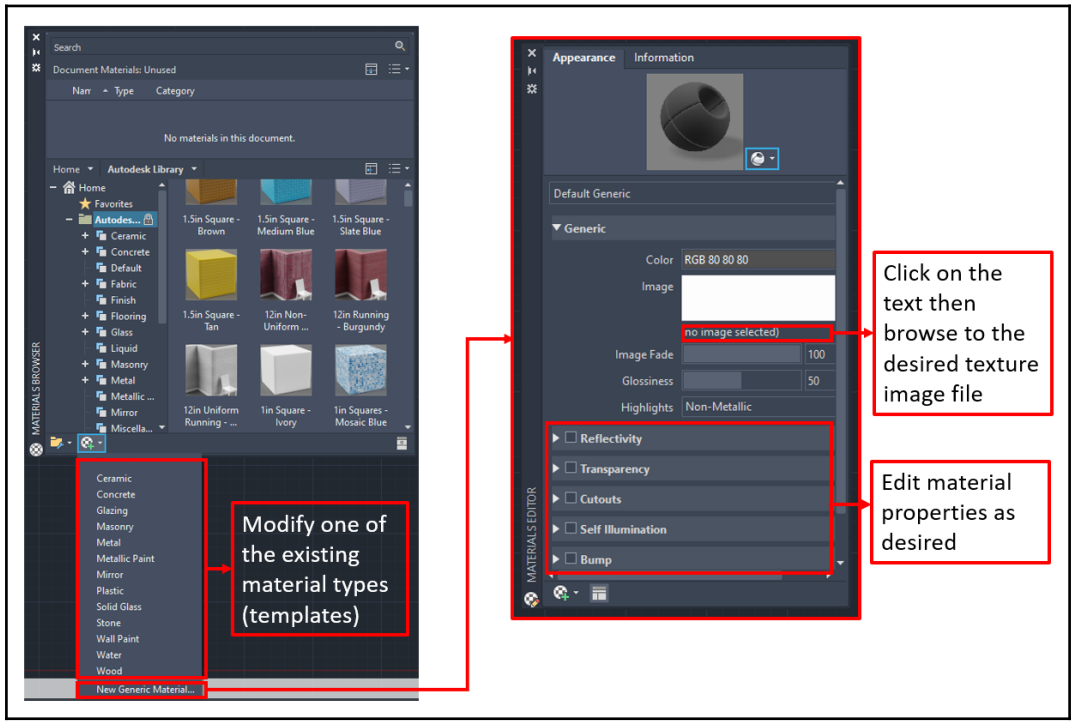


Figure 16.38: Creating a new material

As demonstrated in the previous screenshot, the **Material Editor** palette has two tabs at the top. The first tab was discussed in the previous paragraph. The second tab is called **Information.** There, you can type a name and a description for the newly created material, and even add keywords to easily find it by using the search bar in the **Materials Browser**. After the editing is complete, just close the **Materials Browser** Palette and the newly created material will be saved to the drawing file and can be used.

We have almost completed setting the scene for rendering. We have set the lighting conditions, adjusted the shadow display settings, and applied materials to the objects in the model. The next step in preparing the scene for rendering is to create cameras, which will be discussed in the next section.

Cameras

You may think that after setting the lighting effects and assigning materials to different parts of the model, the next step is to adjust the view angle of the viewport then start rendering. It is possible, but AutoCAD offers a more sophisticated tool to define a 3D view, which is creating cameras. Using a camera allows precise determination of the target point you are aiming at and the position from which you are aiming. In addition, you can adjust the lens to have either a narrow or a wide viewing angle, as we will see in this section. Also, it may be better from an organizational point of view to have the 3D views that will be used in rendering (represented in the cameras created) separate from the views used in modeling (the standard and the user-created views). In this section, we will learn how to create cameras and how to adjust their settings in detail.

Creating a new camera

The command named `CAMERA` can be used to add a new camera to the model. It can be activated by typing `CAMERA` (or just typing `CAM`) and then pressing *Enter*, or alternatively, you can click on the **Create Camera** button found in the **Camera** panel on the **Visualize** tab in the **3D Modeling** workspace, as shown in the following screenshot:

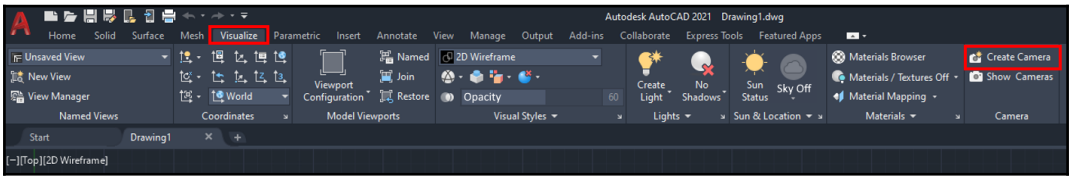


Figure 16.39: The Create Camera button location in the 3D Modeling workspace

Camera creation is very similar to the creation of a Spotlight source, as shown in the following step-by-step example:

1. Click on the **Create Camera** button, and then use the mouse cursor to specify the location of the camera. In the example screenshot, the endpoint of the vertical line in the bottom-left corner will be used as the camera's location.
2. Specify the camera target using the mouse cursor, as shown in the following screenshot:

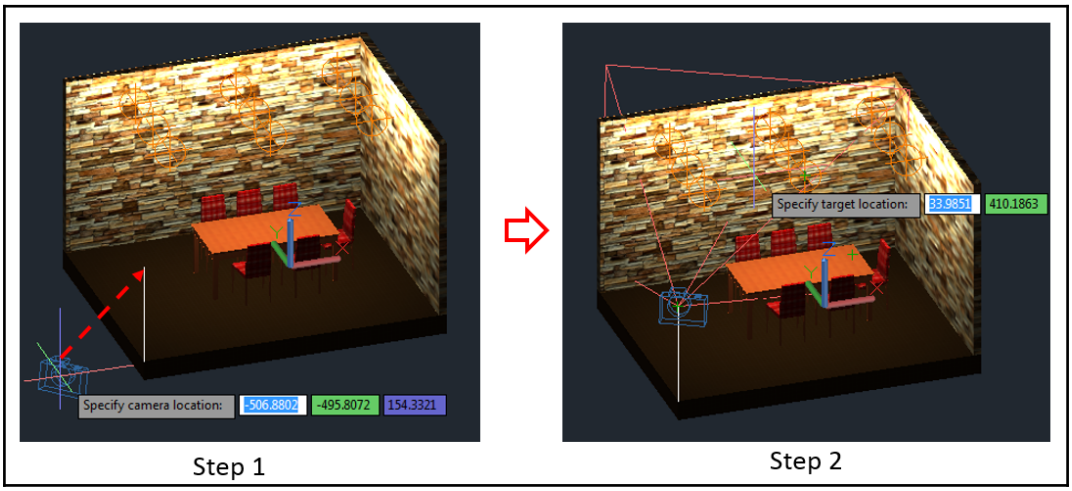


Figure 16.40: Steps for creating a new camera (1/2)

3. After specifying the target location, a settings menu will appear. These settings will be discussed later in this section. To create the camera using the default settings and end the command, click on **End**.

4. The camera is created. Double-clicking on the camera icon will open the camera view in a separate window called camera preview, as shown in the following screenshot:

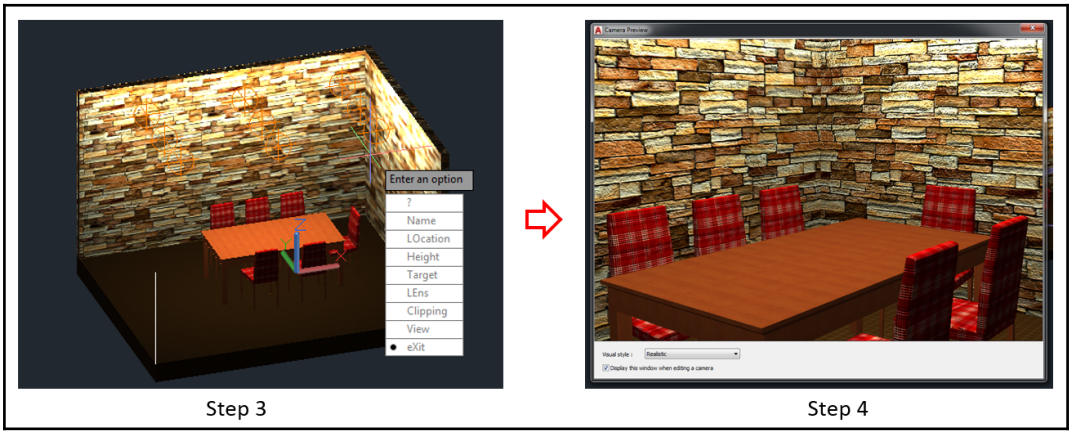


Figure 16.41: Steps for creating a new camera (2/2)

As shown in the previous example, after specifying the camera and the target locations, a settings menu will appear. The settings list contains nine different settings/options to adjust for the created camera, and they are as follows:

- **?:** Click on this option to list all the camera names that are currently present in the model.
- **Name**: Use this option to name the created camera. Just click on the option, type the name, and then press *Enter*.
- **Location**: Click on this option to relocate the camera. You can use either the mouse cursor or type the new location coordinates using the keyboard, and then press *Enter*.
- **Height**: When clicking on this option, the current height of the camera from the **User Coordinate System** (**UCS**) **XY** plane will be displayed. Type the new desired height and then press *Enter*.
- **Target**: Click on this option to relocate the camera target. You can use either the mouse cursor or type the new location coordinates using the keyboard and then press *Enter*.

- **Lens**: Clicking on this option will display the current camera lens' length. Type the new desired lens' length and then press *Enter*.
- **Clipping**: Use this option to set the clipping planes of the created camera.
- **View**: Clicking on this option and choosing **Yes** will switch the model view to the camera view after the command ends.
- **Exit**: Click on this to finish the camera creation and end the command.

Now that we have added cameras to our model, let's have a look at how to adjust/tweak their settings.

Modifying cameras using grab points

If the camera object is selected, a wireframe showing the camera target and the field of view will be displayed as shown in the following screenshot. This wireframe has a number of grab (control) points to quickly modify the camera and target locations, the field of view, and the distance to the target plane:

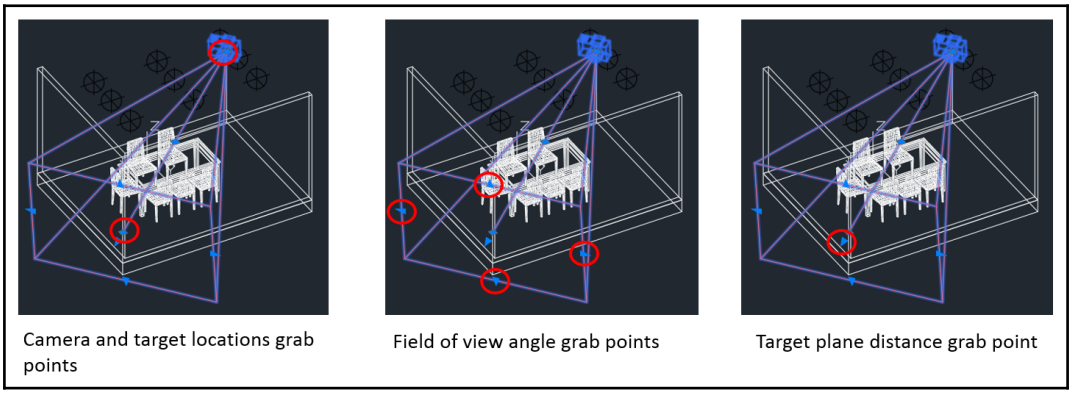


Figure 16.42: Camera grab points

Click on the camera or the target location grab points, move the mouse cursor to the desired new location, and then click again to change either the location of the camera or the target.

The field of view angle can be modified using the grab points, as shown in the following screenshot. The field of view angle is made wider by clicking on one of the grab points and moving outward:

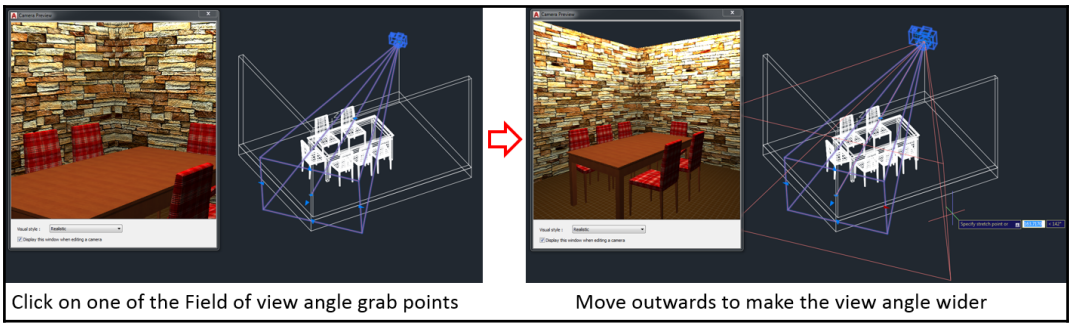


Figure 16.43: Adjusting the camera's field of view angle

The last grab point is the target plane distance grab point. Click on it to change the target plane location along the line between the camera location and the target location. The target plane location is used to place the clipping planes at a relative distance to the camera's position.

We have now completed setting the scene and it is ready to be rendered. Let's have a detailed look at the rendering process.

Rendering scenes

After setting the lighting effects, assigning materials to different parts of the model, and setting the cameras, the final step is to produce a set of high-quality rendered images of your model that can be used for the presentation or as the output of your project. In this section, creating a render and all the related settings will be discussed in detail.

Setting the render quality

Before starting to render, the quality must be set. AutoCAD offers a spectrum of different render qualities, ranging from **Low** (draft) quality, which will just take minutes to complete (however, it will produce a poor quality image), up to a quality called **Overnight Quality**, which will produce a very high-quality render but is very demanding and will take a lot of time to complete. By default, the **Medium** quality is selected, which is not very demanding on the hardware and will produce acceptable quality. So, depending on whether you are creating the final output of the project or still in the working stage, choose the desired render quality from the list found in the **Render** panel on the **Visualize** tab in both the **3D Modeling** and **3D Basics** workspaces. The following screenshot shows the location of the render quality list in the **3D Modeling** workspace:

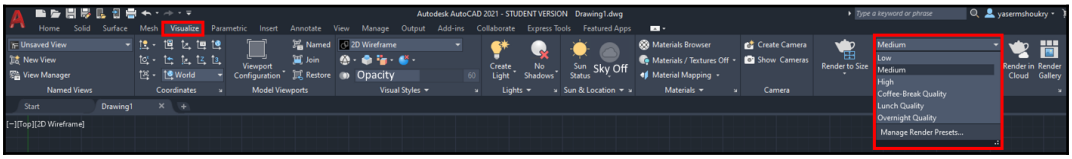


Figure 16.44: The Render Quality location in the 3D Modeling workspace

After setting the render quality, the next thing we want to adjust is the output image resolution.

Setting the rendered image resolution

After adjusting the render quality, the output image resolution should be chosen. AutoCAD has a quick-select list of output resolutions to choose from. This can be found in the **Render** panel on the **Visualize** tab in the **3D Modeling** and **3D Basics** workspaces, as shown in the following image. In addition, clicking on **More Output Settings** will open the **Render to Size Output Settings** window, where a custom output image resolution can be set, either based on the number of pixels or the dpi. There is also an option to automatically save the rendered image to a specific location:

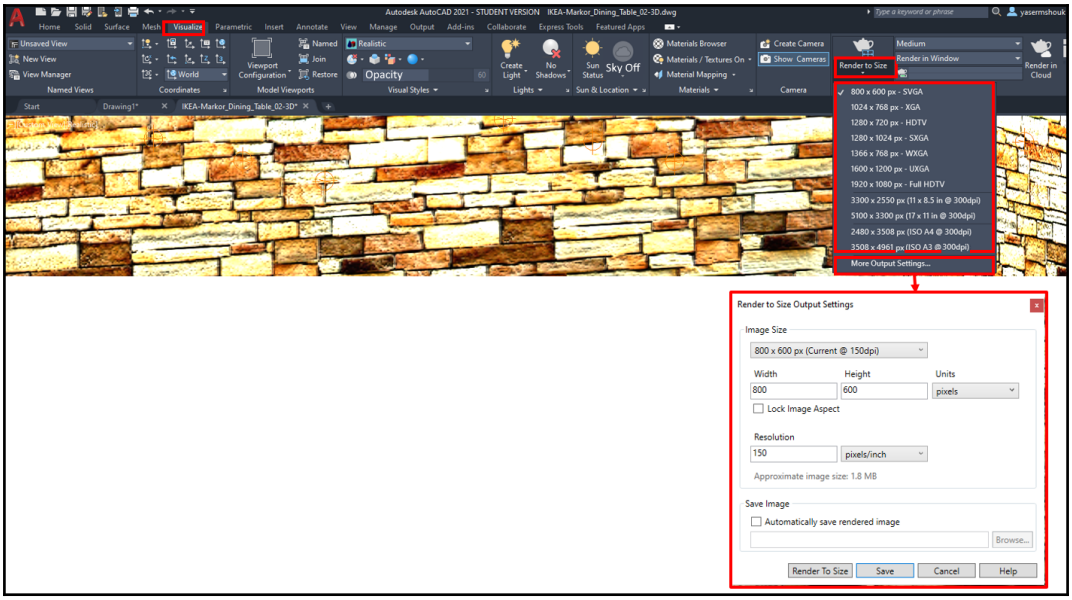


Figure 16.45: Render output size

Now that we are all set to start the rendering process, let's have a more detailed look at it.

Rendering

After setting the render quality and the output image resolution, the final step is to start rendering. Set the view in the drawing area and then click on the **Render** button. The **Render** button can be found in the **Render** panel on the **Visualize** tab in both the **3D Modeling** and **3D Basics** workspaces. The following screenshot shows the location of the Render button in the **3D Modeling** workspace:

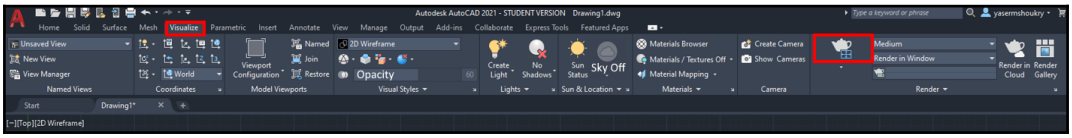


Figure 16.46: The Render button's location in the 3D Modeling workspace

Starting a render will open the **Render** window, as shown in the following screenshot. In the middle of the render window, a preview of the rendered image is shown. At the bottom, you will find a progress bar for the render process. In the top left, there is the **Save** button to save the rendered image in a specified location:

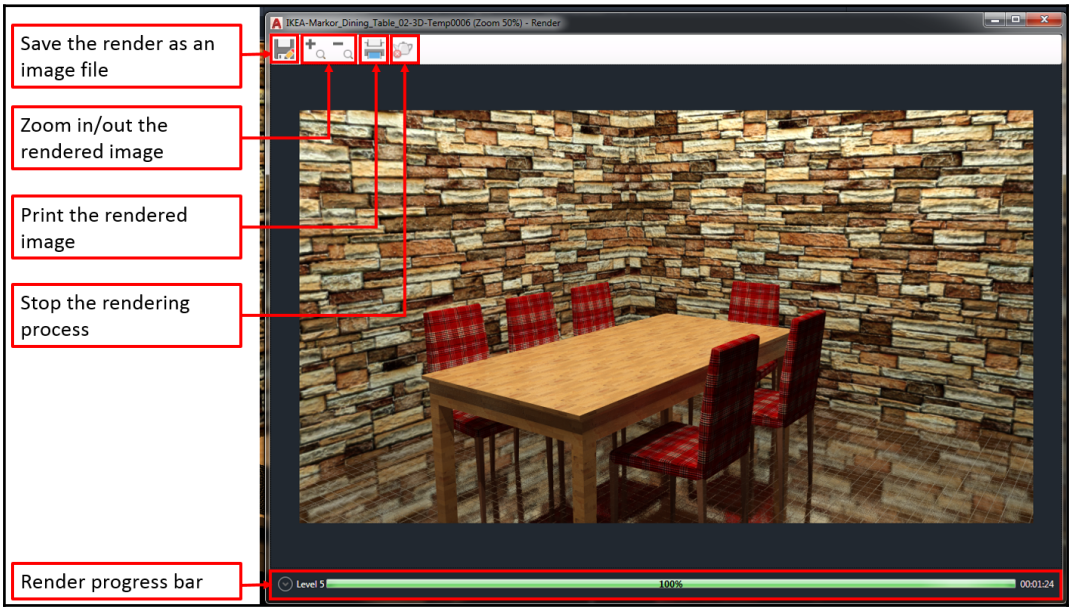


Figure 16.47: Render window

Once rendering is done, you can use buttons with the small magnifying lens to zoom in or out of the rendered image, upi can print it using the **Print** button, and finally, there's a **Stop** render button to interpret the rendering process at any time.

Importing and exporting 3D files

It is common during the 3D modeling process, especially when working on big multi-disciplinary projects, to import 3D models created by other software to your project. AutoCAD can import a variety of different 3D model file types. Simply click on the **Application Menu** button, select **Import**, and then click on **Other formats**. This will open the **Import File** window, where you can browse to the desired file location and open it. There are a variety of different 3D file types that can be imported to AutoCAD, as shown in the following screenshot:

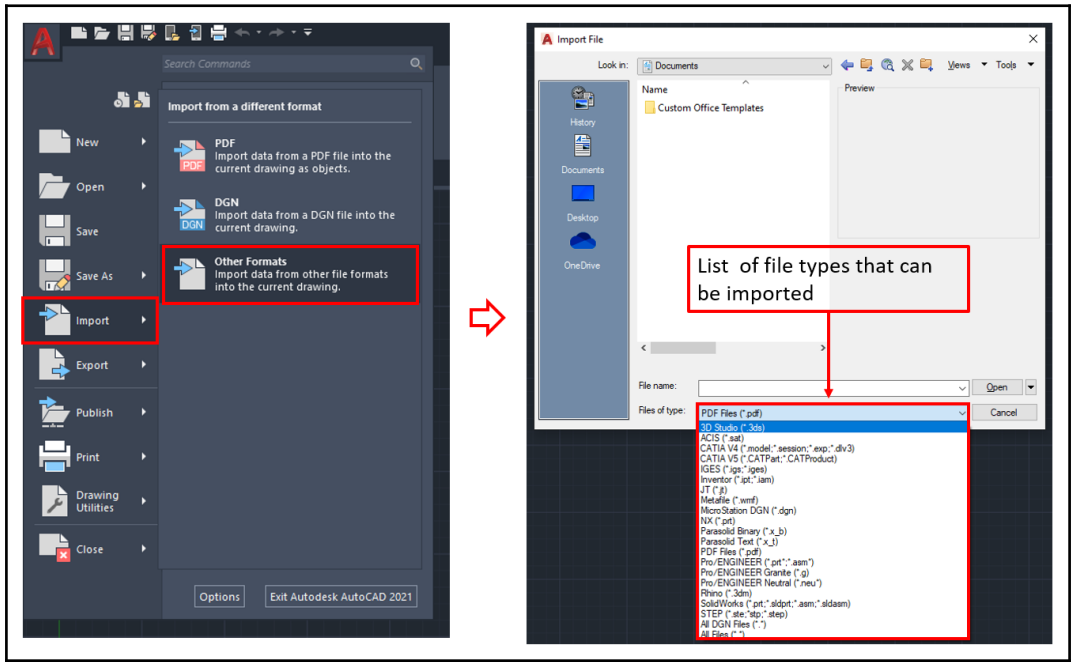


Figure 16.48: Importing external files

The opposite scenario can also occur, where you have to export your 3D model to a different file format. This can be done also by clicking on the **Application Menu**, selecting **Export**, and then clicking on **Other Formats**. This will open the **Export Data** window, as shown in the following screenshot:

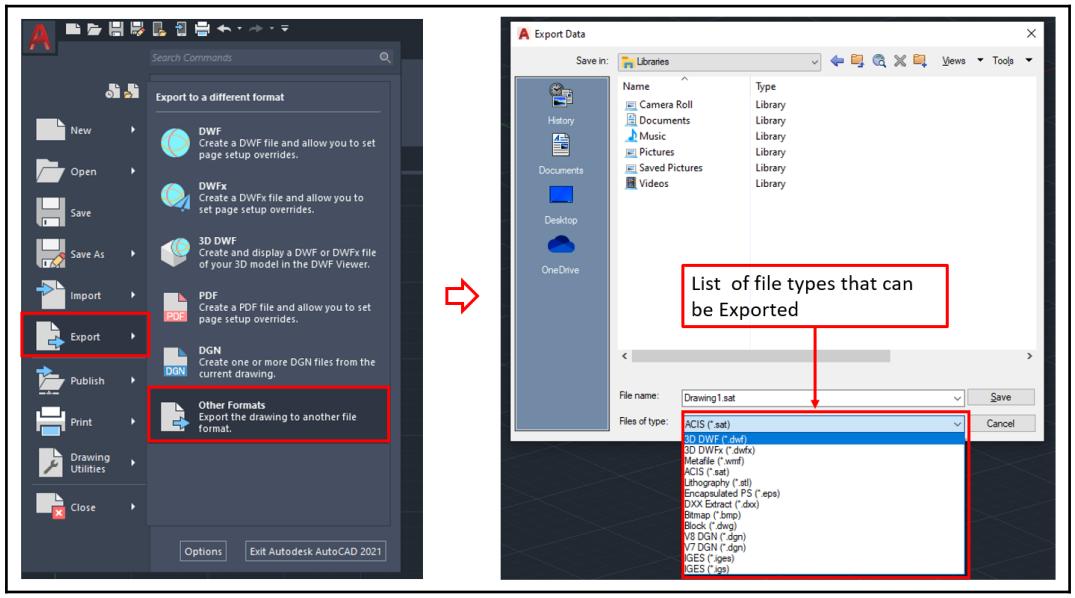


Figure 16.49: Exporting different file formats

As shown in the previous Figure, there are a number of different common 3D file formats that can be used, such as the `ACIS (*.sat)` file format and the `IGES` file format. After selecting the file type, type the desired name of the exported file, browse to the desired location, and click on **Save**.

Summary

In this chapter, the preparation of the scene and producing rendered output images were discussed in detail. The scene preparation discussion started with the addition and modification of the settings of light sources. The different light sources available in AutoCAD were covered in detail, showing the key differences between them, followed by the different settings for displaying shadows.

After that, materials were discussed. The discussion included the application of materials on the model bodies, the creation of new materials, and different related settings, such as material mapping.

The final step in setting the scene is to create the view cameras. The creation of cameras and different related settings were discussed in detail, followed finally by the actual rendering process.

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