Introduction to Drafting and AutoCAD 2D

Introduction to Drafting and AutoCAD 2D

Wally Baumback

BCCAMPUS
VICTORIA, B.C.

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Baumback, W. (2021). *Introduction to drafting and AutoCAD 2D* (B. McGarvie, Ed.). BCcampus. https://opentextbc.ca/autocad2d/

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Ebook ISBN: 978-1-77420-134-3
Print ISBN: 978-1-77420-133-6

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This book was produced with Pressbooks (https://pressbooks.com) and rendered with Prince.

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Accessibility of This Textbook

This textbook is an adaptation of an existing textbook that was not published by us. Due to its size and the complexity of the content, we did not have capacity to remediate the content to bring it up to our accessibility standards at the time of publication. This is something we hope to come back to in the future.

In the mean time, we have done our best to be transparent about the existing accessibility barriers and features below:

Accessibility Checklist

Element	Requirements	Pass?
Headings	Content is organized under headings and subheadings that are used sequentially.	Yes
Images	Images that convey information include alternative text descriptions. These descriptions are provided in the alt text field, in the surrounding text, or linked to as a long description.	No
Images	Images and text do not rely on colour to convey information.	Yes
Images	Images that are purely decorative or are already described in the surrounding text contain empty alternative text descriptions. (Descriptive text is unnecessary if the image doesn't convey contextual content information.)	Yes
Tables	Tables include row and/or column headers that have the correct scope assigned.	Yes
Tables	Tables do not have merged or split cells.	Yes
Tables	Tables have adequate cell padding.	Yes
Links	The link text describes the destination of the link.	Yes
Links	Links do not open new windows or tabs. If they do, a textual reference is included in the link text.	Yes
Links	Links to files include the file type in the link text.	Yes
Font	Font size is 12 point or higher for body text.	Yes
Font	Font size is 9 point for footnotes or endnotes.	Yes
Font	Font size can be zoomed to 200% in the webbook or book formats.	Yes

Known Accessibility Issues and Areas for Improvement

- The book relies heavily on screenshots from the AutoCAD software. These screenshots do not have alt text. While many of the screenshots are described in the surrounding text, the book has not been reviewed to ensure that the surrounding text is an adequate alternative for all images in the book.

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- Web form: BCcampus IT Support
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- **HTML**. An HTML file can be opened in a browser. It has very little style so it doesn't look very nice, but some people might find it useful.

For more information about the accessibility of this textbook, see the Accessibility Statement.

You can access the online webbook and download any of the formats for free here: Introduction to Drafting and AutoCAD 2D. To download the book in a different format, look for the "Download this book" drop-down menu and select the file type you want.

How can I use the different formats?

Format	Internet required?	Device	Required apps	Features
Online webbook	Yes	Computer, tablet, phone	An Internet browser (Chrome, Firefox, Edge, or Safari)	Option to enlarge text, and compatible with browser text-to-speech tools.
PDF	No	Computer, print copy	Adobe Reader (for reading on a computer) or a printer	Ability to highlight and annotate the text. If reading on the computer, you can zoom in.
EPUB	No	Computer, tablet, phone	An eReader app	Option to enlarge text, change font style, size, and colour.
HTML	No	Computer, tablet, phone	An Internet browser (Chrome, Firefox, Edge, or Safari)	Compatible with browser text-to-speech tools.

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    that chapter and use your browser's search feature by hitting **[Cntr] + [f]** on your keyboard if using a Windows computer or **[Command] + [f]** if using a Mac computer.
    - The **[Cntr] + [f]** and **[Command] + [f]** keys will also allow you to search a PDF, HTML, and EPUB files if you are reading them on a computer.
    - If using an book app to read this textbook, the app should have a built-in search tool.
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    - This textbook has a table of contents to help you navigate through the book easier. If using the online webbook, you can find the full table of contents on the book's homepage or by selecting "Contents" from the top menu when you are in a chapter.
- **Annotate the textbook**.
    - If you like to highlight or write on your textbooks, you can do that by getting a print copy, using the Digital PDF in Adobe Reader, or using the highlighting tools in eReader apps.

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Introduction to Drafting and AutoCAD 2D by Wally Baumback was funded by BCcampus Open Education.

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For more information about open education in British Columbia, please visit the BCcampus Open Education website. If you are an instructor who is using this book for a course, please fill out our Adoption of an Open Textbook form.

This book was produced using the following styles: AutoCAD 2D Style Sheet [Word file]

Introduction

Learning Outcomes

When you have completed this introduction, you will be able to:

1. List the software and template files required to complete the AutoCAD 2D book.
2. Describe this book's philosophy and explain how to use the book to learn how to apply the AutoCAD commands and features to draw 2D drawings.
3. Describe the conventions and symbols used in the AutoCAD 2D book.

Software Requirements

Operating System Software Required

This book was designed to be used on AutoCAD software that was designed for the Windows operating system.

AutoCAD Software Required

This book was written to be used with AutoCAD 2015, 2016, 2017, and 2018. You must have one of these software packages installed on your computer to complete the lab exercises in the book.

Template Files Required for the AutoCAD 2D book

To complete the lab exercises in this book, you must have the four template files, that accompany it, on your hard drive. Module 2 will teach you how to use them.

About the Book

Philosophy of the AutoCAD 2D book

The AutoCAD 2D Self-paced book was written as a tool to guide and teach you to master AutoCAD. No two students learn at the same pace, therefore the book was written with competency-based modules. The competency-based modules are bite-size pieces that allow you to work at your own pace. They can be used to learn by distance education, correspondence, online, instructor-lead classes, or by individuals teaching themselves to use AutoCAD in their own home or office.

Scope of the AutoCAD books

There are five AutoCAD Self-paced books. They are AutoCAD 2D, AutoCAD 3D, AutoCAD 2D Advanced, AutoCAD 3D Advanced, and AutoLISP.

Suggested Prerequisites

To get the most from this book, it is suggested that you have a working knowledge of Windows operating system. At the very least, you must have the ability to create folders plus move, copy, rename, and delete files.

Book Structure and Components

The modules in the AutoCAD 2D book were written in a very logical step-by-step order. The following information will help you get the most from this book.

The Modules

The modules in the book were written as bite-size pieces in a precise order and contain the information that is required to learn AutoCAD 2D.

Do not skim through the modules. Read each page and ensure that everything is understood in each module. Do not jump around inside the module or from module to module. Work methodically through each module, page by page.

As you work your way through each module, try to understand all the information it contains. To ensure that it is understood, you must be able to complete the lab exercises to prove it. Only when everything in the module is understood and you can complete the lab exercises, in the specified time limit, should you go to the next module.

Each module may contain Must Know’s, User Tips, Workalongs, Lessons, Key Principles, and Lab Exercises.

Must Knows

Most modules contain user Must Knows. You *must* understand and retain each one of these principles as you work your way through the modules. If you cannot understand any one of them, read back through the module. Do not go on until you fully understand it. Here is an example:

(Do not attempt to understand this Must Know now. It is here as an example only.)

> **MUST KNOW:** The Enter key or the Space bar are used to execute commands. If either one is pressed when there is no active command, AutoCAD will repeat the last command. Right clicking the mouse will produce the same results.

User Tips

User Tips are in the modules to help you draw in AutoCAD. They contain tips, tricks and ways to use commands that will help you draw faster and increase your productivity. Study them and try to use the tips while completing the labs exercises. They are there to help you work smarter, not harder. Here is an example:

(Do not attempt to learn this user tip now. It is here as an example only.)

USER TIP: The function key F3 toggles the AutoSnap feature off and on. Although there are other ways to do this, try to get into the habit of using F3 as it will greatly improve your drawing speed.

Workalongs

A Workalong is an exercise that you complete by working along with the module, tutorial style, to complete a drawing using AutoCAD. Completing the Workalongs will help you understand how the commands and features being taught in that module work by actually using them in AutoCAD. This will prepare you to complete the drawings in the lab exercises without any assistance.

(Do not attempt this workalong now, it is here for an example only.)

WORKALONG: Drawing Lines Using Cartesian Coordinates – Part 1

Step 1

Using the NEW command, start a new drawing using the template 2D English.

Step 2

Save and name the drawing AutoCAD 2D Workalong 04-1. Save it in the folder CAD Courses/AutoCAD 2D/Lab Exercises.

Step 3

Enter the LINE command, as shown below, to draw the object shown in the figure. Keep in mind that you enter what is in bold font, the author’s comments are in italics, and everything else are AutoCAD responses or prompts. (Figure Step 3)

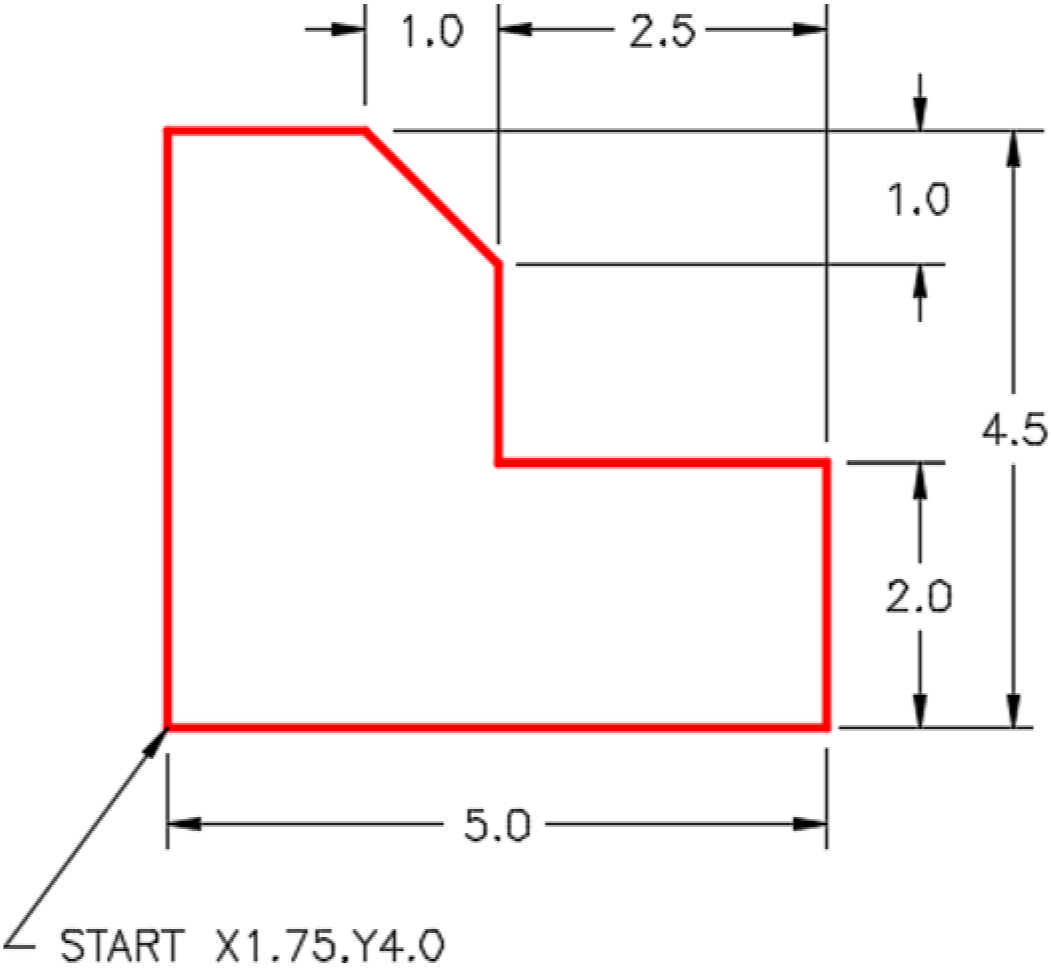


Figure Step 3

Command: **LINE**

Specify first point: **1.75,4**

(Always start with an absolute coordinate. It must be a X then a Y separated with a comma.)

Specify next point or [Undo]: **@5,0**

(Then change to relative coordinates. Note the @ first, then X and Y.)

Specify next point or [Undo]: **@0,2**

Specify next point or [Close/Undo]: **@−2.5,0**

Geometry and Drafting Lessons

Some modules contain drafting and/or geometry lessons. They are included to teach users who do not have any previous drafting/design knowledge or experience. They may also be handy for drafters or designers who need a refresher lesson.

If you already know and understand the theory in the lesson, you can skip it and go on to the next topic in the module. Knowing the theory taught in the lessons will help you complete the lab exercises.

(Do not attempt this geometry lesson now. It is here for an example only.)

Geometry Lesson: Circles

A *circle* is defined as a closed curve in which all points are the same distance from its center point. The center point is a single XY coordinate.

A circle is 360 degrees and can be divided into four *quadrants*. All points on a circle are at a given distance from is center point. The distance between any of the points and the center is called the *radius*.

Study the drawings in Figure 7-1, 7-2, 7-3, and 7-4 for a description of the geometry of a circle.

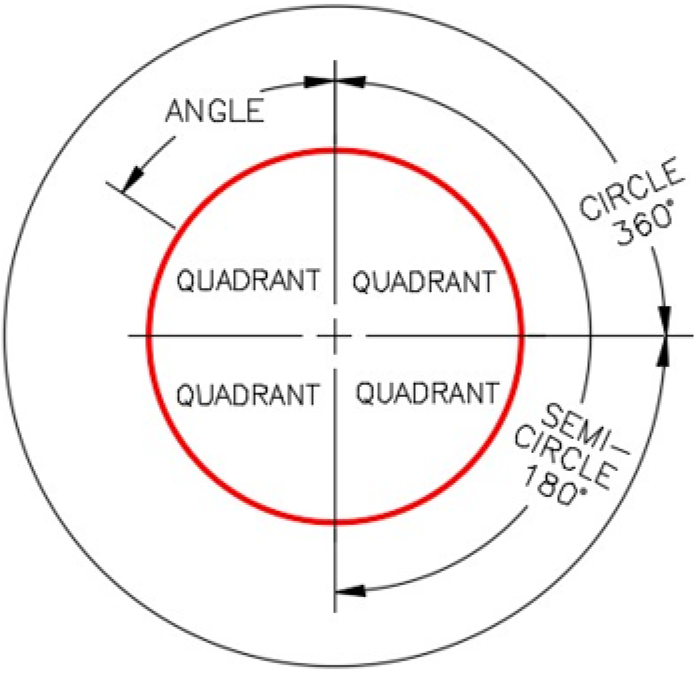


Figure 7-1 Geometry of a Circle – Part 1

Key Principles

Each module contains a list of Key Principles at the end. The Key Principles are principles that you should have learned and understand in that module. It is important that you fully understand each one as you will be required to use them in future AutoCAD work.

(Do not attempt to learn the key principles now. It is here for an example only.)

Key Principles in Module 4

1. The @ symbol means “The last absolute coordinate location,” or sometimes called the lastpoint.
2. To close the last line of a series of lines, enter either C (Close) or the absolute coordinate of the first.

3. To delete existing drawing objects, you can either use the ERASE command or the Delete key on the.
4. Objects can either be selected before or after a command is.
5. To unselect an drawing object, press the Esc Sometimes it has to be pressed twice.

Lab Exercises

All lab exercises have a time limit allocated to them. To complete the lab exercise in the specified time limit, you must be able to complete the drawing, check it for accuracy and make any corrections. If you cannot complete it in the time allowed, redo the entire lab exercise. It is your way of proving that you have mastered the module.

Almost all modules contain at least two lab exercises. It is strongly suggested that you complete all lab exercises in the module.

Checking the Accuracy of Your Drawings

Almost all lab exercises have a key provided that you can use to check the drawing's accuracy. This is taught in Module 3: Checking the Accuracy of Your Drawings.

Competency Tests

Every fifth module is a timed competency test module. A Competency Test Module has multiple choice questions and a comprehensive lab exercise to test your mastery of the last set of modules completed. There is no key supplied with this module since it is meant to be checked by your instructor. If there are any parts of this module that you have trouble completing or cannot complete it in the time allowed, go back and reread the module or modules containing the information that you are having trouble with. If necessary, redo any lab exercises to help you learn the material.

Book Conventions and Symbols

The following conventions and symbols are used in the book to help you understand the material.

New Terms: Red Italics

Words in red italics are new terms being introduced in that module. They will only appear in red italics the first time they are appear and will be defined. Here is an example:

The *current layer* is the layer that AutoCAD will place all newly created drawing objects on.

Bolded Words and Numbers

In Workalongs, all words and numbers in bold font are input by the user. It is your way of knowing if it is your input or an AutoCAD response. Here is an example:

Command: **LINE**
Specify first point: **1.75,4**

```
Specify next point or [Undo]: @5,0
Command:
```

AutoCAD Commands: Uppercase

AutoCAD commands are always in uppercase. Here is an example:

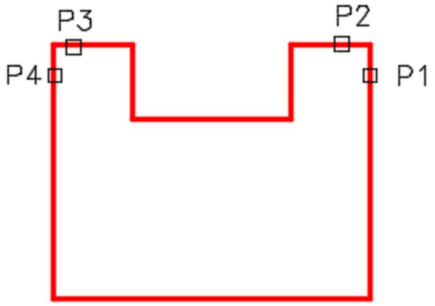
To construct a fillet, use the FILLET command. The rule of thumb to follow is, “If the arc you are drawing is tangent to both objects it is connecting to, use the FILLET command to insert it.”

Picking Objects or Locations on the Drawing

To help understand when an object or a location is picked, the modules use the letter “P” along with a number that indicates which pick it is in the sequence of picks.

(Do not attempt this workalong now. It is here as an example only.)

```
Command: FILLET
Current settings: Mode = TRIM, Radius = 0.6000
Select first object or [Polyline/Radius/Trim]: R
Specify fillet radius <0.6000>: .25
(Set the radius for the fillet.)
Select first object or [Polyline/Radius/Trim]: P1
Select second object: P2
(Pick the two lines to insert the fillet between.)
Command: FILLET
Current settings: Mode = TRIM, Radius = 0.2500
Select first object or [Polyline/Radius/Trim]: P3
Select second object: P4
Command:
```



Author’s Comments: Green, Italic Text and Green Textboxes

Comments by the author, to help explain what is happening in a command, are in green italics and enclosed in parentheses.

(Do not attempt these commands now. They are here for an example only.)

```
Specify first point: 1.75,4
(Always start with an absolute coordinate. It must be X then Y with a comma between them.)
Specify next point or [Undo]: @5,0
(Then change to relative coordinates. Note the @ first, then X and Y.)
Specify next point or [Undo]: @0,2
Specify next point or [Close/Undo]: @−2.5,0
(Note the negative coordinate since we are going in the negative X direction.)
Specify next point or [Close/Undo]: @0,1.5
Specify next point or [Close/Undo]: C
```

(You can use a C to close the last line and return to the first point.)
Command:

There are also author comments in the workalong or lab exercise to help you understand a principle or to complete a task. These will appear in a green textbox and be labelled "Author's comments." Here is an example:

AUTHOR'S COMMENTS: The Insertion scale for the all lab exercises in the AutoCAD 2D book are shown in the Units column as shown in the Figure 3-1.

Key Principles

Key Principles in Module Introduction 1

1. To complete the lab exercises in this book, you must have the four template files, that accompany it, on your hard disk drive.
2. Do not skim through the modules. Read and understand everything in each module. Do not jump around inside the module or from module to module. Work methodically through each module, page by page.
3. The modules were written as competency-based bite-size pieces to allow you to work at your own pace and learn to use AutoCAD. Do not go onto the next module until you fully understood the module you are currently working on and have completed all lab exercises.
4. All lab exercises have a time limit allocated to them. To complete a lab exercise in the specified time limit, you must be able to complete the drawing, check it for accuracy and make any corrections. If you cannot do this in the time allowed, redo the entire lab exercise. It is your way of proving that you mastered the module.

Do not memorize how to use AutoCAD – UNDERSTAND IT.

Have fun on your journey into the fascinating world of mastering AutoCAD.

Using AutoCAD is a never ending learning process and you will continue learning long after completing this book

Configure Your AutoCAD Software

Learning Outcomes

When you have completed this module, you will be able to:

1. Configure your AutoCAD software for the AutoCAD 2D book.

IMPORTANT – PLEASE READ BEFORE COMPLETING THIS MODULE

If you are working on the AutoCAD 2D course in an instructor-led educational institute and only completing the workalongs and lab exercises on the computers at your school or college, skip this module. Go directly to Module 1.

If you are working on the AutoCAD 2D book in an instructor-led educational institute and completing the workalongs and lab exercises on your personal computer, do the following:

1. Go to the BCcampus Open Education website and search for “AutoCAD 2D.” Download the Templates that are listed under “Supplementary Materials”
2. After you download the templates, complete the remainder of this module to configure your personal AutoCAD software. Complete Steps 1 to 48 in the next Workalong. Do not start Module 1 until you completed all 48 steps.

This book was written to be used with AutoCAD running on a computer with Windows software.

WORKALONG: Configuring Your AutoCAD Software

Step 1

Start AutoCAD by clicking the desktop icon. In AutoCAD’s Graphic window, click the New icon located in the upper left corner. It will open the Select Template dialogue box (Figure Step 1).

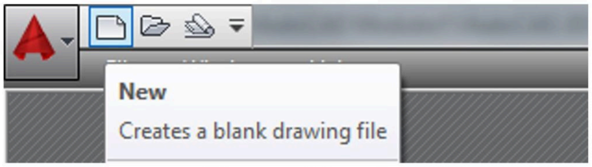


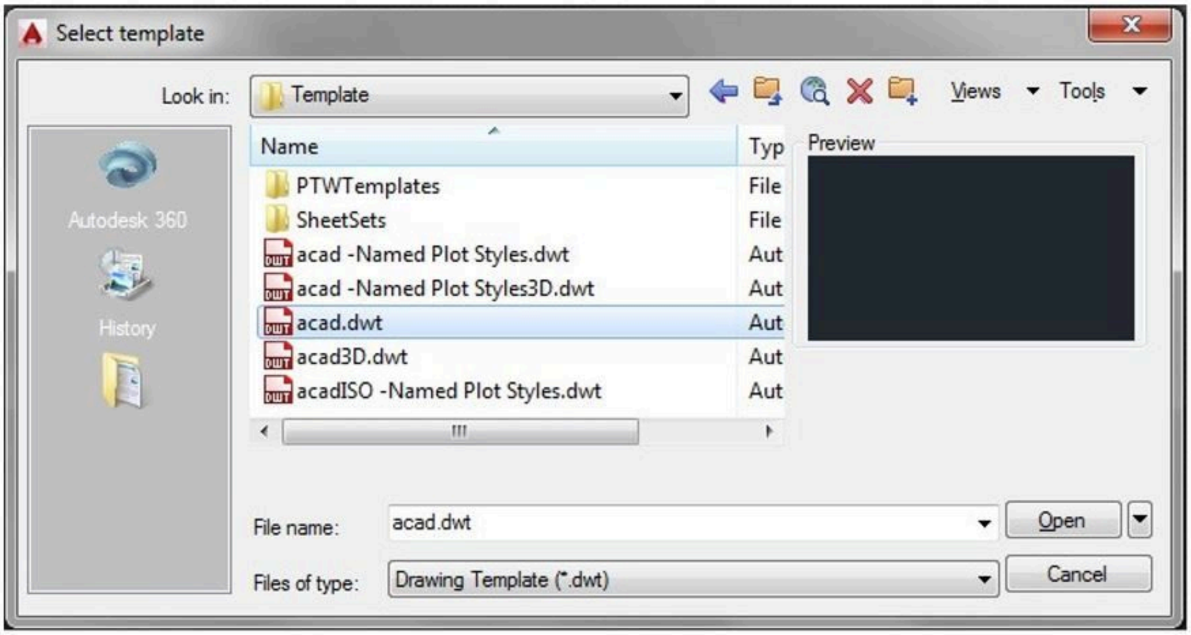
Figure Step 1

Figure Step 1

Step 2

In the Select Template dialogue box, click the template file: acad.dwt (or just acad on some computers) to highlight it. Then click the Open button. This will display AutoCAD's Graphic window. (Figure Step 2)

AUTHOR'S COMMENTS: Your dialogue box may not match the figure exactly.



Look in: Template

Name	Typ
PTWTemplates	File
SheetSets	File
acad -Named Plot Styles.dwt	Aut
acad -Named Plot Styles3D.dwt	Aut
acad.dwt	Aut
acad3D.dwt	Aut
acadISO -Named Plot Styles.dwt	Aut

File name: acad.dwt

Files of type: Drawing Template (*.dwt)

Figure Step 2

Step 3

Disable all features on the Status bar by clicking any that display with a blue background as shown in Figure Step 3A. All features should display with a gray background as shown in Figure Step 3B. (Figure Step 3A and 3B)

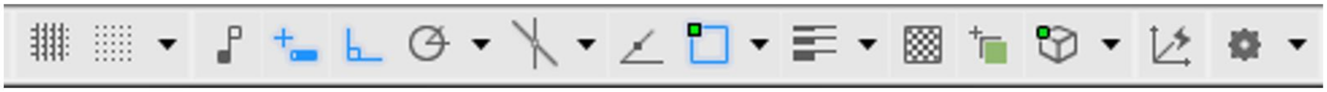


Figure Step 3A

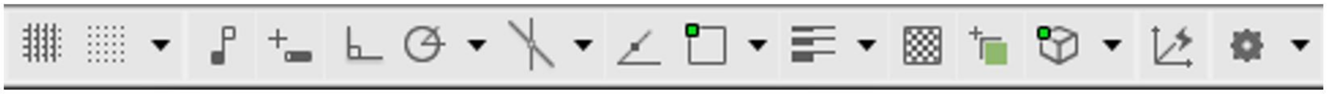
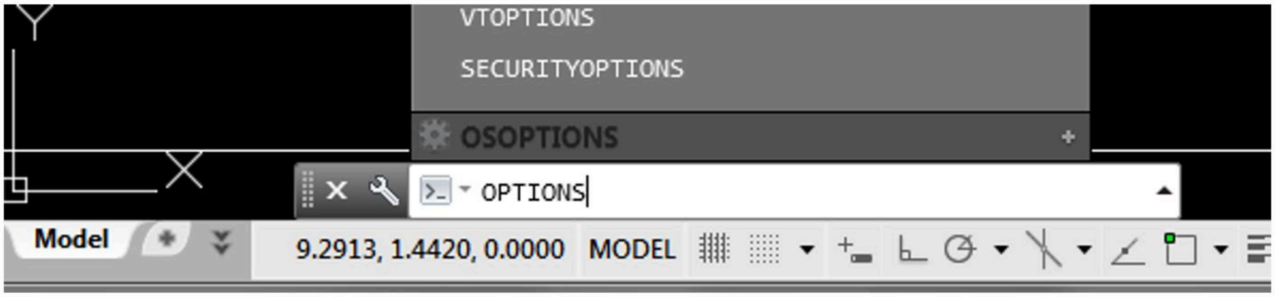


Figure Step 3B

Step 4

In the Command Line window, located on the bottom of the Graphic window, enter the OPTIONS command by typing it on your keyboard. This will open the Options dialogue box. (Figure Step 4)



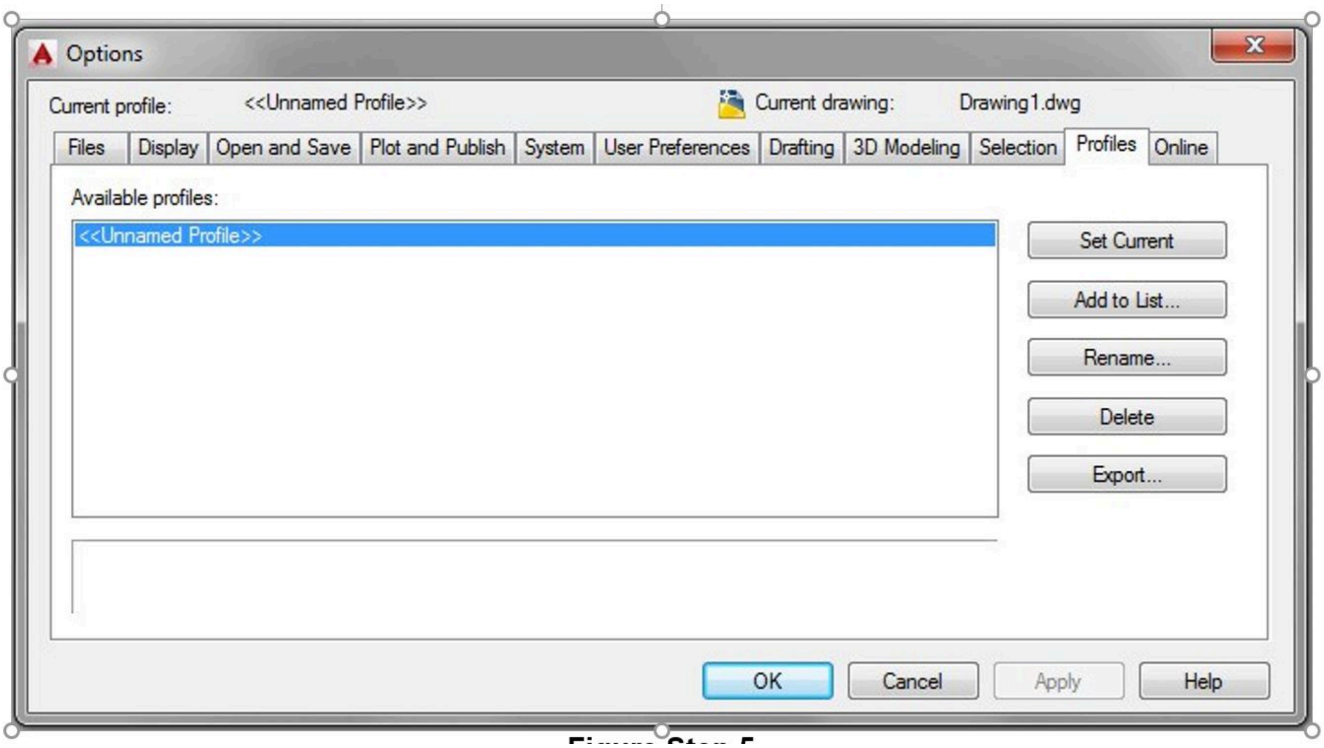
```
OPTIONS
```

VTOPTIONS
SECURITYOPTIONS
OSOPTIONS
Model
9.2913, 1.4420, 0.0000
MODEL

Figure Step 4

Step 5

In the Options dialogue box, enable the Profiles tab. (Figure Step 5)



Current profile: <<Unnamed Profile>>

Current drawing: Drawing1.dwg

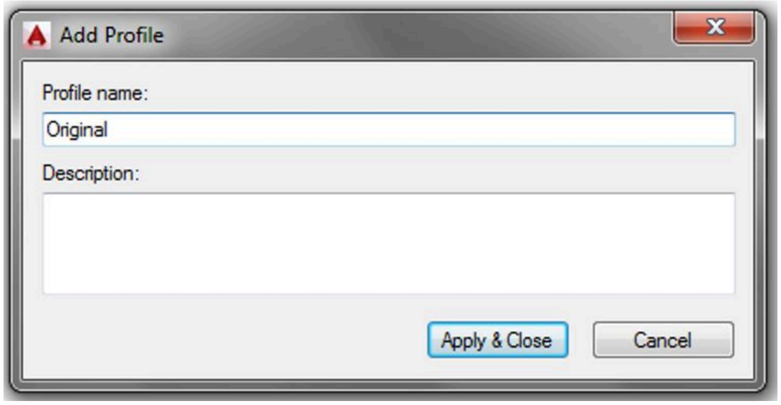
Available profiles: <<Unnamed Profile>>

Options
Files
Display
Open and Save
Plot and Publish
System
User Preferences
Drafting
3D Modeling
Selection
Profiles
Online
Set Current
Add to List...
Rename...
Delete
Export...
OK
Cancel
Apply
Help

Figure Step 5

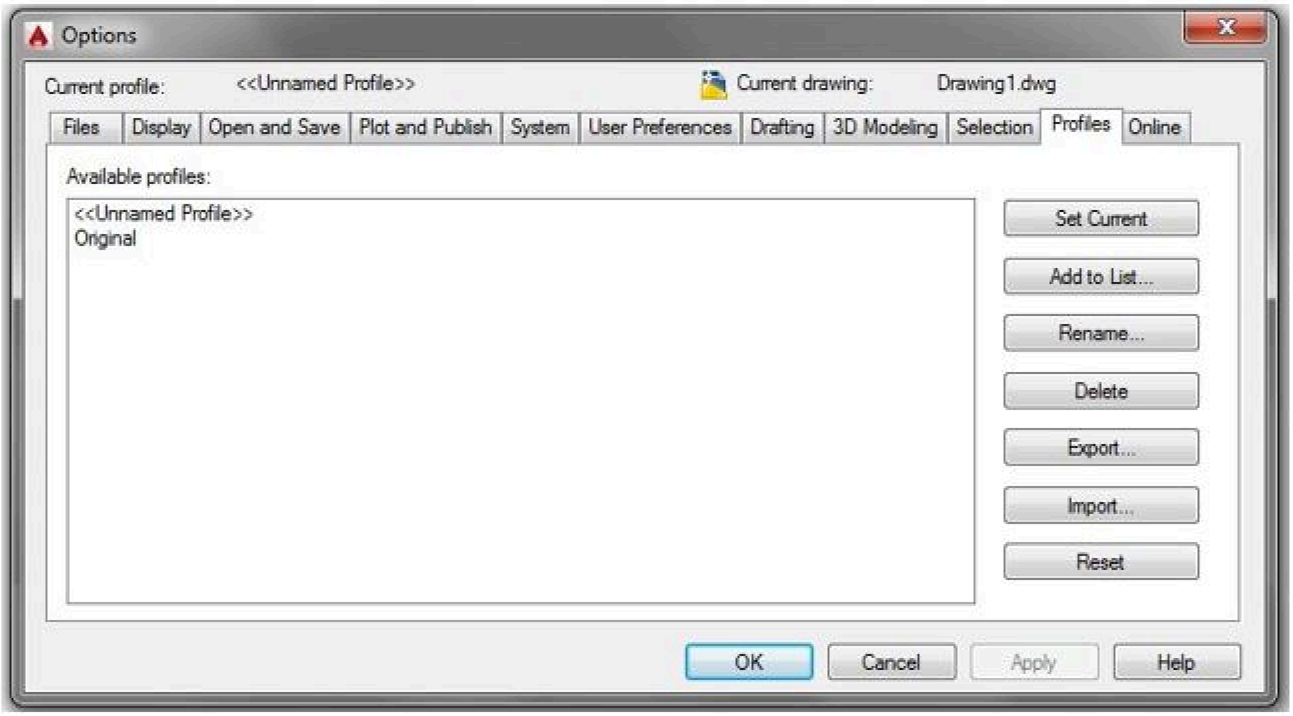
Step 6

Click the Add to List button. This will open the Add Profile dialogue box. Enter the name: Original for the Profile name. (Figure Step 6A and 6B)


Add Profile
Description:
Apply & Close
Cancel

Profile name: Original

Figure Step 6A


Options
Files
Display
Open and Save
Plot and Publish
System
User Preferences
Drafting
3D Modeling
Selection
Profiles
Online
Set Current
Add to List...
Rename...
Delete
Export...
Import...
Reset
OK
Cancel
Apply
Help

Current profile: <<Unnamed Profile>>

Current drawing: Drawing1.dwg

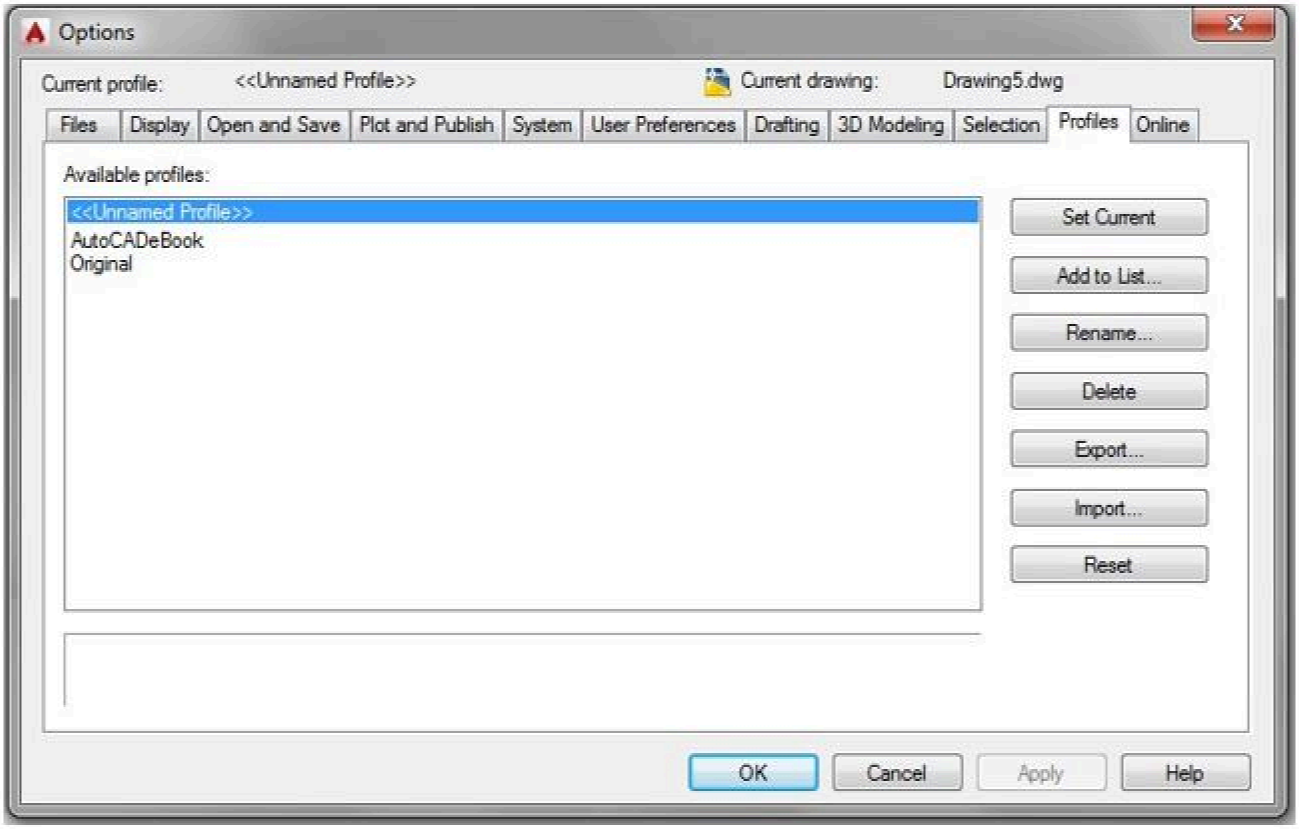
Available profiles:
<<Unnamed Profile>>
Original

Figure Step 6B

AUTHOR'S COMMENTS: I am having you create a profile named Original so that you can go back to the way your AutoCAD was configured when you started this book. You will not be using this profile during the course, but at anytime (or when you complete the book) you can use this original profile to restore the original configuration.

Step 7

Using what you learned in Step 6, create a new profile named: AutoCADbook. (Figure Step 7)

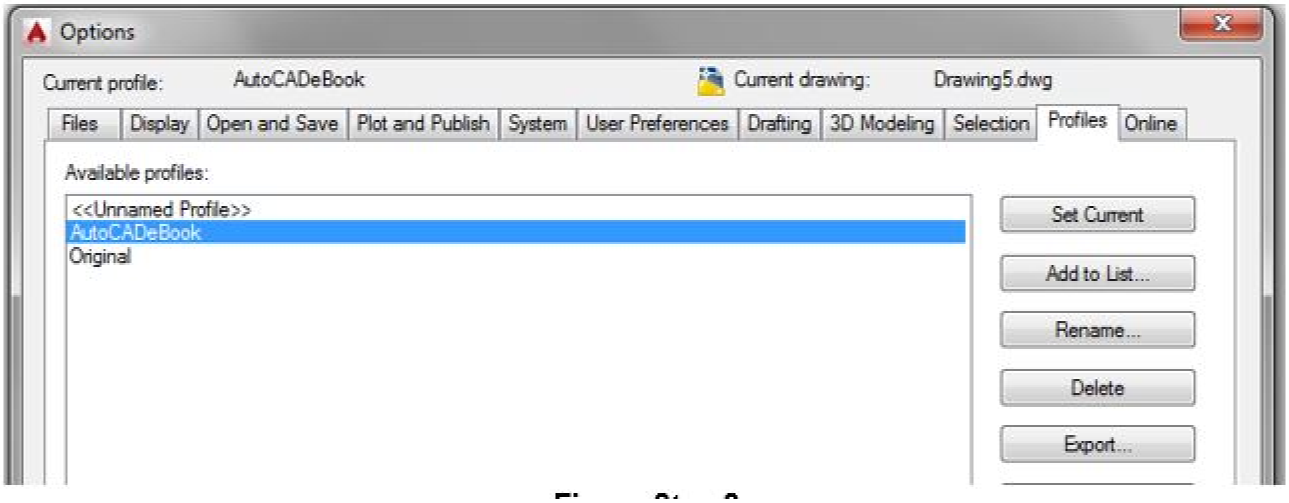


Options
Current profile: <<Unnamed Profile>>
Current drawing: Drawing5.dwg
Files
Display
Open and Save
Plot and Publish
System
User Preferences
Drafting
3D Modeling
Selection
Profiles
Online
Available profiles:
<<Unnamed Profile>>
AutoCADeBook
Original
Set Current
Add to List...
Rename...
Delete
Export...
Import...
Reset
OK
Cancel
Apply
Help

Figure Step 7

Step 8

Click the AutoCADbook profile to highlight it. While it is selected, click the Set Current button. Ensure that the Current profile is AutoCADbook as shown across the top of the dialogue box. (Figure Step 8)


Options
Files
Display
Open and Save
Plot and Publish
System
User Preferences
Drafting
3D Modeling
Selection
Profiles
Online
Available profiles:
<<Unnamed Profile>>
AutoCADeBook
Original
Set Current
Add to List...
Rename...
Delete
Export...

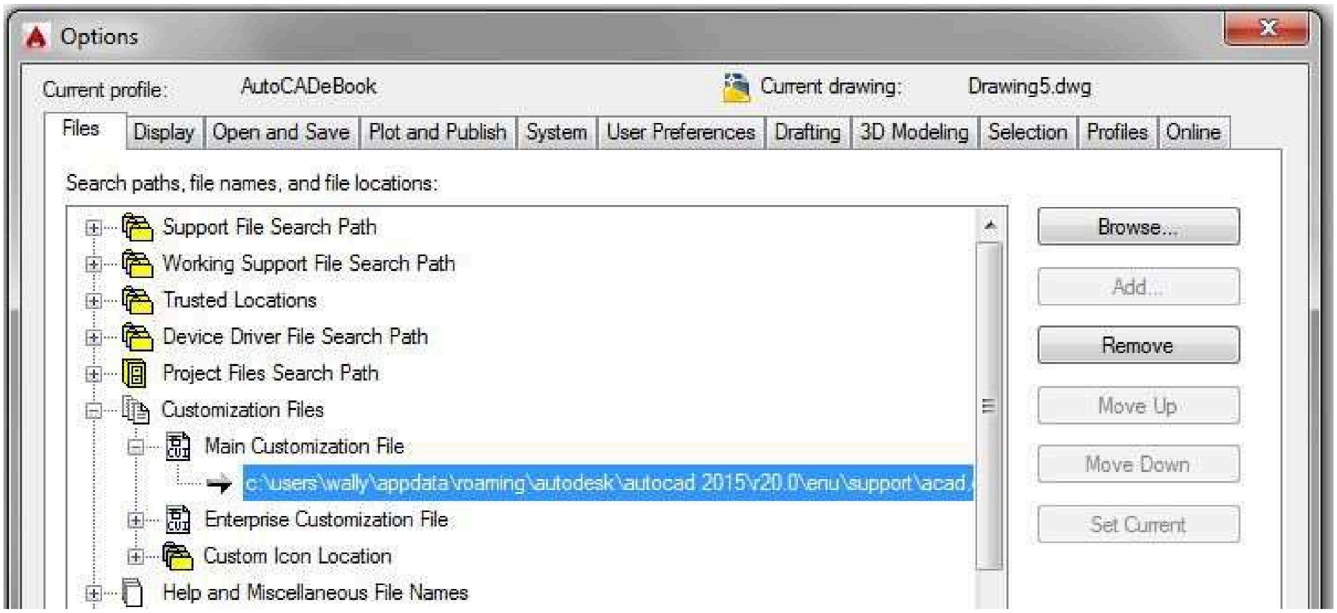
Current profile: AutoCADeBook

Current drawing: Drawing5.dwg

Figure Step 8

Step 9

Enable the display of the Files tab. Click the + on Customization Files to expand it. Click the + on the Main Customization File to expand it. Click the file and location to highlight it as shown in the figure. While it is selected, click the Remove button. (Figure Step 9)


Options
Files
Display
Open and Save
Plot and Publish
System
User Preferences
Drafting
3D Modeling
Selection
Profiles
Online
Search paths, file names, and file locations:
Support File Search Path
Working Support File Search Path
Trusted Locations
Device Driver File Search Path
Project Files Search Path
Customization Files
Main Customization File
Enterprise Customization File
Custom Icon Location
Help and Miscellaneous File Names
Browse...
Add...
Remove
Move Up
Move Down
Set Current

Current profile: AutoCADeBook

Current drawing: Drawing5.dwg

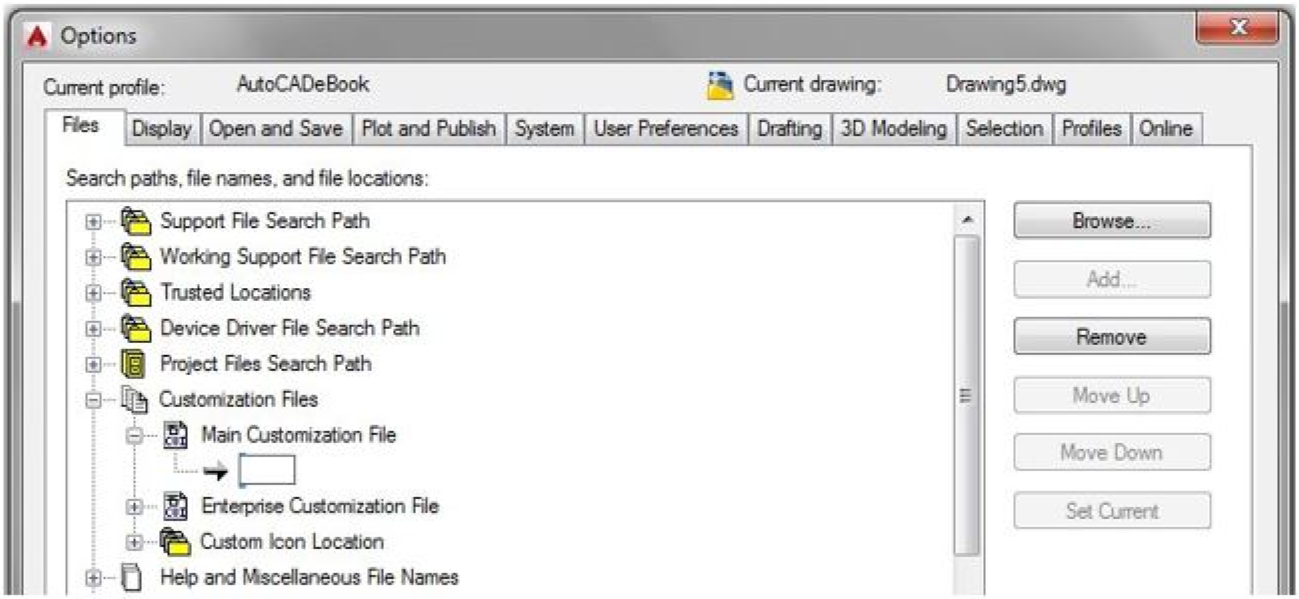
c:\users\wally\appdata\roaming\autodesk\autocad 2015\r20.0\enu\support\acad

Figure Step 9

AUTHOR'S COMMENTS: Your dialogue box may not match the figure exactly.

Step 10

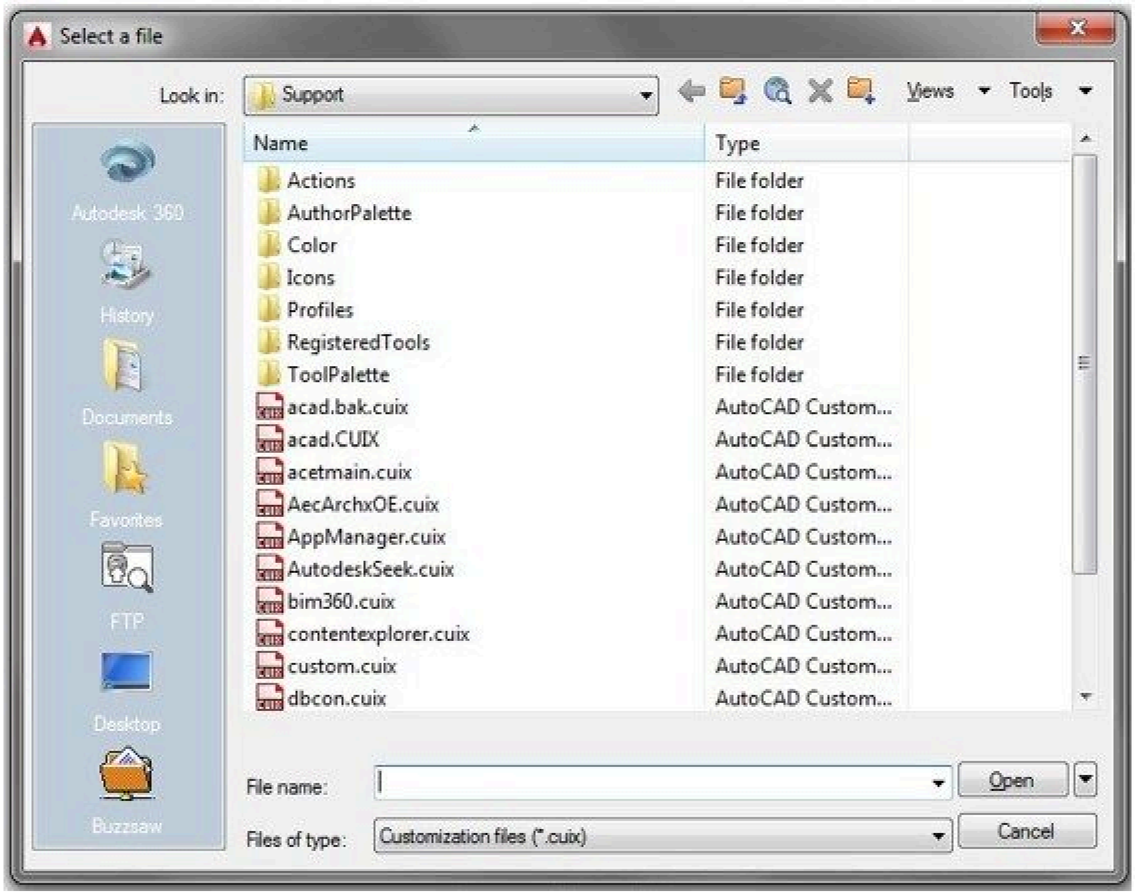
Step 9 will remove the customization file and show a blank box as shown in the figure. Click the Browse button to open the Select a file dialogue box. (Figure Step 10A and 10B)


Options
Files
Display
Open and Save
Plot and Publish
System
User Preferences
Drafting
3D Modeling
Selection
Profiles
Online
Search paths, file names, and file locations:
Support File Search Path
Working Support File Search Path
Trusted Locations
Device Driver File Search Path
Project Files Search Path
Customization Files
Main Customization File
Enterprise Customization File
Custom Icon Location
Help and Miscellaneous File Names
Browse...
Add...
Remove
Move Up
Move Down
Set Current

Current profile: AutoCADeBook

Current drawing: Drawing5.dwg

Figure Step 10A


Select a file
Views
Tools
Autodesk 360
History
Documents
Favorites
FTP
Desktop
Buzzsaw
Open
Cancel

Look in: Support

Name	Type
Actions	File folder
AuthorPalette	File folder
Color	File folder
Icons	File folder
Profiles	File folder
RegisteredTools	File folder
ToolPalette	File folder
acad.bak.cuix	AutoCAD Custom...
acad.CUIX	AutoCAD Custom...
acetmain.cuix	AutoCAD Custom...
AecArchxOE.cuix	AutoCAD Custom...
AppManager.cuix	AutoCAD Custom...
AutodeskSeek.cuix	AutoCAD Custom...
bim360.cuix	AutoCAD Custom...
contentexplorer.cuix	AutoCAD Custom...
custom.cuix	AutoCAD Custom...
dbcon.cuix	AutoCAD Custom...

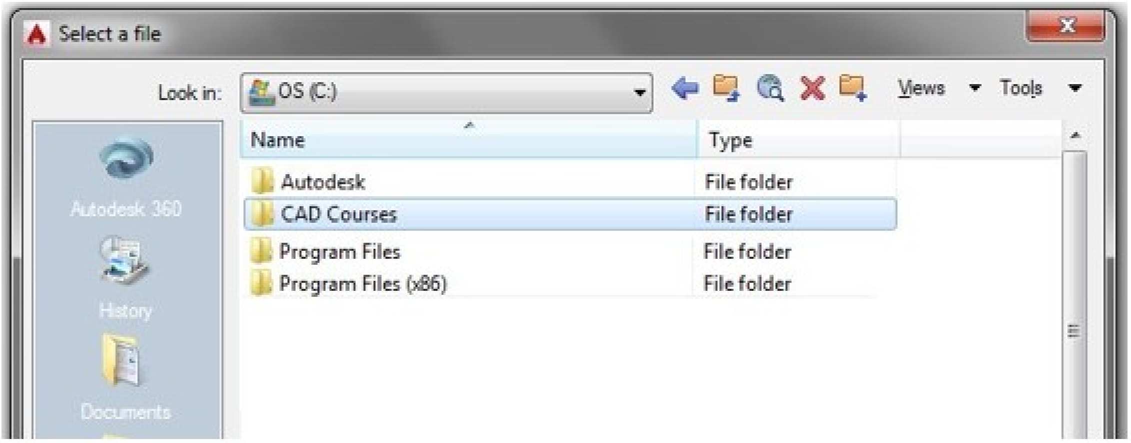
File name:

Files of type: Customization files (*.cuix)

Figure Step 10B

Step 11

In the Select a file dialogue box, pull down the Look in: box and find C drive. Click the folder: CAD Courses. (Figure Step 11)



Select a file

Look in: OS (C:)

Name	Type
Autodesk	File folder
CAD Courses	File folder
Program Files	File folder
Program Files (x86)	File folder

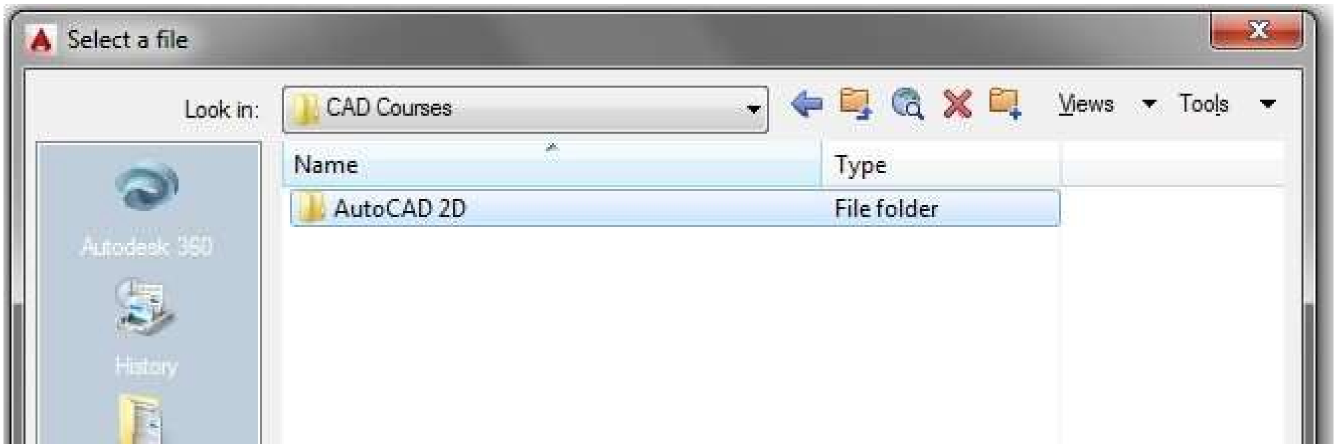
Views
Tools
Autodesk 360
History
Documents

Figure Step 11

AUTHOR'S COMMENTS: If you used a different drive when you downloaded the templates and menu from the website, you will have to find that drive letter instead of drive C.

Step 12

Under CAD Courses, click the folder: AutoCAD 2D. (Figure Step 12)



Select a file

Look in: CAD Courses

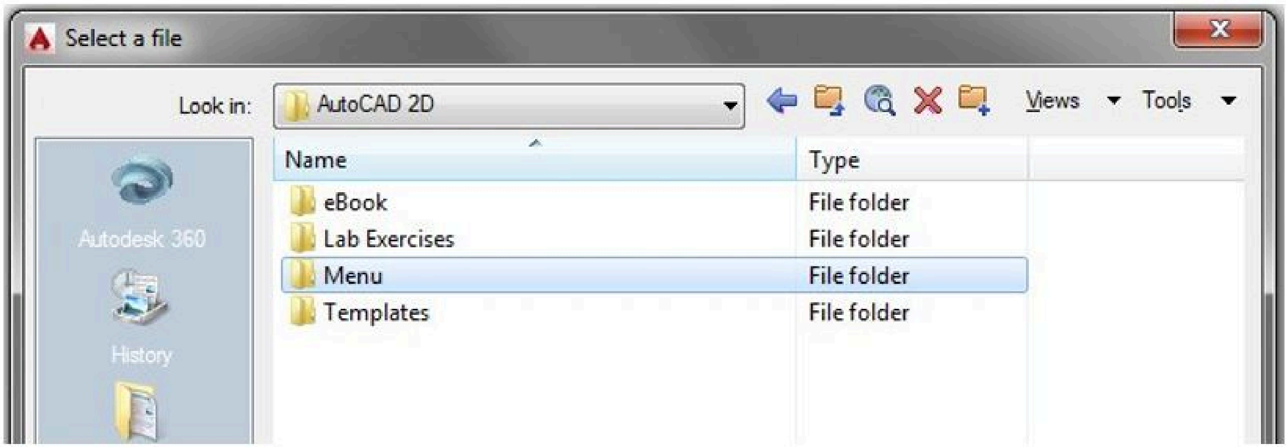
Name	Type
AutoCAD 2D	File folder

Views
Tools
Autodesk 360
History

Figure Step 12

Step 13

Under the folder: AutoCAD 2D, click the folder: Menu. (Figure Step 13)

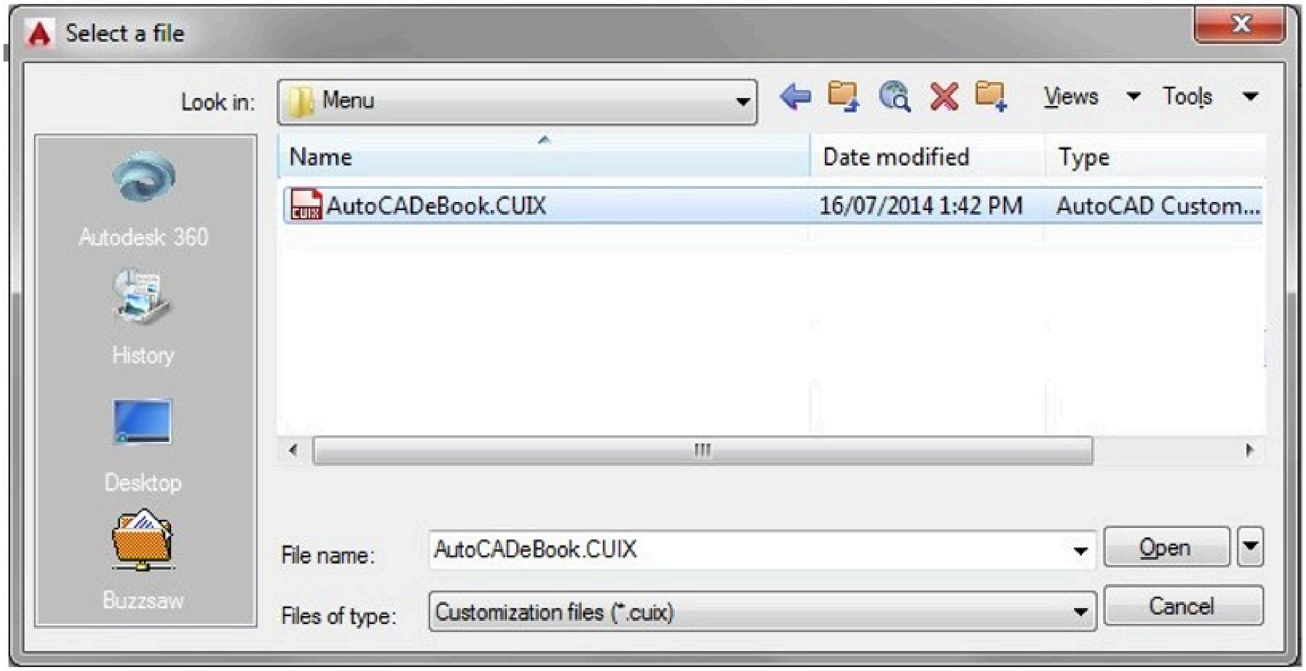

Select a file
Look in:
AutoCAD 2D
Views
Tools
Autodesk 360
History

Name	Type
eBook	File folder
Lab Exercises	File folder
Menu	File folder
Templates	File folder

Figure Step 13

Step 14

In the folder: Menu, click the file: AutoCADbook.CUIX and then the Open box. (Figure Step 14)


Select a file
Look in:
Menu
Views
Tools
Autodesk 360
History
Desktop
Buzzsaw
Open
Cancel

Name	Date modified	Type
AutoCADeBook.CUIX	16/07/2014 1:42 PM	AutoCAD Custom...

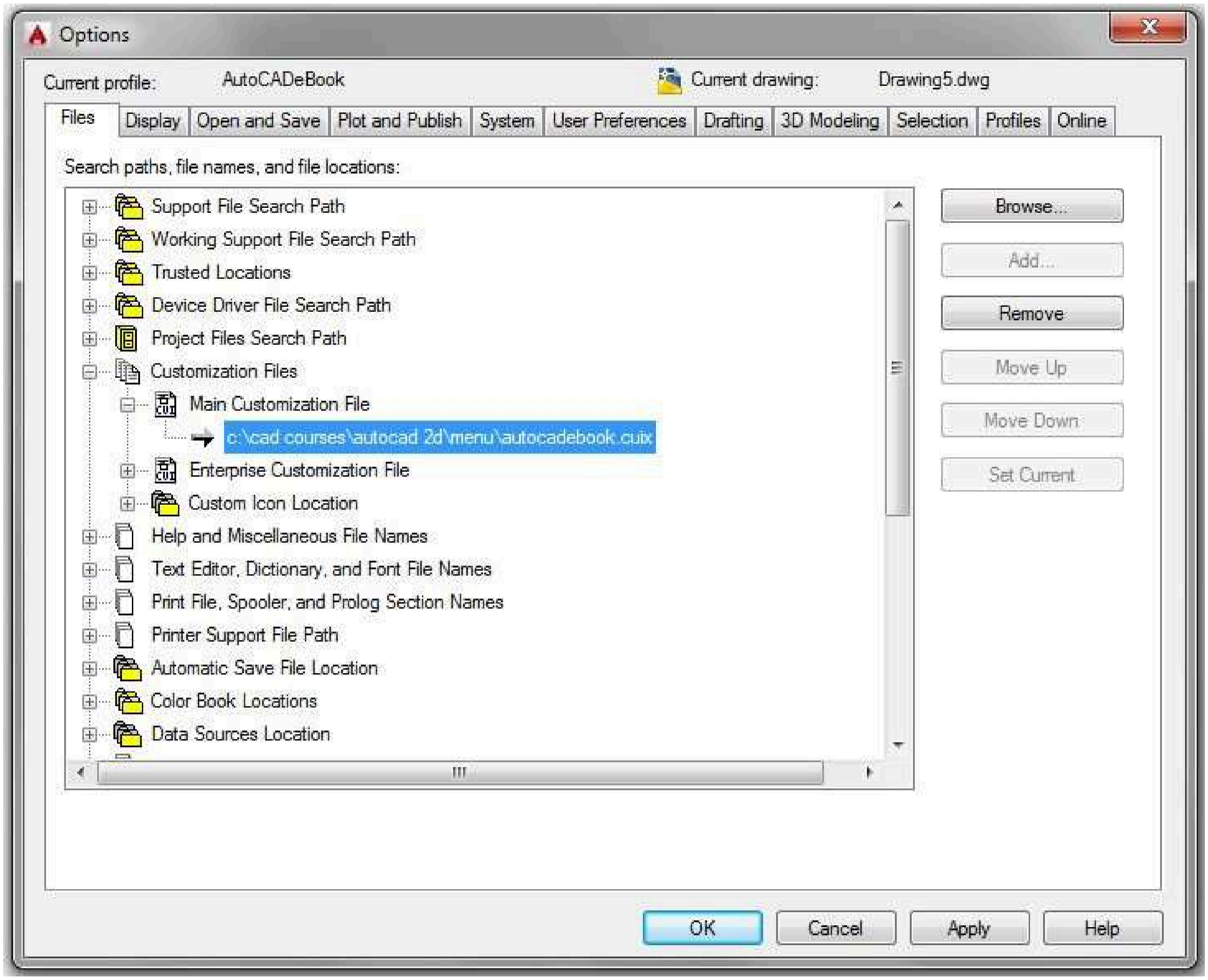
File name: AutoCADeBook.CUIX

Files of type: Customization files (*.cuix)

Figure Step 14

Step 15

You should now see the file location:
C:\CAD Course\AutoCAD 2D\Menu\AutoCADbook.CUIX. (Figure Step 15A and 15B)


Options
Files
Display
Open and Save
Plot and Publish
System
User Preferences
Drafting
3D Modeling
Selection
Profiles
Online
Browse...
Add...
Remove
Move Up
Move Down
Set Current
OK
Cancel
Apply
Help

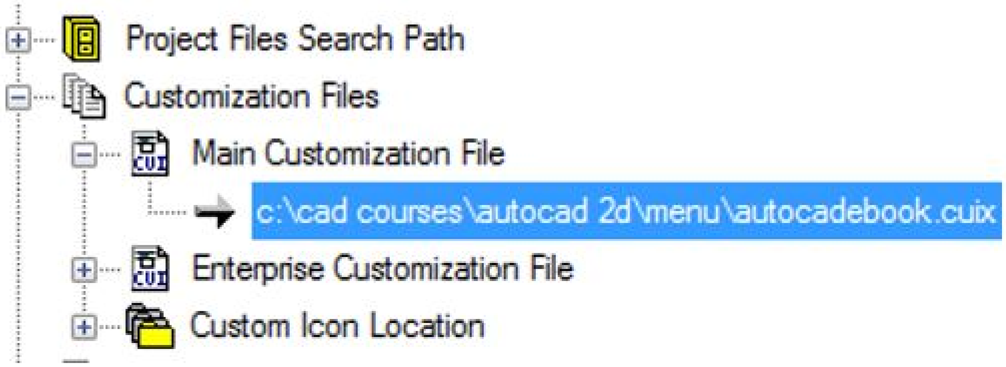
Current profile: AutoCADeBook

Current drawing: Drawing5.dwg

Search paths, file names, and file locations:

- Support File Search Path
- Working Support File Search Path
- Trusted Locations
- Device Driver File Search Path
- Project Files Search Path
- Customization Files
  - Main Customization File
    - c:\cad courses\autocad 2d\menu\autocadebook.cuix
  - Enterprise Customization File
  - Custom Icon Location
- Help and Miscellaneous File Names
- Text Editor, Dictionary, and Font File Names
- Print File, Spooler, and Prolog Section Names
- Printer Support File Path
- Automatic Save File Location
- Color Book Locations
- Data Sources Location

Figure Step 15A



- Project Files Search Path
- Customization Files
  - Main Customization File
    - c:\cad courses\autocad 2d\menu\autocadebook.cuix
  - Enterprise Customization File
  - Custom Icon Location

Figure Step 15B

Step 16

Click the small arrow in the Workplace Switching icon on the right end of the Status bar located along the bottom of the Graphic window. (Figure Step 16)

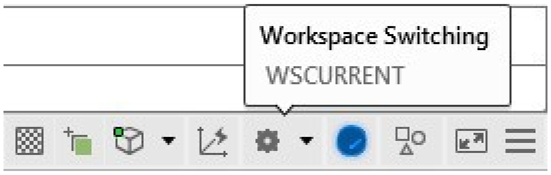

Workspace Switching
WSCURRENT

Figure Step 16

Step 17

In the Workspace pull-up menu, click Workspace Settings. This will open the Workspace Settings dialogue box. (Figure Step 17)

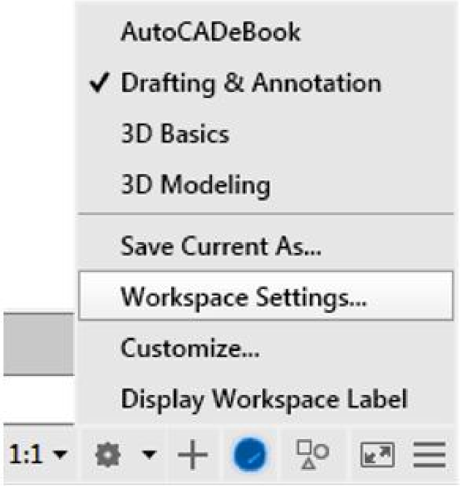

AutoCADeBook
✓ Drafting & Annotation
3D Basics
3D Modeling
Save Current As...
Workspace Settings...
Customize...
Display Workspace Label
1:1

Figure Step 17

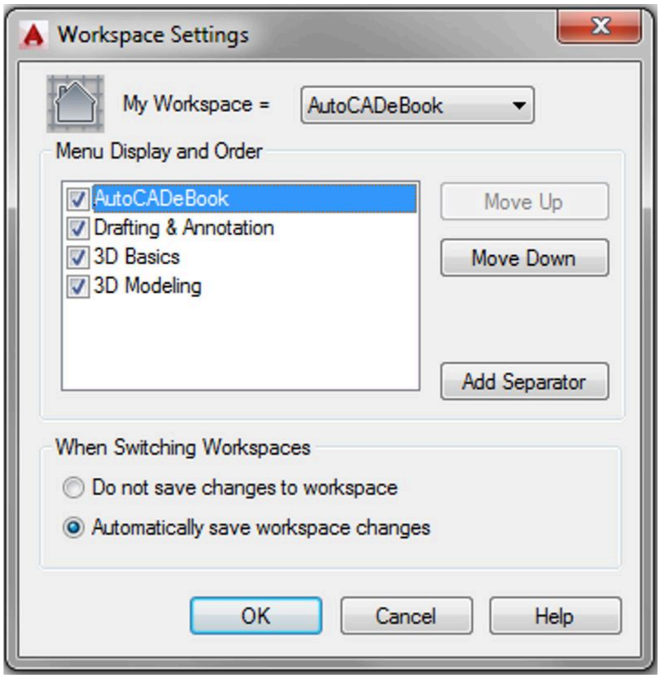
AUTHOR'S COMMENTS: AutoCAD has two basic menu systems, Toolbars and Ribbons. You can switch back and forth between them. Along with using either the Toolbar or Ribbon menu, you should enable the Pull-down menu and use it in conjunction with either menu. The AutoCAD 2D book shows all three menus. I advise the beginner to use the Toolbar menus while working on the AutoCAD 2D book and then try the Ribbon menu after completing it. The AutoCAD 2D Advanced book teaches more about Toolbar, Ribbon, and Pull-down menus as well as how to create and edit them.

If you intend to use Toolbars menus to do this book, complete steps 18a and 19a. If you intend to use Ribbon menus to do this book, complete steps 18b and 19b.

AUTHOR'S COMMENTS: I suggest that you use Toolbar menus while working on this book.

Step 18a (Toolbar Menus)

In the Workspace Settings dialogue box, set My Workspace to: AutoCADbook. Enable Automatically save workspace changes by clicking the radio button. Click OK to close the box. (Figure Step 18a)



Workspace Settings

My Workspace = AutoCADeBook

Menu Display and Order

☑ AutoCADeBook
☑ Drafting & Annotation
☑ 3D Basics
☑ 3D Modeling

When Switching Workspaces

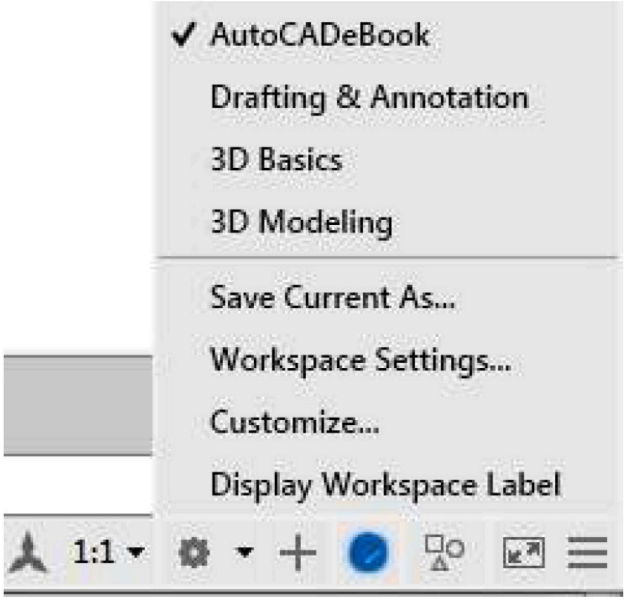
○ Do not save changes to workspace
● Automatically save workspace changes

Move Up
Move Down
Add Separator
OK
Cancel
Help

Figure Step 18a

Step 19a (Toolbar Menus)

Click the small arrow in the Workplace Switching icon again on the Status bar. Click the workspace: AutoCADbook to set it as the current workspace. Make sure that the check mark is beside the workspace: AutoCADbook as shown in the figure. (Figure Step 19a)



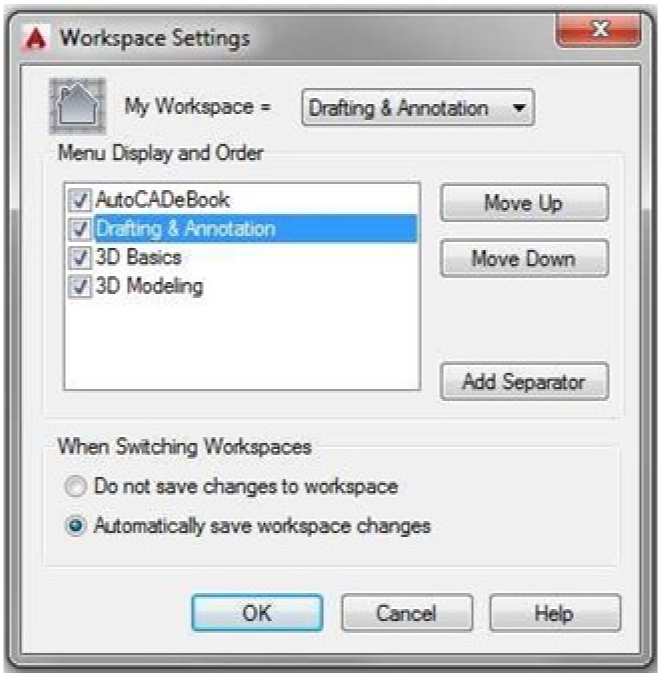
✓ AutoCADeBook
Drafting & Annotation
3D Basics
3D Modeling
Save Current As...
Workspace Settings...
Customize...
Display Workspace Label
1:1

Figure Step 19a

Step 18b (Ribbon Menus)

If you intend to use Ribbon menus to do this book, complete Steps 18b and 19b. If you intend to use Toolbar menus to do this book, skip to Step 20.

In the Workspace Settings dialogue box, set My Workspace to the workspace: Drafting & Annotation. Enable Automatically save workspace changes by clicking the radio button. Click OK to close the box (Figure Step 18b)



Workspace Settings

My Workspace = Drafting & Annotation

Menu Display and Order

☑ AutoCADeBook
☑ Drafting & Annotation
☑ 3D Basics
☑ 3D Modeling

When Switching Workspaces

○ Do not save changes to workspace
● Automatically save workspace changes

Figure Step 18b

Step 19b (Ribbon Menus)

Click the small arrow in the Workplace Switching icon again on the Status bar. Click the workspace: Drafting & Annotation to set it as the current workspace. Make sure that the check mark is beside the workspace: Drafting & Annotation as shown in the figure. (Figure Step 19b)

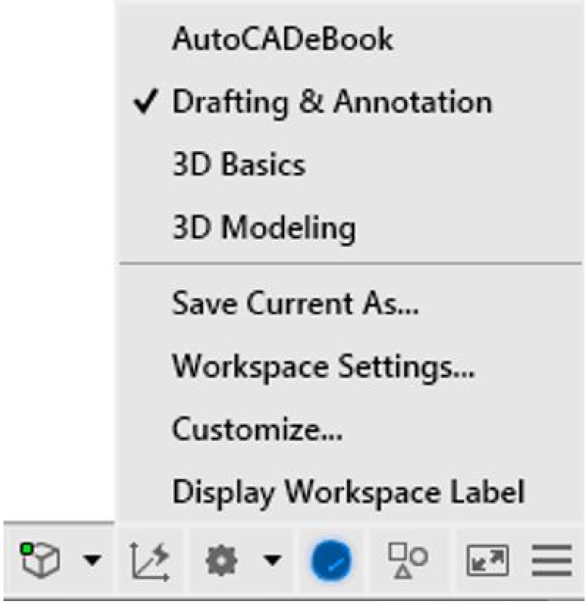
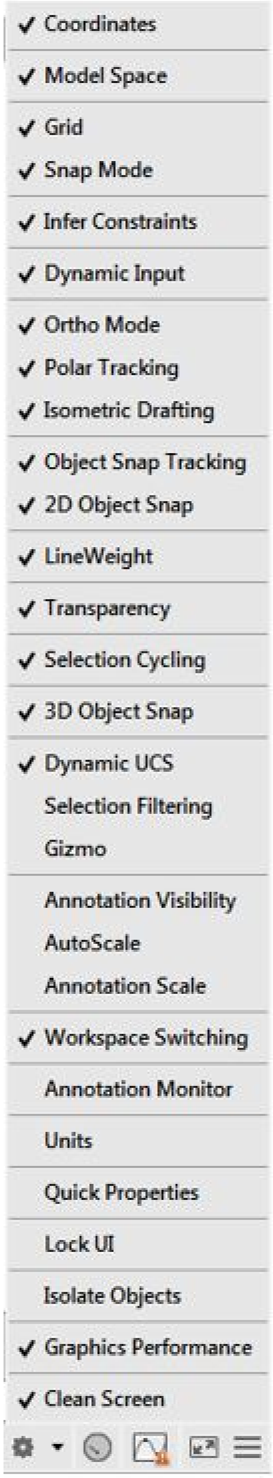

AutoCADeBook
✓ Drafting & Annotation
3D Basics
3D Modeling
Save Current As...
Workspace Settings...
Customize...
Display Workspace Label

Figure Step 19b

Step 20

Click the Hamburger icon located on the far right side of the Status bar. This will display the Customization menu. Enable or disable each item in the menu until it matches the figure. (Figure Step 20)



✓ Coordinates
✓ Model Space
✓ Grid
✓ Snap Mode
✓ Infer Constraints
✓ Dynamic Input
✓ Ortho Mode
✓ Polar Tracking
✓ Isometric Drafting
✓ Object Snap Tracking
✓ 2D Object Snap
✓ LineWeight
✓ Transparency
✓ Selection Cycling
✓ 3D Object Snap
✓ Dynamic UCS
Selection Filtering
Gizmo
Annotation Visibility
AutoScale
Annotation Scale
✓ Workspace Switching
Annotation Monitor
Units
Quick Properties
Lock UI
Isolate Objects
✓ Graphics Performance
✓ Clean Screen

Figure Step 20

Step 21

AUTHOR’S COMMENTS: Enter the following commands in the Command Line window located along the bottom of the Graphic window.

Enter the MENUBAR command and set it to 1, as shown below.

```
Command: MENUBAR
Enter new value for MENUBAR <0>: 1
Command:
```

Step 22

Enter the NAVBARDISPLAY command and set it to 0, as shown below.

```
Command: NAVBARDISPLAY
Enter new value for NAVBARDISPLAY <1>: 0
Command:
```

Step 23

Enter the VPCONTROL command and set it to OFF, as shown below.

```
Command: VPCONTROL
Enter new value for VPCONTROL <ON>: OFF
Command:
```

Step 24 (2015)

If you are using AutoCAD 2015 execute the following command:

Enter the NEWTABMODE command and set it to 0, as shown below.

```
Command: NEWTABMODE
Enter new value for NEWTABMODE <1>: 0
Command:
```

Step 24 (2016-2017)

If you are using AutoCAD 2016 or 2017 execute the following command:

Enter the STARTMODE command and set it to 0, as shown below.

```
Command: STARTMODE
Enter new value for STARTMODE <1>: 0
Command:
```

Step 25

Click Tools in the Pull-down menu. In the Pull-down menu, click Options. (Figure Step 25)

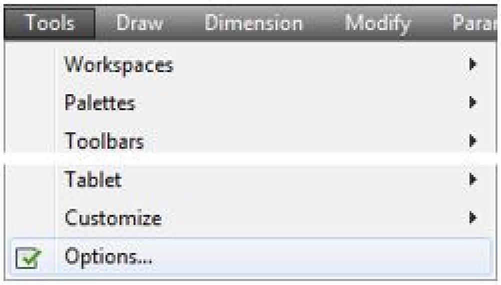
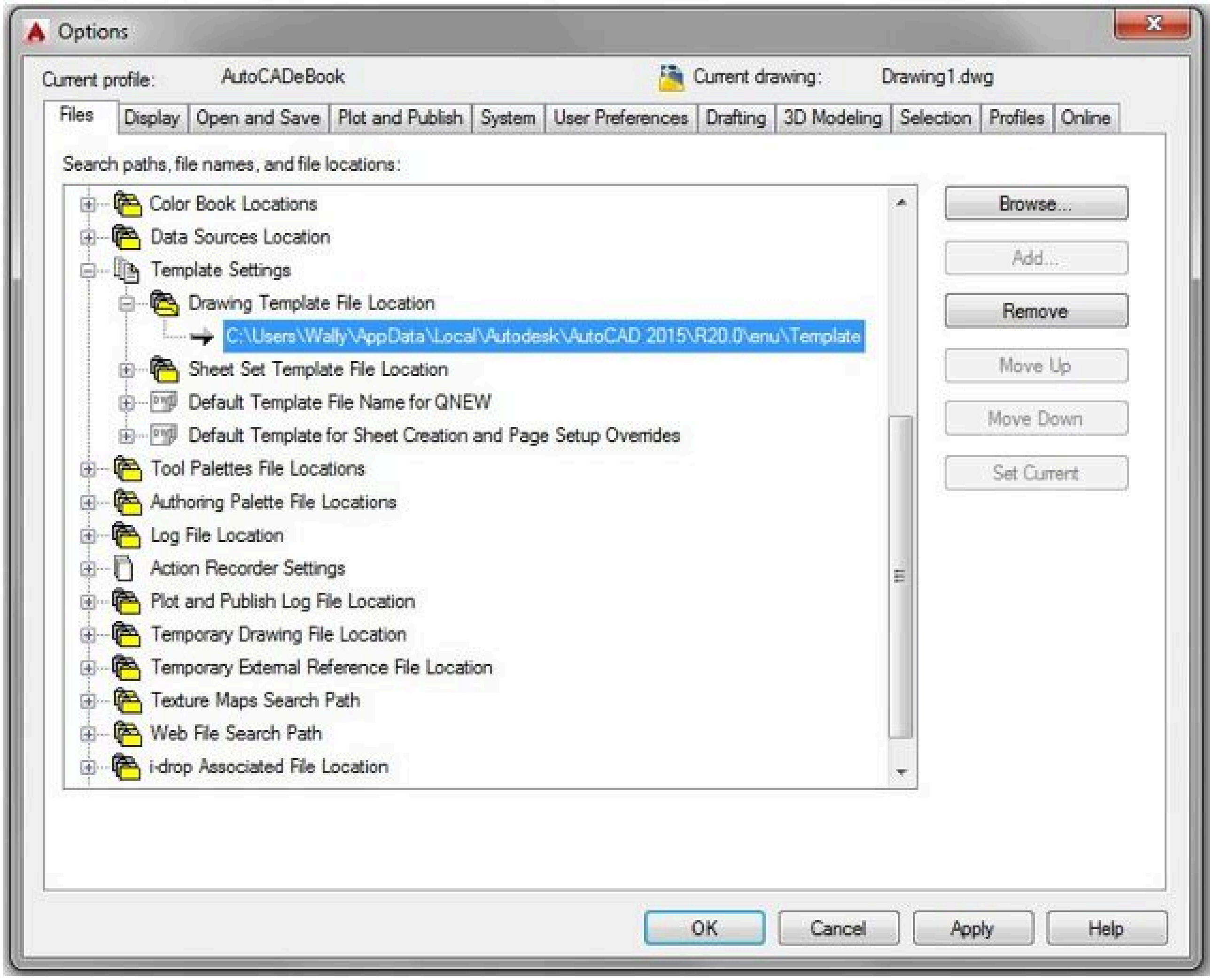

Tools
Draw
Dimension
Modify
Para
Workspaces
Palettes
Toolbars
Tablet
Customize
Options...

Figure Step 25

Step 26

In the Options dialogue box, enable the Files tab. Expand Template Settings and Drawing Template File Location. Highlight the current location by clicking it. Click the Remove button. (Figure Step 26)


Current profile: AutoCADeBook

Current drawing: Drawing1.dwg

C:\Users\Wally\AppData\Local\Autodesk\AutoCAD 2015\R20.0\enu\Template

Options
Files
Display
Open and Save
Plot and Publish
System
User Preferences
Drafting
3D Modeling
Selection
Profiles
Online
Search paths, file names, and file locations:
Color Book Locations
Data Sources Location
Template Settings
Drawing Template File Location
Sheet Set Template File Location
Default Template File Name for QNEW
Default Template for Sheet Creation and Page Setup Overrides
Tool Palettes File Locations
Authoring Palette File Locations
Log File Location
Action Recorder Settings
Plot and Publish Log File Location
Temporary Drawing File Location
Temporary External Reference File Location
Texture Maps Search Path
Web File Search Path
i-drop Associated File Location
Browse...
Add...
Remove
Move Up
Move Down
Set Current
OK
Cancel
Apply
Help

Figure Step 26

Step 27

It will display an empty box as shown in the figure. (Figure Step 27).

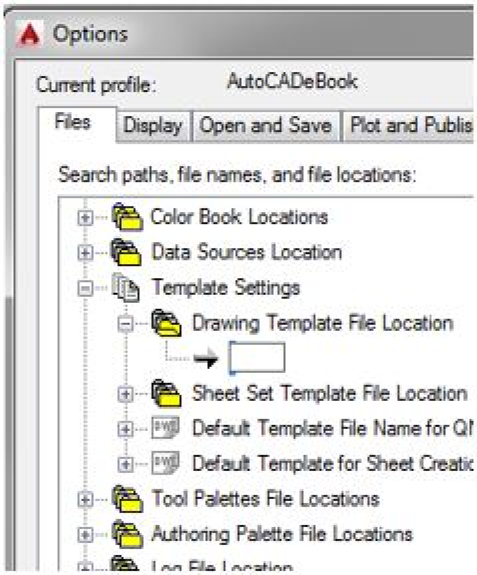
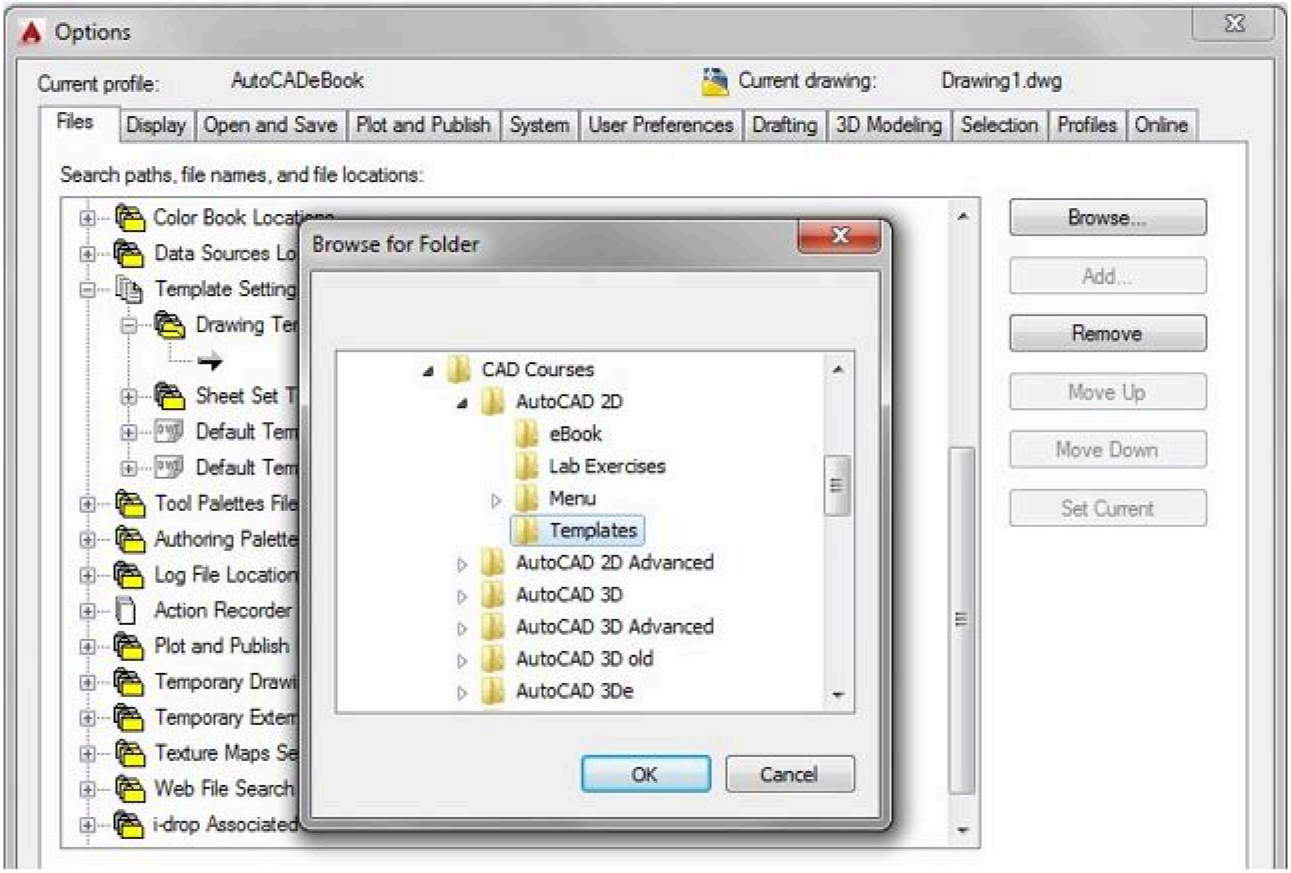

Options
Current profile:
AutoCADeBook
Files
Display
Open and Save
Plot and Publis
Search paths, file names, and file locations:
Color Book Locations
Data Sources Location
Template Settings
Drawing Template File Location
Sheet Set Template File Location
Default Template File Name for QN
Default Template for Sheet Creatic
Tool Palettes File Locations
Authoring Palette File Locations
Log File Location

Figure Step 27

Step 28

Click the Browse button. In the Browse for Folders dialogue box, locate the folder: (Figure Step 28)



Current profile: AutoCADeBook

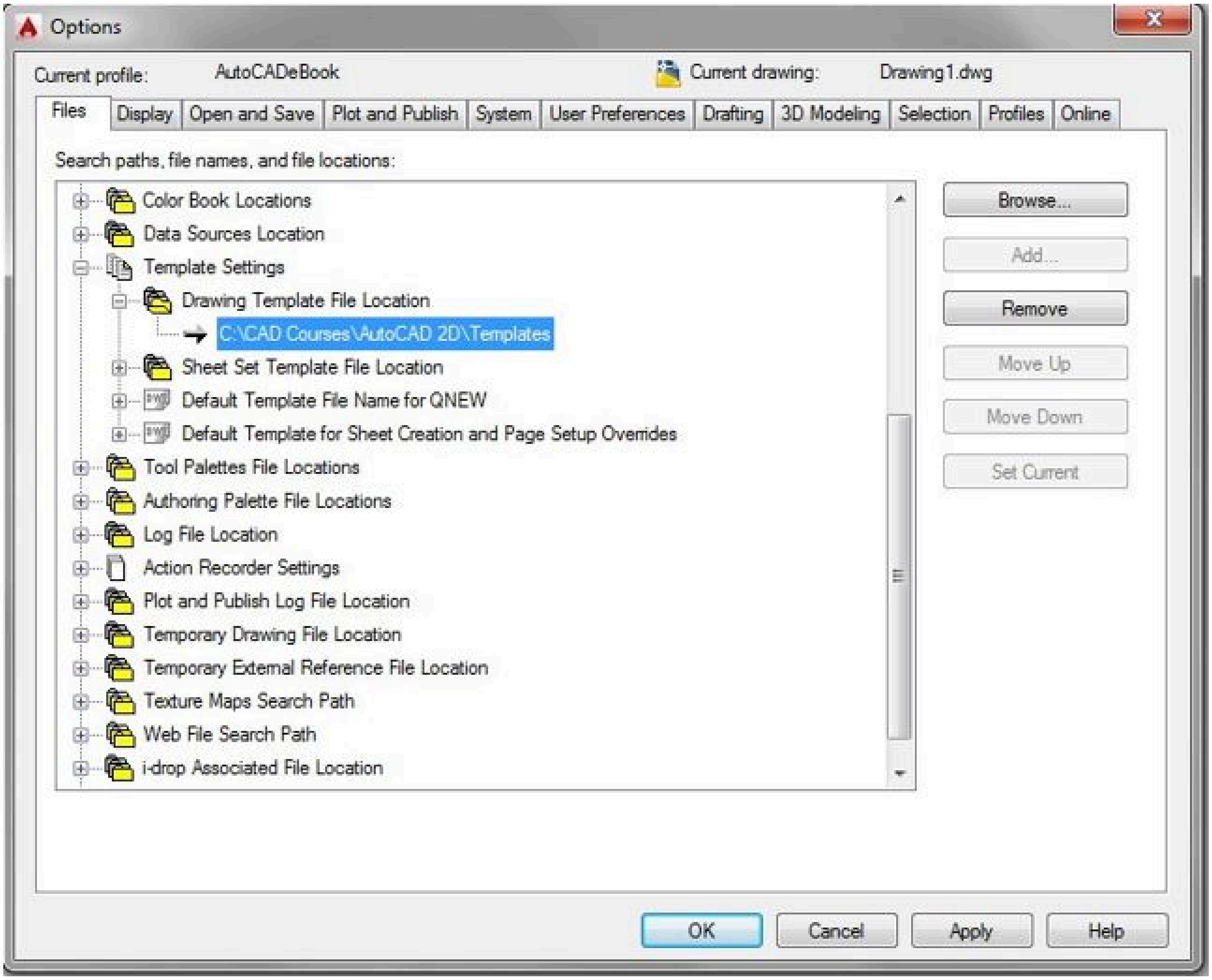
Current drawing: Drawing1.dwg

Options
Files
Display
Open and Save
Plot and Publish
System
User Preferences
Drafting
3D Modeling
Selection
Profiles
Online
Search paths, file names, and file locations:
Color Book Locati
Data Sources Lo
Template Setting
Drawing Te
Sheet Set T
Default Tem
Default Tem
Tool Palettes File
Authoring Palette
Log File Location
Action Recorder
Plot and Publish
Temporary Drawi
Temporary Exter
Texture Maps Se
Web File Search
i-drop Associated
Browse...
Add...
Remove
Move Up
Move Down
Set Current
Browse for Folder
CAD Courses
AutoCAD 2D
eBook
Lab Exercises
Menu
Templates
AutoCAD 2D Advanced
AutoCAD 3D
AutoCAD 3D Advanced
AutoCAD 3D old
AutoCAD 3De
OK
Cancel

Figure Step 28

Step 29

Click OK and the new location will appear as shown in the figure. (Figure Step 29A and 29B)



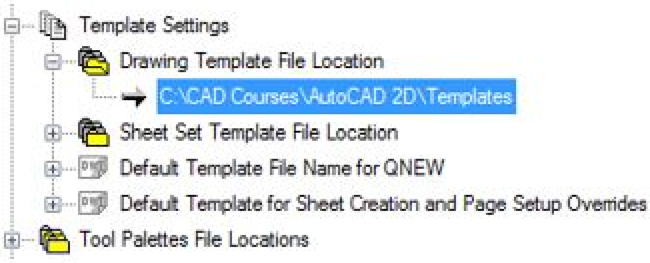
Current profile: AutoCADeBook

Current drawing: Drawing1.dwg

Drawing Template File Location: C:\CAD Courses\AutoCAD 2D\Templates

Options
Files
Display
Open and Save
Plot and Publish
System
User Preferences
Drafting
3D Modeling
Selection
Profiles
Online
Search paths, file names, and file locations:
Color Book Locations
Data Sources Location
Template Settings
Drawing Template File Location
Sheet Set Template File Location
Default Template File Name for QNEW
Default Template for Sheet Creation and Page Setup Overrides
Tool Palettes File Locations
Authoring Palette File Locations
Log File Location
Action Recorder Settings
Plot and Publish Log File Location
Temporary Drawing File Location
Temporary External Reference File Location
Texture Maps Search Path
Web File Search Path
i-drop Associated File Location
Browse...
Add...
Remove
Move Up
Move Down
Set Current
OK
Cancel
Apply
Help

Figure Step 29A



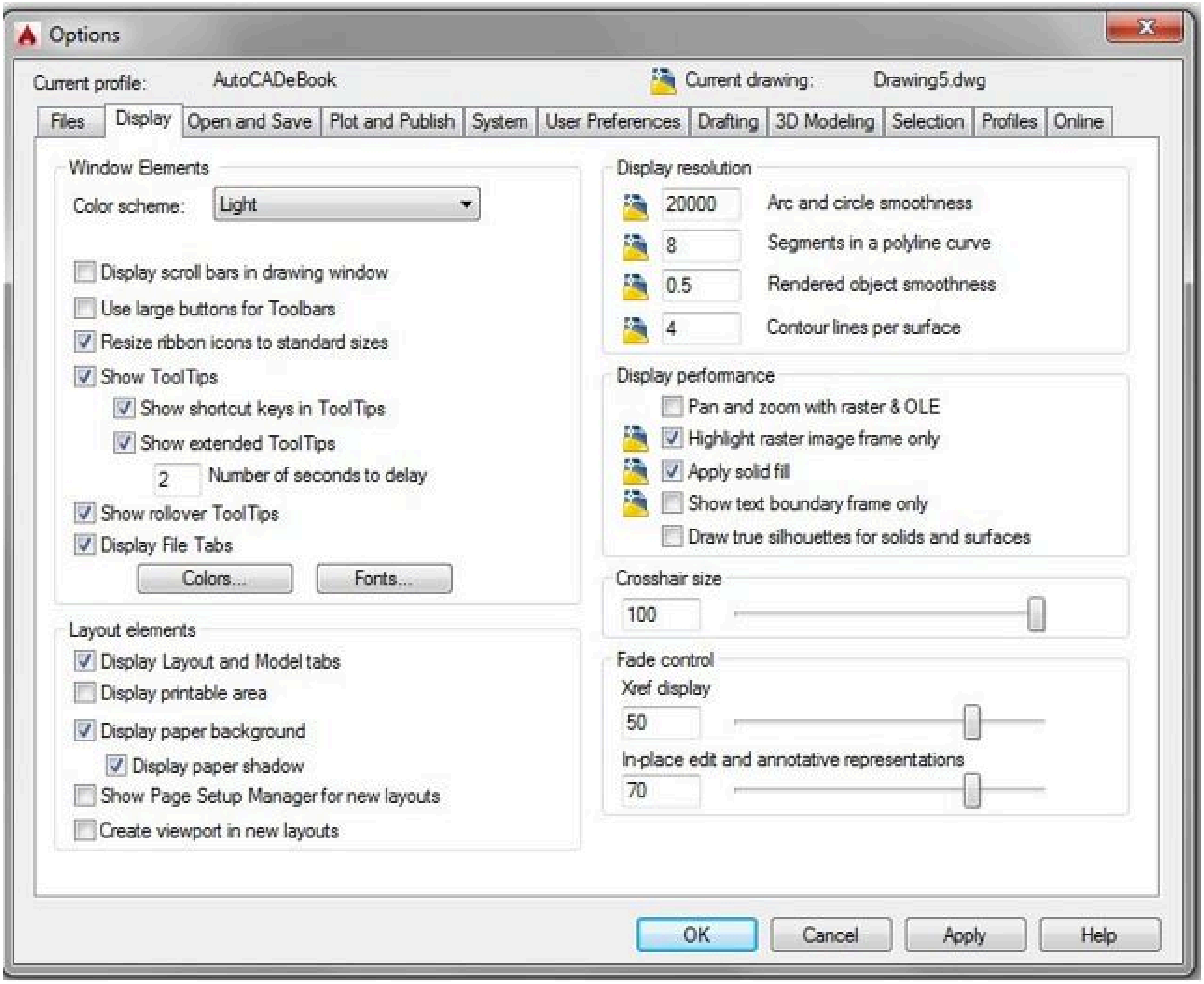
Drawing Template File Location: C:\CAD Courses\AutoCAD 2D\Templates

Template Settings
Drawing Template File Location
Sheet Set Template File Location
Default Template File Name for QNEW
Default Template for Sheet Creation and Page Setup Overrides
Tool Palettes File Locations

Figure Step 29B

Step 30

Enable the display of the Display tab. (Figure Step 30)


Options
Files
Display
Open and Save
Plot and Publish
System
User Preferences
Drafting
3D Modeling
Selection
Profiles
Online
Colors...
Fonts...
OK
Cancel
Apply
Help

Current profile: AutoCADeBook

Current drawing: Drawing5.dwg

Window Elements

Color scheme: Light

☐ Display scroll bars in drawing window
☐ Use large buttons for Toolbars
☑ Resize ribbon icons to standard sizes
☑ Show ToolTips
  ☑ Show shortcut keys in ToolTips
  ☑ Show extended ToolTips
  2 Number of seconds to delay
☑ Show rollover ToolTips
☑ Display File Tabs

Layout elements

☑ Display Layout and Model tabs
☐ Display printable area
☑ Display paper background
  ☑ Display paper shadow
☐ Show Page Setup Manager for new layouts
☐ Create viewport in new layouts

Display resolution

20000 Arc and circle smoothness
8 Segments in a polyline curve
0.5 Rendered object smoothness
4 Contour lines per surface

Display performance

☐ Pan and zoom with raster & OLE
☑ Highlight raster image frame only
☑ Apply solid fill
☐ Show text boundary frame only
☐ Draw true silhouettes for solids and surfaces

Crosshair size

100

Fade control

Xref display

50

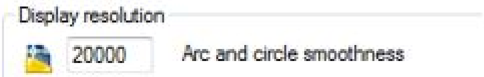
In-place edit and annotative representations

70

Figure Step 30

Step 31

In the Display resolution area, set the Arc and circle smoothness to 20000. (Figure Step 31)



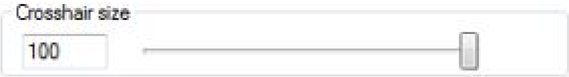
Display resolution

20000 Arc and circle smoothness

Figure Step 31

Step 32

Set the Crosshair size to 100. (Figure Step 32)



Crosshair size: 100

Figure Step 32

Step 33

Click the Colors button to open the Drawing Window Color dialogue box. Highlight 2D model space, Uniform Background and then select the color White as shown in the figure. (Figure Step 33A and 33B)

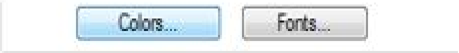
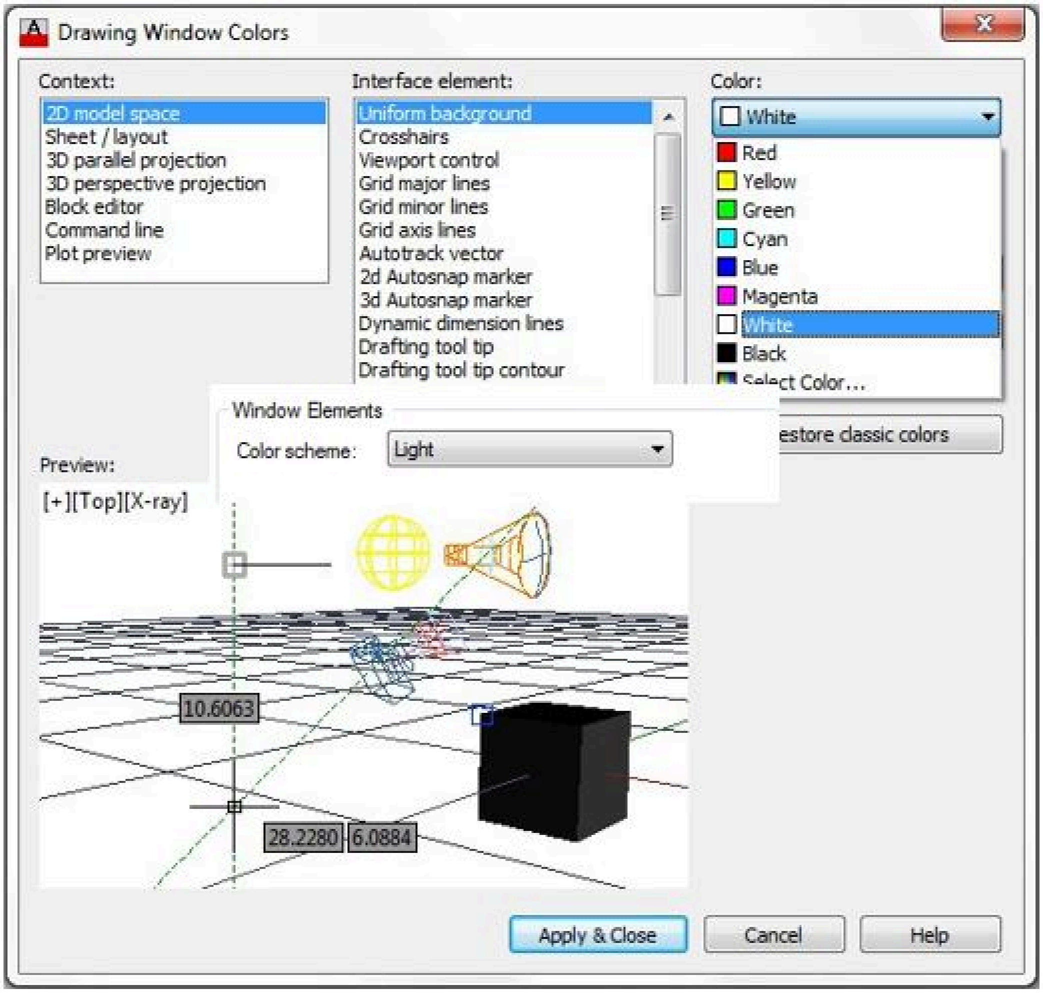

Colors...
Fonts...

Figure Step 33A



Drawing Window Colors

Context: 2D model space

Interface element: Uniform background

Color: White

Color scheme: Light

Sheet / layout
3D parallel projection
3D perspective projection
Block editor
Command line
Plot preview
Crosshairs
Viewport control
Grid major lines
Grid minor lines
Grid axis lines
Autotrack vector
2d Autosnap marker
3d Autosnap marker
Dynamic dimension lines
Drafting tool tip
Drafting tool tip contour
Red
Yellow
Green
Cyan
Blue
Magenta
White
Black
Select Color...
Window Elements
estore classic colors
Preview:
[+][Top][X-ray]
10.6063
28.2280
6.0884
Apply & Close
Cancel
Help

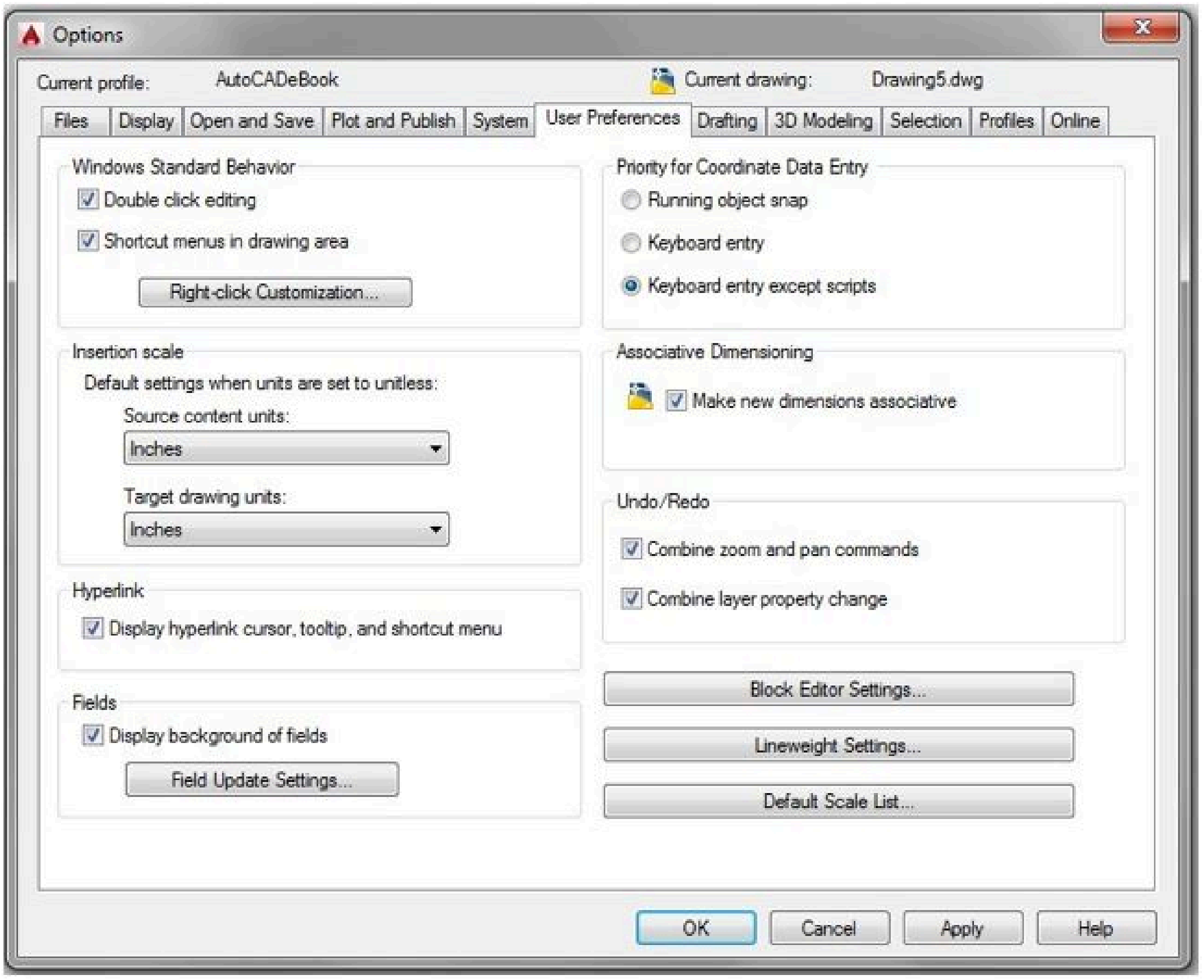
Figure Step 33B

Step 34

Click Apply and Close.

Step 35

Enable the User Preferences tab. (Figure Step 35)


Options
Files
Display
Open and Save
Plot and Publish
System
User Preferences
Drafting
3D Modeling
Selection
Profiles
Online
Right-click Customization...
Field Update Settings...
Block Editor Settings...
Lineweight Settings...
Default Scale List...
OK
Cancel
Apply
Help

Current profile: AutoCADeBook
Current drawing: Drawing5.dwg

Windows Standard Behavior
☑ Double click editing
☑ Shortcut menus in drawing area

Insertion scale
Default settings when units are set to unitless:
Source content units: Inches
Target drawing units: Inches

Hyperlink
☑ Display hyperlink cursor, tooltip, and shortcut menu

Fields
☑ Display background of fields

Priority for Coordinate Data Entry
○ Running object snap
○ Keyboard entry
● Keyboard entry except scripts

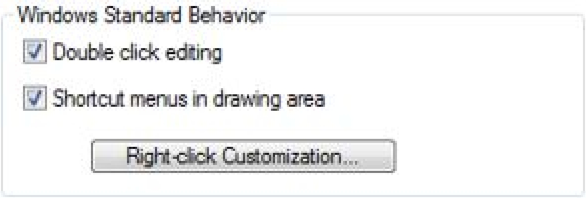
Associative Dimensioning
☑ Make new dimensions associative

Undo/Redo
☑ Combine zoom and pan commands
☑ Combine layer property change

Figure Step 35

Step 36

Ensure that both boxes are enabled in the Windows Standard Behavior area. Click the Right-click Customization button. Enable the box Turn on time sensitive right-click and set the Longer click duration to 250 milliseconds. (Figure Step 36A and 36B)

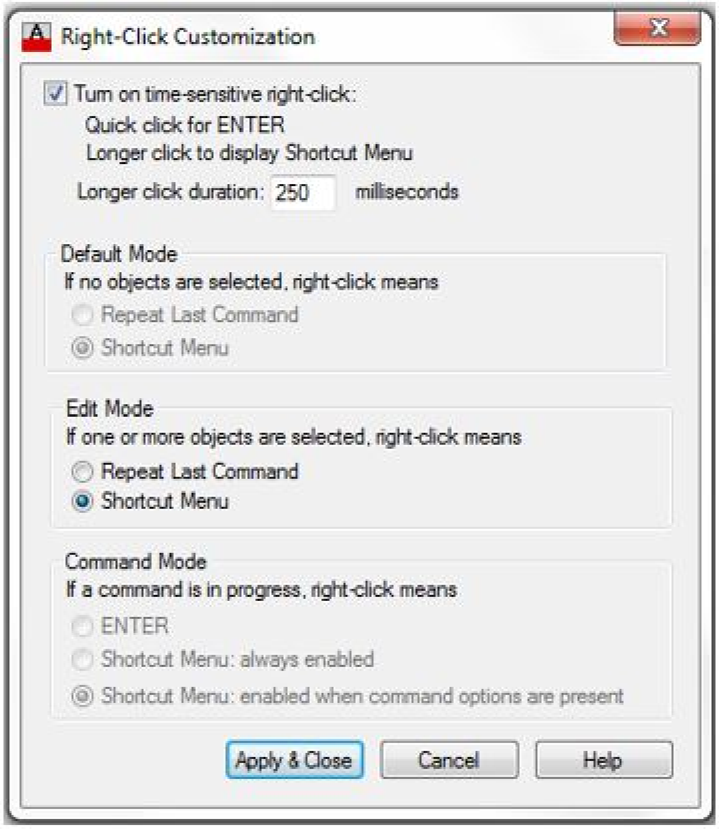


Windows Standard Behavior

☑ Double click editing

☑ Shortcut menus in drawing area

Figure Step 36A



☑ Turn on time-sensitive right-click:

Quick click for ENTER

Longer click to display Shortcut Menu

Longer click duration: 250 milliseconds

Default Mode

If no objects are selected, right-click means

○ Repeat Last Command

● Shortcut Menu

Edit Mode

If one or more objects are selected, right-click means

○ Repeat Last Command

● Shortcut Menu

Command Mode

If a command is in progress, right-click means

○ ENTER

○ Shortcut Menu: always enabled

● Shortcut Menu: enabled when command options are present

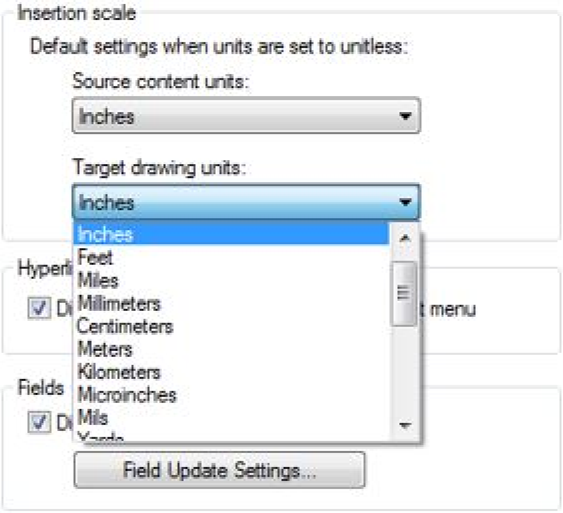
Figure Step 36B

Step 37

Click Apply and Close.

Step 38

In the Insertion scale area, pull down the Source content units menu and select Inches. Do the same for the Target drawing units menu to set both to inches. (Figure Step 38A and 38B)



Insertion scale

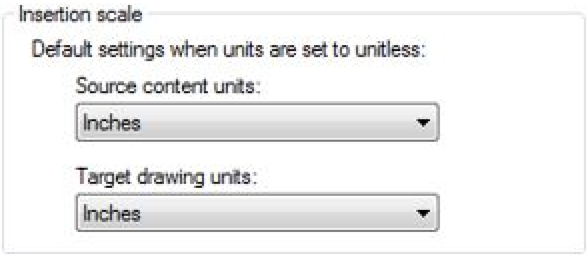
Default settings when units are set to unitless:

Source content units: Inches

Target drawing units: Inches

Inches
Feet
Miles
Millimeters
Centimeters
Meters
Kilometers
Microinches
Mils
Hyperl
Di
t menu
Fields
Di
Field Update Settings...

Figure 38A



Insertion scale

Default settings when units are set to unitless:

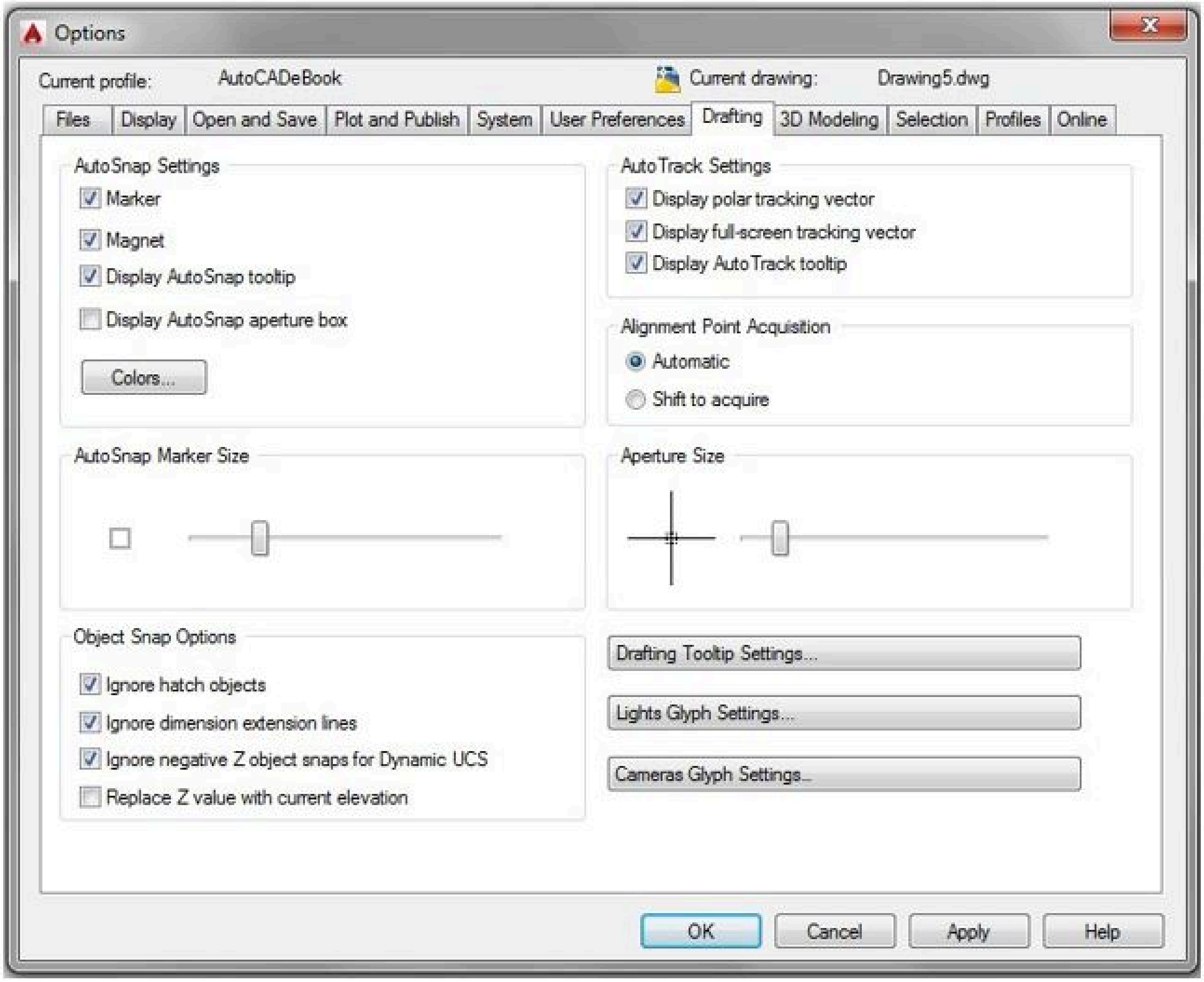
Source content units: Inches

Target drawing units: Inches

Figure Step 38B

Step 39

Enable the Drafting tab. (Figure Step 39)



Options
Current profile: AutoCADeBook
Current drawing: Drawing5.dwg
Files
Display
Open and Save
Plot and Publish
System
User Preferences
Drafting
3D Modeling
Selection
Profiles
Online
AutoSnap Settings
AutoTrack Settings
Colors...
Alignment Point Acquisition
AutoSnap Marker Size
Aperture Size
Object Snap Options
Drafting Tooltip Settings...
Lights Glyph Settings...
Cameras Glyph Settings...
OK
Cancel
Apply
Help

AutoSnap Settings
- ☑ Marker
- ☑ Magnet
- ☑ Display AutoSnap tooltip
- ☐ Display AutoSnap aperture box

AutoTrack Settings
- ☑ Display polar tracking vector
- ☑ Display full-screen tracking vector
- ☑ Display AutoTrack tooltip

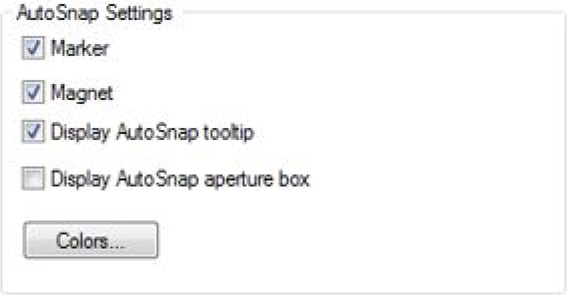
Alignment Point Acquisition
- ● Automatic
- ○ Shift to acquire

Object Snap Options
- ☑ Ignore hatch objects
- ☑ Ignore dimension extension lines
- ☑ Ignore negative Z object snaps for Dynamic UCS
- ☐ Replace Z value with current elevation

Figure Step 39

Step 40

Click the Colors box. In the Color box, pull-down the list and click Select Color. Enable the Index Color tab in the Select Color dialogue box, enter the number 253 and click OK. (Figure Step 40A, 40B, and 40C)



AutoSnap Settings

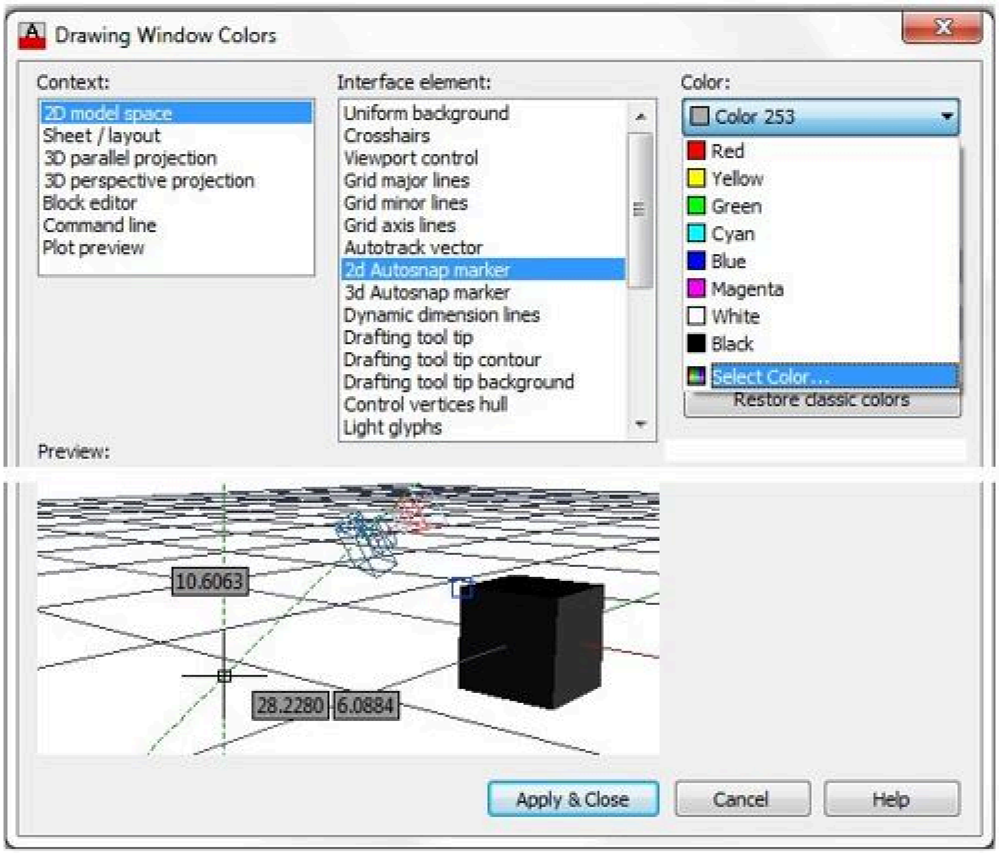
☑ Marker

☑ Magnet

☑ Display AutoSnap tooltip

☐ Display AutoSnap aperture box

Figure Step 40A



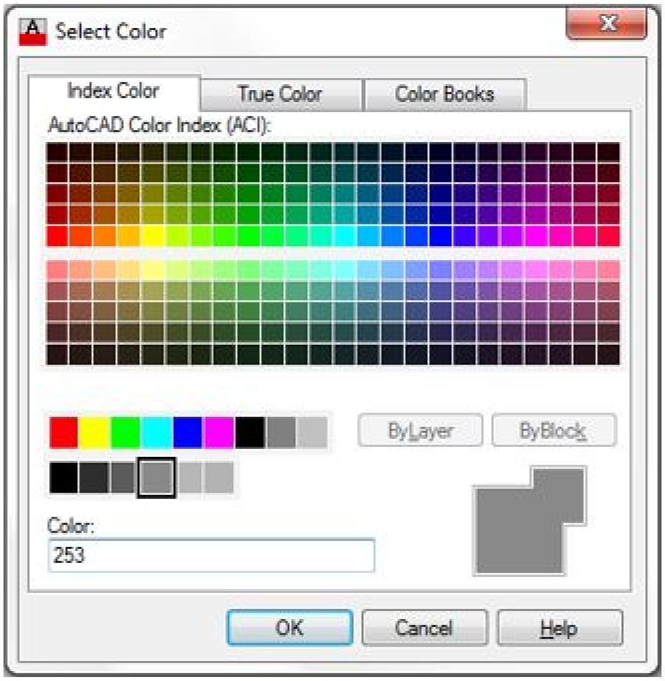
Drawing Window Colors

Context:
- 2D model space
- Sheet / layout
- 3D parallel projection
- 3D perspective projection
- Block editor
- Command line
- Plot preview

Interface element:
- Uniform background
- Crosshairs
- Viewport control
- Grid major lines
- Grid minor lines
- Grid axis lines
- Autotrack vector
- 2d Autosnap marker
- 3d Autosnap marker
- Dynamic dimension lines
- Drafting tool tip
- Drafting tool tip contour
- Drafting tool tip background
- Control vertices hull
- Light glyphs

Color: Color 253
- Red
- Yellow
- Green
- Cyan
- Blue
- Magenta
- White
- Black
- Select Color...

Figure Step 40B


Select Color
Index Color
True Color
Color Books
AutoCAD Color Index (ACI):
ByLayer
ByBlock
OK
Cancel
Help

Color:
253

Figure Step 40C

Step 41

Click OK and then click Apply and Close.

Step 42

Set the AutoSNAP Marker Size and the Aperture Size to approximately the sizes shown in the figures. If required, you can adjust these sizes, when required, as you work your way through the book. (Figure Step 42)

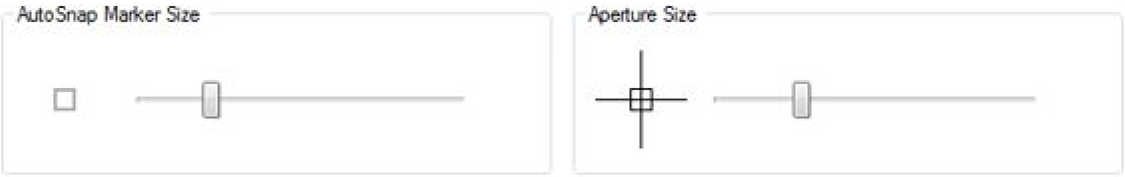
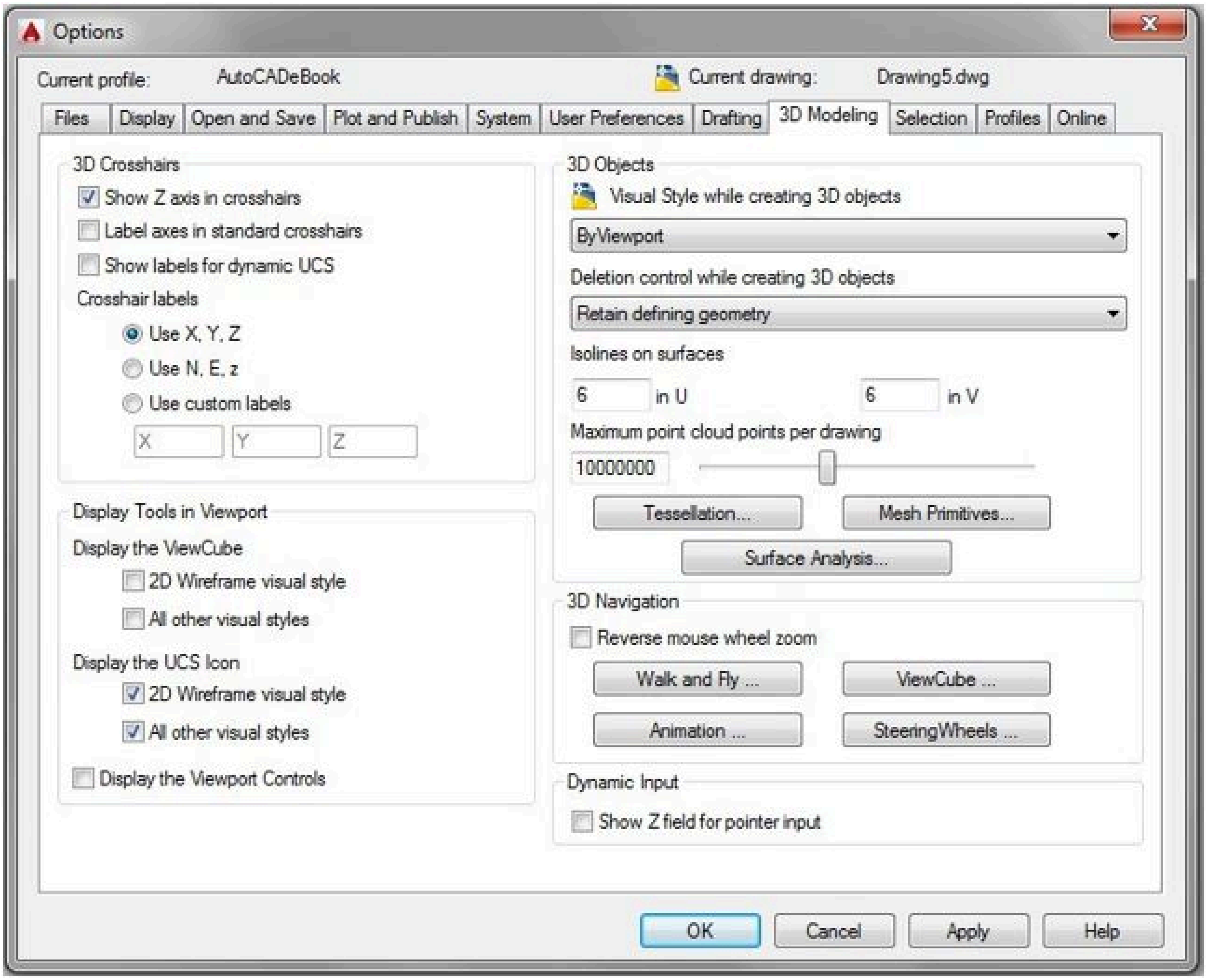

AutoSnap Marker Size
Aperture Size

Figure Step 42

Step 43

Enable the 3D Modeling tab. (Figure Step 43)


Options
Files
Display
Open and Save
Plot and Publish
System
User Preferences
Drafting
3D Modeling
Selection
Profiles
Online
Tessellation...
Mesh Primitives...
Surface Analysis...
Walk and Fly ...
ViewCube ...
Animation ...
SteeringWheels ...
OK
Cancel
Apply
Help

Current profile: AutoCADeBook
Current drawing: Drawing5.dwg

3D Crosshairs
☑ Show Z axis in crosshairs
☐ Label axes in standard crosshairs
☐ Show labels for dynamic UCS
Crosshair labels
● Use X, Y, Z
○ Use N, E, z
○ Use custom labels
X Y Z

Display Tools in Viewport
Display the ViewCube
☐ 2D Wireframe visual style
☐ All other visual styles
Display the UCS Icon
☑ 2D Wireframe visual style
☑ All other visual styles
☐ Display the Viewport Controls

3D Objects
Visual Style while creating 3D objects: ByViewport
Deletion control while creating 3D objects: Retain defining geometry
Isolines on surfaces
6 in U
6 in V
Maximum point cloud points per drawing: 10000000

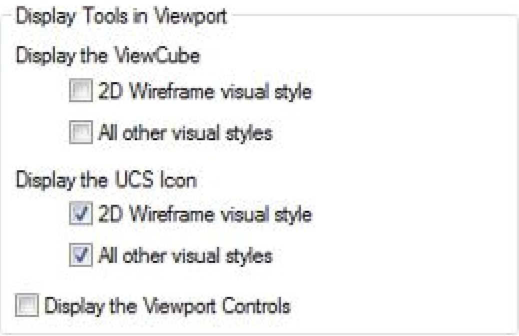
3D Navigation
☐ Reverse mouse wheel zoom

Dynamic Input
☐ Show Z field for pointer input

Figure Step 43

Step 44

In the Display Tools in Viewport area, set it as shown in the figure. (Figure Step 44)

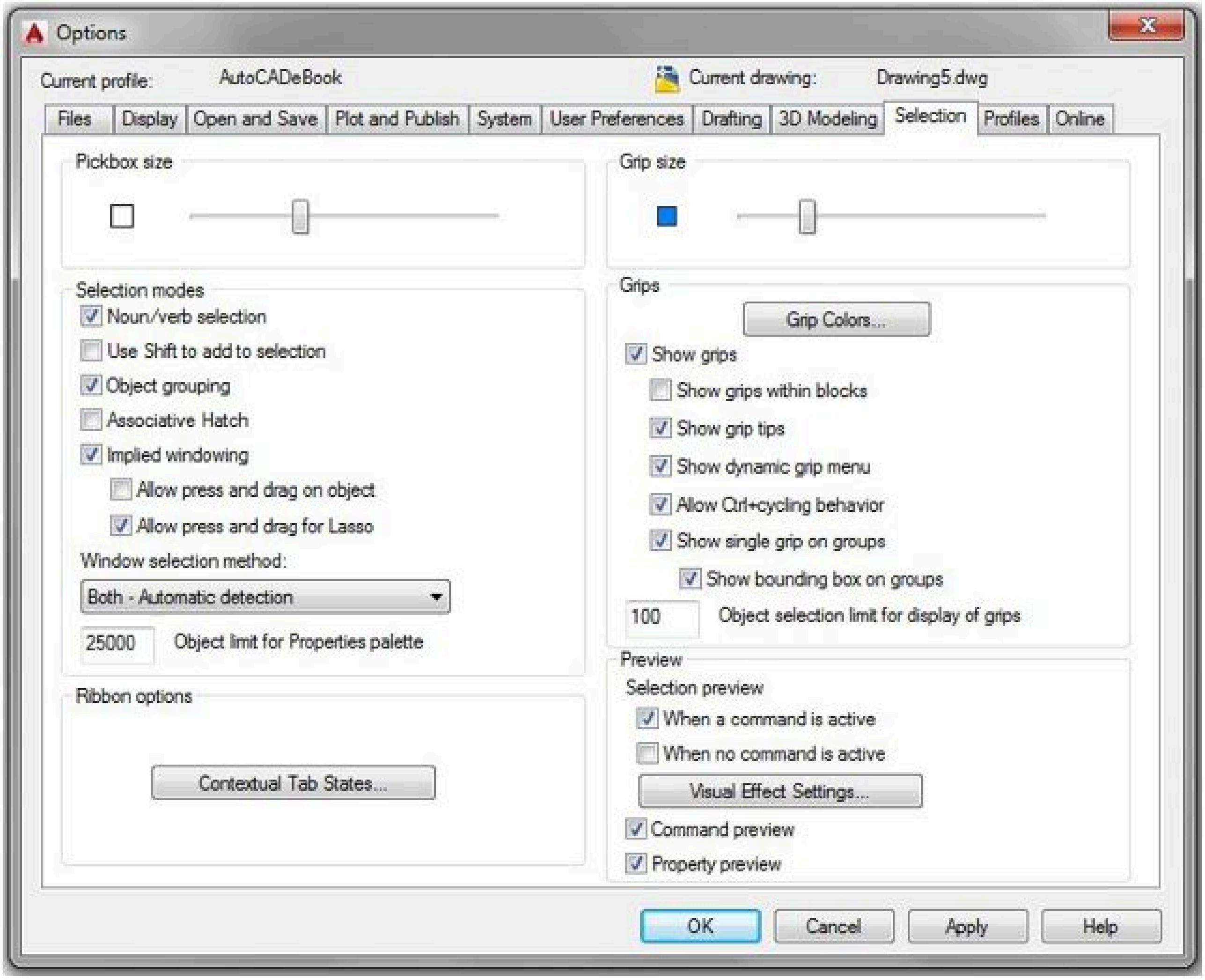


Display Tools in Viewport
Display the ViewCube
☐ 2D Wireframe visual style
☐ All other visual styles
Display the UCS Icon
☑ 2D Wireframe visual style
☑ All other visual styles
☐ Display the Viewport Controls

Figure Step 44

Step 45

Enable the Selection tab. (Figure Step 45)



Options

Current profile: AutoCADeBook

Current drawing: Drawing5.dwg

Files | Display | Open and Save | Plot and Publish | System | User Preferences | Drafting | 3D Modeling | Selection | Profiles | Online

Pickbox size

Grip size

Selection modes

☑ Noun/verb selection
☐ Use Shift to add to selection
☑ Object grouping
☐ Associative Hatch
☑ Implied windowing
  ☐ Allow press and drag on object
  ☑ Allow press and drag for Lasso

Window selection method:
Both - Automatic detection

25000 Object limit for Properties palette

Ribbon options

Contextual Tab States...

Grips

Grip Colors...

☑ Show grips
  ☐ Show grips within blocks
  ☑ Show grip tips
  ☑ Show dynamic grip menu
  ☑ Allow Ctrl+cycling behavior
  ☑ Show single grip on groups
    ☑ Show bounding box on groups

100 Object selection limit for display of grips

Preview

Selection preview
☑ When a command is active
☐ When no command is active

Visual Effect Settings...

☑ Command preview
☑ Property preview

OK | Cancel | Apply | Help

Figure Step 45

Step 46

Set the Pickbox Size and the Grip Size to approximately the sizes as shown in the figure. If required, you can adjust these sizes, when required, as you work your way through the book. (Figure Step 46)

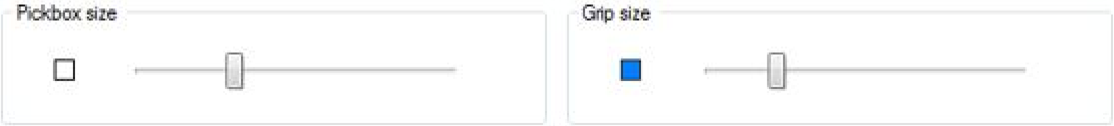


Figure Step 46

Step 47

Click the OK button on the bottom of the Options dialogue box to close it.

Step 48

Your AutoCAD software is now configured for the AutoCAD 2D book. Your Graphic window should closely match the figure. (Figure Step 48)

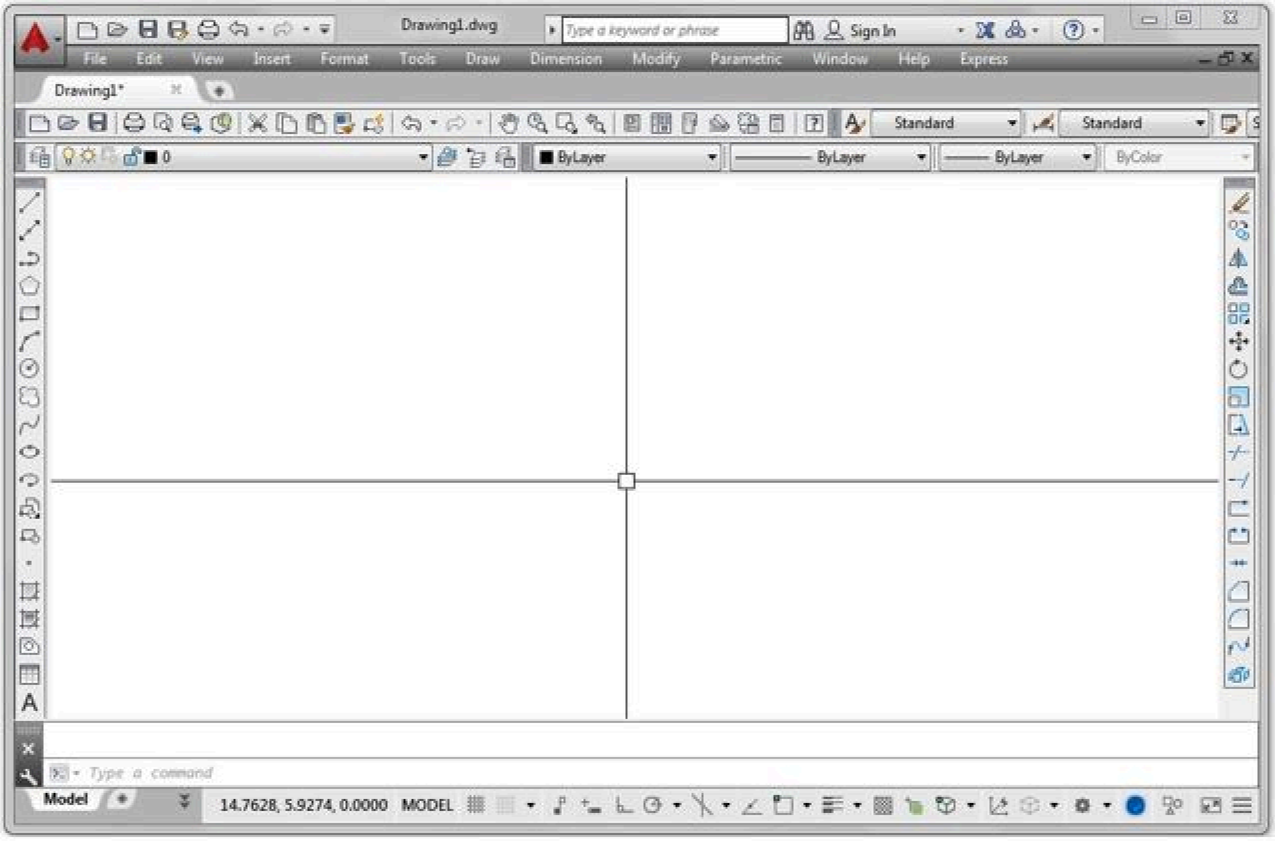

Drawing1.dwg
Type a keyword or phrase
Sign In
File
Edit
View
Insert
Format
Tools
Draw
Dimension
Modify
Parametric
Window
Help
Express
Drawing1*
Standard
Standard
0
ByLayer
ByLayer
ByLayer
ByColor
Type a command
Model
14.7628, 5.9274, 0.0000
MODEL

Figure Step 48

Step 49

Go to Module 1.

Part 1

Module 1: AutoCAD's User Interface

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe and configure AutoCAD's interface including the Graphic window, crosshairs, Graphic cursor, pickbox, Status bar, Model and Layout tabs, Pull-down menu, Toolbar menus, Ribbon menus, dialogue boxes, windows, using of the mouse, Command Line window, and the Text window.
2. List and describe the two basic AutoCAD menus systems; Toolbars and Ribbons.

AutoCAD's Graphic Window

AutoCAD's *Graphic window* is shown in Figure 1-1. The Graphic window is used most of the time when you are working in AutoCAD.

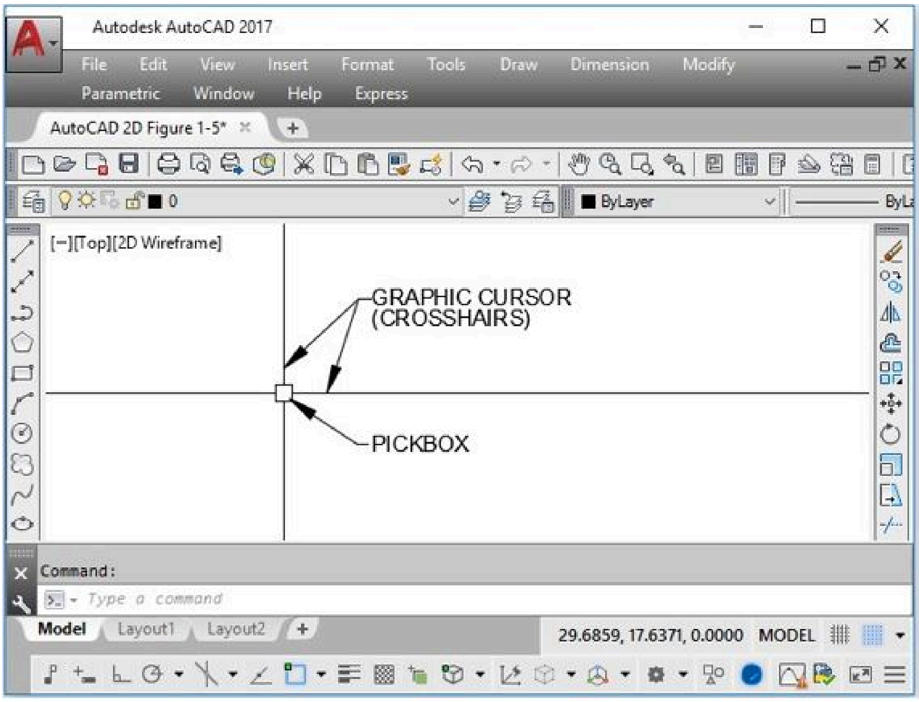

Autodesk AutoCAD 2017
File
Edit
View
Insert
Format
Tools
Draw
Dimension
Modify
Parametric
Window
Help
Express
AutoCAD 2D Figure 1-5*
0
ByLayer
[−][Top][2D Wireframe]
GRAPHIC CURSOR (CROSSHAIRS)
PICKBOX
Command:
Type a command
Model
Layout1
Layout2
29.6859, 17.6371, 0.0000
MODEL

Figure 1-1

Crosshairs and Graphic Cursor

The *crosshairs* indicates your current location on the Graphic window or in 2D space. If you were drawing by hand, think of it as the point of your pencil. It is one of AutoCAD's most important features and is used extensively when you are creating drawings. See Figure 1-1, and 1-3.

When you move the crosshairs onto a menu item, it will change its display to the *Graphic cursor* as shown in Figure 1-2. You use it to select menu items.

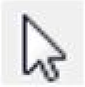


Figure 1-2

Pickbox

The *pickbox* is a small square located at the intersection of the crosshairs as shown in Figure 1-3. It is used to pick drawing objects.

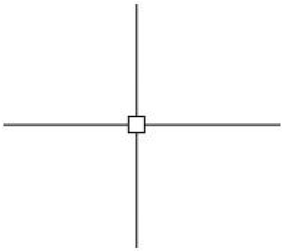


Figure 1-3

Status Bar

The *Status bar*, Figure 1-4, is positioned across the bottom of the Graphic window and becomes a very important part of your day-to-day AutoCAD work.

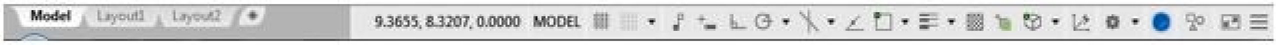


Figure 1-4

The coordinate location or the XYZ coordinates of the current location of the crosshairs is displayed near the center of the Status bar. The coordinate location display can be enabled or disabled by clicking it with the left mouse button. Toggle buttons for AutoCAD's features are located immediately to the right of the coordinate display. Many of these features will be taught in the AutoCAD 2D book.

Model and Layout Tabs

The *Model and Layout tabs*, Figure 1-5, are used to toggle the display of Model space or one of the layouts. This will be discussed in greater detail in Module 18.

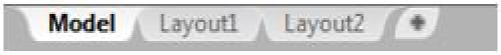


Figure 1-5

USER TIP: The size of the crosshairs can be changed. For the AutoCAD 2D book and most other AutoCAD work, it is best to have them full screen as shown in Figure 1-1.

AutoCAD's Menu Systems

AUTHOR'S COMMENTS: AutoCAD has two basic menu systems, Toolbars and Ribbons. You can switch back and forth between them. Along with using either the Toolbar or Ribbon menu, you should enable the Pull-down menu and use it in conjunction with either menu. The AutoCAD 2D book shows all three menus. I advise the beginner to use the Toolbar menus while working on the AutoCAD 2D book and then try the Ribbon menu after completing it.

AutoCAD has two different menu systems that allows you to communicate with it. The two menu systems that you can choose from are the Toolbar menus and the Ribbon menus. See Figure 1-6. You can and should add the Pull-down menu to either the Toolbar or the Ribbon menu.

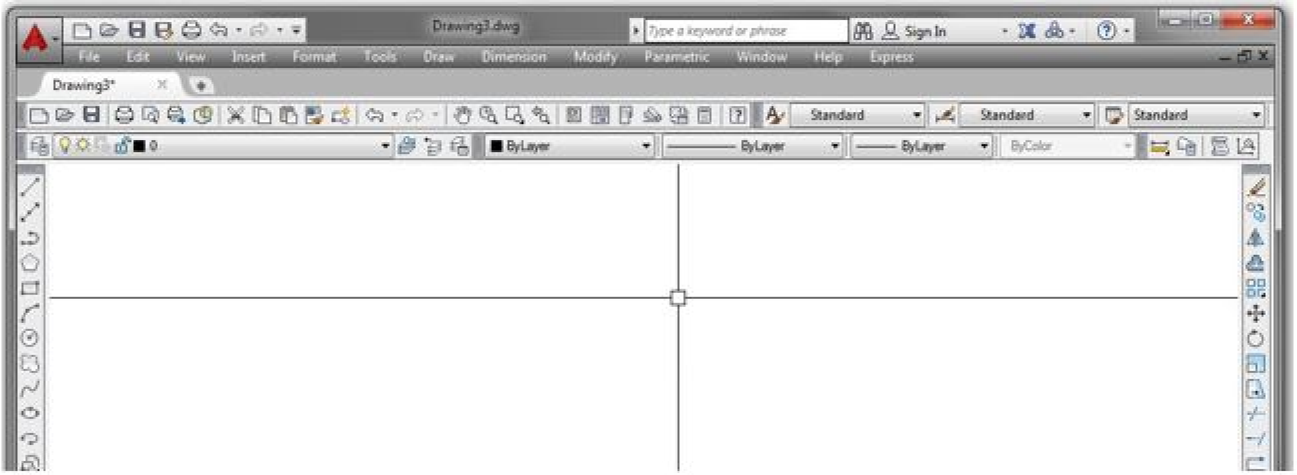

Drawing3.dwg
Type a keyword or phrase
Sign In
File
Edit
View
Insert
Format
Tools
Draw
Dimension
Modify
Parametric
Window
Help
Express
Drawing3*
Standard
Standard
Standard
0
ByLayer
ByLayer
ByLayer
ByColor

Figure 1-6a The Toolbar Menus

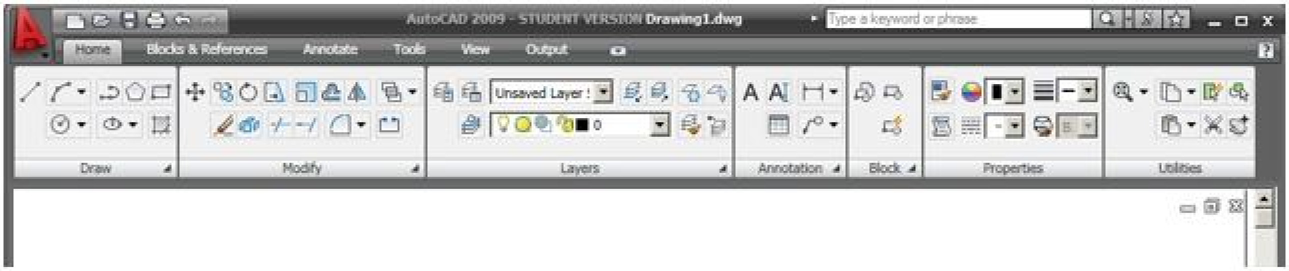

AutoCAD 2009 - STUDENT VERSION Drawing1.dwg
Type a keyword or phrase
Home
Blocks & References
Annotate
Tools
View
Output
Unsaved Layer
0
Draw
Modify
Layers
Annotation
Block
Properties
Utilities

Figure 1-6b The Ribbon Menus

Pull-down Menus

The *Pull-down menus* are located across the top of the Graphic window. When you click one of the items on the menu, with the left mouse button, the menu will pull down as shown in Figure 1-7 and 1-8.

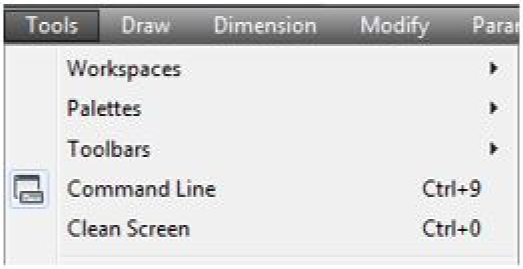
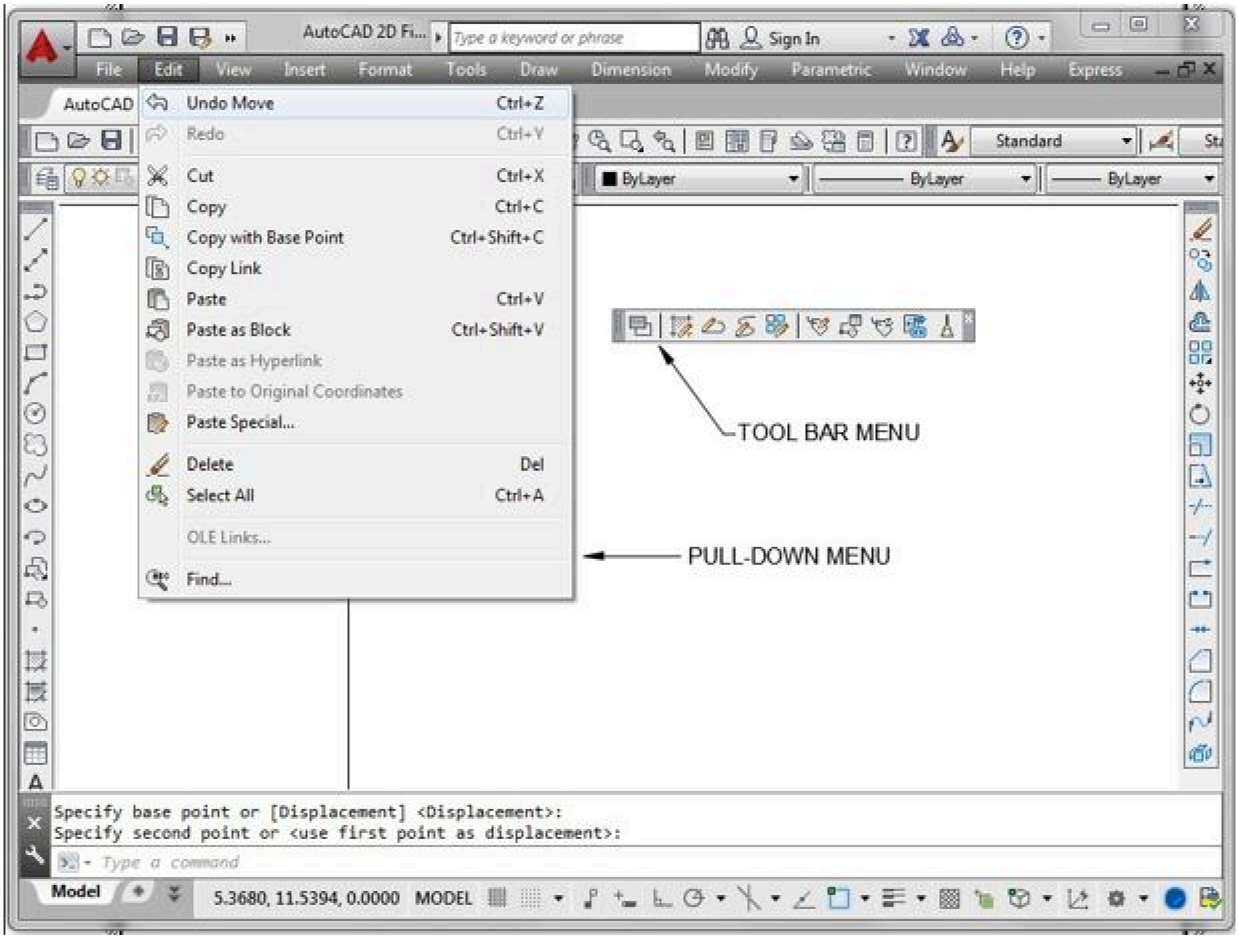

Tools
Draw
Dimension
Modify
Para
Workspaces
Palettes
Toolbars
Command Line Ctrl+9
Clean Screen Ctrl+0

Figure 1-7 Pull-Down Menu


AutoCAD 2D Fi...
Type a keyword or phrase
Sign In
File
Edit
View
Insert
Format
Tools
Draw
Dimension
Modify
Parametric
Window
Help
Express
AutoCAD
Undo Move Ctrl+Z
Redo Ctrl+Y
Cut Ctrl+X
Copy Ctrl+C
Copy with Base Point Ctrl+Shift+C
Copy Link
Paste Ctrl+V
Paste as Block Ctrl+Shift+V
Paste as Hyperlink
Paste to Original Coordinates
Paste Special...
Delete Del
Select All Ctrl+A
OLE Links...
Find...
Standard
ByLayer
ByLayer
ByLayer
Model
5.3680, 11.5394, 0.0000
MODEL

TOOL BAR MENU

PULL-DOWN MENU

```
Specify base point or [Displacement] <Displacement>:
Specify second point or <use first point as displacement>:
Type a command
```

Figure 1-8 AutoCAD's Pull-down and Toolbar Menus

A Pull-down Menu Flyout

If an item on the Pull-down menu has a small solid triangle at the end, it has a Flyout menu associated with it. Place the Graphic cursor on the triangle to force the Flyout menu to display as shown in Figure 1-9.

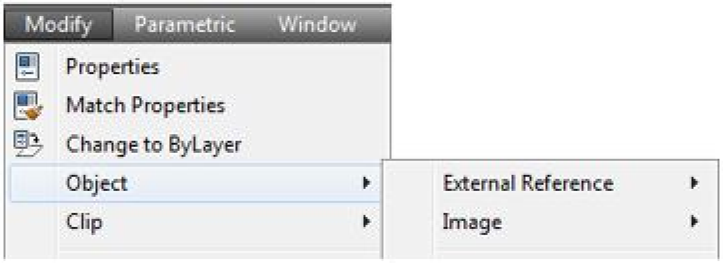

Modify
Parametric
Window
Properties
Match Properties
Change to ByLayer
Object
Clip
External Reference
Image

Figure 1-9 Flyout Menu

Toolbar Menus

Toolbar menus are a series of icons that are located on a collection of menu bars. See Figure 1-10. The display of each toolbar can be enabled or disabled at your discretion. If they were all displayed at the same time, they would virtually fill the Graphic window. Therefore, their display must be enabled or disabled, as required, by the current operation in the drawing.

Toolbars can float inside the Graphic window or they can be docked along any one of its four sides. If the Graphic cursor is placed on an icon in a toolbar, a *tooltip* will display indicating the purpose of the icon as shown in Figure 1-11. AutoCAD allows you to create or edit existing toolbar menus. This is taught in the AutoCAD 2D Advanced book.

Figure 1-10 An AutoCAD Toolbar

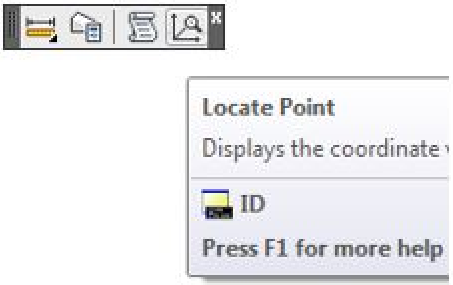

Locate Point
Displays the coordinate
ID
Press F1 for more help

Figure 1-11 A Toolbar Menu – Showing a Tooltip

Docked and Floating Toolbar Menus

In Figure 1-8, the toolbars across the top of the Graphic window are *docked* and the ones in the centre are *floating*.

Displaying Toolbar Menus

To enable or disable the display of a toolbar menu, place the Graphic cursor on any icon on a currently displayed toolbar and right-click the mouse. A list of all the available toolbar menus will display as shown in Figure 1-12. Click the name on the toolbar menu to toggle its display. A checkmark beside the toolbar's name indicates that it is currently displayed in the Graphic window.

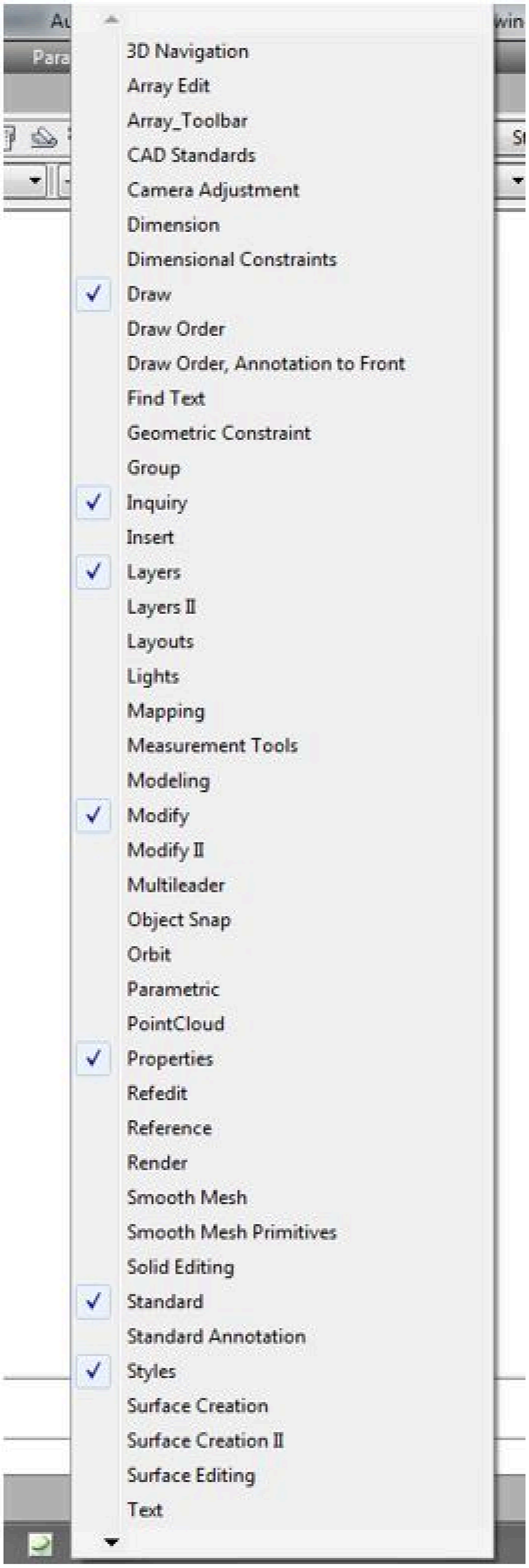

3D Navigation
Array Edit
Array_Toolbar
CAD Standards
Camera Adjustment
Dimension
Dimensional Constraints
✓ Draw
Draw Order
Draw Order, Annotation to Front
Find Text
Geometric Constraint
Group
✓ Inquiry
Insert
✓ Layers
Layers II
Layouts
Lights
Mapping
Measurement Tools
Modeling
✓ Modify
Modify II
Multileader
Object Snap
Orbit
Parametric
PointCloud
✓ Properties
Refedit
Reference
Render
Smooth Mesh
Smooth Mesh Primitives
Solid Editing
✓ Standard
Standard Annotation
✓ Styles
Surface Creation
Surface Creation II
Surface Editing
Text

Figure 1-12 Toolbar Menu Pull-down

Dialogue Boxes and Windows

When a menu item has 3 dots after it as shown in Figure 1-13 it indicates that there is more. When clicked, it will open a dialogue box or a window.

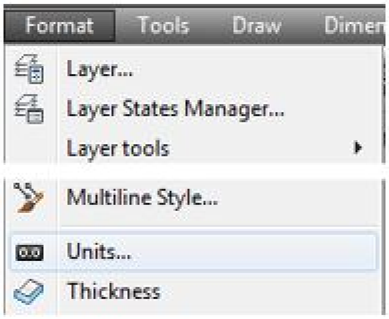
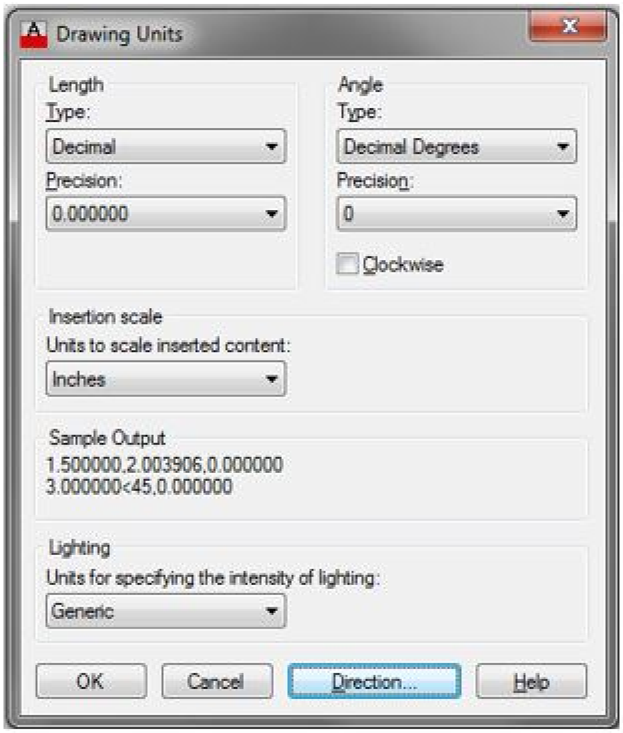

Format
Tools
Draw
Dimen
Layer...
Layer States Manager...
Layer tools
Multiline Style...
Units...
Thickness

Figure 1-13 Pull Down Menu – More

Dialogue Boxes

AutoCAD uses many different dialogue boxes to speed the drawing process. A *dialogue box* is an effective and efficient method for AutoCAD to allow you to modify or enter information, settings, and sizes. While a dialogue box is open, no other work can be done in the current drawing. It must be closed before continuing to the next command. If you close a dialogue box by clicking the X in the top right corner, it will cancel your current input in the box. If you want to save your input, you must click the OK button. A typical dialogue box is shown in Figure 1-14.


Drawing Units

Length
Type: Decimal
Precision: 0.000000

Angle
Type: Decimal Degrees
Precision: 0
☐ Clockwise

Insertion scale
Units to scale inserted content: Inches

Sample Output
1.500000,2.003906,0.000000
3.000000<45,0.000000

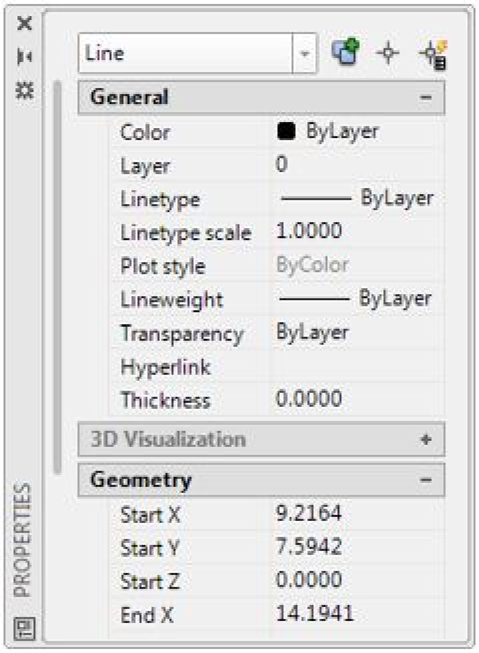
Lighting
Units for specifying the intensity of lighting: Generic

OK
Cancel
Direction...
Help

Figure 1-14 A Dialogue Box

Windows

AutoCAD uses many different *windows* to speed the drawing process. Compared to a dialogue box, a window is a more efficient method for AutoCAD and you to communicate. Windows are interactive and display data and information about the current drawing or objects in that drawing as the drawing is being worked on. Unlike a dialogue box, a window updates automatically and can remain on the screen as you work on the drawing. You must close a window by clicking the X in the upper corner. This will not cancel you input. A typical window is shown in Figure 1-15.

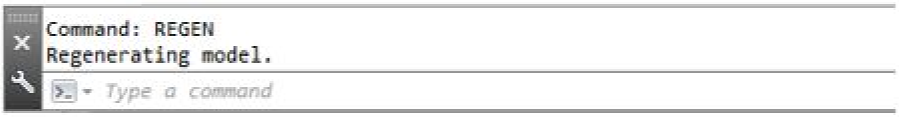


Color	ByLayer
Layer	0
Linetype	ByLayer
Linetype scale	1.0000
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	
Thickness	0.0000
Start X	9.2164
Start Y	7.5942
Start Z	0.0000
End X	14.1941

Figure 1-15 AutoCAD's Property Window

Command Line Window

The *Command Line window* is used, by you, to enter keyboard commands and user inputs. It is also used by AutoCAD to output information to you. See Figure 1-16. It is one of the ways that you communicate with AutoCAD and is a very important way for AutoCAD to communicate with you. It is important for you to watch this window closely when you are drawing. It is used by AutoCAD to ask for information, instruct you to what data it requires, or display answers to your inquiries. This will be discussed in much greater detail in Module 2.

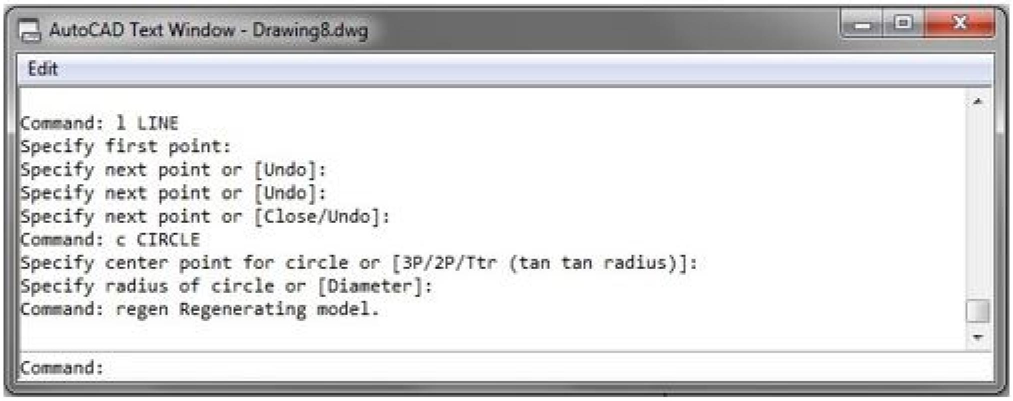


```
Command: REGEN
Regenerating model.
```

Figure 1-16 Command Line Window

Text Window

The *Text window*, Figure 1-17, displays all of the commands that were entered by you and AutoCAD's responses to those command for the duration that the current drawing is open. This is one way for you to look back at the commands, user input, and AutoCAD's responses. To toggle the display of the Text window, press F2.



```
Command: l LINE
Specify first point:
Specify next point or [Undo]:
Specify next point or [Undo]:
Specify next point or [Close/Undo]:
Command: c CIRCLE
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
Specify radius of circle or [Diameter]:
Command: regen Regenerating model.

Command:
```

Figure 1-17 AutoCAD's Text Window

The Mouse

AutoCAD is programed to use the three buttons on a mouse as follows:

Left Button: This is the pick button. Use it to pick objects, pick menu items, or to select locations on the drawing.

Middle Button: The middle button, or wheel, is used to zoom and pan around the drawing. This will be discussed in detail in Module 9.

Right Button: The right button has two uses. If pressed and released quickly, it repeats the last command. If pressed and held in, it displays the cursor menu. See Figure 1-18.

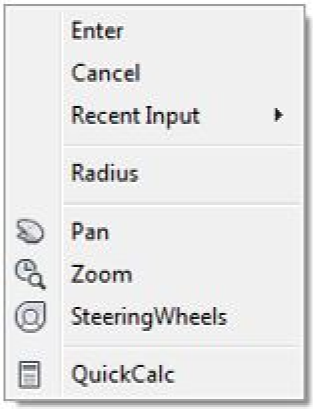


Figure 1-18 Cursor Menu for the FILLET Command

The cursor menu displays differently depending on the current or last command entered. It can be a very helpful menu once you become a more experienced user.

WORKALONG: Learning to Use AutoCAD’s Interface Using Toolbar Menus

If you are using Ribbon menus, skip forward to WORKALONG: Learning to Use AutoCAD’s Interface using Ribbon Menus.

Step 1

Launch AutoCAD. AutoCAD’s blank window will display. (Figure Step 1)

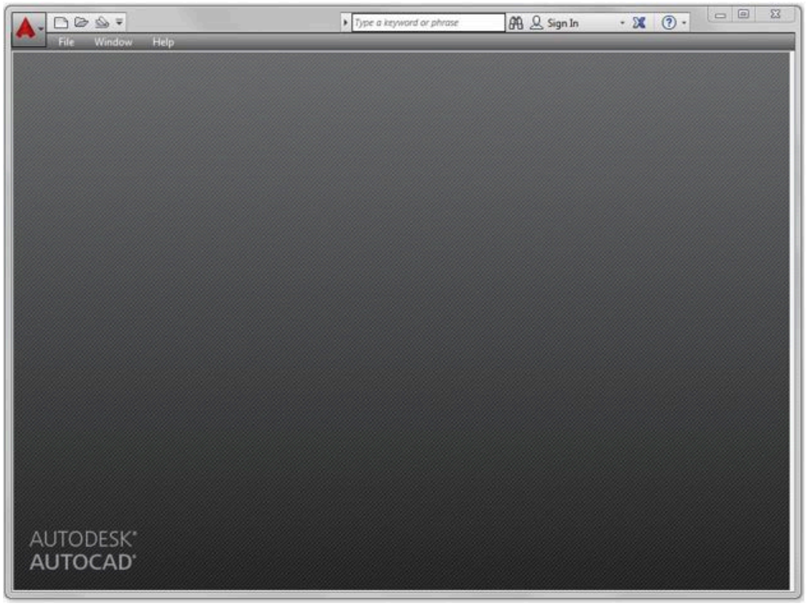

File
Window
Help
Type a keyword or phrase
Sign In
AUTODESK
AUTOCAD

Figure Step 1

AUTHOR’S COMMENTS: The blank window displays grey when there is no active drawing. Your window should closely match the figure.

Step 2

Click the New icon located in the upper right corner as shown in the figure. This will open the Select Template dialogue box. (Figure Step 2)

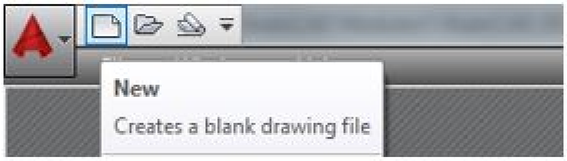
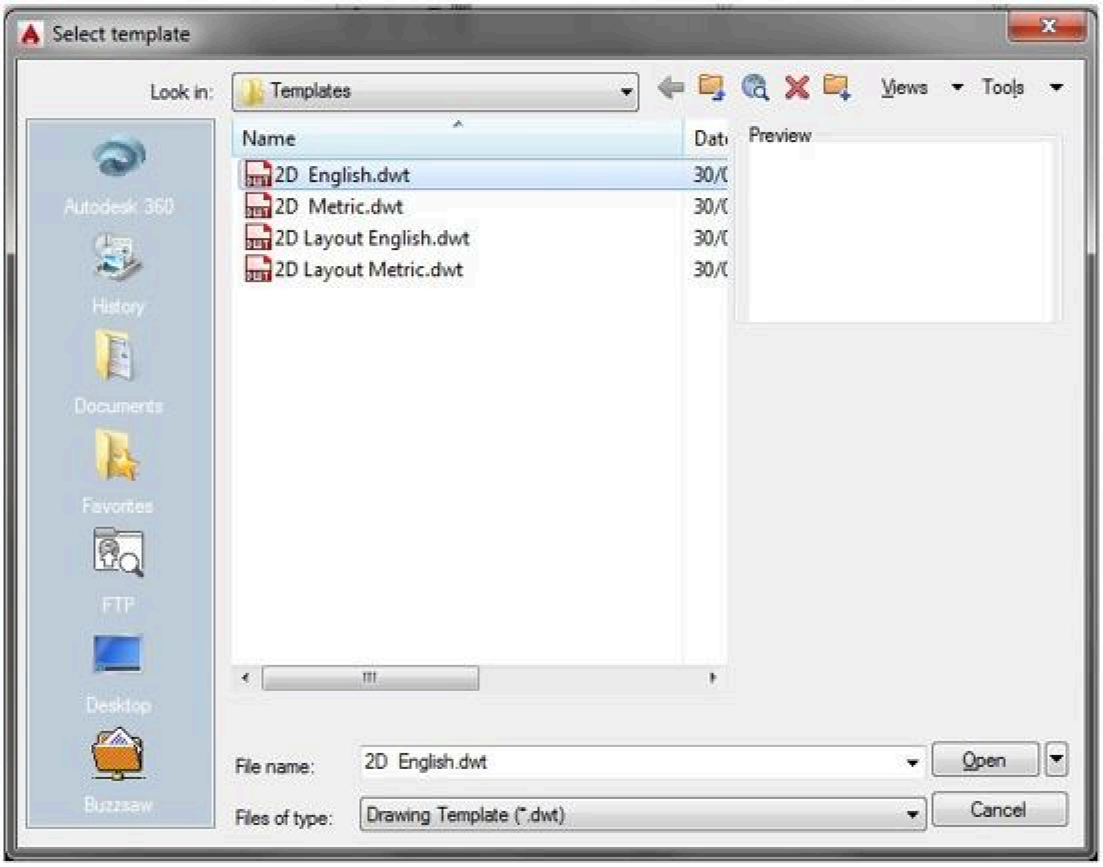

New
Creates a blank drawing file

Figure Step 2

Step 3

In the Select Template dialogue box, click the template file: 2D English to highlight it. Then click the Open button. This will open AutoCAD's Graphic window. (Figure Step 3)


Select template
Views
Tools
Name
Date
Preview
Autodesk 360
History
Documents
Favorites
FTP
Desktop
Buzzsaw
Open
Cancel

Look in: Templates

2D English.dwt 30/(
2D Metric.dwt 30/(
2D Layout English.dwt 30/(
2D Layout Metric.dwt 30/(

File name: 2D English.dwt

Files of type: Drawing Template (*.dwt)

Figure Step 3

AUTHOR'S COMMENTS: The four templates are part of the AutoCAD 2D book and must be used to complete all workalongs and lab exercises. If you do NOT see the four templates files shown in Figure Step 3, you must go back and redo module Introduction 2. If you only see the name and not the .dwt extension, that is OK. Do not continue in this module until your dialogue box shows the four templates. If you have trouble getting it to match and have an instructor, email him/her for help.

Step 4

Click the small arrow in the Workplace Switching icon again on the Status bar. Click the workspace: AutoCADbook to set it as the current workspace. Make sure that the check mark is beside AutoCADbook as shown in the figure. Your Graphic window should match the figure. (Figure Step 4A and 4B)

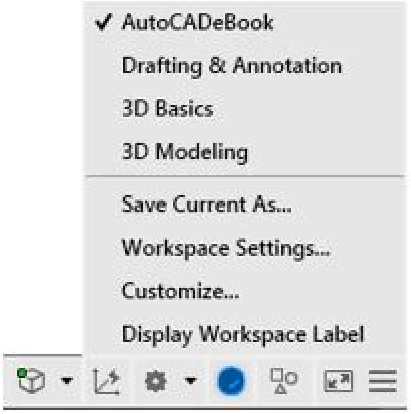
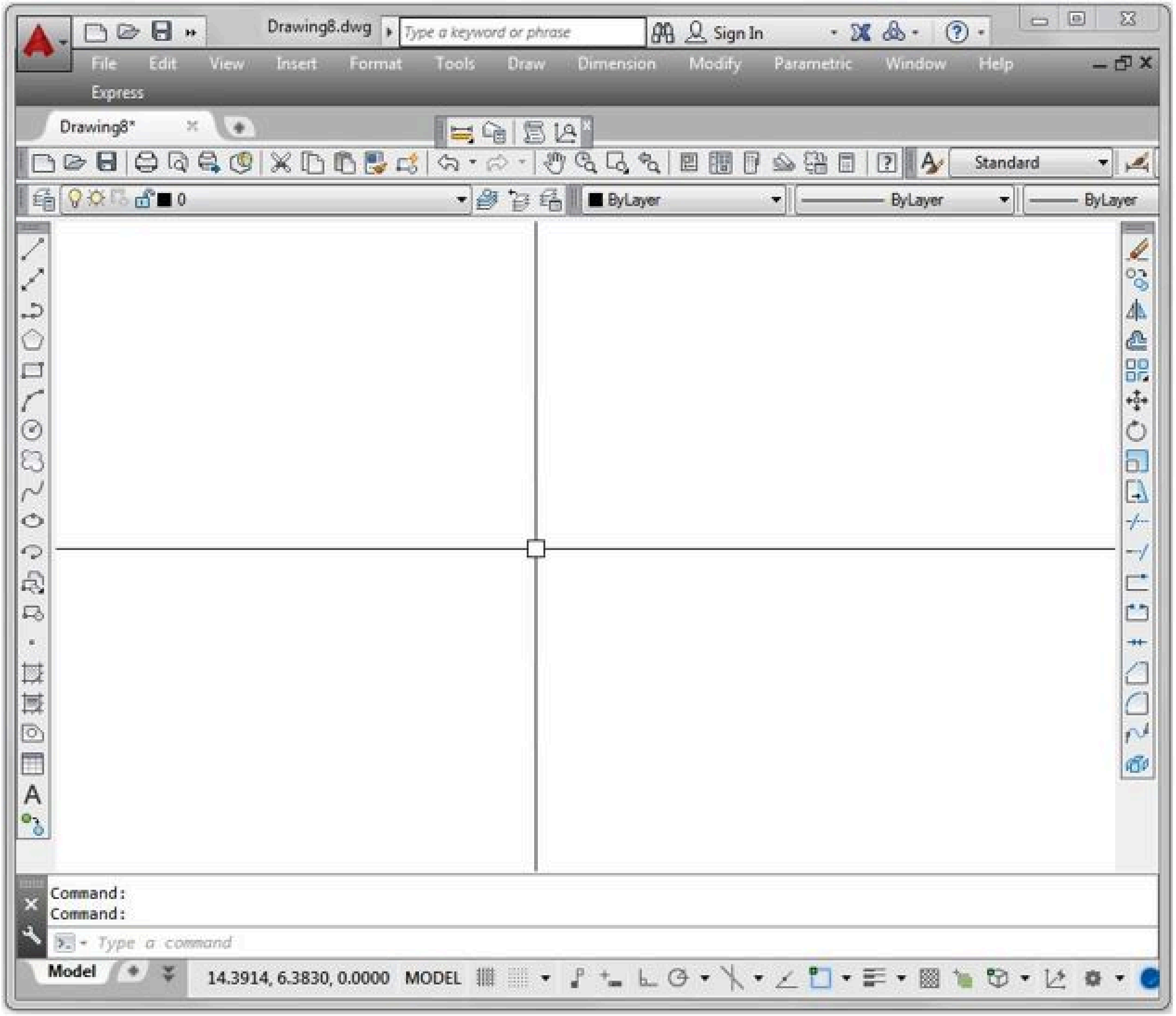

✓ AutoCADeBook
Drafting & Annotation
3D Basics
3D Modeling
Save Current As...
Workspace Settings...
Customize...
Display Workspace Label

Figure Step 4A


Drawing8.dwg
Type a keyword or phrase
Sign In
File
Edit
View
Insert
Format
Tools
Draw
Dimension
Modify
Parametric
Window
Help
Express
Drawing8*
Standard
0
ByLayer
ByLayer
ByLayer
Type a command
Model
14.3914, 6.3830, 0.0000
MODEL

```
Command:
Command:
```

Figure Step 4B

AUTHOR’S COMMENTS: Your Graphic window should closely match the figure.

Step 5

Check to ensure that the Model tab is enabled. It is located on the bottom left corner of the window. If it is not enabled, click it with the left mouse button. (Figure Step 5)

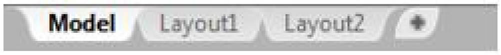

Model
Layout1
Layout2

Figure Step 5

AUTHOR’S COMMENTS: The Model tab should be enabled at all times, while working on workalongs and lab exercises in the AutoCAD 2D book, unless you are instructed otherwise. Module 18 will teach you how work in the Layout tabs.

Step 6

Disable all features on the Status bar by clicking any that display with a blue background as shown in Figure Step 6A. All features should display with a gray background as shown in Figure Step 6B. (Figure Step 6A and 6B)

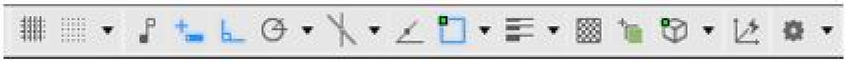


Figure Step 6A

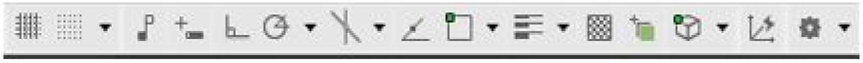
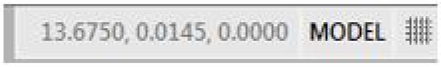


Figure Step 6B

AUTHOR'S COMMENTS: Important Unless otherwise instructed, each time you start a workalong or lab exercise while working on the AutoCAD 2D book, all features on the Status bar should be disabled as shown in Figure Step 6B.

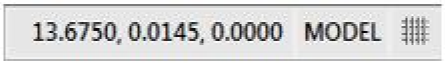
Step 7

Click the coordinate display, located on the left side of the Status bar, to disable it shown in Figure Step 7A. Click it again to enable it as shown in Figure Step 7B. (Figure Step 7A and 7B)



13.6750, 0.0145, 0.0000 MODEL

Figure Step 7A

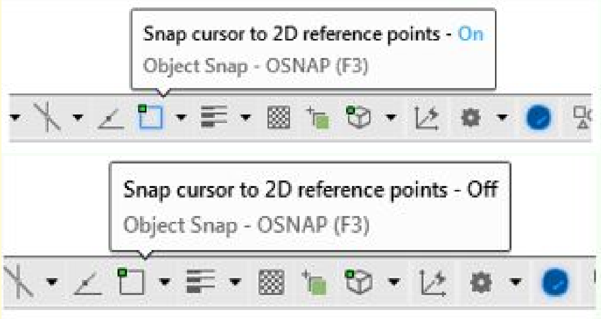


13.6750, 0.0145, 0.0000 MODEL

Figure Step 7B

AUTHOR'S COMMENTS: The coordinate display should be enabled while working in AutoCAD. While it is not used very much in day-to-day drawing, it can be very handy at times.

USER TIP: You can easily check to see if a feature on the Status bar is enabled or disabled by holding your graphic cursor on the icon. A pop up window will display indicating the name of the feature and whether it is currently on or off as shown in the figures.



Snap cursor to 2D reference points - On
Object Snap - OSNAP (F3)

Snap cursor to 2D reference points - Off
Object Snap - OSNAP (F3)

Step 8

Click the Draw item in the Pull-down menu to pull down the menu. Hold the Graphic cursor over the flyout triangle at the end of the Circle command to force the flyout to display. (Figure Step 8)

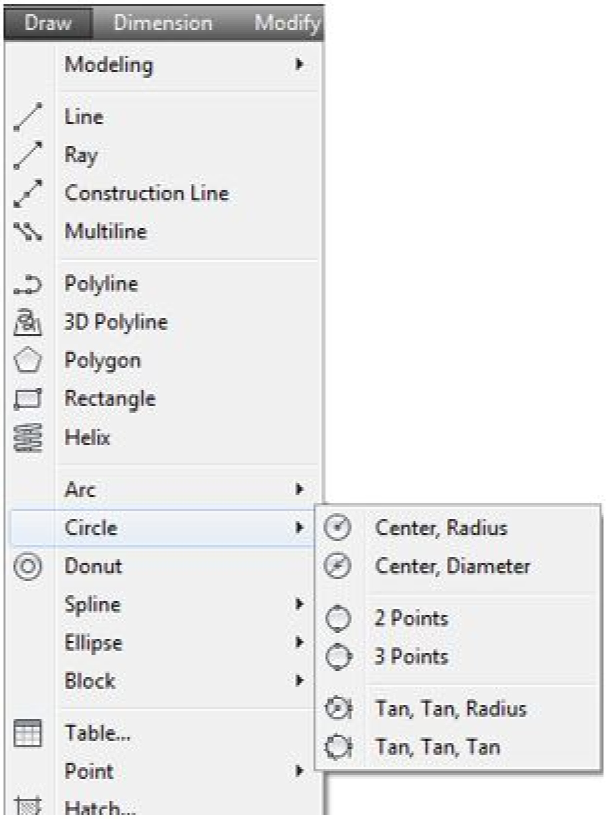

Draw
Dimension
Modify
Modeling
Line
Ray
Construction Line
Multiline
Polyline
3D Polyline
Polygon
Rectangle
Helix
Arc
Circle
Donut
Spline
Ellipse
Block
Table...
Point
Hatch...
Center, Radius
Center, Diameter
2 Points
3 Points
Tan, Tan, Radius
Tan, Tan, Tan

Figure Step 8

Step 9

Move the Graphic cursor onto any toolbar icon and right click the mouse. This will pull down a list of all available toolbar menus. The ones that are preceded with a checkmark are enabled and are currently displayed in the Graphic window. Ensure that the toolbars Draw, Inquiry, Layers, Modify, Properties, Standard, and Styles are the only ones enabled. Click on the name to toggle the display of the toolbar. (Figure Step 9)

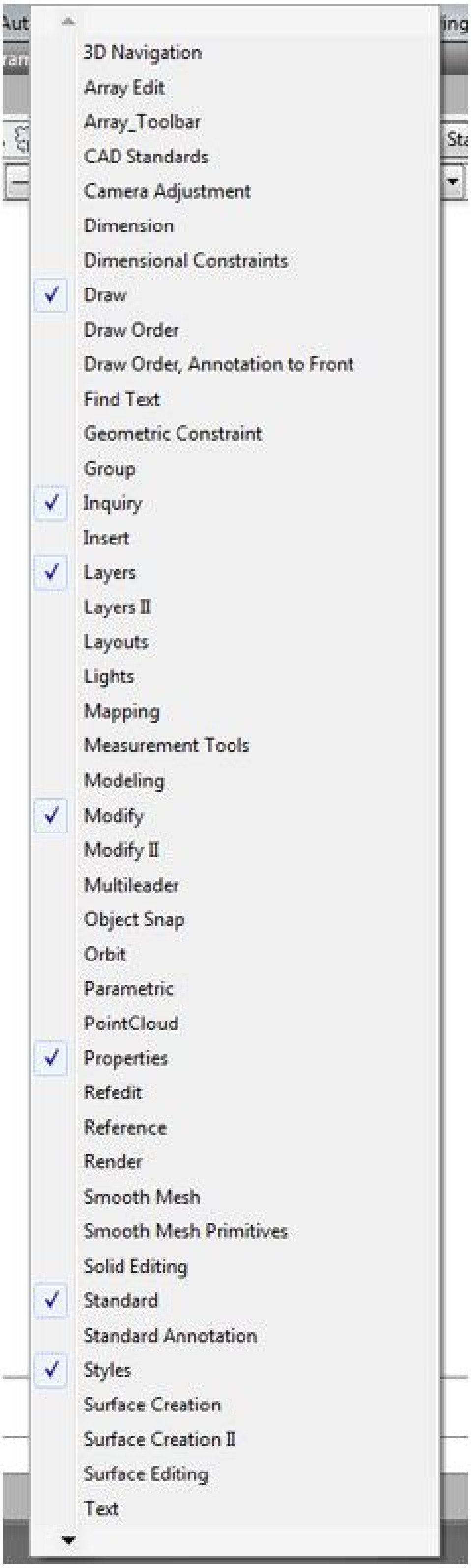

3D Navigation
Array Edit
Array_Toolbar
CAD Standards
Camera Adjustment
Dimension
Dimensional Constraints
✓ Draw
Draw Order
Draw Order, Annotation to Front
Find Text
Geometric Constraint
Group
✓ Inquiry
Insert
✓ Layers
Layers II
Layouts
Lights
Mapping
Measurement Tools
Modeling
✓ Modify
Modify II
Multileader
Object Snap
Orbit
Parametric
PointCloud
✓ Properties
Refedit
Reference
Render
Smooth Mesh
Smooth Mesh Primitives
Solid Editing
✓ Standard
Standard Annotation
✓ Styles
Surface Creation
Surface Creation II
Surface Editing
Text

Figure Step 9

Step 10

Move the Graphic cursor onto the lines at the end of the Layers toolbar. Press and hold the left mouse button down and drag the toolbar somewhere into the Graphic window and release the button. (Figure Step 10A, 10B, and 10C)

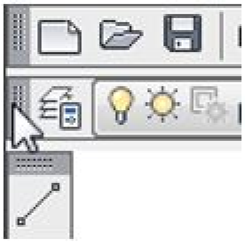


Figure Step 10A

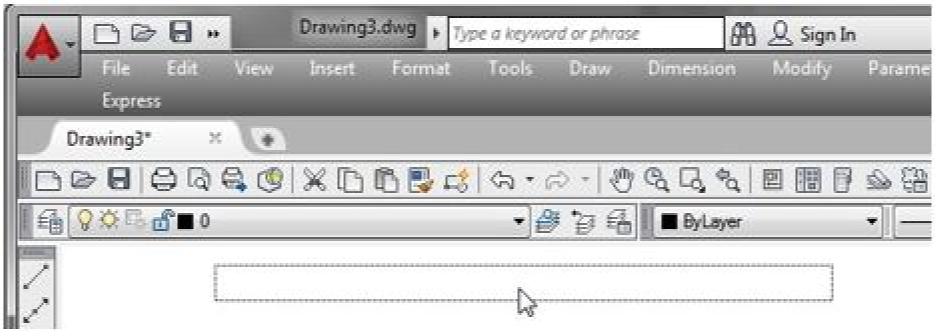

Drawing3.dwg
Type a keyword or phrase
Sign In
File
Edit
View
Insert
Format
Tools
Draw
Dimension
Modify
Parame
Express
Drawing3*
0
ByLayer

Figure Step 10B

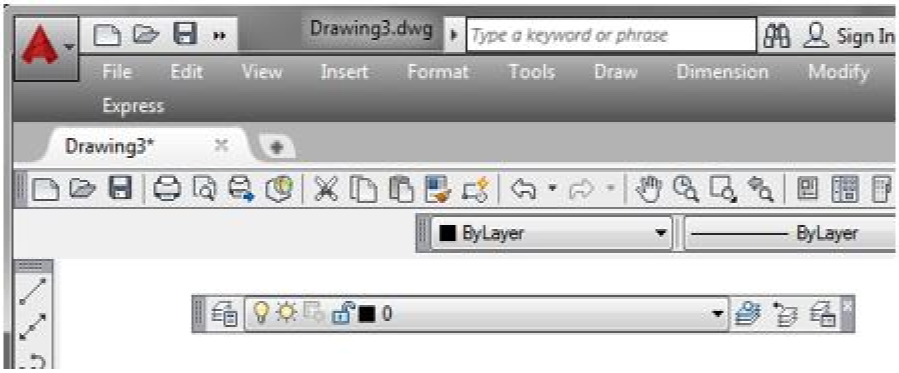

Drawing3.dwg
Type a keyword or phrase
Sign In
File
Edit
View
Insert
Format
Tools
Draw
Dimension
Modify
Express
Drawing3*
ByLayer
ByLayer
0

Figure Step 10C

AUTHOR'S COMMENTS: This step may be a little tricky the first few tries. Practice it until you can do it easily.

Step 11

Move the cursor onto bars at the end of the Layers toolbar and hold it for a few seconds. The toolbar's name, Layers, will display as a tooltip. (Figure Step 11)

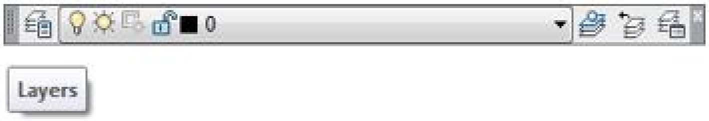


Figure Step 11

Step 12

Using what was taught in Step 9, drag all six toolbars into the Graphic window as shown in the figure. (Figure Step 12A and 12B)

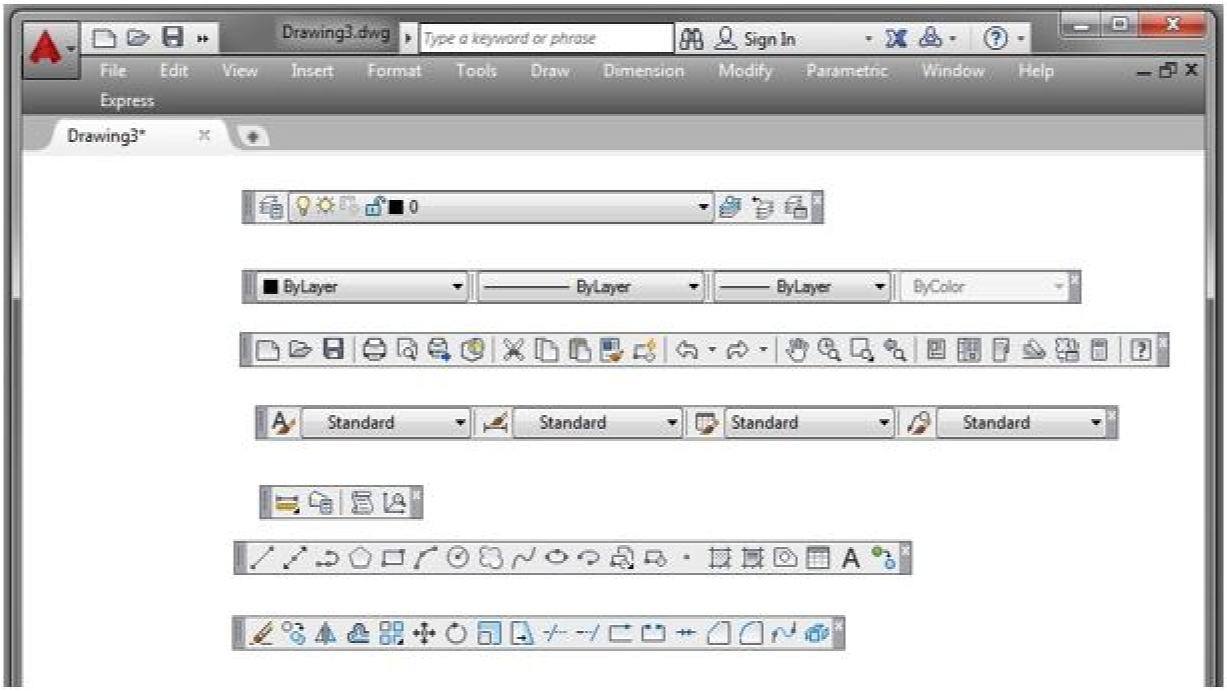

Drawing3.dwg
Type a keyword or phrase
Sign In
File
Edit
View
Insert
Format
Tools
Draw
Dimension
Modify
Parametric
Window
Help
Express
Drawing3*
0
ByLayer
ByLayer
ByLayer
ByColor
Standard
Standard
Standard
Standard
A

Figure Step 12A

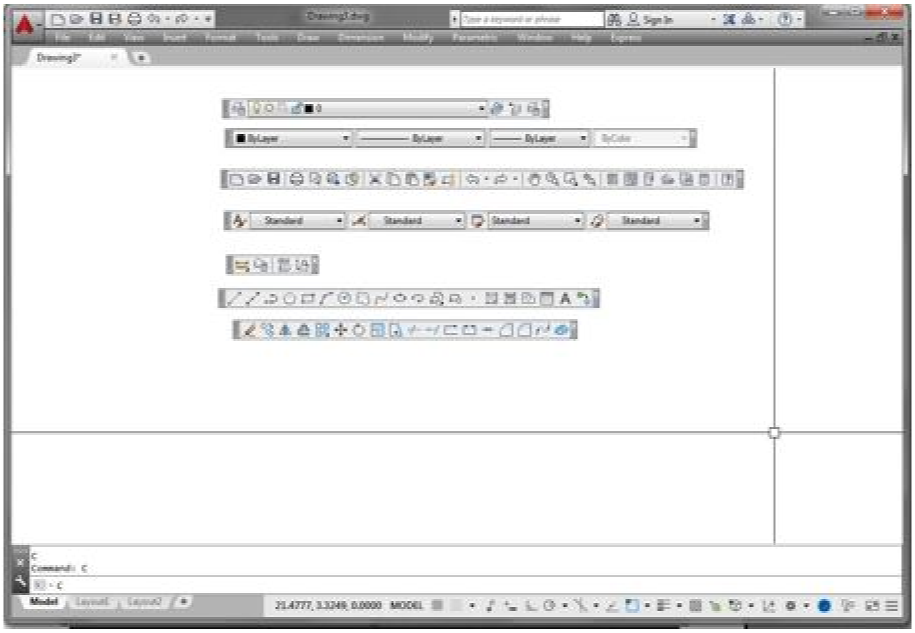


Figure Step 12B

Step 13

Using what you learned doing Steps 9 and 12, drag all six toolbars back into position by docking them to the locations shown in the figure. (Figure Step 13)

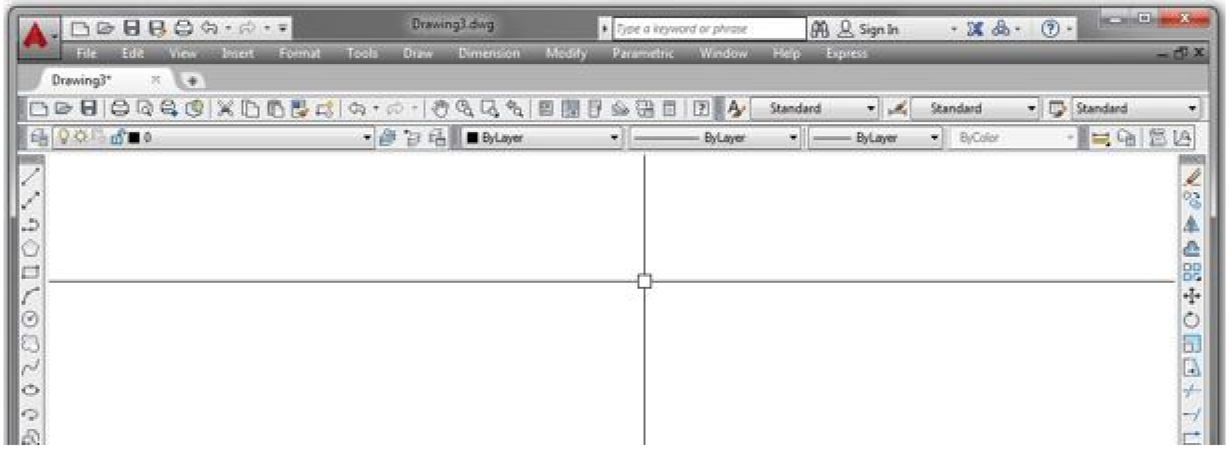


Figure Step 13

> **AUTHOR'S COMMENTS:** Dragging and docking the toolbar can be a little tricky at first. Practice it until it is easy for you to do. The six toolbars are now all docked.

Step 14

Using what you learned earlier in the workalong, enable the display of the Modify II toolbar. It will display as a floating toolbar. (Figure Step 14A and 14B)

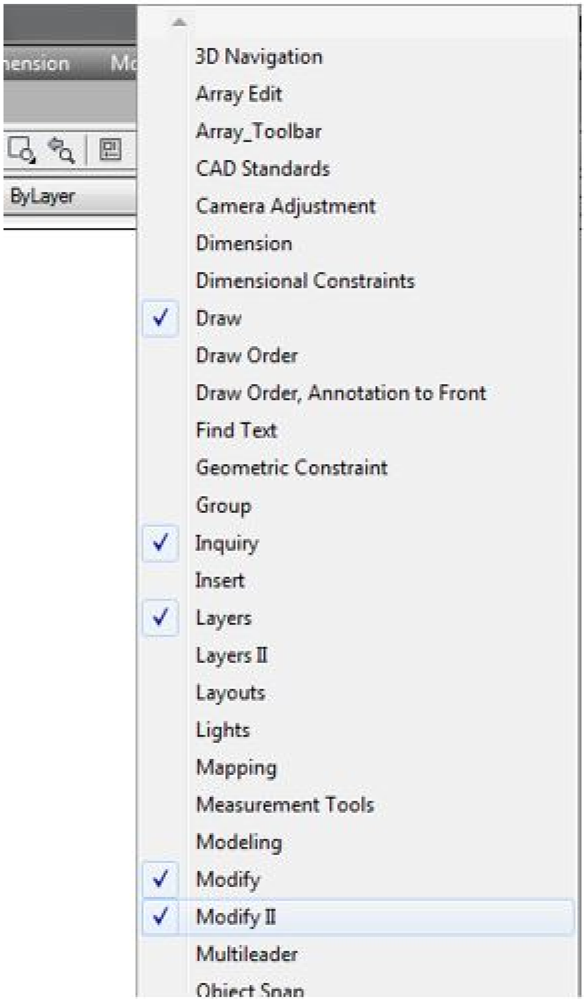
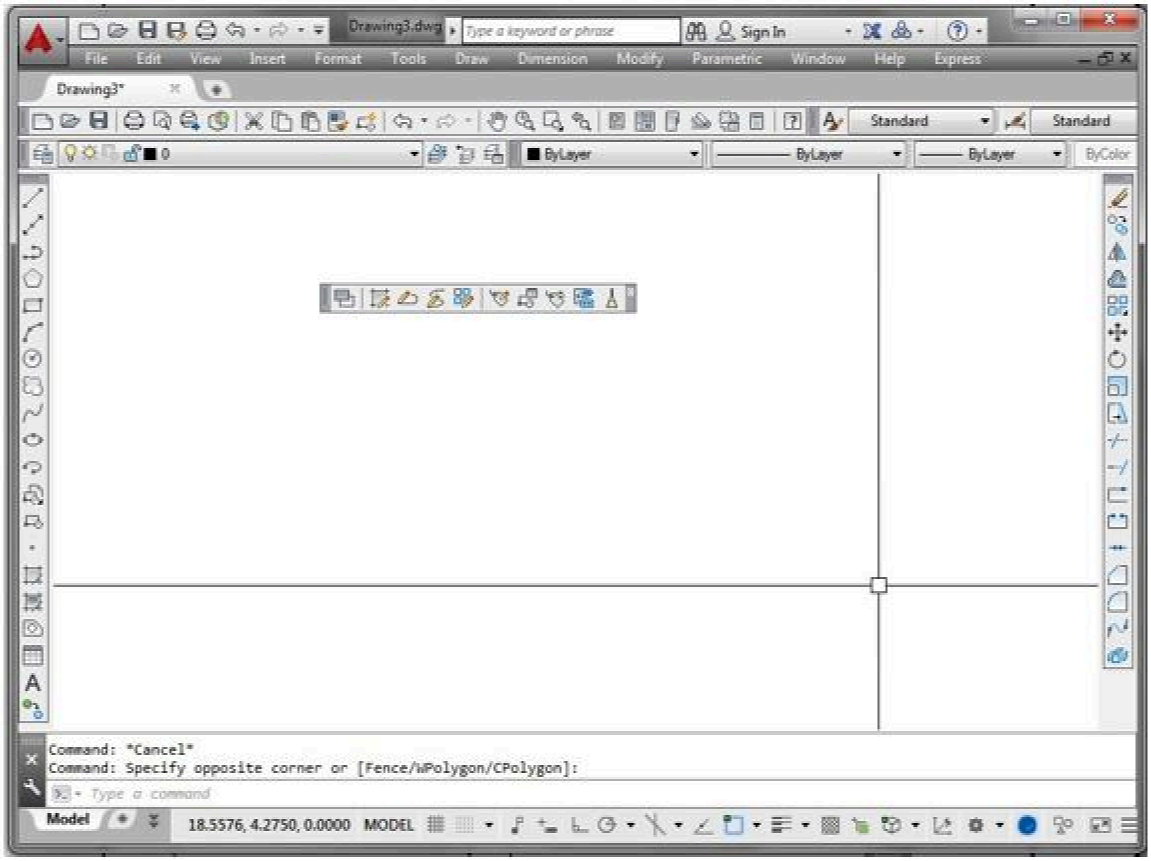

ension
Mo
ByLayer
3D Navigation
Array Edit
Array_Toolbar
CAD Standards
Camera Adjustment
Dimension
Dimensional Constraints
✓ Draw
Draw Order
Draw Order, Annotation to Front
Find Text
Geometric Constraint
Group
✓ Inquiry
Insert
✓ Layers
Layers II
Layouts
Lights
Mapping
Measurement Tools
Modeling
✓ Modify
✓ Modify II
Multileader
Object Snap

Figure Step 14A


Drawing3.dwg
Type a keyword or phrase
Sign In
File
Edit
View
Insert
Format
Tools
Draw
Dimension
Modify
Parametric
Window
Help
Express
Drawing3*
Standard
Standard
0
ByLayer
ByLayer
ByLayer
ByColor
Model
MODEL

```
Command: *Cancel*
Command: Specify opposite corner or [Fence/WPolygon/CPolygon]:
Type a command
```

18.5576, 4.2750, 0.0000

Figure Step 14B

Step 15

Move the Graphic cursor onto the top edge of the Modify II toolbar until the cursor displays as double arrows. See Figure Step 15A. Press the left mouse button and while holding it down, drag the toolbar upwards until it matches Figure Step 15C. (Figure Step 15 A, 15B, 15C)

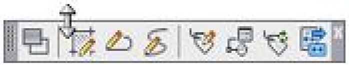


Figure Step 15A

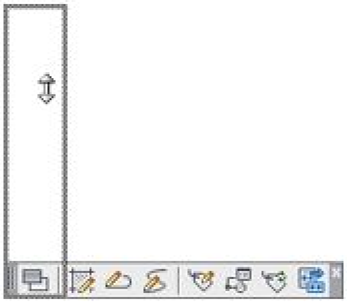


Figure Step 15B

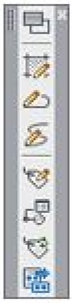


Figure Step 15C

AUTHOR'S COMMENTS: Changing the shape of a toolbar can be a little tricky at first. Practice it until it is easy for you to do.

Step 16

Using what was taught in Step 14 and 15, enable the display of the Object Snap toolbar and change its appearance to match the figure. (Figure Step 16)

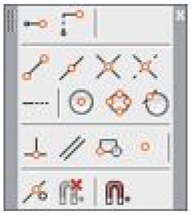


Figure Step 16

Step 17

Close the Object Snap and Modify II toolbars by clicking the X in the top right corner of each toolbar.

AUTHOR'S COMMENTS: As you can see, toolbars can easily be displayed, have there shape changed, or moved around the Graphic window as needed. You can enable their display, when required, in the drawing process and closed them when the process is completed. While working on the AutoCAD 2D book, leave the 6 toolbar displayed and docked at all times as shown in Figure Step 13.

Step 18

Click Format on the Pull-down menu and then click Units to open the Drawing Units dialogue box. Click OK to close the dialogue box. (Figure Step 18A and 18B)

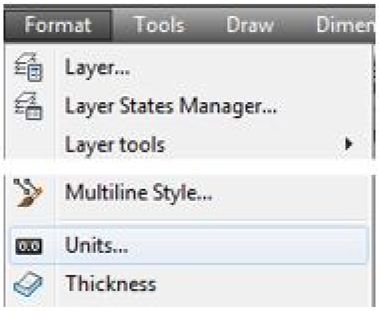
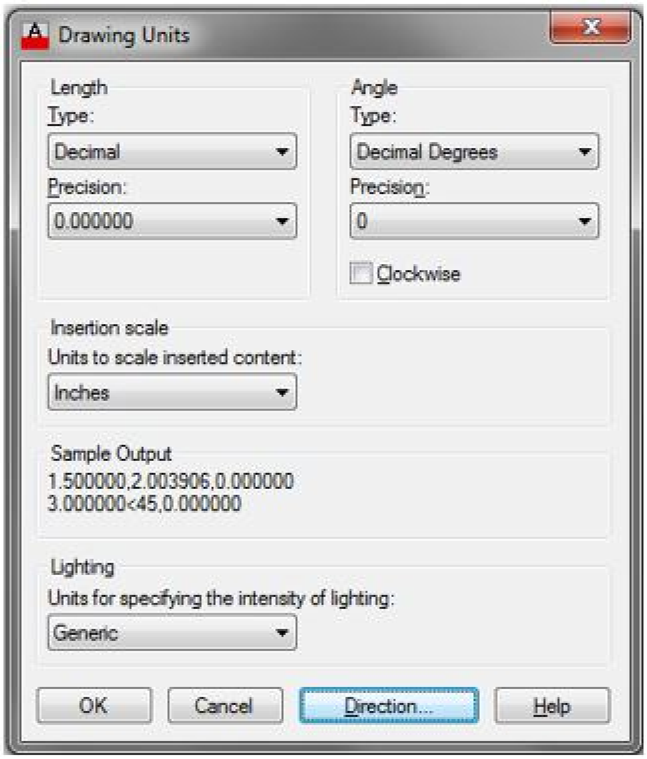

Format
Tools
Draw
Dimen
Layer...
Layer States Manager...
Layer tools
Multiline Style...
Units...
Thickness

Figure Step 18A


Drawing Units

Length
Type: Decimal
Precision: 0.000000

Angle
Type: Decimal Degrees
Precision: 0
☐ Clockwise

Insertion scale
Units to scale inserted content: Inches

Sample Output
1.500000,2.003906,0.000000
3.000000<45,0.000000

Lighting
Units for specifying the intensity of lighting: Generic

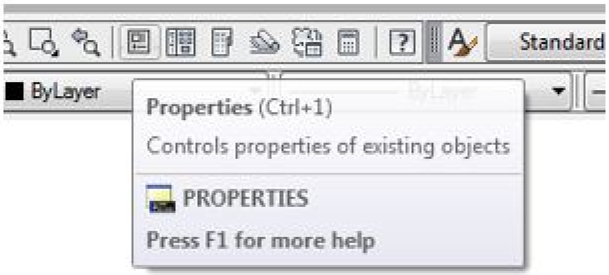
OK
Cancel
Direction...
Help

Figure Step 18B

AUTHOR'S COMMENTS: Always close a dialogue box using the OK or the Cancel button. If you close it by clicking the X in the top right corner, you are cancelling all changes that you made while the box was open. If you want to cancel the changes, click the Cancel button to close the dialogue box. When you click OK, you are closing it and saving all the changes you made.

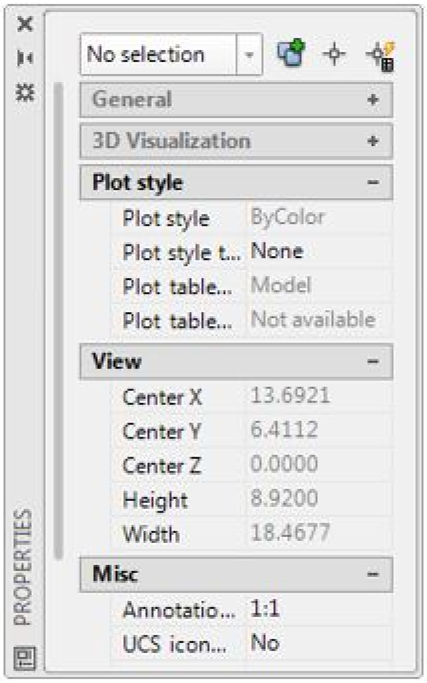
Step 19

Move the Graphic cursor onto the icons in the Standard toolbar. As you move it over the icons, tooltips with the toolbar's name will display. Find the Properties icon. Click it to open the Properties window. (Figure Step 19A and 19B)



Properties (Ctrl+1)
Controls properties of existing objects
PROPERTIES
Press F1 for more help

Figure Step 19A



No selection

Plot style
Plot style: ByColor
Plot style t...: None
Plot table...: Model
Plot table...: Not available

View
Center X: 13.6921
Center Y: 6.4112
Center Z: 0.0000
Height: 8.9200
Width: 18.4677

Misc
Annotatio...: 1:1
UCS icon...: No

Figure Step 19B

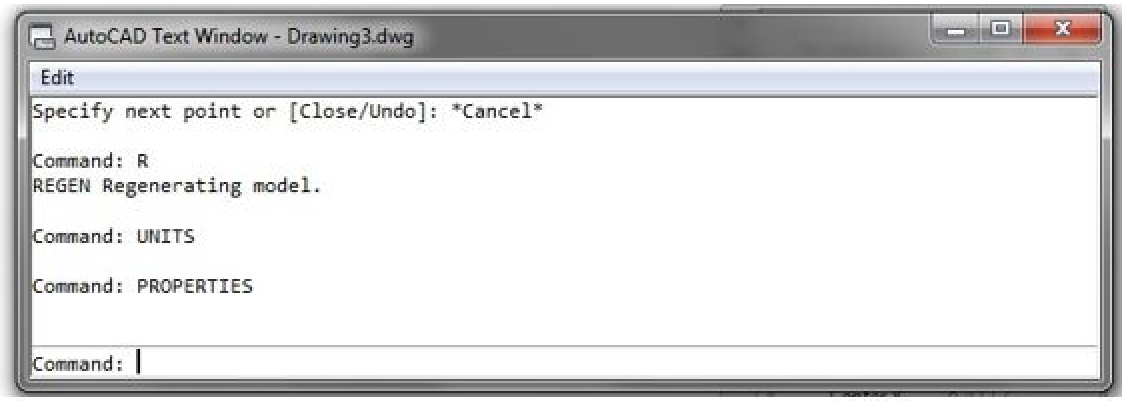
Step 20

Click the X in the top corner of the window to close it.

AUTHOR'S COMMENTS: Windows are different than dialogue box as they are interactive. Any changes made, while they are open, are saved immediately. That makes it OK to close them by clicking the X.

Step 21

Press the function key F2. It will open the Text window as shown in the figure. (Figure Step 21)


AutoCAD Text Window - Drawing3.dwg
Edit

```
Specify next point or [Close/Undo]: *Cancel*

Command: R
REGEN Regenerating model.

Command: UNITS

Command: PROPERTIES


Command:
```

Figure Step 21

AUTHOR’S COMMENTS: Your Text window may not match the figure exactly.

Step 22

Press the function key F2 again to close the Text window.

Step 23

Click File on the Pull-down menu. Click Exit to close AutoCAD. If you are asked if you want to save the drawing, click No. (Figure Step 23)

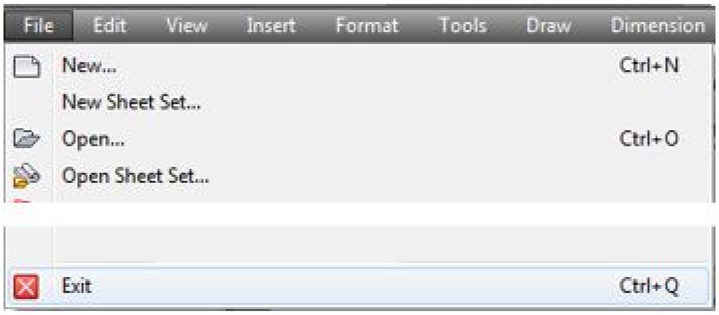

File
Edit
View
Insert
Format
Tools
Draw
Dimension
New... Ctrl+N
New Sheet Set...
Open... Ctrl+O
Open Sheet Set...
Exit Ctrl+Q

Figure Step 23

Step 24

Go to Module 2.

WORKALONG: Learning to Use AutoCAD's Interface using Ribbon Menus

If you are using Toolbar menus, go to Module 2.

Step 1

Launch AutoCAD. AutoCAD's blank window will display. (Figure Step 1)

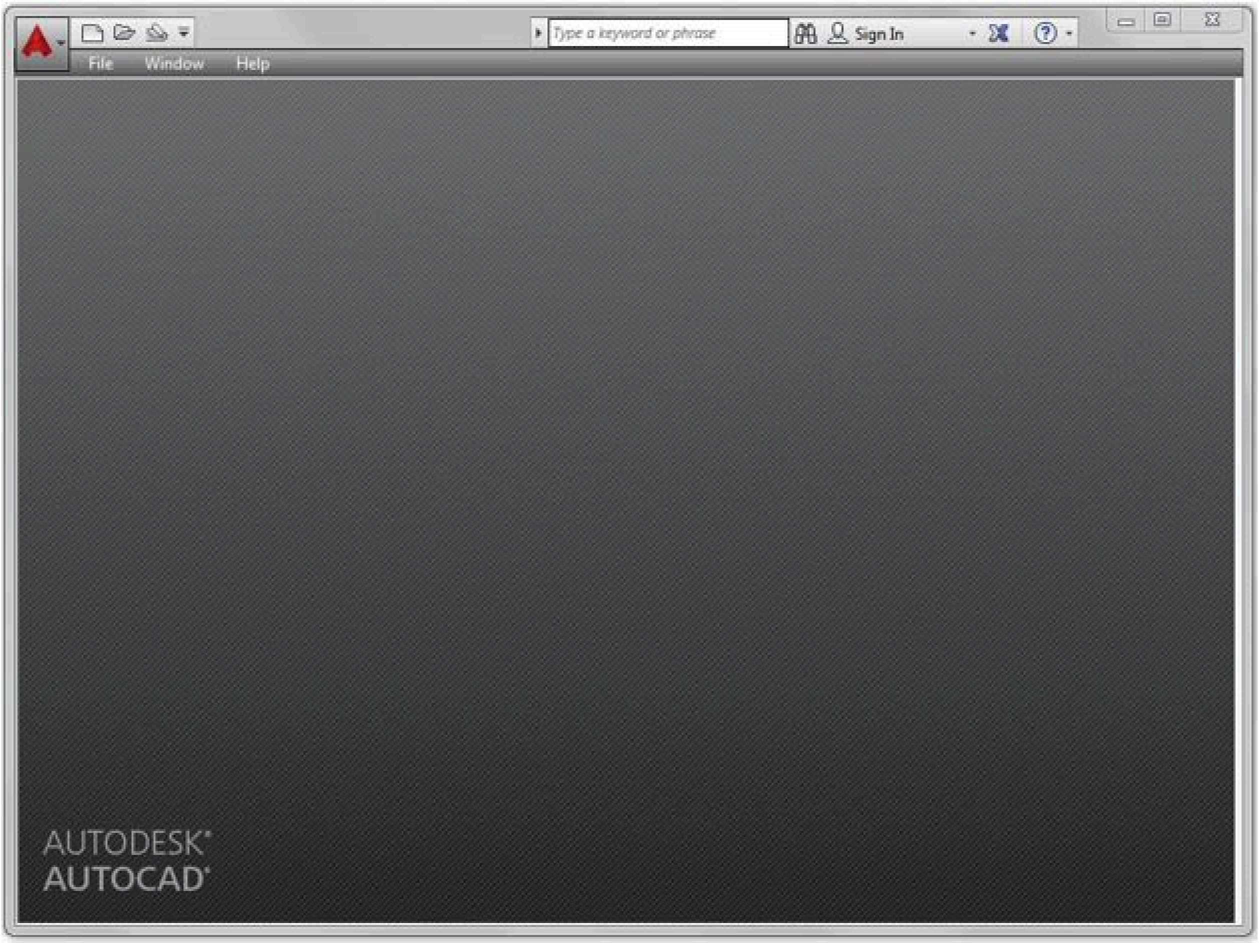

Type a keyword or phrase
Sign In
File
Window
Help
AUTODESK
AUTOCAD

Figure Step 1

AUTHOR'S COMMENTS: The blank window displays grey when there is no active drawing. Your window should closely match the figure.

Step 2

Click the New icon located in the upper right corner as shown in the figure. This will open the Select Template dialogue box. (Figure Step 2)

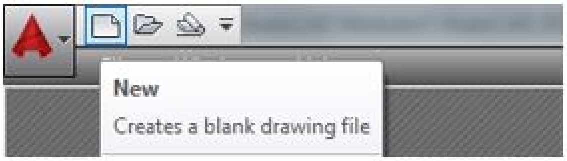
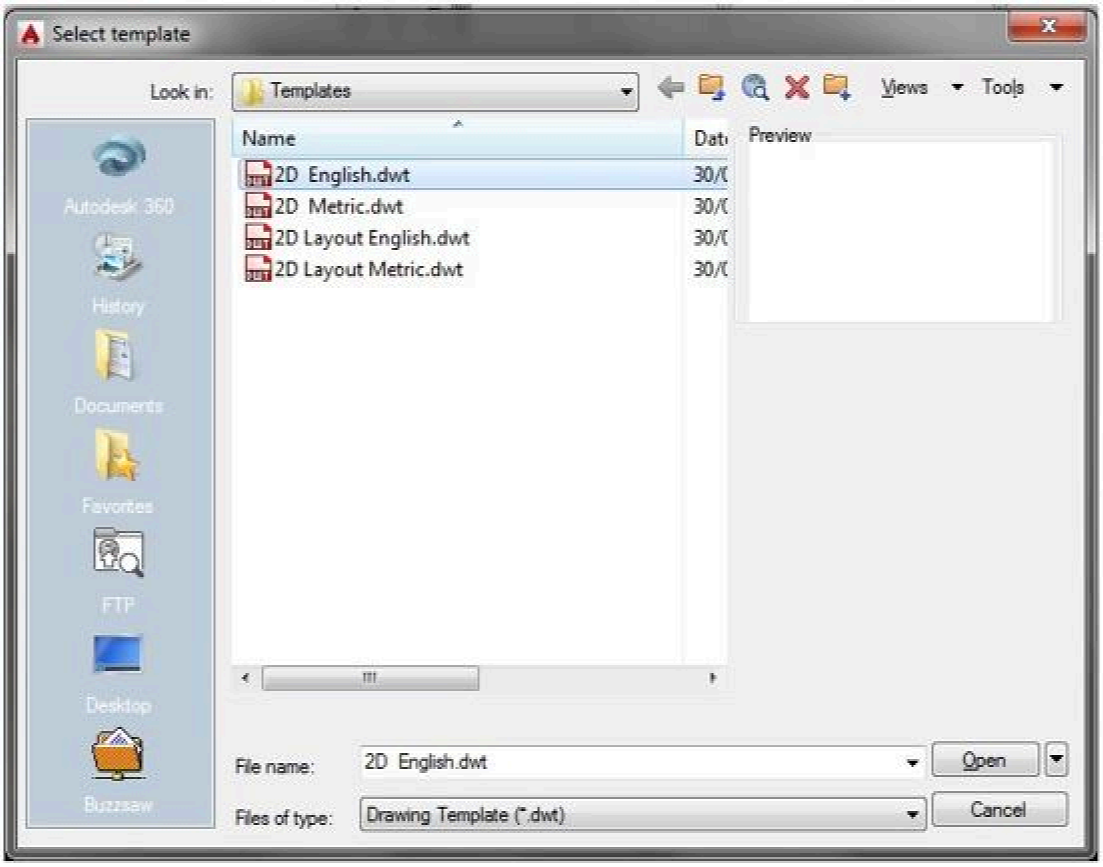

New
Creates a blank drawing file

Figure Step 2

Step 3

In the Select Template dialogue box, click the template file: 2D English to highlight it. Then click the Open button. This will open AutoCAD's Graphic window. (Figure Step 3)


Look in: Templates

2D English.dwt
2D Metric.dwt
2D Layout English.dwt
2D Layout Metric.dwt

File name: 2D English.dwt

Files of type: Drawing Template (*.dwt)

Select template
Views
Tools
Name
Dat
30/(
30/(
30/(
30/(
Preview
Autodesk 360
History
Documents
Favorites
FTP
Desktop
Buzzsaw
Open
Cancel

Figure Step 3

AUTHOR'S COMMENTS: The four templates are part of the AutoCAD 2D book and must be used to complete all workalongs and lab exercises. If you do NOT see the four templates files shown in Figure Step 3, you must go back and redo module Introduction 2. If you only see the name and not the .dwt extension, that is OK. Do not continue in this module until your dialogue box shows the four templates. If you have trouble getting it to match and have an instructor, email him/her for help.

Step 4

Click the small arrow in the Workplace Switching icon again on the Status bar. Click the workspace:

Drafting & Annotation to set it as the current workspace. Make sure that the check mark is beside Drafting & Annotation as shown in the figure. Your Graphic window should closely match the figure. (Figure Step 4A and 4B)

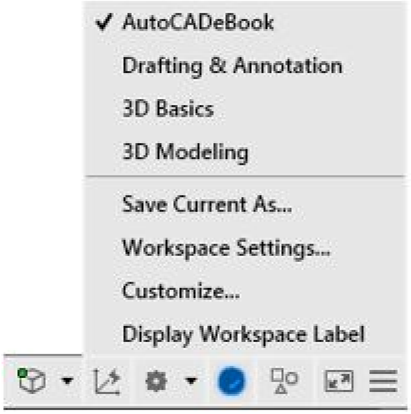
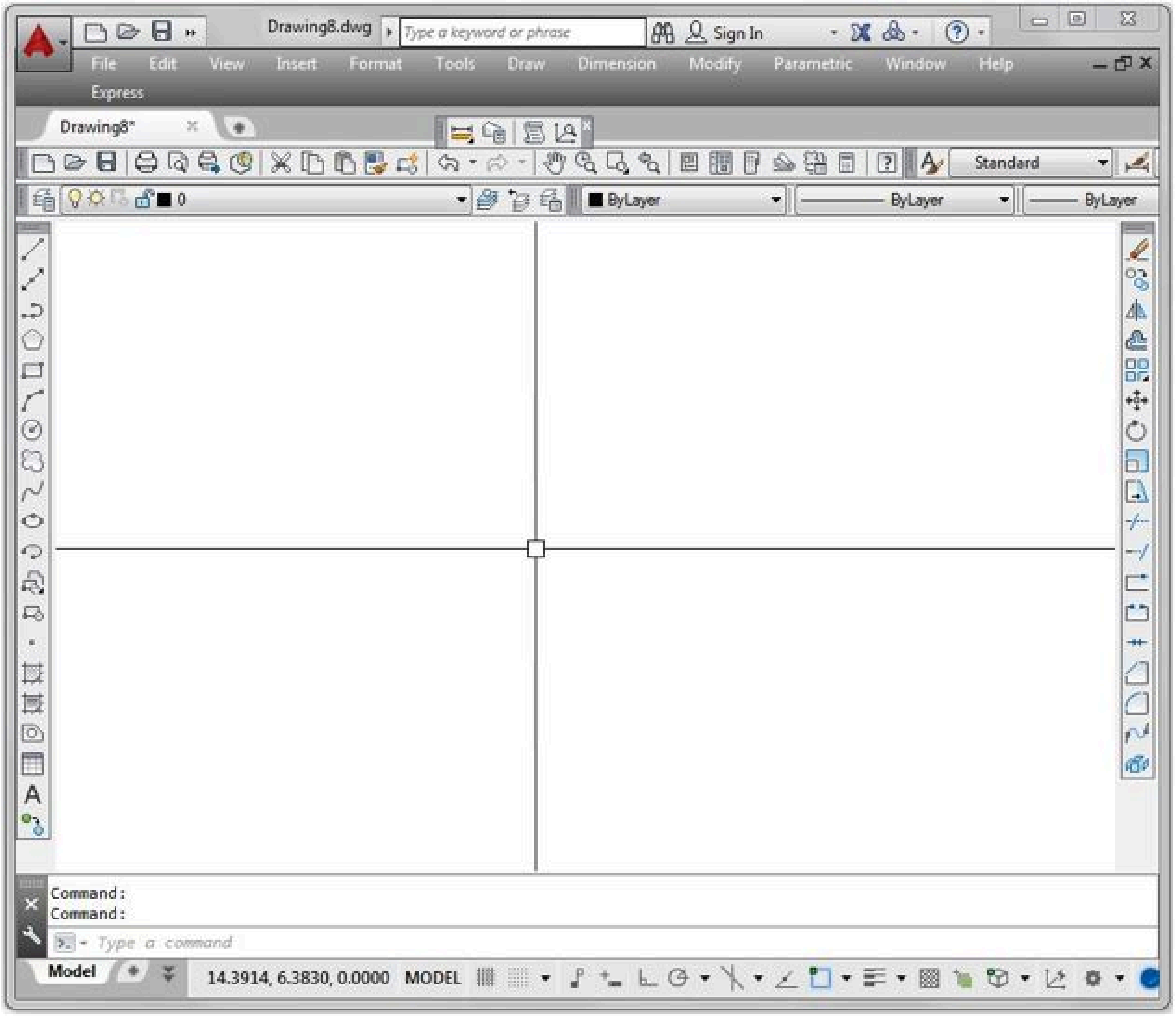

✓ AutoCADeBook
Drafting & Annotation
3D Basics
3D Modeling
Save Current As...
Workspace Settings...
Customize...
Display Workspace Label

Figure Step 4A


Drawing8.dwg
Type a keyword or phrase
Sign In
File
Edit
View
Insert
Format
Tools
Draw
Dimension
Modify
Parametric
Window
Help
Express
Drawing8*
Standard
0
ByLayer
ByLayer
ByLayer
Type a command
Model
14.3914, 6.3830, 0.0000
MODEL

```
Command:
Command:
```

Figure Step 4B

AUTHOR'S COMMENTS: Your Graphic window should closely match the figure.

Step 5

Check to ensure that the Model tab is enabled. It is located on the bottom left corner of the window. If it is not enabled, click it with the left mouse button. (Figure Step 5)

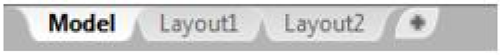

Model
Layout1
Layout2

Figure Step 5

AUTHOR'S COMMENTS: The Model tab should be enabled at all times, while working on workalongs and lab exercises in the AutoCAD 2D book, unless you are instructed otherwise. Module 18 will teach you how work in the Layout tabs.

Step 6

Disable all features on the Status bar by clicking any that display with a blue background as shown in Figure Step 6A. All features should display with a gray background as shown in Figure Step 6B. (Figure Step 6A and 6B)

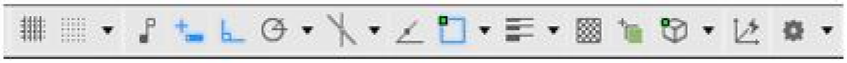


Figure Step 6A

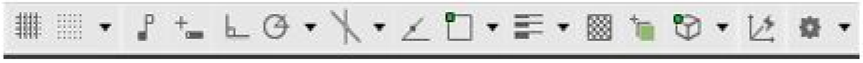
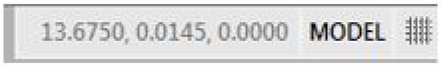


Figure Step 6B

AUTHOR'S COMMENTS: Important. Unless otherwise instructed, each time you start a workalong or lab exercise while working on the AutoCAD 2D book all features on the Status bar should be disabled as shown in Figure Step 6B.

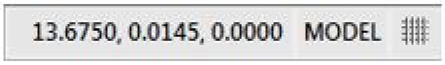
Step 7

Click the coordinate display, located on the left side of the Status bar, to disable it shown in Figure Step 7A. Click it again to enable it as shown in Figure Step 7B. (Figure Step 7A and 7B)



13.6750, 0.0145, 0.0000 MODEL

Figure Step 7A

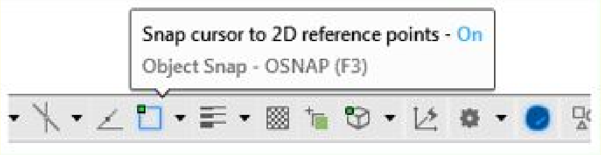


13.6750, 0.0145, 0.0000 MODEL

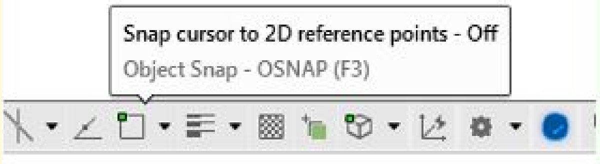
Figure Step 7B

AUTHOR'S COMMENTS: The coordinate display should be enabled while working in AutoCAD. While it is not used very much in day-to-day drawing, it can be very handy at times.

USER TIP: You can easily check to see if a feature on the Status bar is enabled or disabled by holding your graphic cursor on the icon. A pop up window will display indicating the name of the feature and whether it is currently on or off as shown in the figures.



Snap cursor to 2D reference points - On
Object Snap - OSNAP (F3)



Snap cursor to 2D reference points - Off
Object Snap - OSNAP (F3)

Step 8

Click Format on the Pull-down menu and then click Units to open the Drawing Units dialogue box. Click OK to close the dialogue box. (Figure Step 8A and 8B)

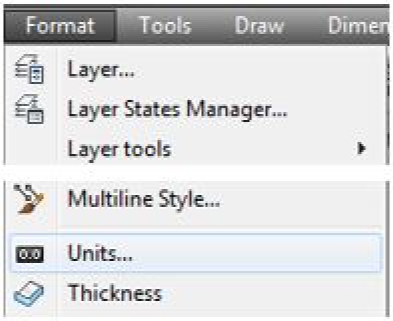
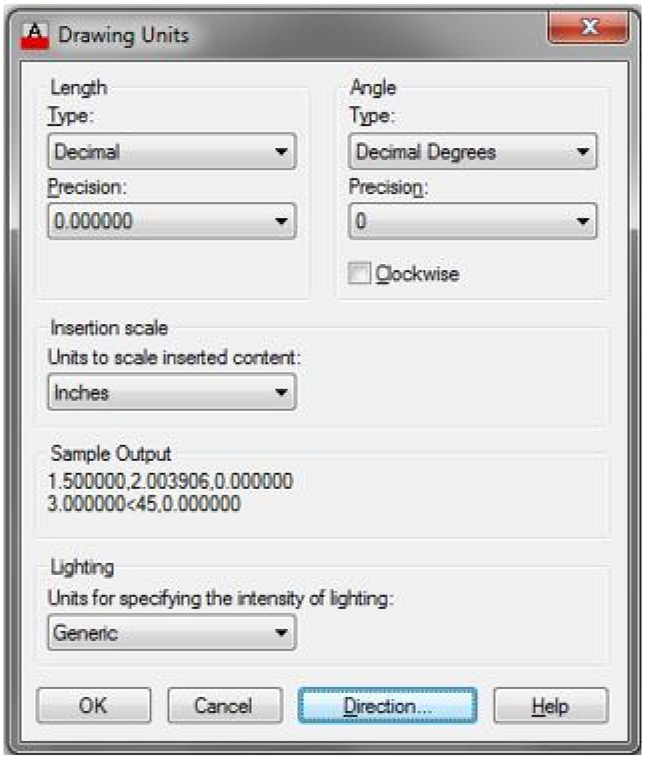

Format
Tools
Draw
Dimen
Layer...
Layer States Manager...
Layer tools
Multiline Style...
Units...
Thickness

Figure Step 8A



Drawing Units

Length
Type: Decimal
Precision: 0.000000

Angle
Type: Decimal Degrees
Precision: 0
☐ Clockwise

Insertion scale
Units to scale inserted content: Inches

Sample Output
1.500000,2.003906,0.000000
3.000000<45,0.000000

Lighting
Units for specifying the intensity of lighting: Generic

OK
Cancel
Direction...
Help

Figure Step 8B

AUTHOR'S COMMENTS: Always close a dialogue box using the OK or the Cancel button. If you close it by clicking the X in the top right corner, you are cancelling all changes that you made while the box was open. If you want to cancel the changes, click the Cancel button to close the dialogue box. When you click OK, you are closing it and saving all the changes you made.

Step 9

Click the View tab on the Ribbon menu. Find the Properties icon. Click it to open the Properties window. (Figure Step 9A and 9B)

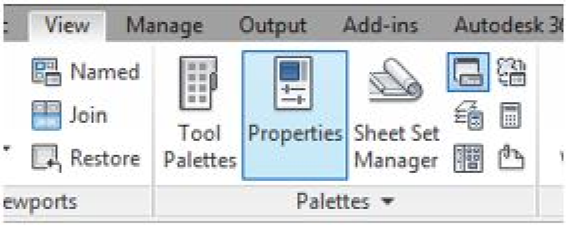
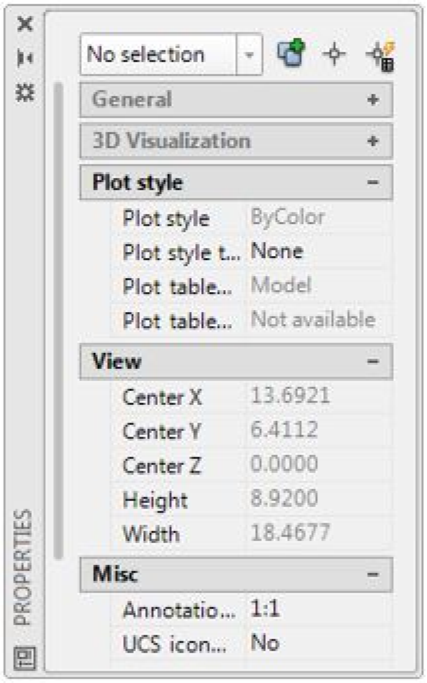

View
Manage
Output
Add-ins
Autodesk 3
Named
Join
Restore
Tool Palettes
Properties
Sheet Set Manager
ewports
Palettes

Figure Step 9A


PROPERTIES

No selection

General +

3D Visualization +

Plot style –

Plot style	ByColor
Plot style t...	None
Plot table...	Model
Plot table...	Not available

View –

Center X	13.6921
Center Y	6.4112
Center Z	0.0000
Height	8.9200
Width	18.4677

Misc –

Annotatio...	1:1
UCS icon...	No

Figure Step 9B

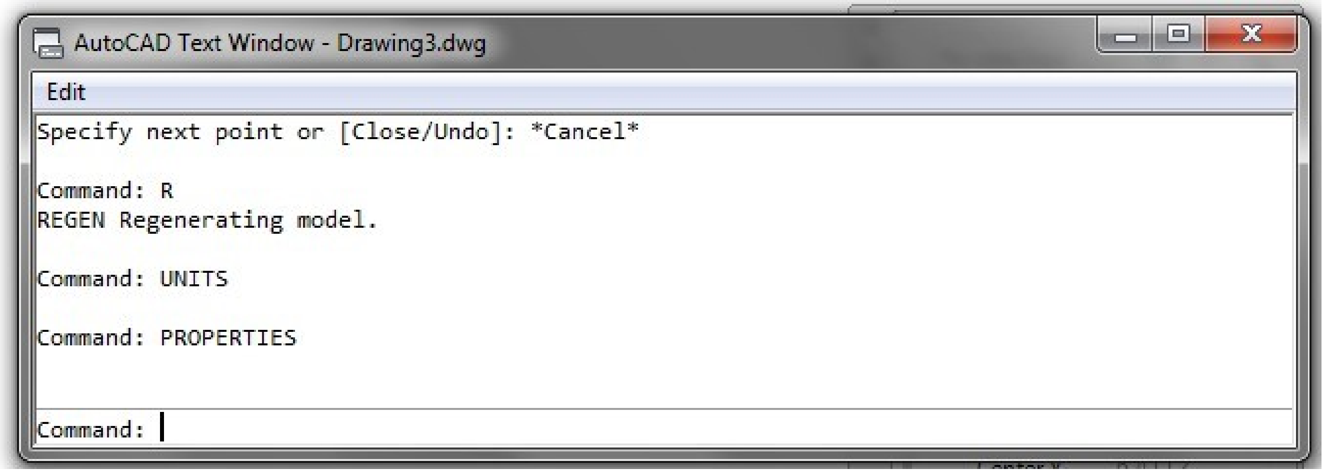
Step 10

Click the X in the top corner of the window to close it.

AUTHOR'S COMMENTS: Windows are different than dialogue box as they are interactive. Any changes made, while they are open, are saved immediately. That makes it OK to close them by clicking the X.

Step 11

Press the function key F2. It will open the Text window as shown in the figure. (Figure Step 11)


AutoCAD Text Window - Drawing3.dwg
Edit

```
Specify next point or [Close/Undo]: *Cancel*

Command: R
REGEN Regenerating model.

Command: UNITS

Command: PROPERTIES


Command:
```

Figure Step 11

Step 12

Press the function key F2 again to close the Text window.

Step 13

Click File on the Pull-down menu. Click Exit to close AutoCAD. If you are asked if you want to save the drawing, click No. (Figure Step 13)

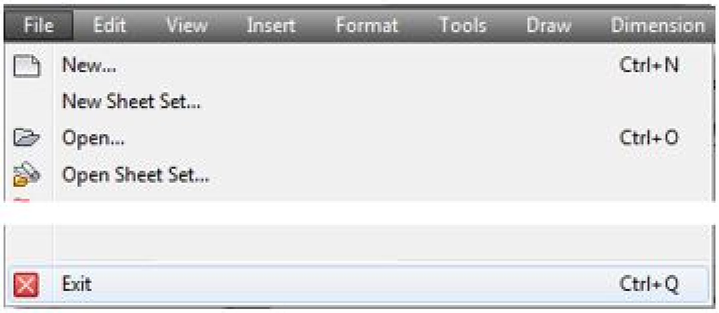

File
Edit
View
Insert
Format
Tools
Draw
Dimension
New... Ctrl+N
New Sheet Set...
Open... Ctrl+O
Open Sheet Set...
Exit Ctrl+Q

Figure Step 13

Step 14

Go to Module 2.

Key Principles

Key Principles in Module 1

1. The Graphic window is used most of the time when working in AutoCAD.
2. The crosshairs indicates your current location. Think of it as the point of the pencil. It is used extensively while working in AutoCAD. It is one of the most important tools.
3. The pickbox is a small square located at the intersection of the crosshairs.
4. The Status bar is positioned across the bottom of the Graphic window and becomes a very important part of your day-to-day AutoCAD work.
5. The Model and Layout tabs are used to enable the display of the model or a layout.
6. The Command Line window is used to enter keyboard commands and user inputs. It is also used by AutoCAD to output information to you.
7. The Pull-down menu Is located across the top of the Graphic window. If an item on the Pulldown menu has a small solid triangle at the end, it has a flyout menu associated with it.
8. The toolbar menus are a series of icons that are located on a collection of menu bars.
9. AutoCAD uses many different dialogue boxes to speed the drawing process. A dialogue box is an effective and efficient method for AutoCAD to allow you to modify or enter information, settings, and sizes.
10. AutoCAD uses many different windows to speed the drawing process. Compared to a dialogue box, a window is a more efficient method for AutoCAD and you to communicate.
11. AutoCAD is programed to use the three buttons on a mouse. The left button is the pick button.
12. AutoCAD’s Text window displays all of the commands that were entered by you and AutoCAD’s responses to those command for the duration that the current drawing is open. To toggle the display of the Text window, press F2.

Module 2: Getting Started

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe a drawing, a drawing template, drawing names, and drawing file extensions.
2. Apply the NEW, QSAVE, OPEN, SAVEAS, and CLOSE commands to start a new drawing, name and save the current drawing into a specified folder, open a filed drawing, save the current drawing with a different name, and close the current drawing.
3. Describe AutoCAD commands, option prompts, and defaults.
4. Describe how to execute, repeat, and abort commands.
5. Describe how the Space, Enter, and Esc keyboard keys are used in AutoCAD.

An AutoCAD Drawing

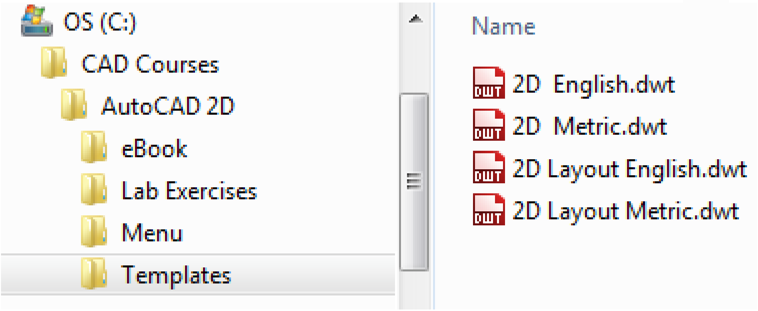
An AutoCAD *drawing* is a database file containing graphical and non-graphical data. The file must have the extension .dwg. To be able to draw anything in AutoCAD, you must have an open drawing. Multiple drawings can be opened at the same time and you can switch between them, as required. Only one drawing can be current at a time. The*current drawing* is the drawing currently being affected by the commands you are entering. It is also called the*active drawing*.

Drawing Templates

A*drawing template* is a drawing file that may contain layouts, defaults, blocks, and standards set by the creator of the template. A template file must be used to start every new drawing. A template drawing is simply an AutoCAD drawing saved as a template file and must have the extension .dwt. Creating template files is taught in the AutoCAD 2D Advanced book.

Drawing Templates for the AutoCAD 2D book

There are four drawing template files that are an important part of the AutoCAD 2D book. One of these templates must be used to start all new drawings for all workalongs and lab exercise throughout the book. These templates are supplied to each user of the AutoCAD 2D book and should be saved in the Templates folder as shown in Figure 2-1.



OS (C:)
CAD Courses
AutoCAD 2D
eBook
Lab Exercises
Menu
Templates

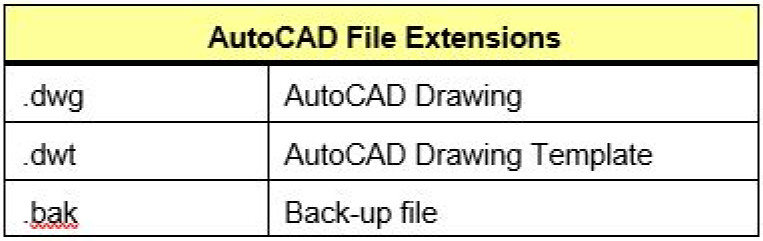
Name
2D English.dwt
2D Metric.dwt
2D Layout English.dwt
2D Layout Metric.dwt

Figure 2-1
AutoCAD 2D book Templates

AutoCAD File Extensions

AutoCAD uses many different file extension names. The file extensions used in the AutoCAD 2D book are .dwg, .dwt, and .bak. See Figure 2-2. Many other AutoCAD file extensions are taught in the AutoCAD 2D Advanced book.

AutoCAD creates a .bak file when a previously saved drawing is saved. When the current drawing is saved, AutoCAD changes the extension of the existing drawing from .dwg to .bak and then creates a new file with the extension .dwg. If a drawing file ever gets lost or corrupted, the extension name of the .bak file can be changed to .dwg and opened in AutoCAD.



AutoCAD File Extensions	
.dwg	AutoCAD Drawing
.dwt	AutoCAD Drawing Template
.bak	Back-up file

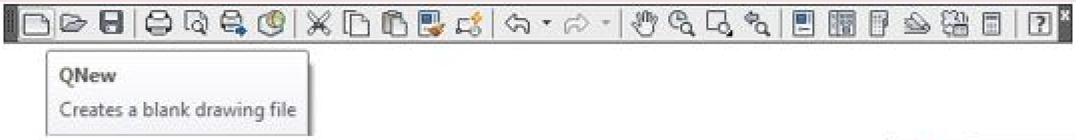
Figure 2-2 AutoCAD File Extension Names

The Drawing Name

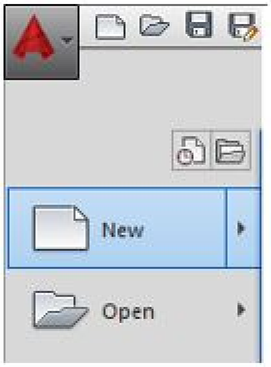
Each drawing should be assigned a unique name. It is important for you to name a new drawing as soon as possible after you start it. Getting into a habit of doing this will save you many hours of searching the disk drive looking for existing drawings or redrawing lost drawings.

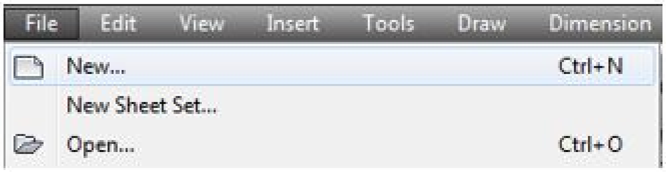
AutoCAD Command: NEW

The NEW command is used to start a new drawing. Shortcut: **CNTL+N**


QNew
Creates a blank drawing file

Standard Toolbar



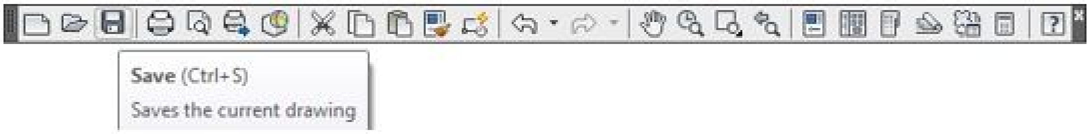

File
Edit
View
Insert
Tools
Draw
Dimension
New... Ctrl+N
New Sheet Set...
Open... Ctrl+O

File Pull-down

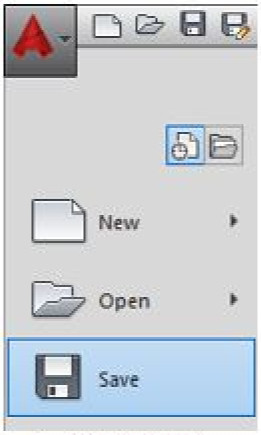
AutoCAD Pull-down

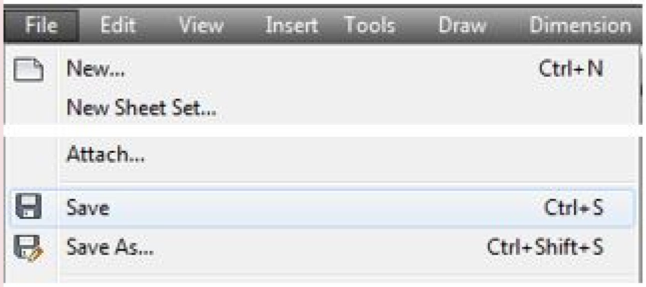
AutoCAD Command: QSAVE

The QSAVE command is used to save the current drawing. Shortcut: **CTRL+S**


Save (Ctrl+S)
Saves the current drawing

Standard Toolbar


New
Open
Save

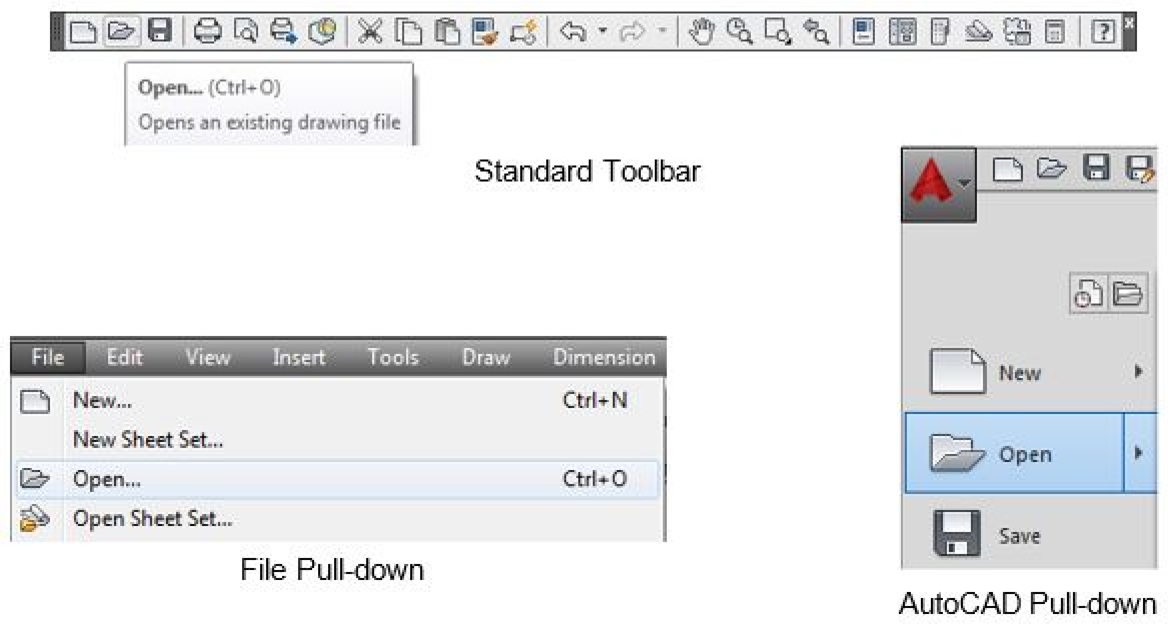

File
Edit
View
Insert
Tools
Draw
Dimension
New... Ctrl+N
New Sheet Set...
Attach...
Save Ctrl+S
Save As... Ctrl+Shift+S

File Pull-down

AutoCAD Pull-down

AutoCAD Command: OPEN

The OPEN command is used to open a saved drawing. Shortcut: **CTRL+O**


Open... (Ctrl+O)
Opens an existing drawing file
File
Edit
View
Insert
Tools
Draw
Dimension
New... Ctrl+N
New Sheet Set...
Open... Ctrl+O
Open Sheet Set...
New
Open
Save

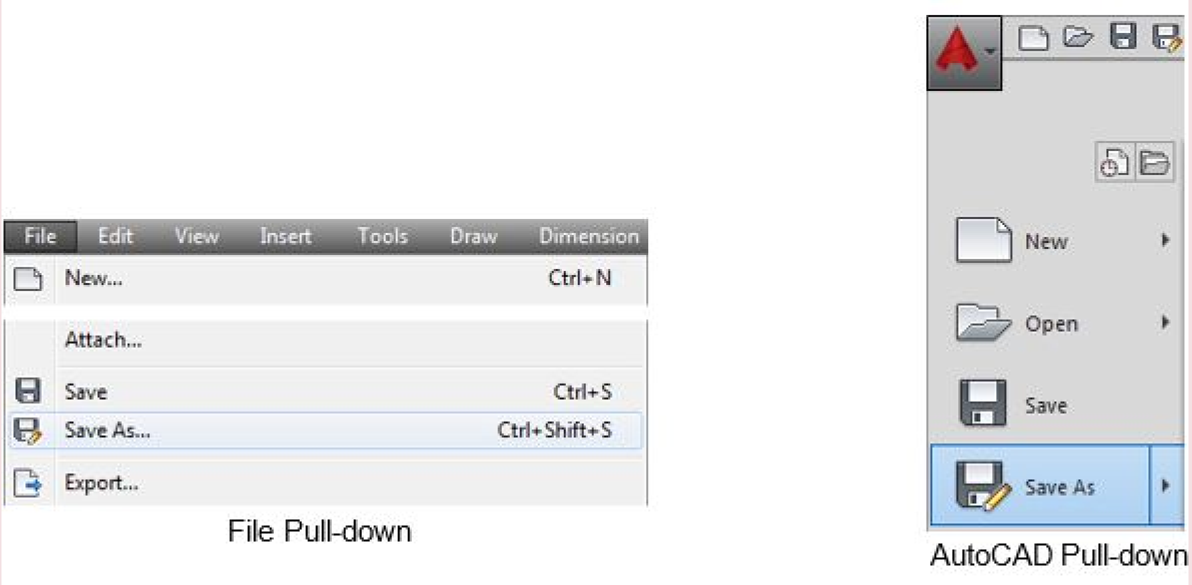
Standard Toolbar

File Pull-down

AutoCAD Pull-down

AutoCAD Command: SAVEAS

The SAVEAS command is used to save the current drawing with a new name. Shortcut: **CTRL+SHIFT+S**

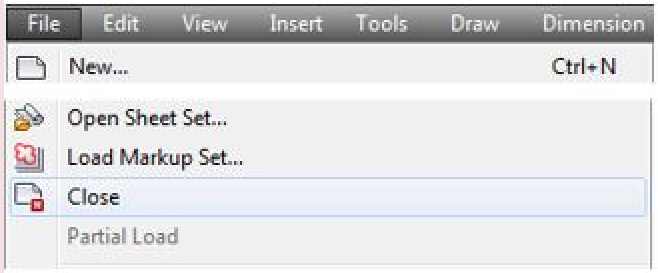

File
Edit
View
Insert
Tools
Draw
Dimension
New... Ctrl+N
Attach...
Save Ctrl+S
Save As... Ctrl+Shift+S
Export...
New
Open
Save
Save As

File Pull-down

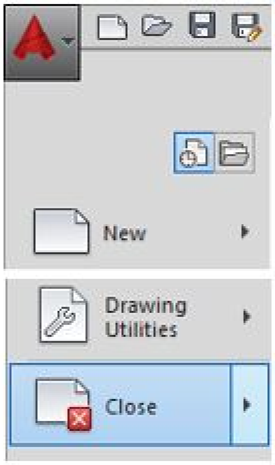
AutoCAD Pull-down

AutoCAD Command: CLOSE

The CLOSE command is used to close the current drawing. Shortcut: none


File
Edit
View
Insert
Tools
Draw
Dimension
New...
Ctrl+N
Open Sheet Set...
Load Markup Set...
Close
Partial Load

File Pull-down


New
Drawing Utilities
Close

AutoCAD Pull-down

MUST KNOW: When you save a drawing for the first time, AutoCAD adds the extension .dwg to the drawing's name. When you save a drawing that was previously saved, AutoCAD changes the extension of the existing drawing file to .bak and creates a new drawing file with the extension .dwg. If a drawing file ever gets lost or corrupted, the extension name of the .bak file can be changed to .dwg and opened in AutoCAD.

WORKALONG: Creating, Naming, Saving, Opening, and Closing Drawings

Step 1

Launch AutoCAD.

Step 2

Click the New icon. This will open the Select template dialogue box. Select the template: 2D English and then click Open (Figure Step 2)

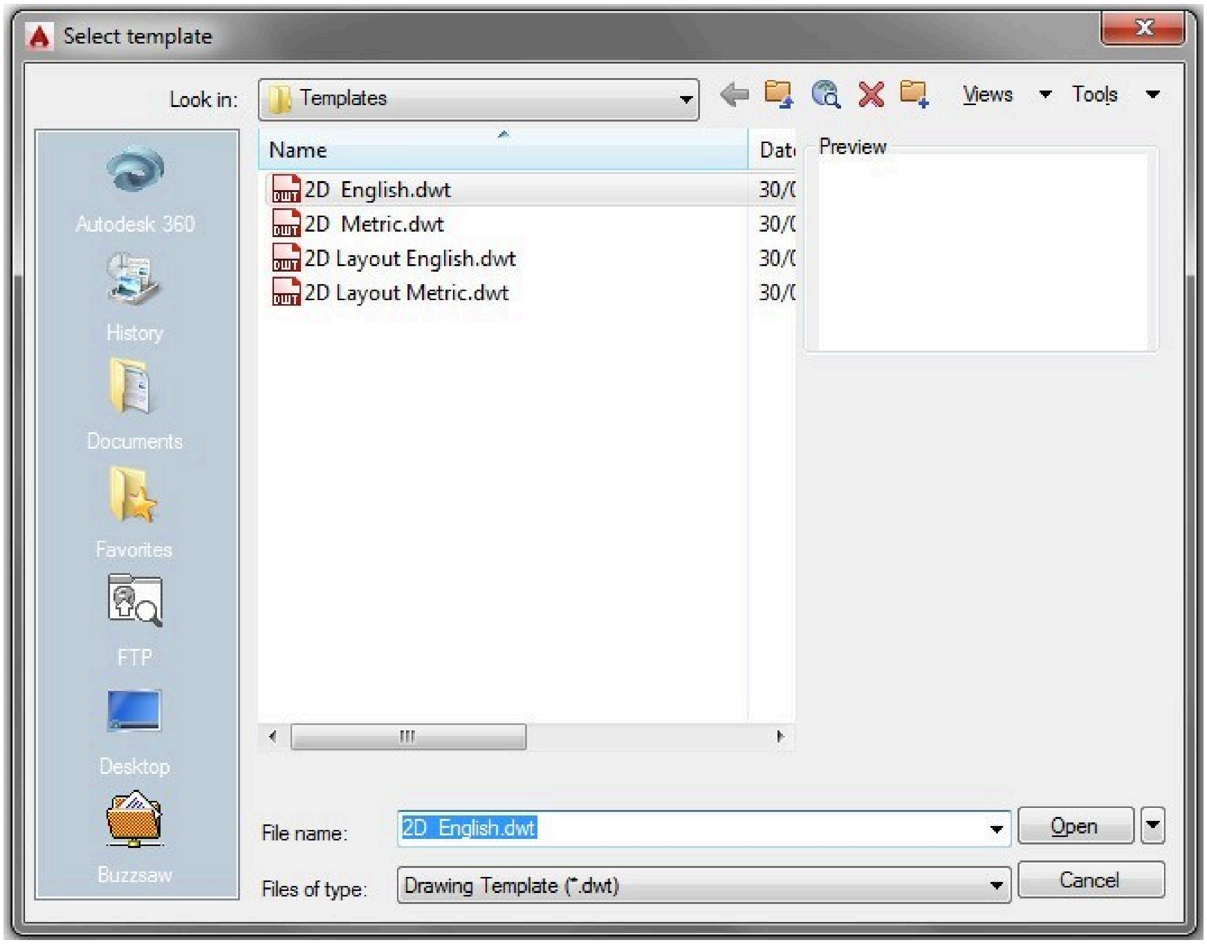

Select template
Look in:
Templates
Views
Tools
Name
Dat
Preview
2D English.dwt
2D Metric.dwt
2D Layout English.dwt
2D Layout Metric.dwt
30/(
30/(
30/(
30/(
Autodesk 360
History
Documents
Favorites
FTP
Desktop
Buzzsaw
File name:
2D English.dwt
Files of type:
Drawing Template (*.dwt)
Open
Cancel

Figure Step 2

AUTHOR'S COMMENTS: If you do NOT see the four templates files shown in Figure Step 2, you must go back and redo module Introduction 2. If you only see the name and not the .dwt extension, that is OK. Do not continue in this module until your dialogue box shows the four template shown in Figure Step 2 with or without the extension. If you have trouble getting it to match and have an instructor, email him/her for help.

Step 3

Note the drawing name: Drawing1.dwg displays on the top bar of the Graphic window. (Figure Step 3)

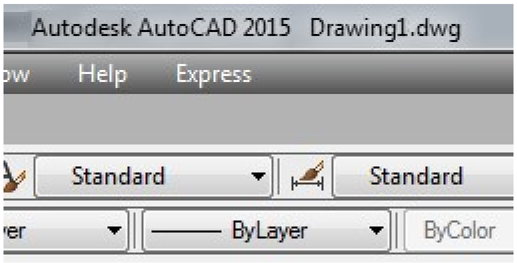

Autodesk AutoCAD 2015 Drawing1.dwg
ow
Help
Express
Standard
Standard
er
ByLayer
ByColor

Figure Step 3

AUTHOR'S COMMENTS: AutoCAD automatically assigns this name.

Step 4

Enter the QSAVE command to open the Save Drawing As dialogue box. Find and select the folder: CAD Courses. Under it, select the folder: AutoCAD 2D and under that, the folder: Lab Exercise. (Figure Step 4A, 4B, and 4C)

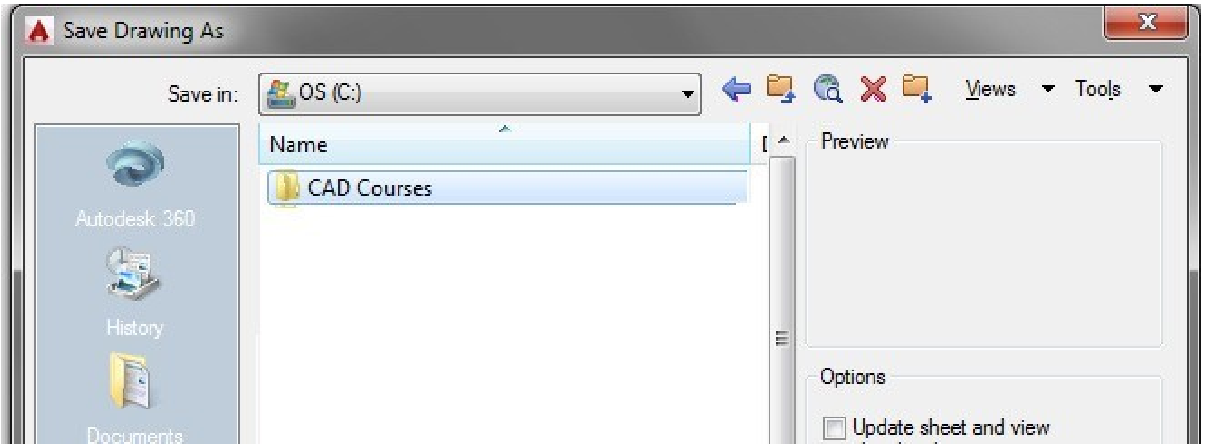
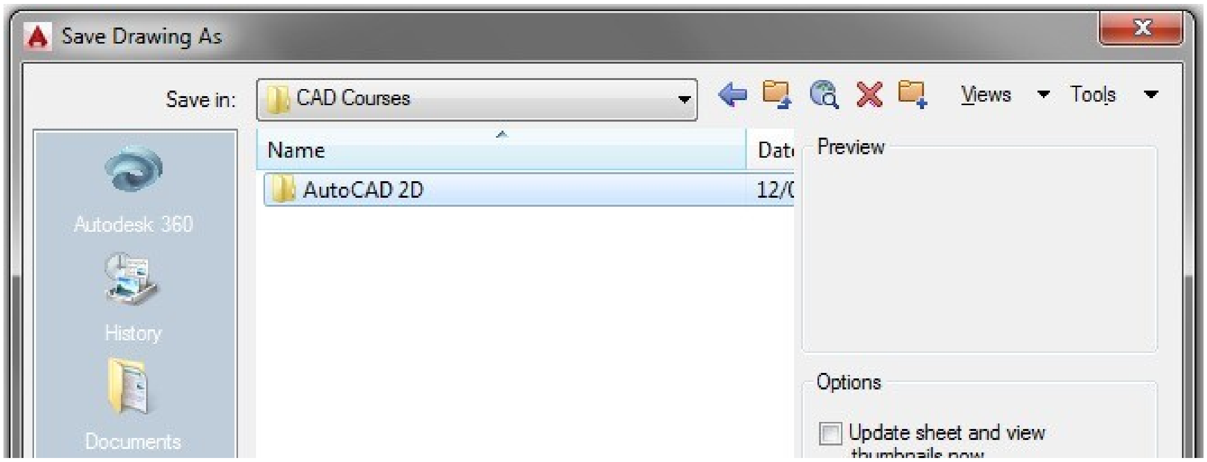
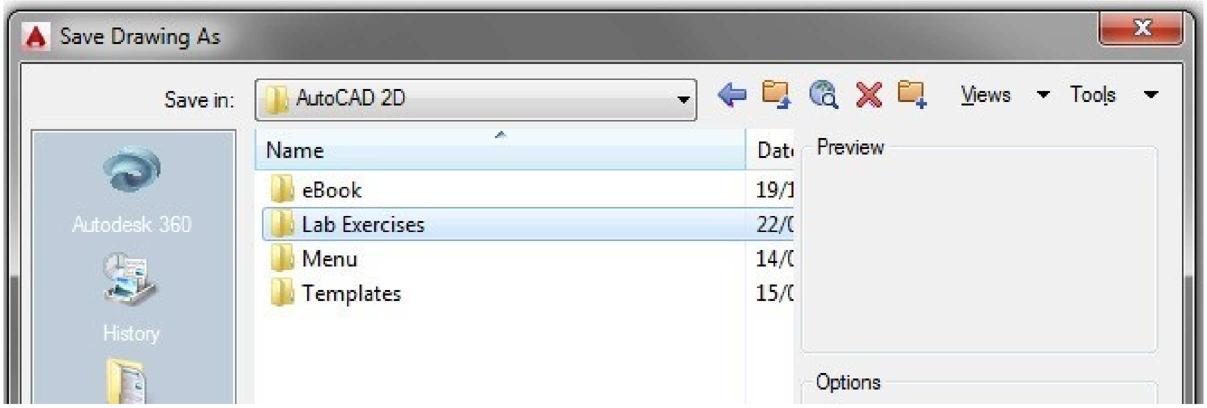

Save Drawing As
Save in:
OS (C:)
Views
Tools
Autodesk 360
History
Documents
Name
CAD Courses
Preview
Options
Update sheet and view

Figure Step 4A


Save in: CAD Courses

Save Drawing As
Views
Tools
Name
Dat
AutoCAD 2D
12/(
Preview
Autodesk 360
History
Documents
Options
☐ Update sheet and view thumbnails now

Figure Step 4B

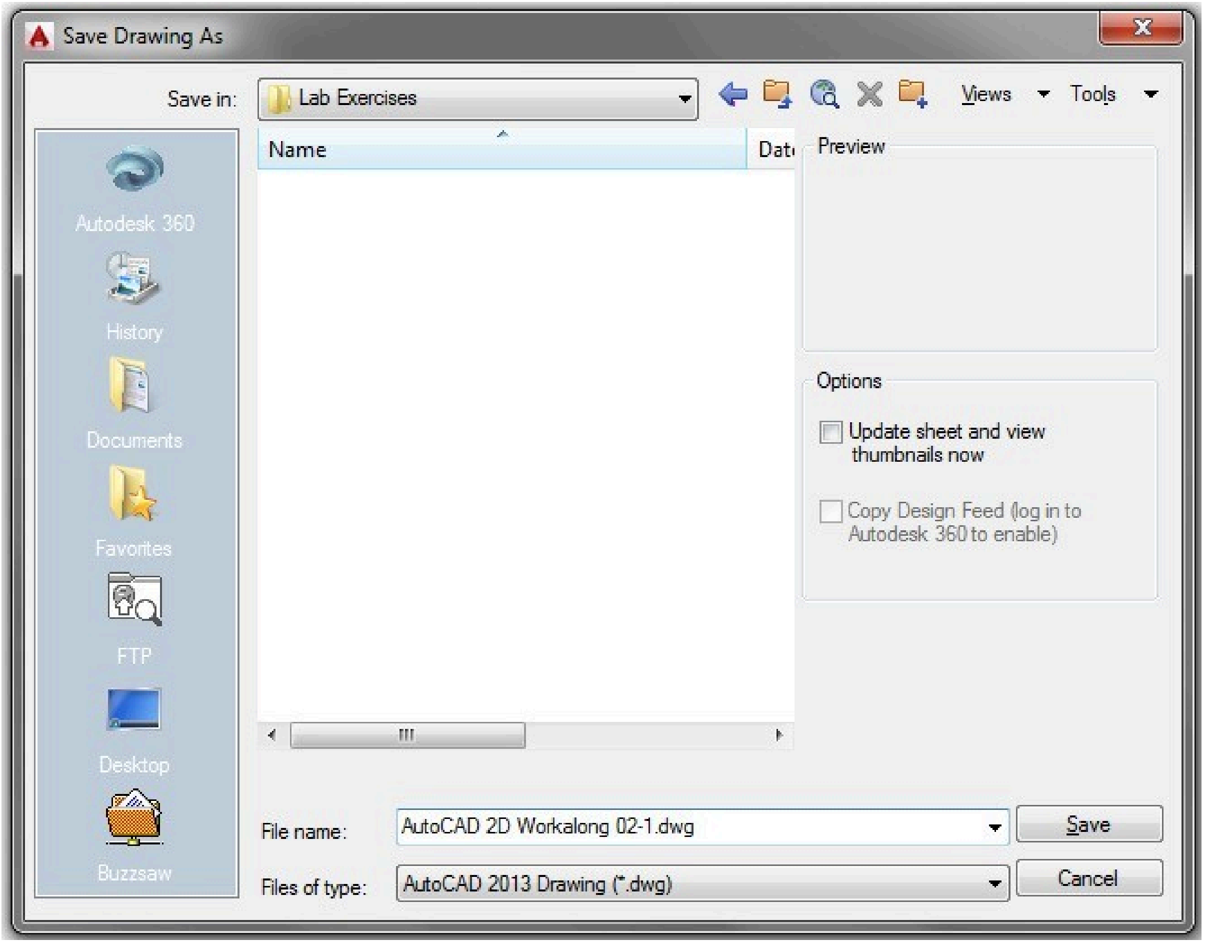

Save in: AutoCAD 2D

Save Drawing As
Views
Tools
Name
Dat
eBook
19/1
Lab Exercises
22/(
Menu
14/(
Templates
15/(
Preview
Autodesk 360
History
Options

Figure Step 4C

Step 5

With Lab Exercises in the Save in box, enter the name: AutoCAD 2D Workalong 02-1 in the File name: box. (Figure Step 5)


Save Drawing As
Views
Tools
Name
Dat
Preview
Options
Autodesk 360
History
Documents
Favorites
FTP
Desktop
Buzzsaw
Save
Cancel

Save in: Lab Exercises

☐ Update sheet and view thumbnails now

☐ Copy Design Feed (log in to Autodesk 360 to enable)

File name: AutoCAD 2D Workalong 02-1.dwg

Files of type: AutoCAD 2013 Drawing (*.dwg)

Figure Step 5

AUTHOR'S COMMENTS: I recommend that you save and name a new drawing immediately after you start it and then save it every 15 minutes after that. If there is a power failure or your computer freezes, the maximum amount of drawing time you can lose is 15 minutes. The first time you save your drawing, you will be asked to assign a name. Once it is named, you can save it quickly with the QSAVE command.

Step 6

Note that the folder location and the drawing name:

C:\CAD Courses\AutoCAD 2D\Lab Exercises\AutoCAD 2D Workalong 02-1.dwg displays on the top bar of the Graphic window. (Figure Step 6)

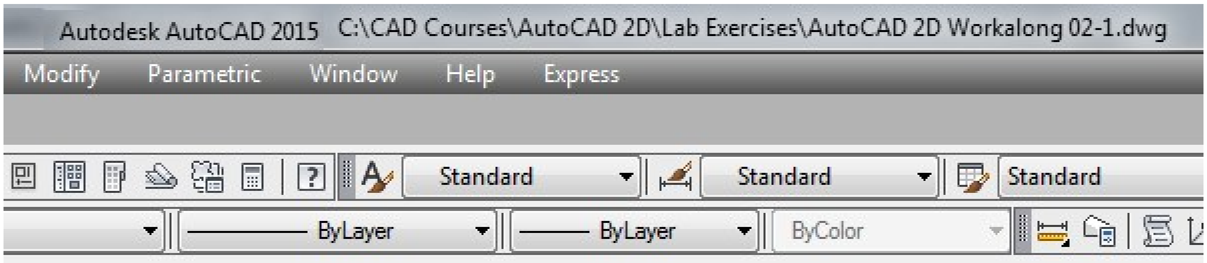

Autodesk AutoCAD 2015 C:\CAD Courses\AutoCAD 2D\Lab Exercises\AutoCAD 2D Workalong 02-1.dwg
Modify
Parametric
Window
Help
Express
Standard
Standard
Standard
ByLayer
ByLayer
ByColor

Figure Step 6

Step 7

Check to ensure that the Model tab is enabled. If not, enable it by clicking it with the left mouse button. (Figure Step 7)

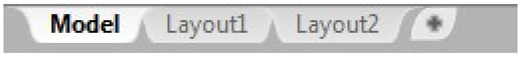

Model
Layout1
Layout2

Figure Step 7

AUTHOR'S COMMENTS: Until you are instructed otherwise, the Model tab should be enabled at all times while working on workalongs and lab exercises in the AutoCAD 2D book. Module 18 will teach you how to work in the layouts.

Step 8

Disable all features on the Status bar by clicking any that are enabled. Your Status bar should match the figure. (Figure Step 8)

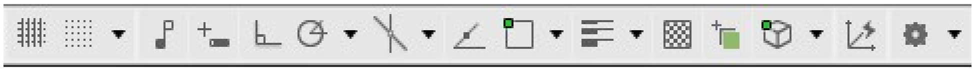
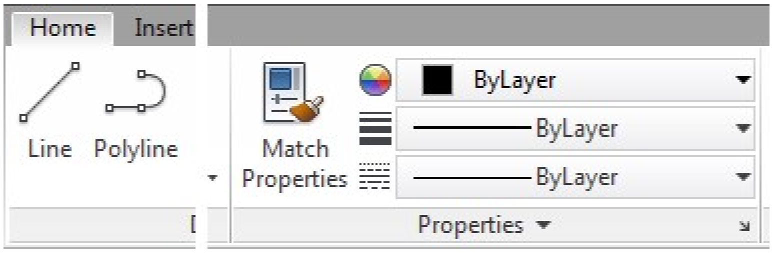


Figure Step 8

Step 9

Ensure that ByLayer is enabled in the first 3 boxes on the Properties toolbar or the Properties tab in the Ribbon menu. (Figure Step 9)


Home
Insert
Line
Polyline
Match Properties
ByLayer
ByLayer
ByLayer
Properties

Ribbon Menu

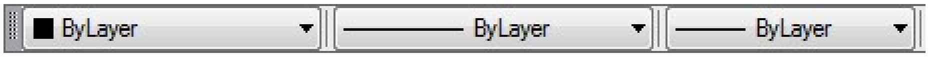

ByLayer
ByLayer
ByLayer

Figure Step 9 Toolbar Menu

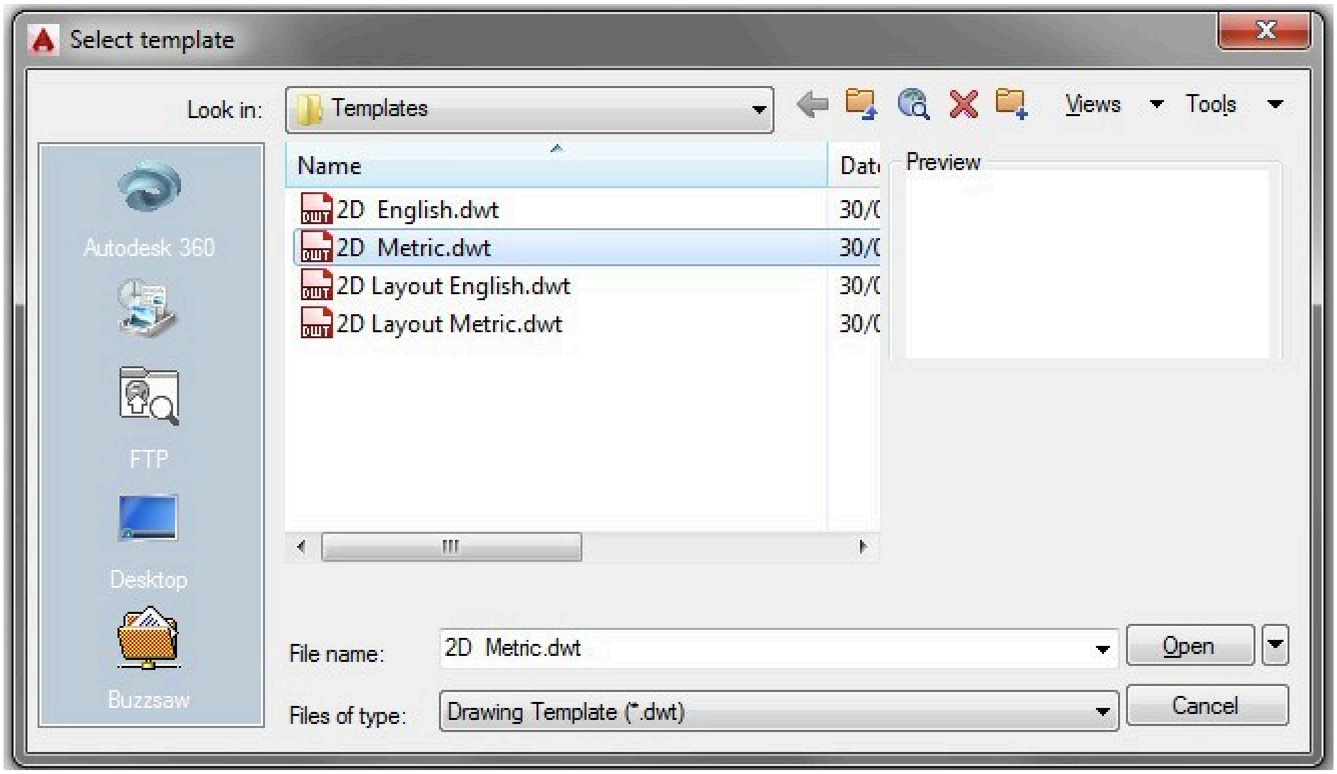
AUTHOR'S COMMENTS: **Important** Unless otherwise instructed, each time you start a workalong or a lab exercise while working on the AutoCAD 2D book, the Model tab should be enabled, all features on the Status bar should be disabled, and ByLayer should be enabled in the first three boxes on the Properties toolbar or Ribbon menu.

Step 10

Enter the CLOSE command to close the current drawing. If you are asked if you want to save the drawing, answer Yes.

Step 11

Using what you learned in the first part of the workalong, start a new drawing using the template: 2D Metric. (Figure Step 11)



Look in: Templates

File name: 2D Metric.dwt

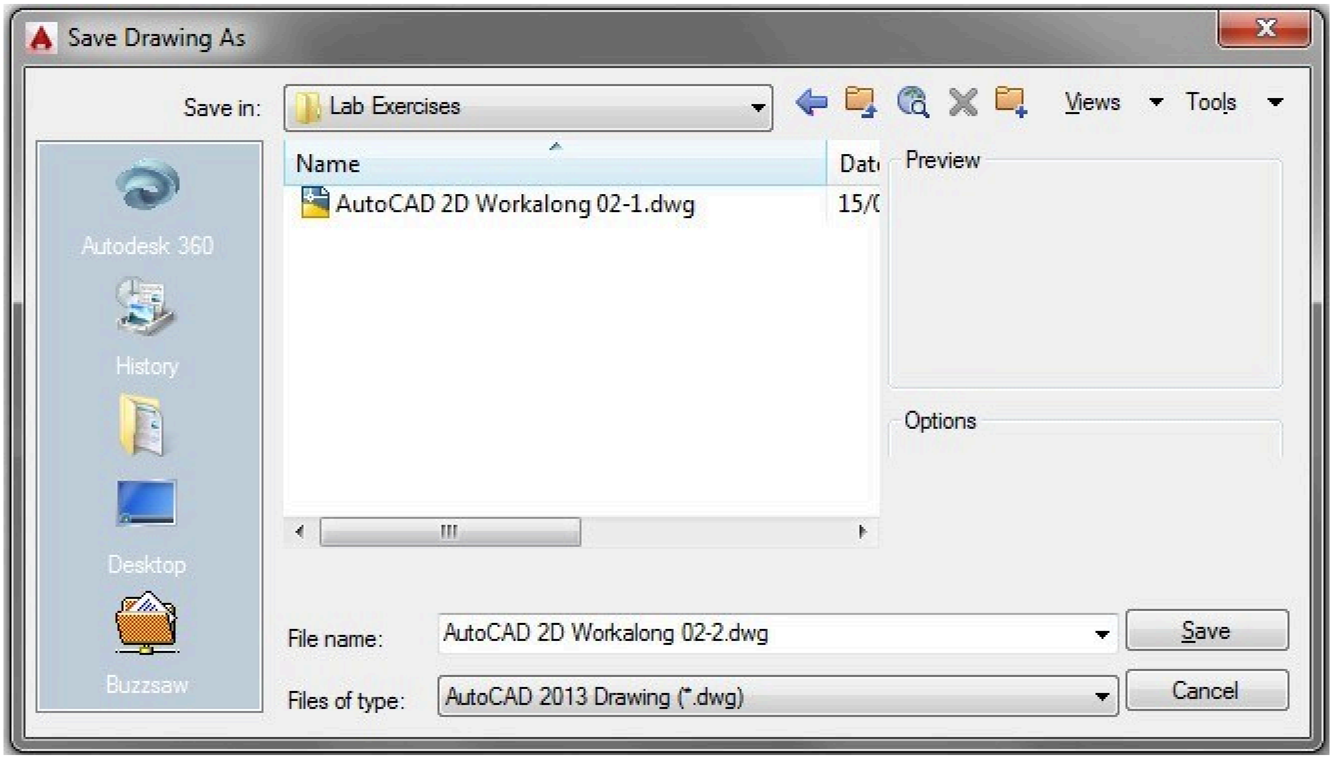
Files of type: Drawing Template (*.dwt)

Select template
Views
Tools
Name
Date
Preview
2D English.dwt
2D Metric.dwt
2D Layout English.dwt
2D Layout Metric.dwt
Autodesk 360
FTP
Desktop
Buzzsaw
Open
Cancel

Figure Step 11

Step 12

Save and name the drawing: AutoCAD 2D Workalong 02-2 in the folder: C:\CAD Courses\AutoCAD 2D\Lab Exercises. (Figure Step 12)


Save Drawing As
Save in:
Lab Exercises
Views
Tools
Name
Dat
AutoCAD 2D Workalong 02-1.dwg
15/(
Preview
Options
Autodesk 360
History
Desktop
Buzzsaw
Save
Cancel

File name: AutoCAD 2D Workalong 02-2.dwg

Files of type: AutoCAD 2013 Drawing (*.dwg)

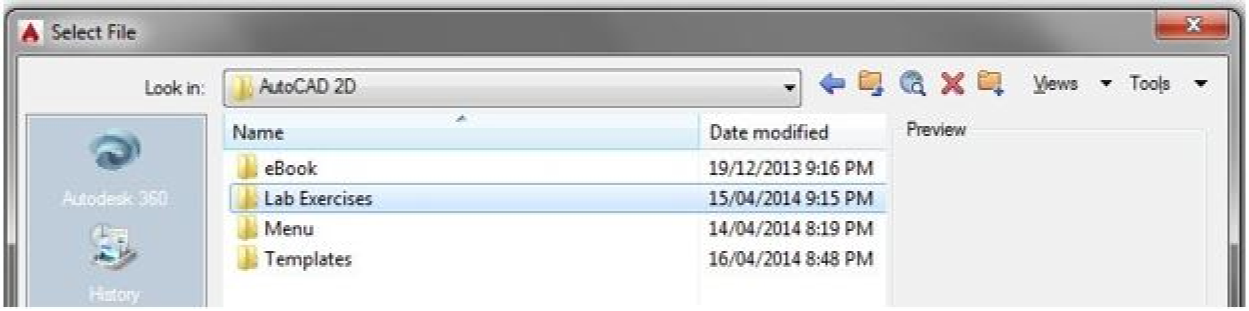
Figure Step 12

Step 13

Enter the CLOSE command to close the current drawing. If you are asked if you want to save the drawing, answer Yes.

Step 14

Enter the OPEN command to open the Select File dialogue box. In it, locate the folder: C:\CAD Courses\AutoCAD 2D\Lab Exercises. (Figure Step 14)


Select File
Views
Tools
Preview
Autodesk 360
History

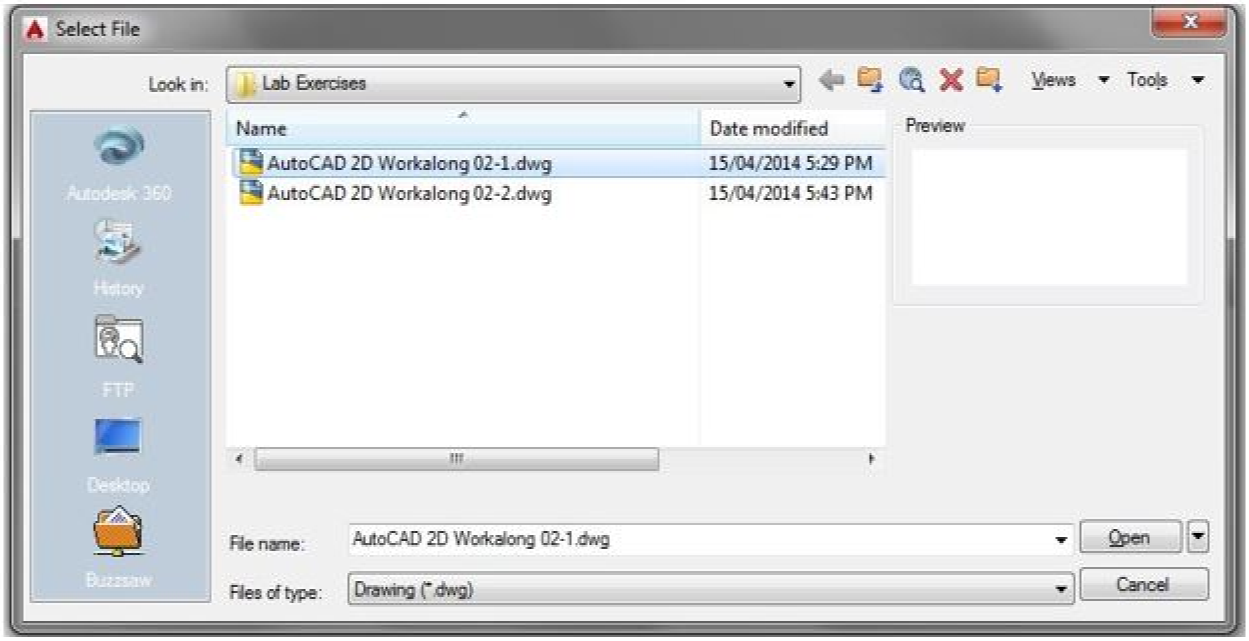
Look in: AutoCAD 2D

Name	Date modified
eBook	19/12/2013 9:16 PM
Lab Exercises	15/04/2014 9:15 PM
Menu	14/04/2014 8:19 PM
Templates	16/04/2014 8:48 PM

Figure Step 14

Step 15

With Lab Exercise in the Look in box, select the drawing: AutoCAD 2D Workalong 02-1. (Figure Step 15)


Select File
Views
Tools
Preview
Autodesk 360
History
FTP
Desktop
Buzzsaw
Open
Cancel

Look in: Lab Exercises

Name	Date modified
AutoCAD 2D Workalong 02-1.dwg	15/04/2014 5:29 PM
AutoCAD 2D Workalong 02-2.dwg	15/04/2014 5:43 PM

File name: AutoCAD 2D Workalong 02-1.dwg

Files of type: Drawing (*.dwg)

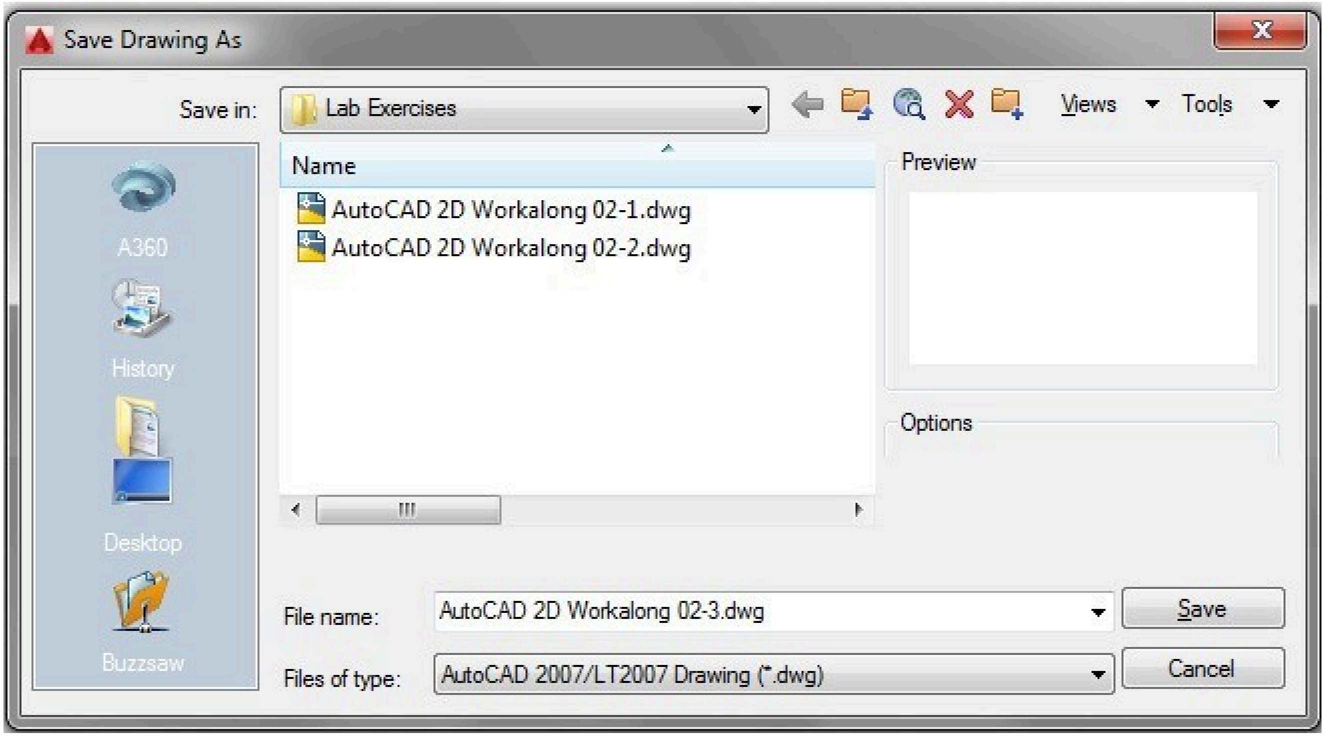
Figure Step 15

USER TIP: If the Select File dialogue box does not open when you enter the NEW command, enter the FILEDIA command and set the value to 1 as shown below:

```
Command: FILEDIA
Enter new value for FILEDIA <0>: 1
Command:
```

Step 16

Enter the SAVEAS command. It will open the Save Drawing As dialogue box. In the File name: box, enter the name: AutoCAD 2D Workalong 02-3. (Figure Step 16)



Save in: Lab Exercises

Name
AutoCAD 2D Workalong 02-1.dwg
AutoCAD 2D Workalong 02-2.dwg

File name: AutoCAD 2D Workalong 02-3.dwg

Files of type: AutoCAD 2007/LT2007 Drawing (*.dwg)

Save Drawing As
Views
Tools
A360
History
Desktop
Buzzsaw
Preview
Options
Save
Cancel

Figure Step 16

Step 17

Enter the CLOSE command to close the current drawing. If you are asked if you want to save the drawing, answer Yes.

Step 18

Minimize AutoCAD and open File Explorer. Find the folder:

C:\CAD Courses\AutoCAD 2D\Lab Exercises. You should see three drawing files either with or without the .dwg extension as shown in the figures. (Figure Step 18A and 18B)

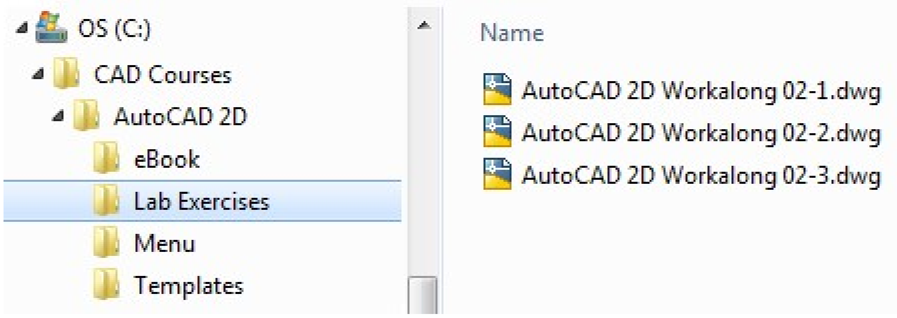

OS (C:)
CAD Courses
AutoCAD 2D
eBook
Lab Exercises
Menu
Templates
Name
AutoCAD 2D Workalong 02-1.dwg
AutoCAD 2D Workalong 02-2.dwg
AutoCAD 2D Workalong 02-3.dwg

Figure Step 18A

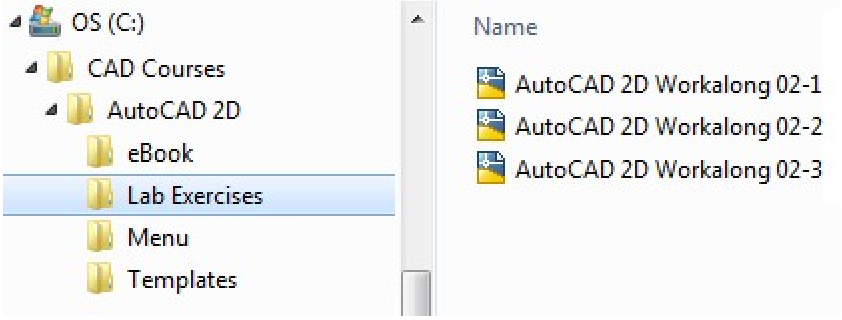

OS (C:)
CAD Courses
AutoCAD 2D
eBook
Lab Exercises
Menu
Templates
Name
AutoCAD 2D Workalong 02-1
AutoCAD 2D Workalong 02-2
AutoCAD 2D Workalong 02-3

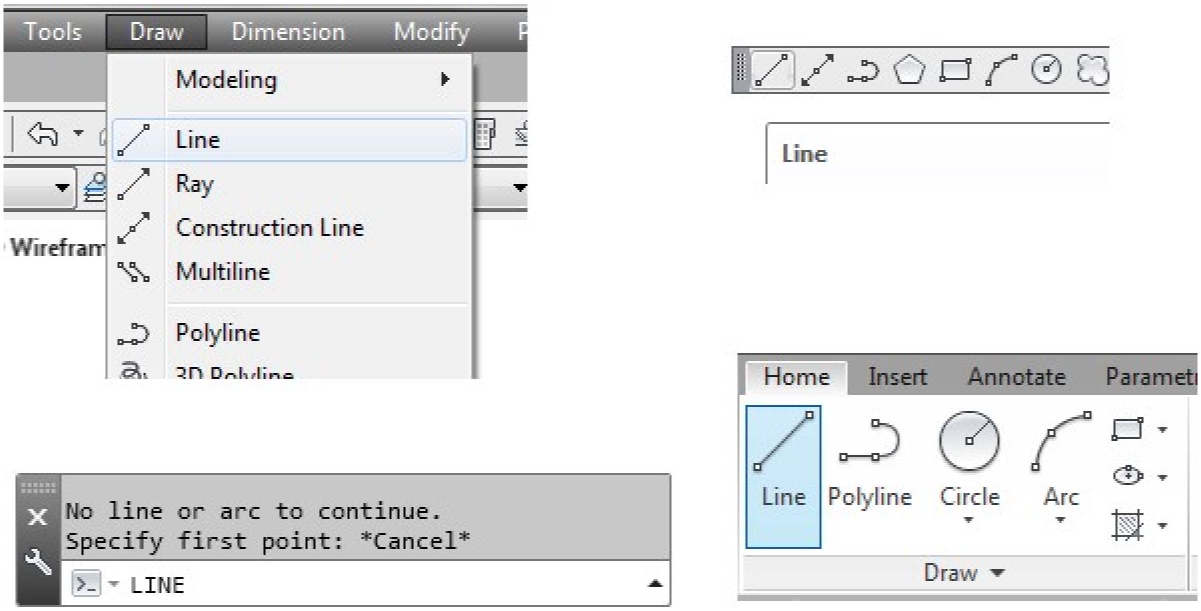
Figure Step 18B

AUTHOR'S COMMENTS: You may see some .bak files in your list with the same name as the drawing files. Note that drawing files have the AutoCAD icon preceding the drawing name. Figure Step 18A and 18B only shows the drawing files. If you see some .bak files, ignore them.

AutoCAD Commands

Most work in AutoCAD consists of you issuing instructions to AutoCAD. Those instructions are called commands. Commands are usually entered using the Command Line window, a Pull-down menu, a Toolbar menu, or a Ribbon menu. Figure 2-3 shows the LINE command being entered using all four methods.

Regardless of the way you enter a command, AutoCAD does not know or care where the command was initiated. To AutoCAD, all commands look alike, therefore you can choose which method you want to use. There is no right or wrong method to enter a command. Throughout the AutoCAD 2D book, the different methods of entering commands is shown and the you can decided which method to use.


Tools
Draw
Dimension
Modify
Modeling
Line
Ray
Construction Line
Multiline
Polyline
3D Polyline
Wirefram
Line
Home
Insert
Annotate
Parametr
Line
Polyline
Circle
Arc
Draw

```
No line or arc to continue.
Specify first point: *Cancel*
LINE
```

Figure 2-3 Different Methods of Executing the LINE Command

Executing Commands

Some menu items execute commands automatically but most of the time it is up to you to execute the command. *Executing a command* is your way of telling AutoCAD that you are finished and AutoCAD should do what was commanded. A command can be executed by either pressing the Enter key or the Space Bar.

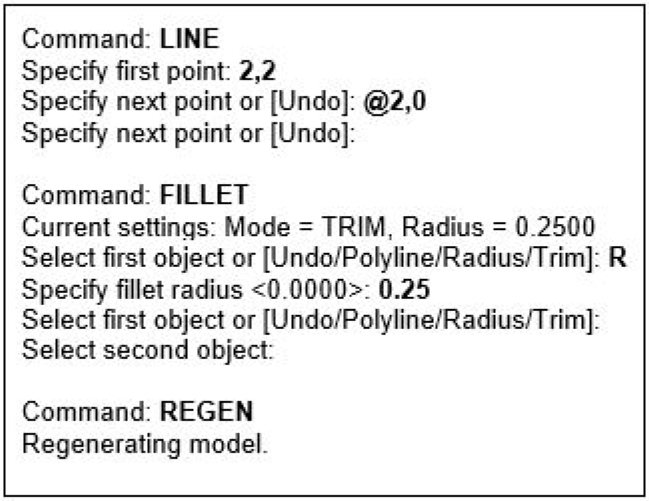
Repeating the Last Command

AutoCAD can be very repetitive. To work faster, the last command that was executed can be repeated by pressing the Enter key, the Space bar, or by right clicking the mouse.

Option Prompts

After most commands are executed, AutoCAD will respond with an option prompt. An *option prompt* is AutoCAD asking you a question in response to a command or requesting some user input. Figure 2-4 shows three commands and the option prompts returned by each one. Your input is shown in **bold.** In the LINE command, AutoCAD's option prompt is

‘ Specify first point ‘ and you respond 2,2. AutoCAD then prompts ‘ Specify next point or [Undo]: ‘. It is important for you to watch the Command Line window to see what AutoCAD is asking from you next.

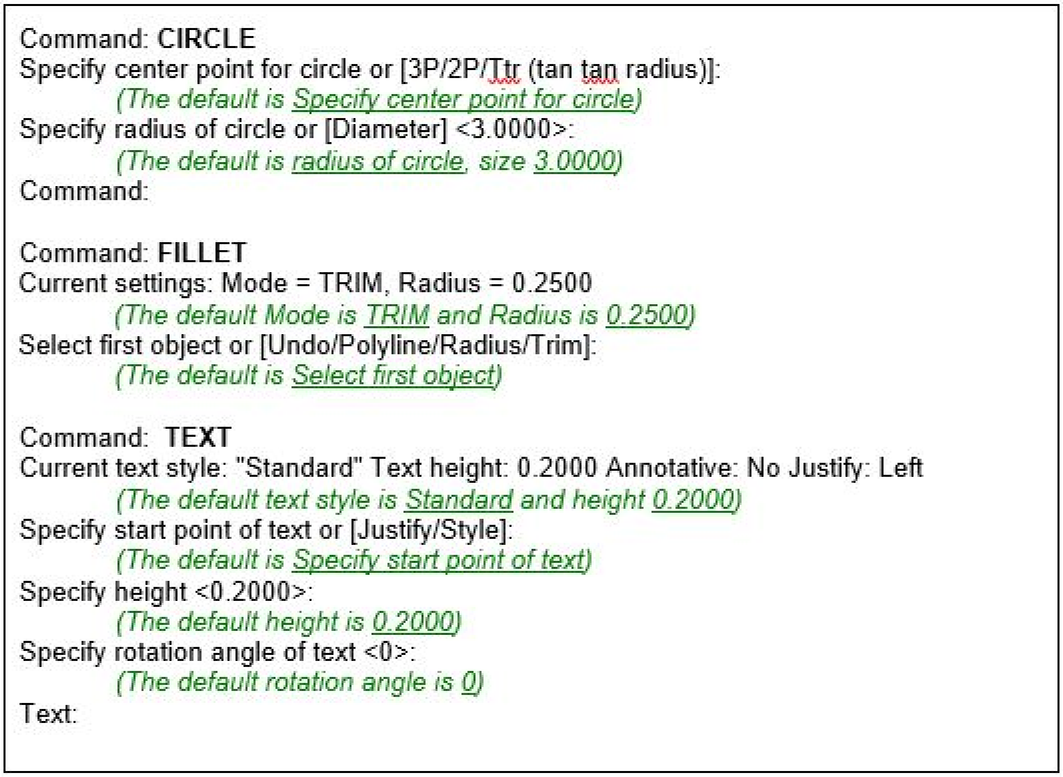


```
Command: LINE
Specify first point: 2,2
Specify next point or [Undo]: @2,0
Specify next point or [Undo]:

Command: FILLET
Current settings: Mode = TRIM, Radius = 0.2500
Select first object or [Undo/Polyline/Radius/Trim]: R
Specify fillet radius <0.0000>: 0.25
Select first object or [Undo/Polyline/Radius/Trim]:
Select second object:

Command: REGEN
Regenerating model.
```

Figure 2-4 Option Prompts

Defaults

AutoCAD sets defaults for almost everything. A *default* is the value AutoCAD uses, if you do not supply one. Defaults are enclosed inside angle brackets ‘ < > ‘. If you want to use the default value, simply respond by pressing the Enter key or the Space bar. See Figure 2-5.



```
Command: CIRCLE
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
          (The default is Specify center point for circle)
Specify radius of circle or [Diameter] <3.0000>:
          (The default is radius of circle, size 3.0000)
Command:

Command: FILLET
Current settings: Mode = TRIM, Radius = 0.2500
          (The default Mode is TRIM and Radius is 0.2500)
Select first object or [Undo/Polyline/Radius/Trim]:
          (The default is Select first object)

Command:  TEXT
Current text style: "Standard" Text height: 0.2000 Annotative: No Justify: Left
          (The default text style is Standard and height 0.2000)
Specify start point of text or [Justify/Style]:
          (The default is Specify start point of text)
Specify height <0.2000>:
          (The default height is 0.2000)
Specify rotation angle of text <0>:
          (The default rotation angle is 0)
Text:
```

Figure 2-5 AutoCAD Defaults

Command Options

Command options are selections that you can choose from to change the default. Command options are enclosed inside ‘ [ ] ‘ as shown in Figure 2-5. An example of the defaults and options for the CIRCLE command are as follows:

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:

Specify center point for circle – is the default.

[3P/2P/Ttr (tan tan radius)] – are the options.

Cancelling or Aborting a Command

Commands can be cancelled or aborted by pressing the Esc key on the keyboard. Sometimes it must be pressed twice to completely abort the command and return to the Command: prompt.

MUST KNOW: The Enter key or the Space bar are used to execute commands. If either one is pressed when there is no active command, AutoCAD will repeat the last command. Right clicking the mouse will produce the same results.

USER TIP: AutoCAD’s option prompts are a combination of upper and lower case letters. To help you work faster, only enter the uppercase character(s) when entering an option. For example:

Command: LENGTHEN
Select an object or [Delta/Percent/Total/DYnamic]:
(For Delta, enter D, for Percent enter P, for Total enter T, and for DYnamic enter DY.)

WORKALONG: Entering, Executing, and Aborting Commands

Step 1

Using what you learned earlier in this module, start a new drawing using template: 2D English.

Step 2

Save the drawing with the name: AutoCAD 2D Workalong 02-4 in the Lab Exercises folder.

Step 3

Check to ensure that the Model tab is enabled. If it is not, enable it by clicking it with the left mouse button. (Figure Step 3)

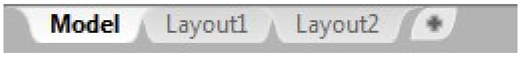


Figure Step 3

Step 4

Disable all features on the Status bar by clicking any that are enabled. Your Status bar should match the figure. (Figure Step 4)

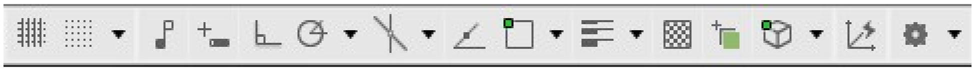
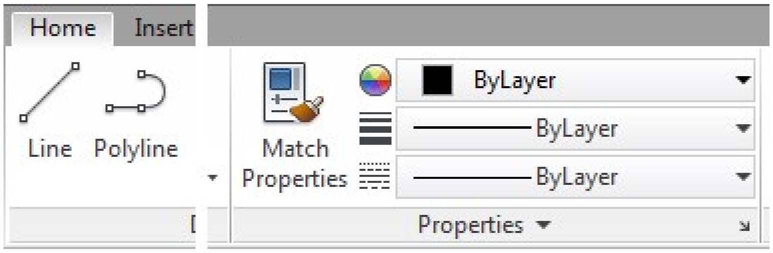


Figure Step 4

Step 5

Ensure that ByLayer is enabled in the first 3 boxes on the Properties toolbar or the Properties tab in the Ribbon menu. (Figure Step 5)


Home
Insert
Line
Polyline
Match Properties
ByLayer
ByLayer
ByLayer
Properties

Ribbon Menu

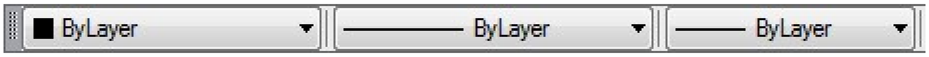
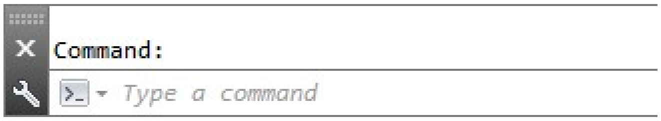

ByLayer
ByLayer
ByLayer

Figure Step 5

AUTHOR'S COMMENTS: You will not be drawing anything in this module. You are learning how to enter commands and understand AutoCAD's prompts and defaults. You will be aborting all commands after you enter them. Be patient, you will start drawing in Module 4.

Step 6

Look in the Command Line window. Note that when there is no active command, AutoCAD displays ' Type a command ' prompt. The prompt is AutoCAD asking you to enter a command. (Figure Step 6)



```
Command:
Type a command
```

Figure Step 6

AUTHOR'S COMMENTS: If the prompt does not display as shown in Figure Step 6, press the Esc key until it does.

Step 7

Enter the LINE command by selecting Draw on the Pull-down menu. In the Draw Pull- down, select Line. (Figure Step 7)

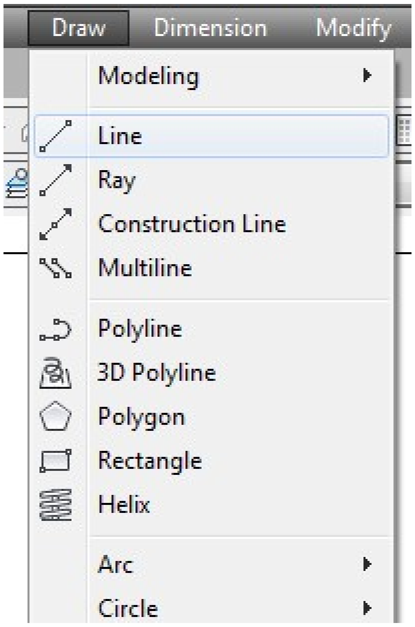
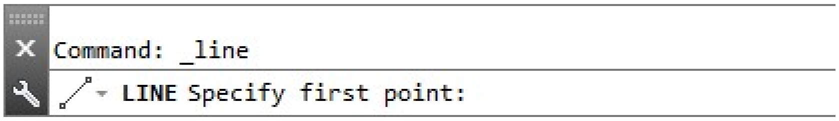

Draw
Dimension
Modify
Modeling
Line
Ray
Construction Line
Multiline
Polyline
3D Polyline
Polygon
Rectangle
Helix
Arc
Circle

Figure Step 7

Step 8

Note in the Command Line window AutoCAD is now prompting ‘ Specify first point: ‘. (Figure Step 8)

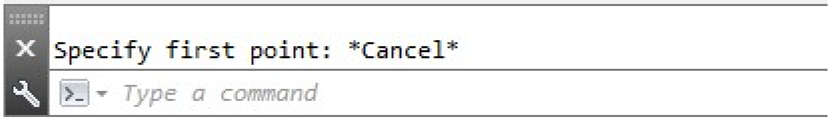


```
Command: _line
LINE Specify first point:
```

Figure Step 8

Step 9

Press Esc to abort (cancel) the command. (Figure Step 9)

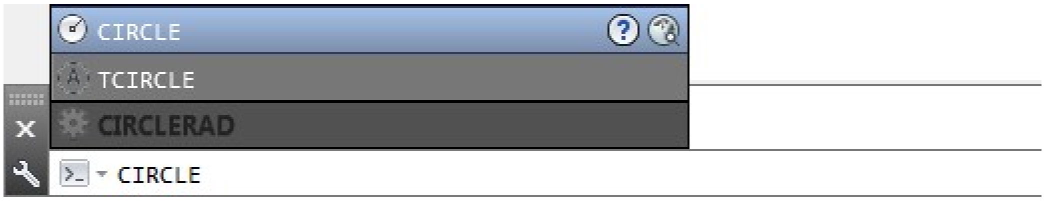


```
Specify first point: *Cancel*
Type a command
```

Figure Step 9

Step 10

At the Command: prompt, enter the CIRCLE command by typing it on the keyboard. (Figure Step 10)



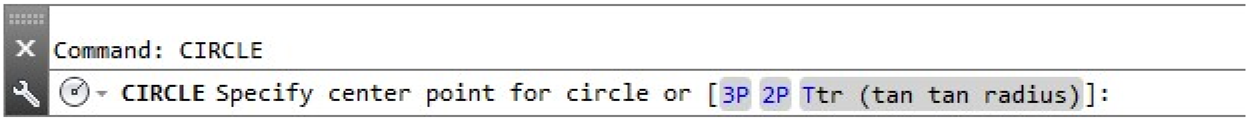
```
CIRCLE
```

Figure Step 10

AUTHOR’S COMMENTS: Note that as you type the command, AutoCAD will suggest possible commands. If you see the command you want and it is blue, press the Enter key. If is grey, click it with the Graphic cursor to complete the full command.

Step 11

Press Enter to execute the command. Note how AutoCAD prompts ‘ Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: ‘ (Figure Step 11)



```
Command: CIRCLE
CIRCLE Specify center point for circle or [3P 2P Ttr (tan tan radius)]:
```

Figure Step 11

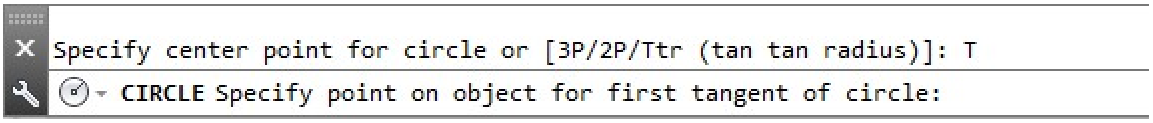
AUTHOR’S COMMENTS: The AutoCAD prompt means the following:

Specify center point for circle – The default.

[3P/2P/Ttr (tan tan radius)] – This is the option prompt. If you do not want to use the default, you can enter 3P, 2P or T to select one of the options.

Step 12

Enter a T and press Enter. Take note of the new prompt ‘ Specify point on object for first tangent of circle: ‘. (Figure Step 12)

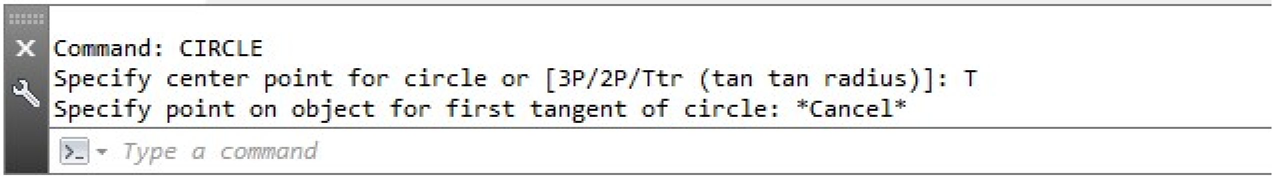


```
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: T
CIRCLE Specify point on object for first tangent of circle:
```

Figure Step 12

Step 13

Press Esc to abort the command. (Figure Step 13)

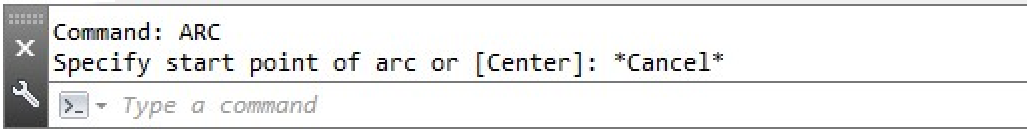


```
Command: CIRCLE
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: T
Specify point on object for first tangent of circle: *Cancel*
```

Figure Step 13

Step 14

Enter the ARC command by typing it in on the keyboard. Press the Space bar to execute command. Press Esc to abort the command. (Figure Step 14)



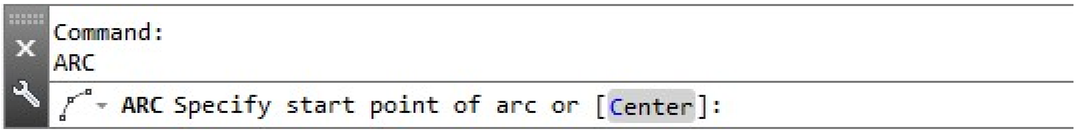
```
Command: ARC
Specify start point of arc or [Center]: *Cancel*
```

Figure Step 14

AUTHOR'S COMMENTS: Note that a command can be executed by either pressing the Enter key or the Space bar.

Step 15

Press Enter without entering a command. Note how AutoCAD will repeat the last command. (Figure Step 15)

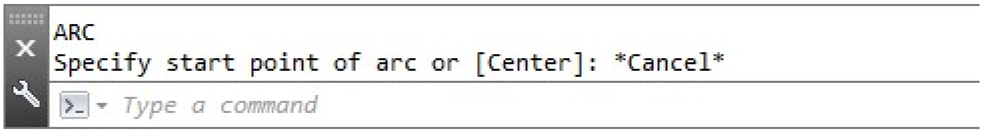


```
Command:
ARC
ARC Specify start point of arc or [Center]:
```

Figure Step 15

Step 16

Press Esc to abort the command to return to the Command prompt. (Figure Step 16)

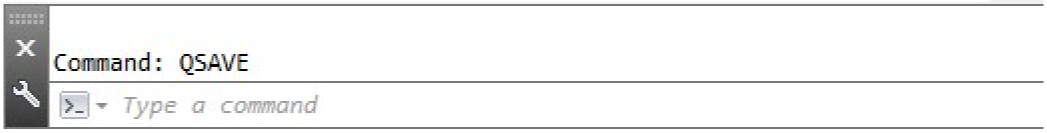


```
ARC
Specify start point of arc or [Center]: *Cancel*
```

Figure Step 16

Step 17

Enter the QSAVE command by either typing it on the keyboard. (Figure Step 17)



```
Command: QSAVE
```

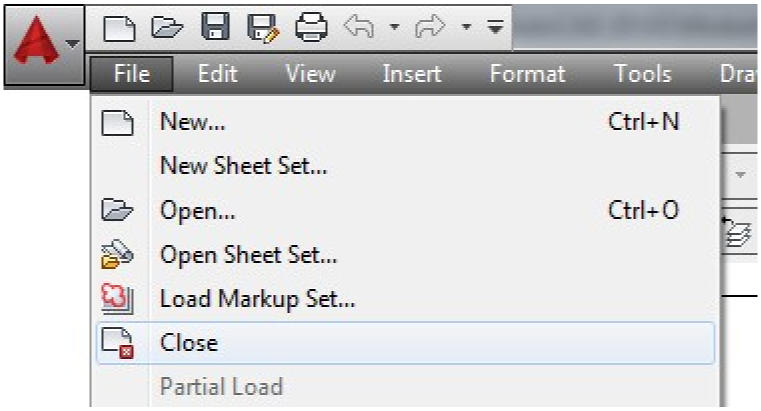
Type a command

Figure Step 17

AUTHOR'S COMMENTS: Note how some commands do not prompt you for anything.

Step 18

Enter the CLOSE command to close the drawing. (Figure Step 18)



File
Edit
View
Insert
Format
Tools
Dra
New... Ctrl+N
New Sheet Set...
Open... Ctrl+O
Open Sheet Set...
Load Markup Set...
Close
Partial Load

Figure Step 18

MUST KNOW: Command options are selections that you can choose from to change the command default. Command options are enclosed inside square brackets ' [ ] '.Select first object [Polyline/Radius/Trim] – Enter either a P, R or T to change the default.

AUTHOR'S COMMENTS: To sum up:

1. Either the Enter key or the Space bar can be used to execute a
2. If there is no active command, AutoCAD will repeat the last command if you press the Enter key, the Space bar, or right clicks the
3. The Esc key will abort (cancel) any command or process and return you to the Command: prompt.
4. Most AutoCAD command prompts contain a default and an option
5. A default is the value AutoCAD uses, if you do not supply one. Defaults are enclosed inside angle brackets ' < > '. If you want to use the default value, simply respond by pressing the Enter key or the Space

6. Option prompts are enclosed inside square brackets ‘ [ ] ‘.

MUST KNOW: The Esc key is a very important when drawing in AutoCAD. When pressed, it will abort or cancel any command or process and return you to the command prompt. You will be using this key frequently while working in AutoCAD.

Key Principles

Key Principles in Module 2

1. An AutoCAD drawing is a database file containing graphical and non-graphical data and must have the extension .dwg.

2. A drawing template is a drawing file that may contain layouts, defaults, blocks, and standards set by the creator of the template and must be used to start a new drawing.

3. The AutoCAD file extensions taught in the AutoCAD 2D book are .dwg, .dwt, and .bak. .dwg is a drawing file. .dwt is a template file. .bak is a backup file.

4. Regardless of the way commands are entered, AutoCAD doesn’t really know or care where the command was initiated.

5. A default is the value AutoCAD uses, if you do not supply one. Defaults are enclosed inside angle brackets ‘ < > ‘. If you want to use the default value, simply respond by pressing the Enter key or the Space bar.

6. In an AutoCAD command prompt, the options are enclosed inside square brackets ‘ [ ] ‘.

7. The Enter key or the Space bar are used to execute a command. If there is no active command and either the Enter key or Space bar are pressed, the last command will repeat. The same thing can be achieved by right clicking the mouse.

8. The Esc key will cancel or abort any AutoCAD command or process.

Lab Exercise 2-1

Time allowed: 30 minutes.

Step 1

Start a new drawing using the template: 2D English.

Step 2 IMPORTANT STEP

Unless otherwise instructed, each time you start a workalong or a lab exercise in the AutoCAD 2D book, the Model tab should be enabled. (Figure Step 2)

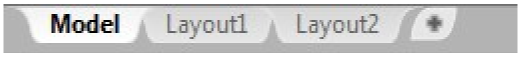

Model
Layout1
Layout2

Figure Step 2

Step 3 IMPORTANT STEP

Unless otherwise instructed, each time you start a workalong or a lab exercise in the AutoCAD 2D book, all features on the Status bar should be disabled as shown in the figure. (Figure Step 3)

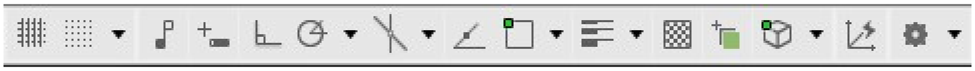


Figure Step 3

Step 4

If any icons in the Status bar are enabled as shown in Figure Step 4, click them until they are disabled as shown in Figure Step 3. (Figure Step 4)

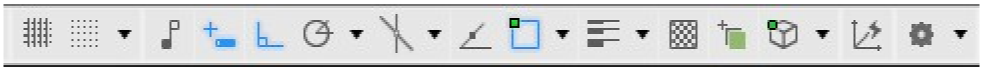
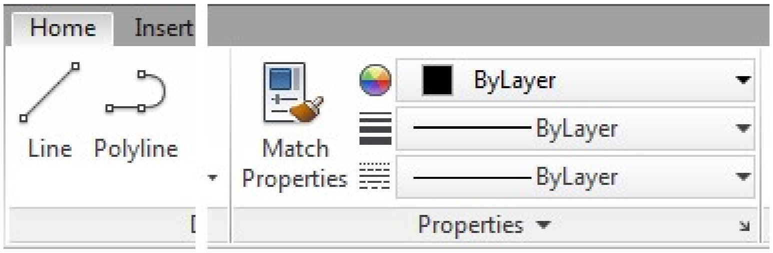


Figure Step 4

Step 5 IMPORTANT STEP

Unless otherwise instructed, each time you start a workalong or a lab exercise in the AutoCAD 2D book, the first three box of the Properties toolbar or the Properties tab in Ribbon menu should be set ByLayer . (Figure Step 5)


Home
Insert
Line
Polyline
Match Properties
ByLayer
ByLayer
ByLayer
Properties

Ribbon Menu

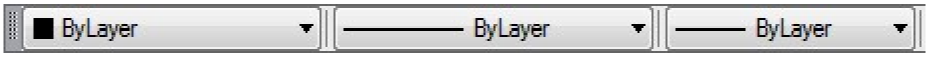

ByLayer
ByLayer
ByLayer

Figure Step 5 Toolbar Menu

Step 6

Save and name the drawing with the name: AutoCAD 2D Lab 02-1. Save the drawing in the folder: C:\CAD Courses/AutoCAD 2D/Lab Exercises. Using the CLOSE command, close the drawing.

Step 7

Complete the following:

- A Using the NEW command, start a new drawing using the template: 2D Metric
- B Save and name the drawing with the name: AutoCAD 2D Lab 02-2
- C Save the drawing in the folder:

C:\CAD Courses/AutoCAD 2D/Lab Exercises

- D Using the CLOSE command, close the

Step 8

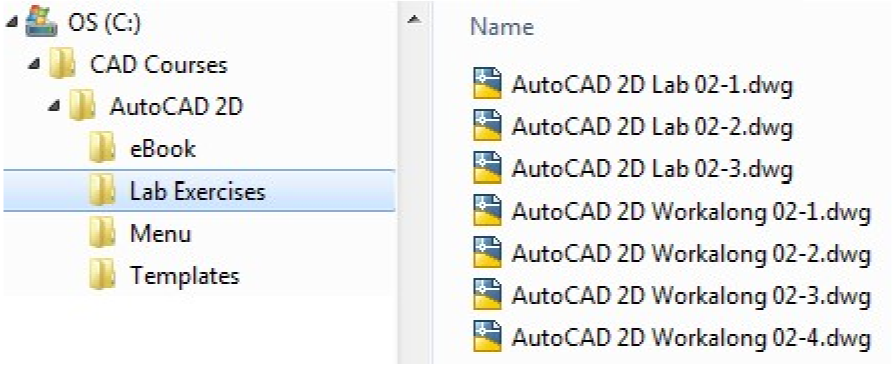
Complete the following:

- A Using the OPEN command, open the drawing: AutoCAD 2D Lab 02-2
- B Using the SAVEAS command, save and name the drawing: AutoCAD 2D Lab 02-3
- D Using the CLOSE command, close the drawing.

Step 9

Minimize AutoCAD and open File Explorer. Locate the folder:

C:\CAD Courses/AutoCAD 2D/Lab Exercises. It should contain 7 drawing files, with or without the extension .dwg, as shown in the figure. (Figure Step 9A and 9B)



AutoCAD 2D Lab 02-1.dwg
AutoCAD 2D Lab 02-2.dwg
AutoCAD 2D Lab 02-3.dwg
AutoCAD 2D Workalong 02-1.dwg
AutoCAD 2D Workalong 02-2.dwg
AutoCAD 2D Workalong 02-3.dwg
AutoCAD 2D Workalong 02-4.dwg

Figure Step 9A

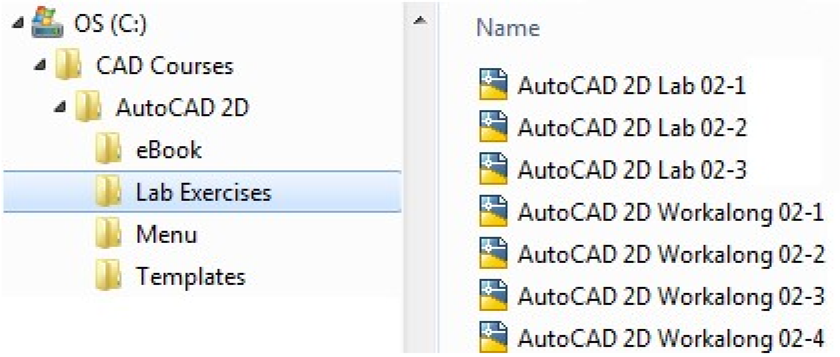

OS (C:)
CAD Courses
AutoCAD 2D
eBook
Lab Exercises
Menu
Templates
Name
AutoCAD 2D Lab 02-1
AutoCAD 2D Lab 02-2
AutoCAD 2D Lab 02-3
AutoCAD 2D Workalong 02-1
AutoCAD 2D Workalong 02-2
AutoCAD 2D Workalong 02-3
AutoCAD 2D Workalong 02-4

Figure Step 9B

AUTHOR'S COMMENTS: You may see some .bak files in your list with the same name as the drawing files. Note that drawing files have the AutoCAD icon preceding the drawing name. Figure Step 9 only shows the drawing files. Your list may show some .bak files, ignore them.

Module 3: Checking the Accuracy of Your Drawings

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe how you can check the lab exercise drawings you complete in the AutoCAD 2D book for accuracy.

AUTHOR’S COMMENTS: It is extremely important to create all drawings with 100% accuracy. This cannot be stressed strongly enough as it is essential that each object in the drawing is positioned and sized perfectly. Most lab exercises in the AutoCAD 2D book can be checked for accuracy by overlaying the drawing with a *key*. This helps you to know when your drawing is accurate and will display any objects that are drawn incorrectly. As you become more experienced drawing in AutoCAD 2021, you will not require the use of keys. Interface varies slightly for each release.

WORKALONG: Checking the Accuracy of Your Drawings

Step 1

Using the NEW command, start a new drawing using the template: 2D English.

Step 2

Save and name the drawing: AutoCAD 2D Lab 03-1. Save it in the folder: CAD Courses/AutoCAD 2D/Lab Exercises.

Step 3

Check to ensure that the Model tab is enabled. If it is not, enable it by clicking it with the left mouse button. (Figure Step 3)

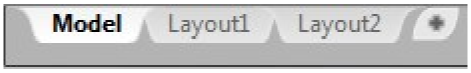


Figure Step 3

AUTHOR’S COMMENTS: The Model tab should be enabled at all times, while working on workalongs and lab exercises in the AutoCAD 2D book, unless you are instructed otherwise. Module 18 will teach you how work in the Layout tabs.

Step 4

Ensure that all features on the Status bar are disabled by clicking any that are enabled. Your Status bar should match the figure. (Figure Step 4)

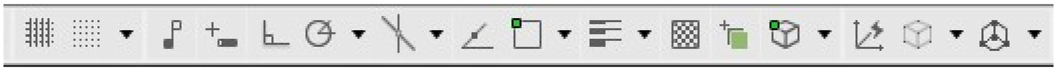
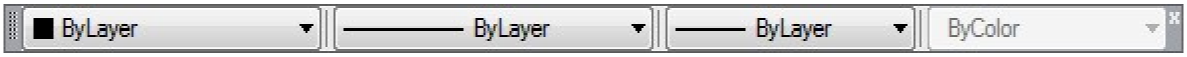


Figure Step 4

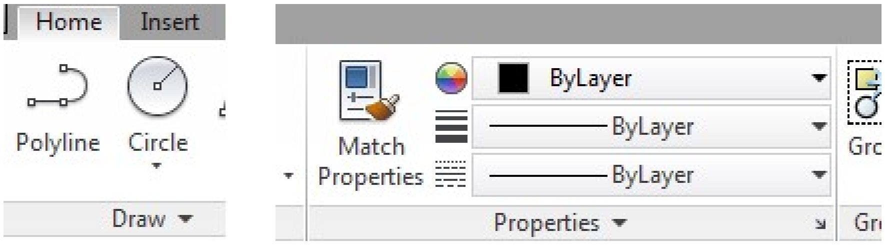
AUTHOR'S COMMENTS: Unless otherwise instructed, each time you start a workalong or lab exercise while working on the AutoCAD 2D book, all features on the Status bar should be disabled.

Step 5

Ensure that ByLayer is enabled in the first 3 boxes on the Properties menu. (Figure Step 5)


ByLayer
ByLayer
ByLayer
ByColor

Properties Toolbar


Home
Insert
Polyline
Circle
Draw
Match
Properties
ByLayer
ByLayer
ByLayer
Properties
Grc
Gr

Home Ribbon
Figure Step 5

AUTHOR'S COMMENTS: Unless otherwise instructed, each time you work on a workalong or a lab exercise while working on the AutoCAD 2D book, ByLayer should be enabled in the first three boxes on the Properties toolbar or Ribbon menu.

Step 6

Click Format on the Pull-down menu and click Units to open the Drawing Units dialogue box. (Figure Step 6A and 6B)

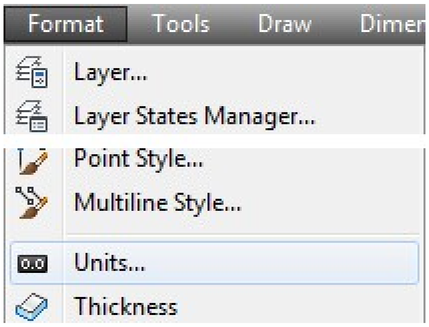
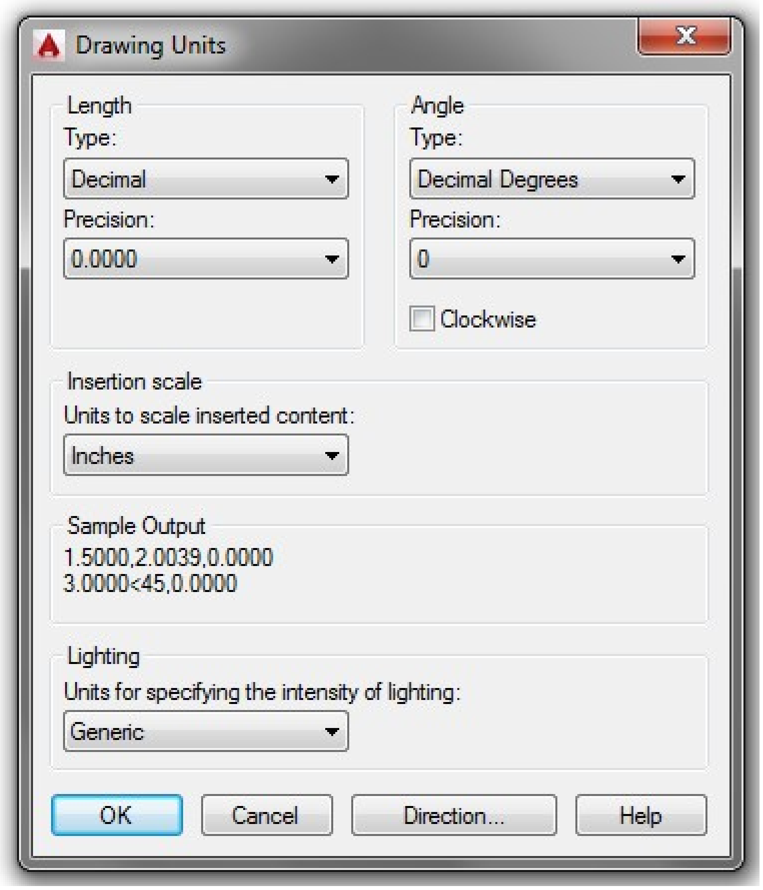

Format
Tools
Draw
Dimer
Layer...
Layer States Manager...
Point Style...
Multiline Style...
Units...
Thickness

Figure Step 6A



Drawing Units

Length
Type: Decimal
Precision: 0.0000

Angle
Type: Decimal Degrees
Precision: 0
☐ Clockwise

Insertion scale
Units to scale inserted content: Inches

Sample Output
1.5000,2.0039,0.0000
3.0000<45,0.0000

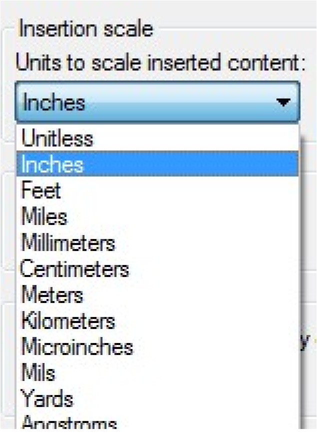
Lighting
Units for specifying the intensity of lighting: Generic

OK
Cancel
Direction...
Help

Figure Step 6B

Step 7

Pull down the Insertion scale list and select Inches. Click OK to close the Drawing Units dialogue box. (Figure Step 7)



Insertion scale
Units to scale inserted content: Inches

Unitless
Inches
Feet
Miles
Millimeters
Centimeters
Meters
Kilometers
Microinches
Mils
Yards
Angstroms

Figure Step 7

Step 8

Click Insert on the Pull-down menu and select Block. This will open the Insert dialogue box. (Figure Step 8A and 8B)

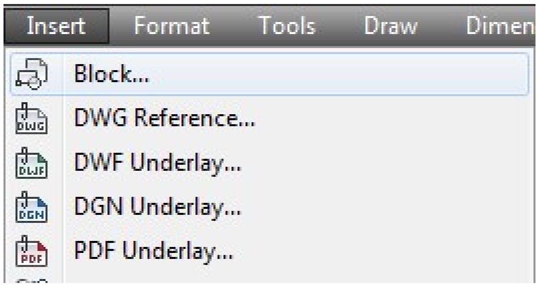
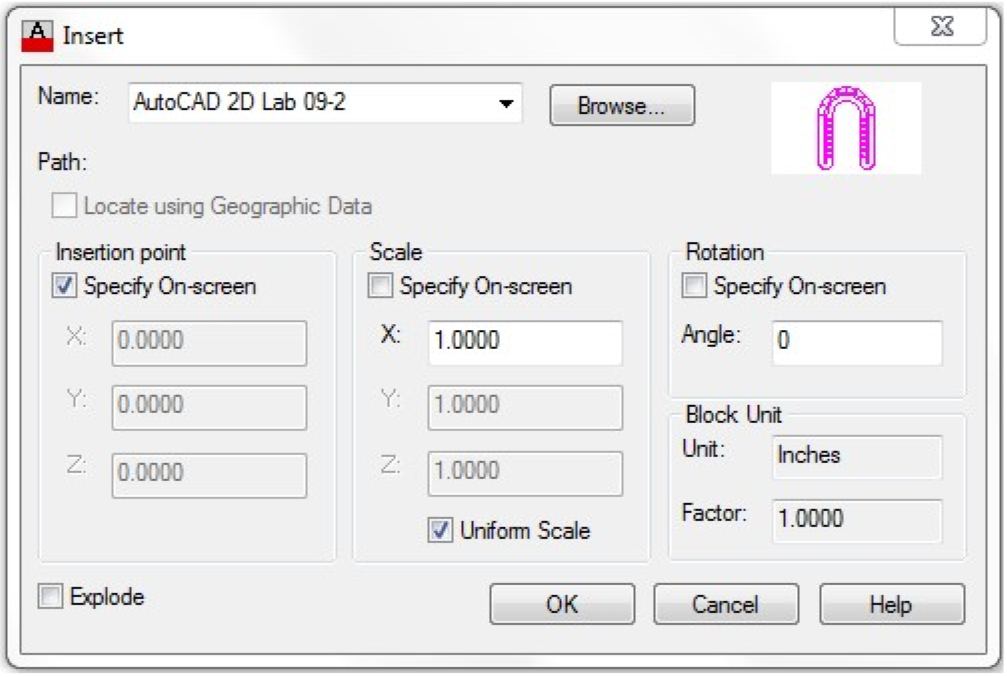

Insert
Format
Tools
Draw
Dimen
Block...
DWG Reference...
DWF Underlay...
DGN Underlay...
PDF Underlay...

Figure Step 8A


Insert
Browse...
OK
Cancel
Help

Name: AutoCAD 2D Lab 09-2

Path:

☐ Locate using Geographic Data

Insertion point

☑ Specify On-screen

X: 0.0000

Y: 0.0000

Z: 0.0000

Scale

☐ Specify On-screen

X: 1.0000

Y: 1.0000

Z: 1.0000

☑ Uniform Scale

Rotation

☐ Specify On-screen

Angle: 0

Block Unit

Unit: Inches

Factor: 1.0000

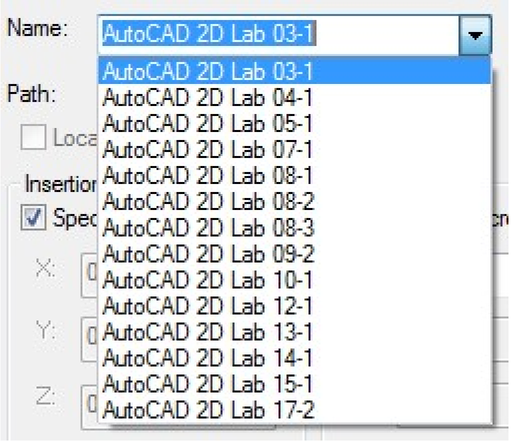
☐ Explode

Figure Step 8B

AUTHOR'S COMMENTS: The dialogue box in your AutoCAD may not match the figure exactly.

Step 9

Click the small triangle at the end of the Name: pull-down list to pull down the list of block names. (Figure Step 9)



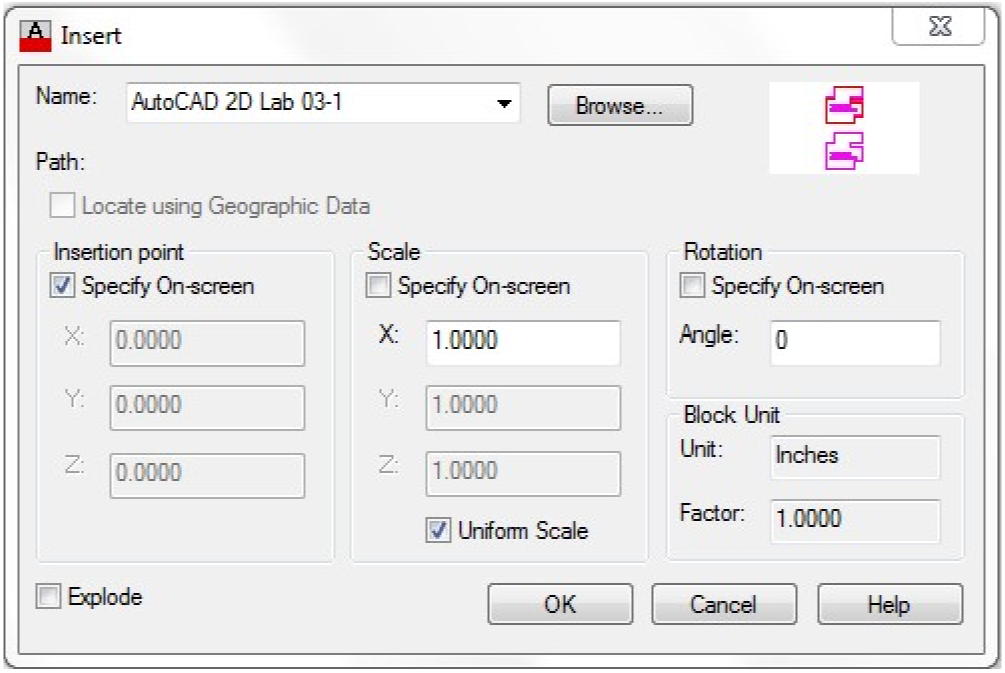
Name: AutoCAD 2D Lab 03-1

AutoCAD 2D Lab 03-1
AutoCAD 2D Lab 04-1
AutoCAD 2D Lab 05-1
AutoCAD 2D Lab 07-1
AutoCAD 2D Lab 08-1
AutoCAD 2D Lab 08-2
AutoCAD 2D Lab 08-3
AutoCAD 2D Lab 09-2
AutoCAD 2D Lab 10-1
AutoCAD 2D Lab 12-1
AutoCAD 2D Lab 13-1
AutoCAD 2D Lab 14-1
AutoCAD 2D Lab 15-1
AutoCAD 2D Lab 17-2

Figure Step 9

Step 10

Select the name: AutoCAD 2D Lab 03-1. (Figure Step 10)



Insert

Name: AutoCAD 2D Lab 03-1

Path:

☐ Locate using Geographic Data

Insertion point
☑ Specify On-screen
X: 0.0000
Y: 0.0000
Z: 0.0000

Scale
☐ Specify On-screen
X: 1.0000
Y: 1.0000
Z: 1.0000
☑ Uniform Scale

Rotation
☐ Specify On-screen
Angle: 0

Block Unit
Unit: Inches
Factor: 1.0000

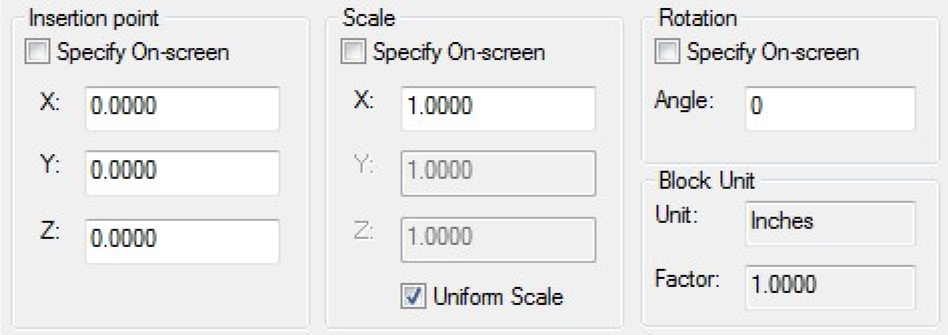
☐ Explode

Figure Step 10

AUTHOR'S COMMENTS: Always select the name of the current drawing.

Step 11

Ensure that the three Specify On-screen boxes are disabled as shown in the figure. (Figure Step 11)



Insertion point
☐ Specify On-screen
X: 0.0000
Y: 0.0000
Z: 0.0000

Scale
☐ Specify On-screen
X: 1.0000
Y: 1.0000
Z: 1.0000
☑ Uniform Scale

Rotation
☐ Specify On-screen
Angle: 0

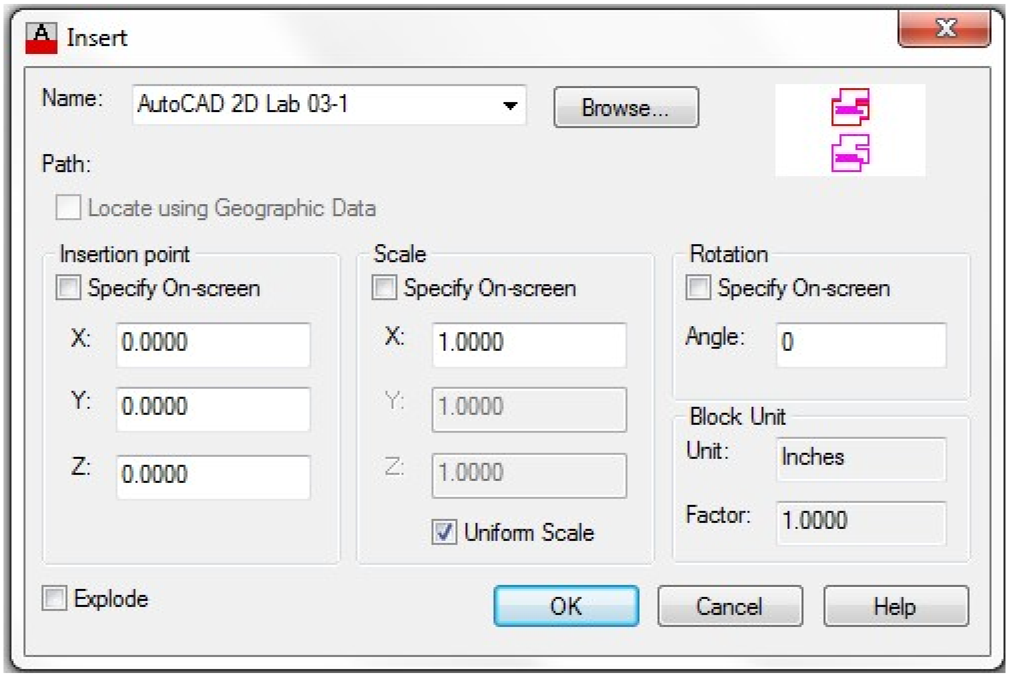
Block Unit
Unit: Inches
Factor: 1.0000

Figure Step 11

> **AUTHOR'S COMMENTS:** After you do this step once, it will keep the same settings unless you intentionally change it.

Step 12

Ensure that the Insert dialogue box matches the figure and click the OK button. (Figure Step 12)



Name: AutoCAD 2D Lab 03-1
Path:
☐ Locate using Geographic Data

Insertion point
☐ Specify On-screen
X: 0.0000
Y: 0.0000
Z: 0.0000

Scale
☐ Specify On-screen
X: 1.0000
Y: 1.0000
Z: 1.0000
☑ Uniform Scale

Rotation
☐ Specify On-screen
Angle: 0

Block Unit
Unit: Inches
Factor: 1.0000

☐ Explode

Figure Step 12

> **AUTHOR'S COMMENTS:** This is just a demo, therefore there is nothing for you to do but learn the process of checking your drawing for accuracy with a key. You will be checking your first drawing for accuracy in Module 4.

Step 13

This will insert a magenta colored key on your drawing. (Figure Step 13)

AUTHOR’S COMMENTS: If you see double objects or places where the red objects and the key’s magenta objects do not match, your drawing is inaccurate. If there is only one object, even though it may share the colors magenta and red, your drawing is accurate. If the drawing is inaccurate, disable the key’s display and correct the drawing. Enable the key’s display and recheck it for accuracy. When your drawing matches the key, save and close the drawing and go on to the next lab exercise or module. Module 5 will teach you how to disable and enable the key’s display.

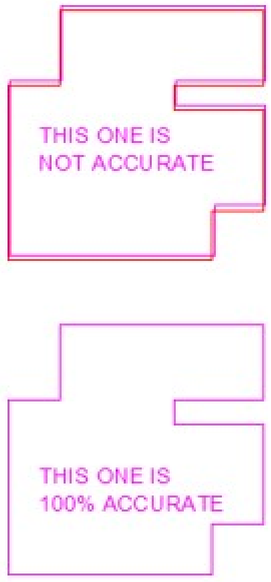


Figure Step 13

Step 14

Save and close the drawing.

Key Principles

Key Principles in Module 3

1. AutoCAD drawings must be drawn 100% accurate.
2. You can check the accuracy of most lab exercise drawings in the AutoCAD 2D book using a key.
3. The name of the key that you select to check your drawing for accuracy is always the same as the name of the current drawing.

4. Your drawing is not accurate if the magenta objects in the key do not match your drawing objects. If you only see one object, either magenta or red, your drawing is 100% accurate.

Module 4: Drawing Lines Using Cartesian Coordinates

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe the geometry of points and lines.
2. Describe the Cartesian Coordinate System.
3. Define and explain the terms lastpoint, absolute coordinates, and relative coordinates.
4. Apply the LINE command to draw lines using the XY Cartesian Coordinate System using both absolute and relative coordinates.

Geometry Lesson: Points and Lines

A *point* is defined as a single XY coordinate. It does not have a width, height, or depth. A *line* is the shortest distance between two XY coordinates. Lines can be horizontal, *vertical*, or *inclined*. Lines that are the same distance apart are called *parallel* lines. *Perpendicular* lines are at right angles to each other or 90 degrees apart. See Figure 4-1 and 4-2.

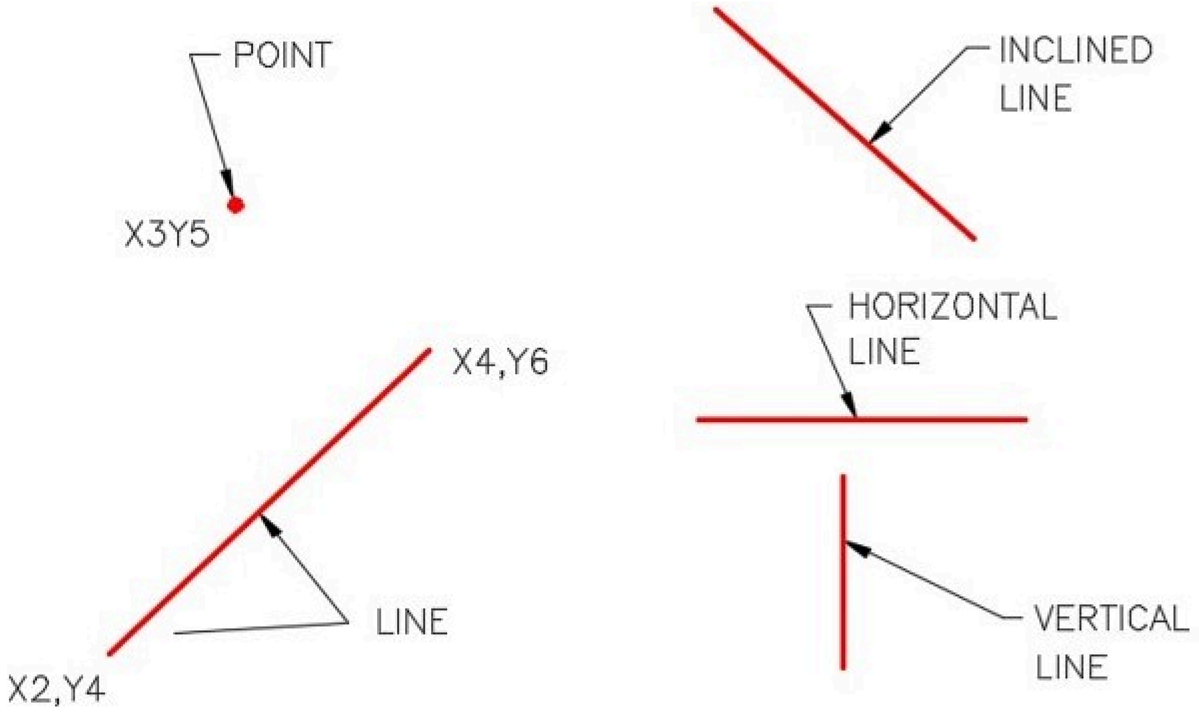


Figure 4-1 Points and Lines

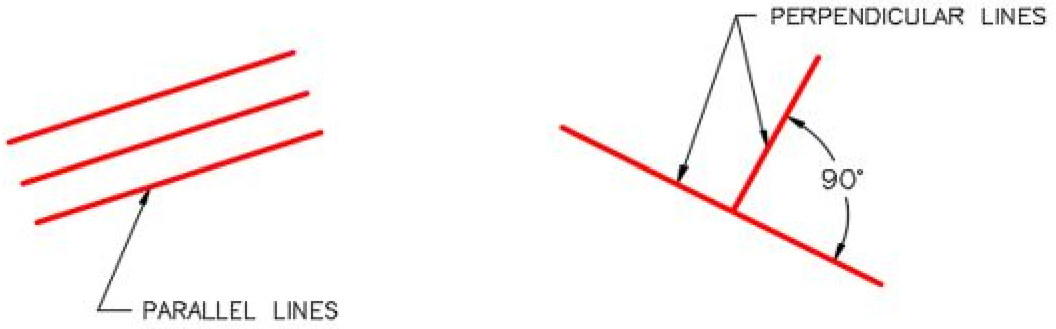

PERPENDICULAR LINES
90°
PARALLEL LINES

Figure 4-2 Parallel and Perpendicular Lines

The Cartesian Coordinate System

To accurately draw an AutoCAD two dimensional (2D) drawing, you must enter XY coordinate locations. These XY coordinates are based on the Cartesian Coordinate System.

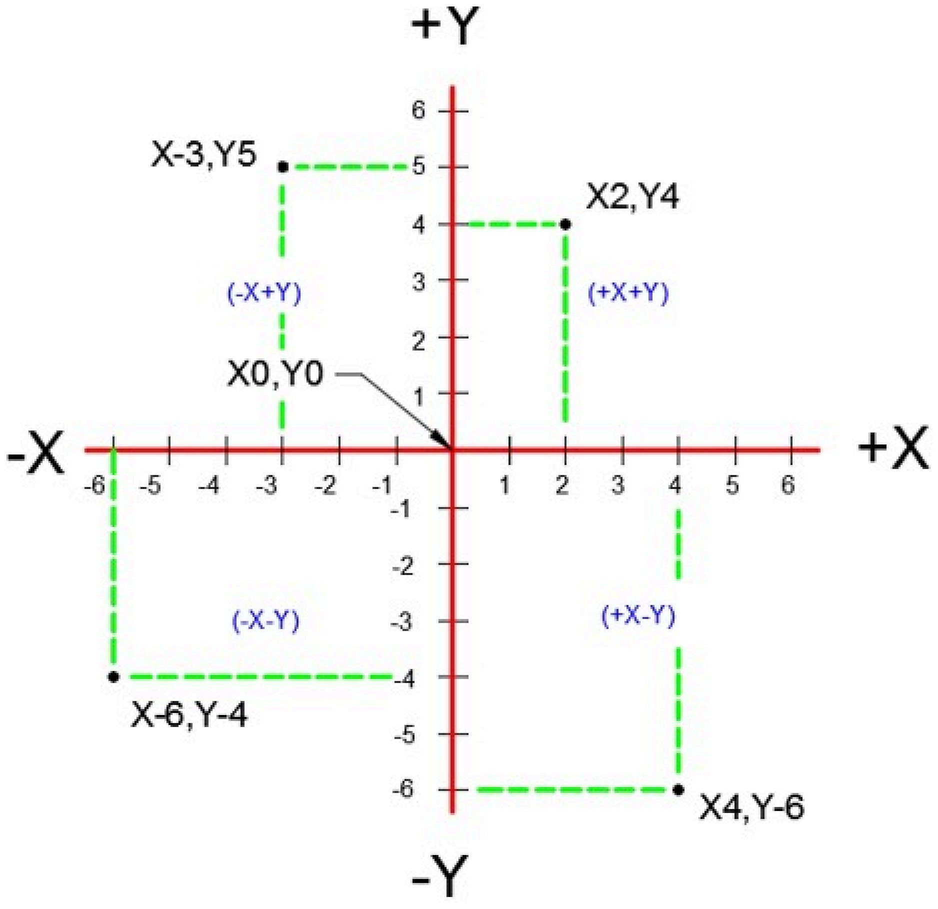

+Y
X-3,Y5
X2,Y4
(-X+Y)
(+X+Y)
X0,Y0
-X
+X
(-X-Y)
(+X-Y)
X-6,Y-4
X4,Y-6
-Y
6
5
4
3
2
1
-1
-2
-3
-4
-5
-6

Figure 4-3 The Cartesian Coordinate System

The *Cartesian Coordinate System* consists of two numbered lines crossing perpendicular to one another at their zero values. The horizontal axis is the X axis and the vertical axis is the Y axis. See Figure 4-3. A coordinate value is assigned to each location on the current construction plane. Throughout the AutoCAD 2D book you will be working on the same construction plane. Construction planes are fully explained in the AutoCAD 3D book.

When you are using a AutoCAD 2D command, each coordinate value consists of a pair of numbers, the first is the X coordinate and the second is the Y coordinate, written X,Y. The X and Y values must be separated by a comma. For example, X2,Y4 (entered in AutoCAD as 2,4) is the location 2 units to the right (positive) and 4 units up (positive) from X0,Y0 or 0,0.

The values can be either positive or negative. Positive numbers are default so the plus sign is not required. If the value is negative, the minus sign must precede the number. For example if entered in a an AutoCAD command, -3,5 is X minus 3 and Y positive 5.

Cartesian coordinates can be entered in a AutoCAD command as either absolute coordinates or relative coordinates.

Absolute Cartesian Coordinates

Absolute Cartesian Coordinates are always referenced to the absolute origin 0,0. The Absolute Cartesian Coordinate 3,4 (X3Y4) is 3 units to the right and 4 units upwards from 0,0 (X0Y0).

Negative values can also be used. The Absolute Cartesian Coordinate -4,2 (X-4,Y2) is 4 units to the left and 2 units upwards from 0,0 (X0Y0).

Relative Cartesian Coordinates

Relative Cartesian Coordinates are incremental to the lastpoint. To indicate to AutoCAD that the coordinate being entered is relative, the @ symbol must precede the coordinate value. For example, @2,6, which means ‘ from the lastpoint go 2 units in the positive X and 6 units in the positive Y ‘. Another example using negative values @4,-2 which means ‘ from the lastpoint go 4 units in the positive X and 2 units in the negative Y.

Lastpoint

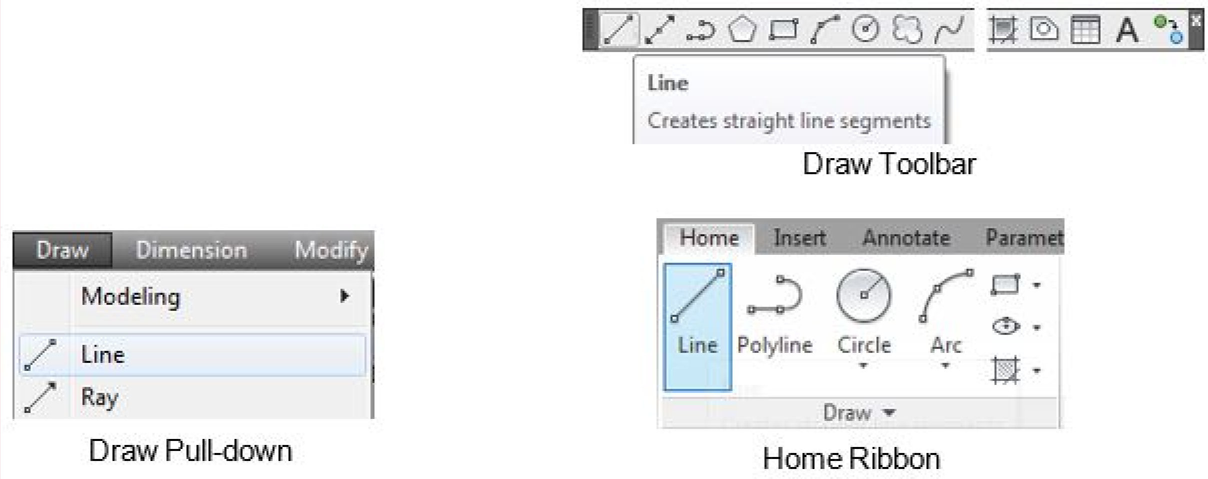
The *lastpoint* is the last XY location that was used in an AutoCAD command. The lastpoint is very important to you when drawing in AutoCAD. AutoCAD remembers the lastpoint entered and saves it in the @ symbol. The @ symbol means ‘ The last absolute coordinate location ‘.

USER TIP: When you start a drawing, the first XY location is always specified using an absolute coordinate. After that, relative coordinates are used. It would be too difficult to draw using all absolute coordinates.

AutoCAD Command: LINE

The LINE command is used to draw lines.

Shortcut: **L**


Line
Creates straight line segments
Draw
Dimension
Modify
Modeling
Line
Ray
Home
Insert
Annotate
Paramet
Line
Polyline
Circle
Arc
Draw

Draw Toolbar

Draw Pull-down

Home Ribbon

WORKALONG: Drawing Lines Using Cartesian Coordinates – Part 1

Step 1

Using the NEW command, start a new drawing using the template: 2D English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 04-1. Save it in the folder: CAD Courses/ AutoCAD 2D/Lab Exercises

Step 3

Enter the LINE command, as shown below, to draw the object shown in the figure. Keep in mind that you enter what is in bold font, the author's comments are in italics, and everything else are AutoCAD responses or prompts. (Figure Step 3)

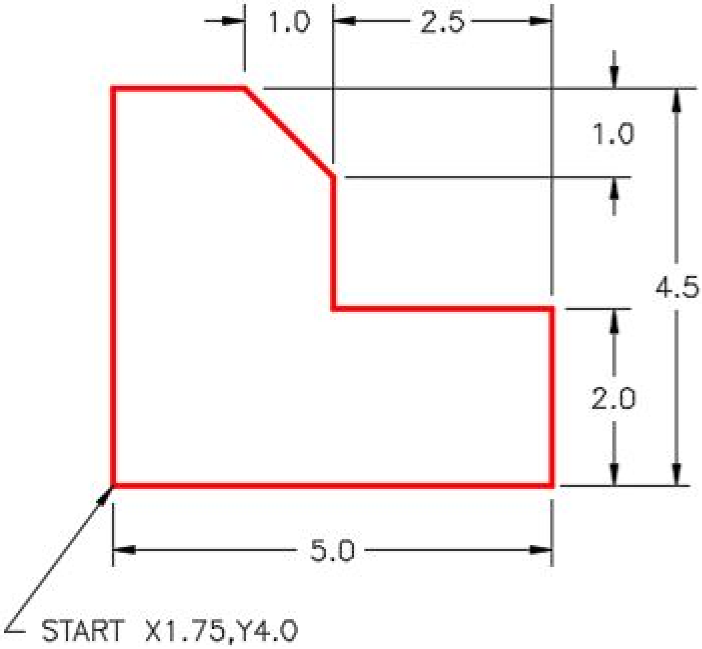


Figure Step 3

Command: **LINE**

Specify first point: **1.75,4**

(Always start with an absolute coordinate. It must be a X then a Y separated with a comma.)

Specify next point or [Undo]: **@5,0**

(Then change to relative coordinates. Note the @ first, then X and Y.)

Specify next point or [Undo]: **@0,2**

Specify next point or [Close/Undo]: **@-2.5,0**

(A negative coordinate is used since the line is going in the negative X direction.)

Specify next point or [Close/Undo]: **@0,1.5**

Specify next point or [Close/Undo]: **@-1,1**

(When both the X and Y coordinates have a value other then zero, the line will be inclined.)

Specify next point or [Close/Undo]: **@-1.5,0**

Specify next point or [Close/Undo]: **C**

(You can use a C or 1.75,4 to close the last line and return to the first point.)

Command:

AUTHOR'S COMMENTS: Rather then just entering the coordinate values, try to understand the values you are entering by studying the Figure Step 3.

Step 4

Your completed drawing should match the figure. (Figure Step 4)

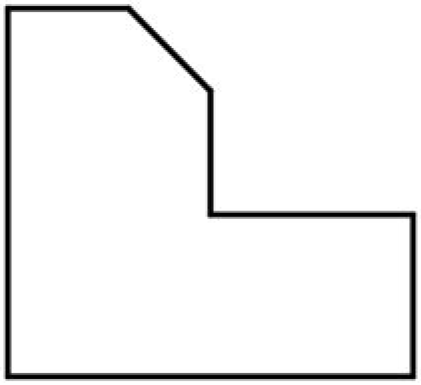


Figure Step 4

AUTHOR'S COMMENTS: If you have trouble drawing this object the first try, do not be concerned. Start it over again, from scratch, until you can complete it. The more practice you get drawing, the easier it will get.

Step 5

Save and close the drawing.

MUST KNOW: When drawing inclined lines using Cartesian Coordinates, the value of both the X and the Y coordinates cannot be zero. See example below.

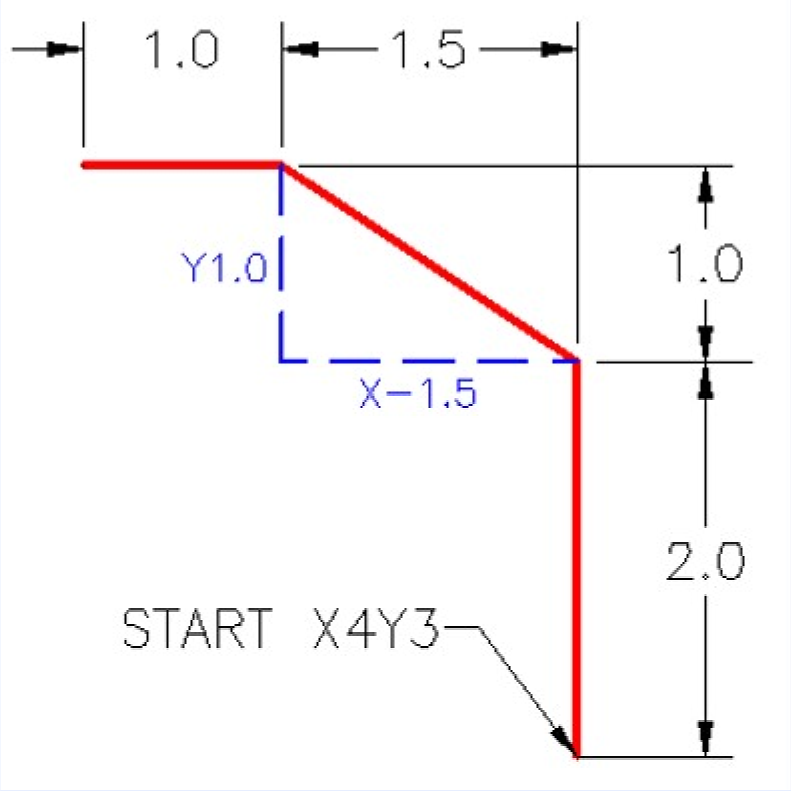


Figure 4-4 Drawing Inclined Lines Using Cartesian Coordinates

```
Command: LINE
Specify first point: 4,3
Specify next point or [Undo]: @0,2
Specify next point or [Undo]: @-1.5,1
Specify next point or [Close/Undo]: @-1,0
Specify next point or [Close/Undo]:
Command:
```

Note to draw the inclined line, you have to enter a X value and a Y value that is not zero. In this example, @-1.5,1 If either X or the Y is zero, then the line would be either horizontal or vertical. See Figure 4-4.

In Module 10, you will be taught how to draw incline lines using Polar Coordinates.

WORKALONG: Drawing Lines Using Cartesian Coordinates – Part 2

Step 1

Using the NEW command, start a new drawing using the template: 2D English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 04-2. Save it in the folder: CAD Courses/AutoCAD 2D/Lab Exercises

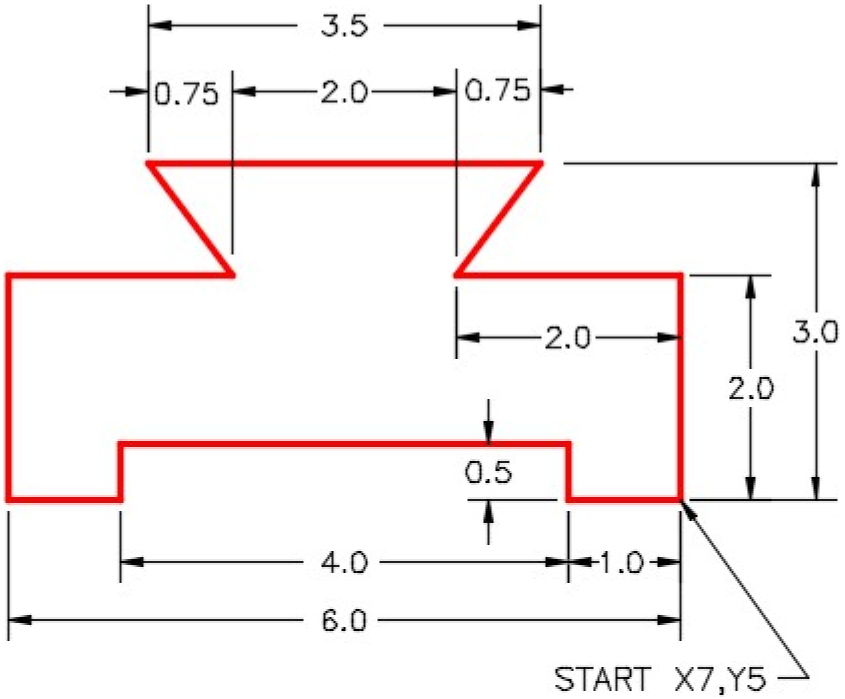


Figure Step 3

Step 3

Enter the LINE command, as shown below, to draw the object shown in the figure. (Figure Step 3)

Command: **L**

(L is the shortcut for the LINE command)

Specify first point: **7,5**

Specify next point or [Undo]: **@0,2**

Specify next point or [Undo]: **@-2,0**

Specify next point or [Close/Undo]: **@.75,1**

(You can draw an inclined line by entering a number other then zero for both the X and Y.)

Specify next point or [Close/Undo]: **@-3.5,0**

Specify next point or [Close/Undo]: **@-.75,-1**

Specify next point or [Close/Undo]: **U**

(When you make an input error, enter a U to go back one step. More than one U can be entered to step back further. Ensure that you press the ENTER or SPACE after each one.)

Specify next point or [Close/Undo]: **@.75,-1**

Specify next point or [Close/Undo]: **@-2,0**

Specify next point or [Close/Undo]: **@0,-2**

Specify next point or [Close/Undo]: **@1,0**

Specify next point or [Close/Undo]: **@0,.5**

Specify next point or [Close/Undo]: **@4,0**

Specify next point or [Close/Undo]: **@0,-.5**

Specify next point or [Close/Undo]: **7,5**

(The object was closed by entering the absolute coordinate of the first point.)

Specify next point or [Close/Undo]: Command:

Step 4

Your completed object should match the figure. (Figure Step 4)

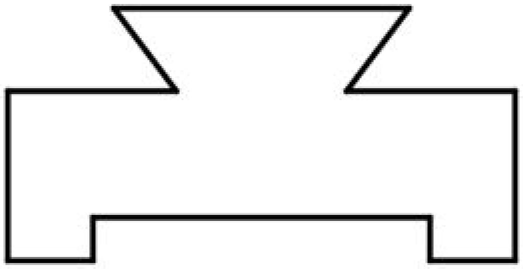


Figure Step 4

AUTHOR'S COMMENTS: If you have trouble drawing this object the first try, do not be concerned. Start it over again, from scratch, until you can complete it. The more practice you get drawing, the easier it will get.

Step 5

Save and close the drawing.

USER TIP: When entering a decimal number the ends in a zero, for example 4.0, enter the number up to the zero only. In this case, 4. If the number is 3.6700, all you have enter is 3.67. AutoCAD will automatically adds the zeros.

Deleting Objects

To delete existing drawing objects, you can either use the ERASE command or the Delete key.

Using the ERASE Command

When deleting drawing objects using the ERASE command, the drawing objects can either be selected before or after the command is entered. If the command is entered before selecting the objects, select the objects when prompted by the ‘ Select Object ‘ prompt, as shown below. If the objects are selected before entering the ERASE command, there is no prompt.

Command: **ERASE**

Select Object:

Command:

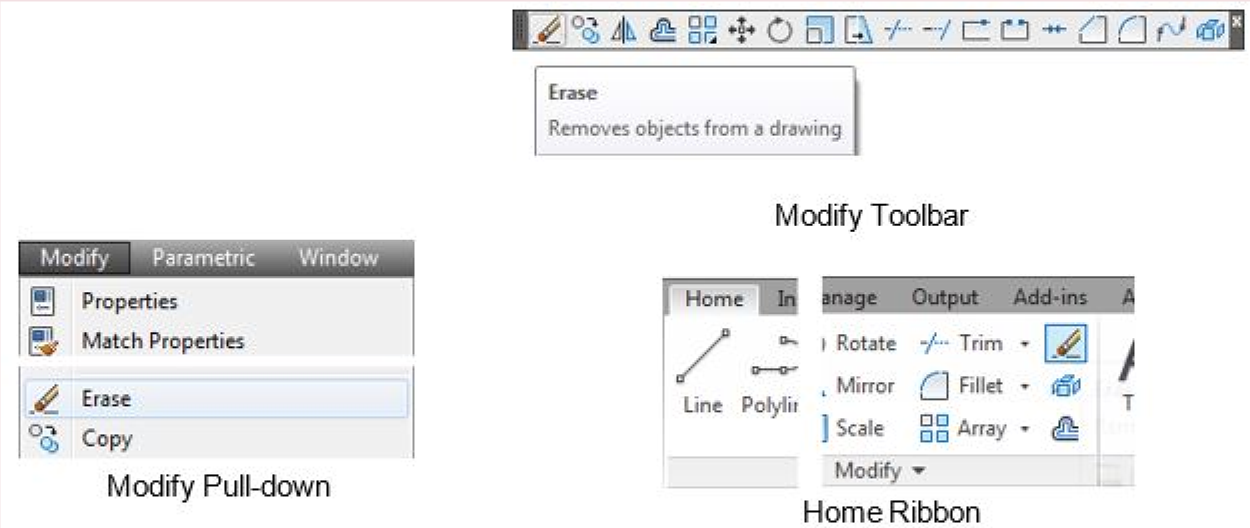
Using the Delete Key

When deleting objects using the Delete key, select the object or objects before pressing the key.

AutoCAD Command: ERASE

The ERASE command is used to permanently remove drawing objects from the drawing.

Shortcut: **E**


Erase
Removes objects from a drawing
Modify Toolbar
Modify
Parametric
Window
Properties
Match Properties
Erase
Copy
Modify Pull-down
Home
In
anage
Output
Add-ins
Line
Polylir
Rotate
Trim
Mirror
Fillet
Scale
Array
Modify
Home Ribbon

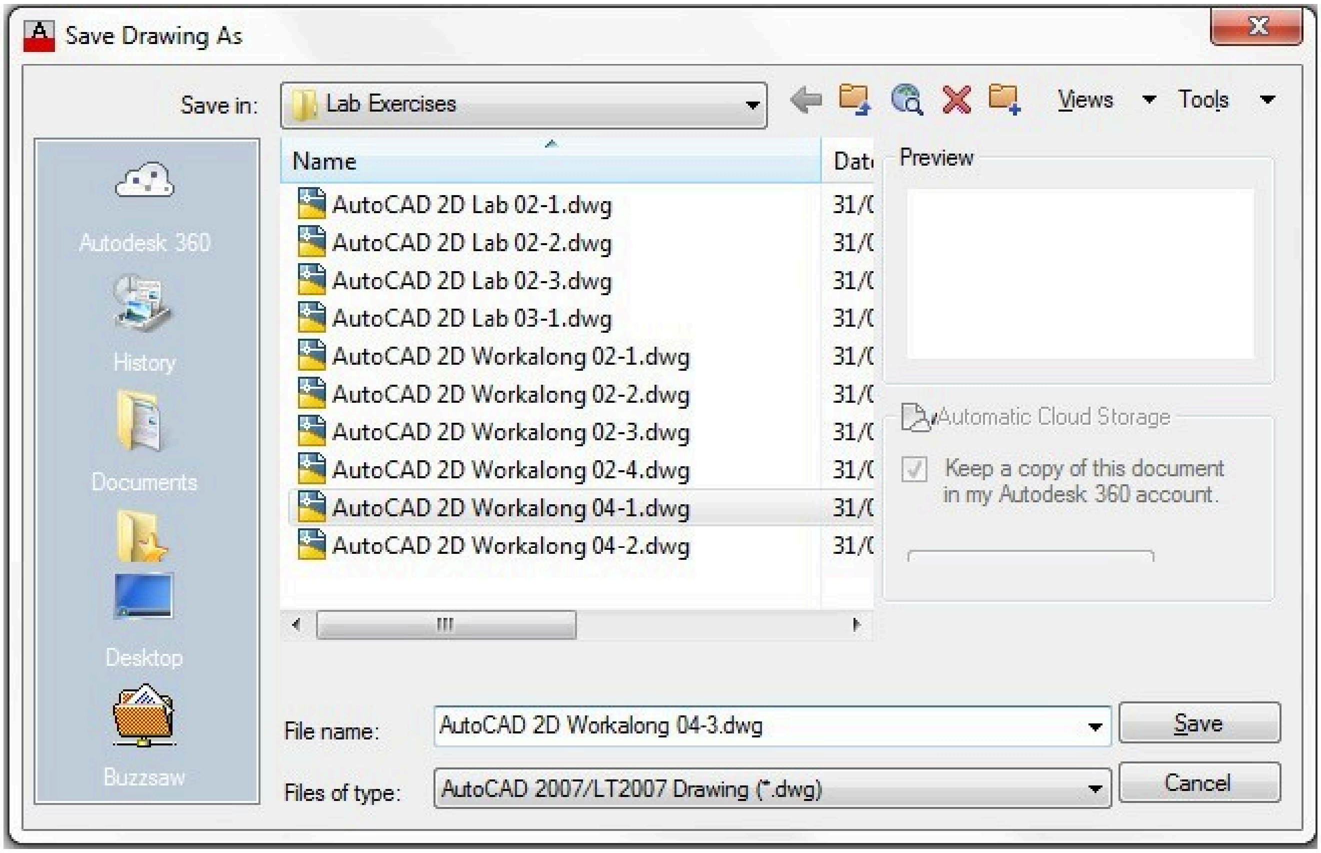
WORKALONG: Deleting Drawing Objects

Step 1

Open the drawing: AutoCAD 2D Workalong 04-1.

Step 2

Using the SAVEAS command, save the drawing with the name: AutoCAD 2D Workalong 04-3. (Figure Step 2)


Save Drawing As
Views
Tools
Name
Dat
Preview
Automatic Cloud Storage
Autodesk 360
History
Documents
Desktop
Buzzsaw
Save
Cancel

Save in: Lab Exercises

AutoCAD 2D Lab 02-1.dwg
AutoCAD 2D Lab 02-2.dwg
AutoCAD 2D Lab 02-3.dwg
AutoCAD 2D Lab 03-1.dwg
AutoCAD 2D Workalong 02-1.dwg
AutoCAD 2D Workalong 02-2.dwg
AutoCAD 2D Workalong 02-3.dwg
AutoCAD 2D Workalong 02-4.dwg
AutoCAD 2D Workalong 04-1.dwg
AutoCAD 2D Workalong 04-2.dwg

☑ Keep a copy of this document in my Autodesk 360 account.

File name: AutoCAD 2D Workalong 04-3.dwg

Files of type: AutoCAD 2007/LT2007 Drawing (*.dwg)

Figure Step 2

Step 3

Enter the ERASE command as shown below. Move the pickbox onto the horizontal line and select it by pressing the left mouse button. Move the cursor onto the vertical line and select it. Press the Enter key to execute the command. (Figure Step 3A, 3B, and 3C)

Command: **ERASE**

Select objects: 1 found

Select objects: 1 found, 2 total Select objects:

Command:

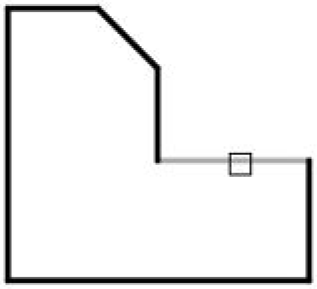


Figure Step 3A

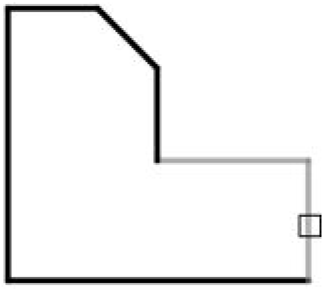


Figure Step 3B

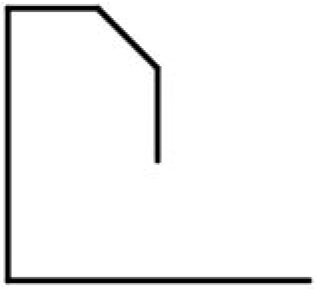


Figure Step 3C

AUTHOR'S COMMENTS: When you select a drawing object inside a command they will highlight and appear dashed.

MUST KNOW: The lastpoint is the last XY location that was used in an AutoCAD command. The lastpoint is very important to you when drawing in AutoCAD. AutoCAD remembers the lastpoint entered and saves it in the @ symbol. The @ symbol means ‘ The last absolute coordinate location ‘.

Step 4

To delete drawing objects without entering a command, move the pickbox onto the lines,

as shown in the figure, and select them by clicking the left mouse button. They will highlight and appear dashed with small blue squares on them. When the lines display as shown in the figure, press the Delete key on the keyboard. (Figure Step 4A and 4B)

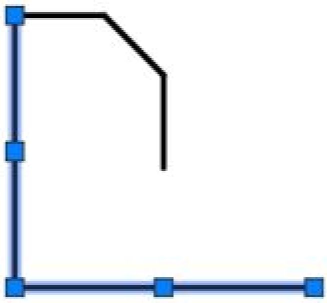


Figure Step 4A

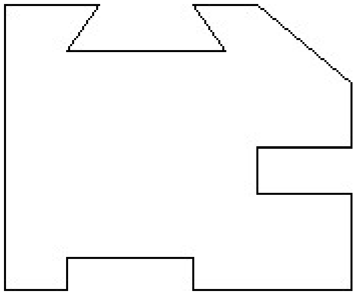


Figure Step 3A
Completed Drawing

AUTHOR’S COMMENTS: The small blue squares that display on a selected object when selected are called grips. You will learn more about them later in the book and how to use them in the AutoCAD 2D Advanced book.

AUTHOR’S COMMENTS: If you want to unselect one or more drawing objects, press the Esc key. Sometime you have to press it twice to totally unselect the selected objects.

Step 5

Save and close the drawing.

MUST KNOW: To enter a positive number in AutoCAD, enter the number only. Positive is the AutoCAD default. If the number is negative, the ‘ – ‘ sign must precede the number. For example, if the number is 4.0, enter 4. If the number is -4.0, enter -4.

Key Principles

Key Principles in Module 4

1. The Cartesian Coordinate System consists of two numbered lines crossing perpendicular to one another at their zero values. The horizontal axis is the X axis and the vertical axis is the Y axis.
2. Absolute Cartesian Coordinates are always referenced to the absolute origin 0,0.
3. The @ symbol means ‘ The last absolute coordinate location ‘ or sometimes called the lastpoint. Relative Cartesian Coordinates must be preceded with @ symbol.
4. To close the last line of a series of lines, enter either ‘ C ‘ (Close) or the absolute coordinate of the first point.
5. To delete existing drawing objects, you can either use the ERASE command or the Delete key.
6. Objects can either be selected before or after a command is entered.
7. To unselect a selected drawing object, press the Esc key. Sometimes it has to be pressed twice.

Lab Exercise 4-1

Time allowed: 30 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 04-1	2D English	Inches

Step 1

Start a new drawing using the template shown above.

Step 2

Save and name the drawing: AutoCAD 2D Lab 04-1 in the folder: CAD Courses/AutoCAD 2D/Lab Exercises.

Step 3

Using the LINE command, draw the object shown in the figure. (Figure Step 3A and 3B)

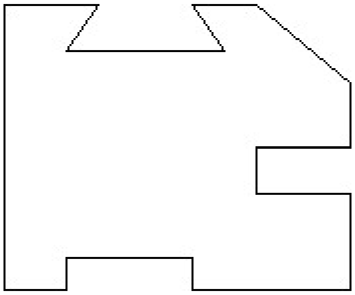


Figure Step 3A
Completed Drawing

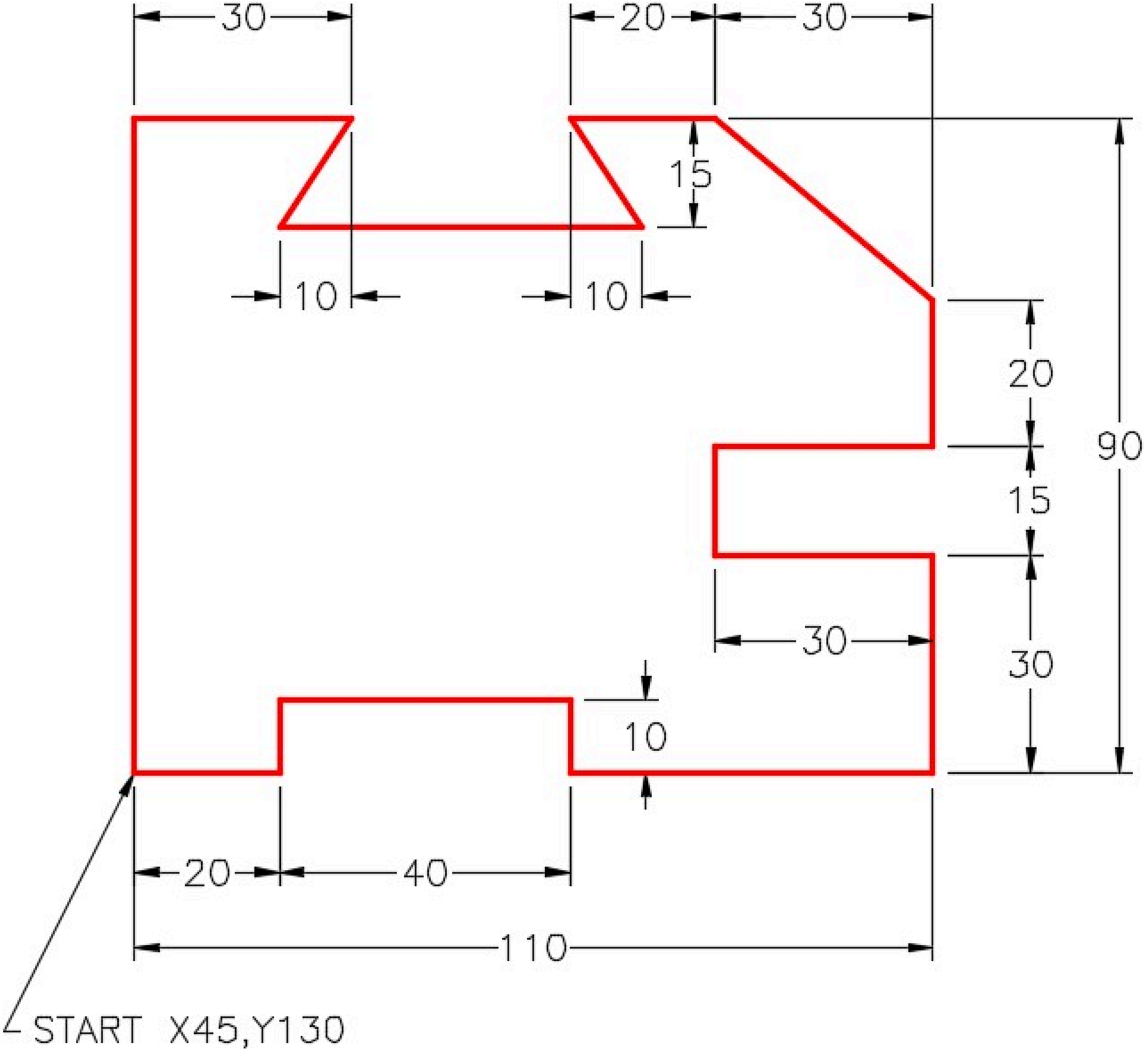
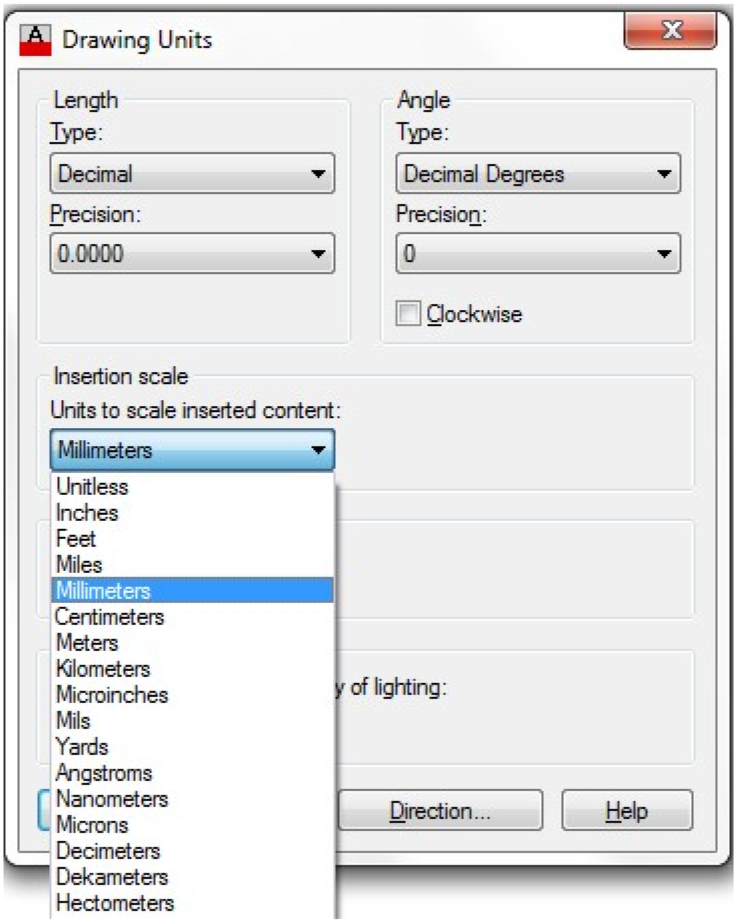

30
20
30
15
10
10
20
90
15
30
30
10
20
40
110
START X45,Y130

Figure Step 3B

Step 4

Enter the UNITS command. In the Drawing Units dialogue box, set the Insertion Units to Inches. (Figure Step 4)



Drawing Units

Length

Type: Decimal

Precision: 0.0000

Angle

Type: Decimal Degrees

Precision: 0

☐ Clockwise

Insertion scale

Units to scale inserted content: Millimeters

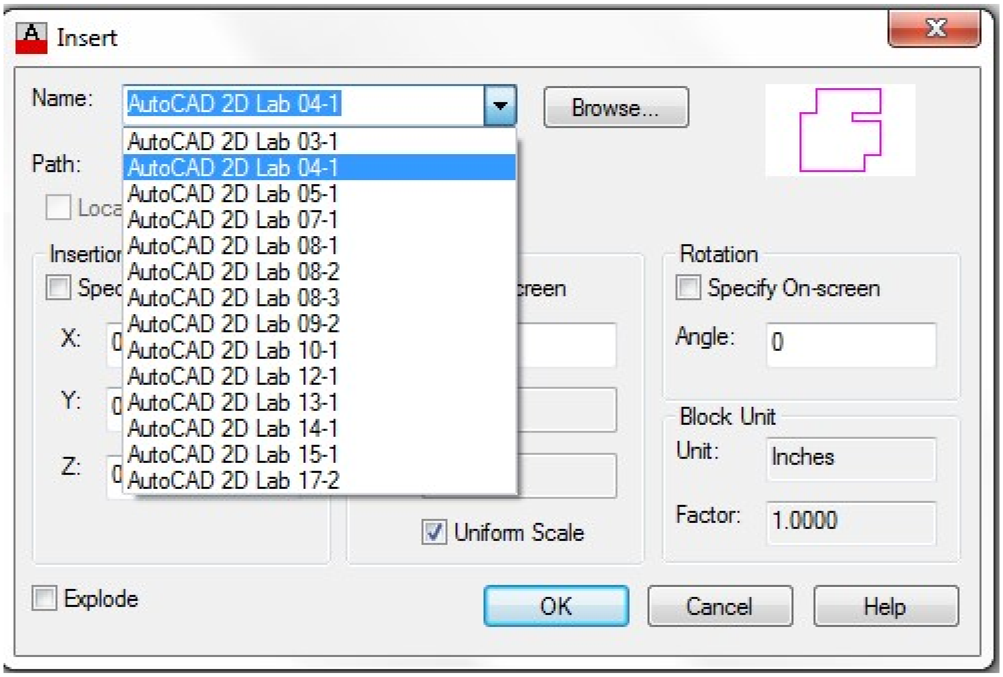
Unitless
Inches
Feet
Miles
Millimeters
Centimeters
Meters
Kilometers
Microinches
Mils
Yards
Angstroms
Nanometers
Microns
Decimeters
Dekameters
Hectometers
y of lighting:
Direction...
Help

Figure Step 4

AUTHOR’S COMMENTS: You can find the Insertion scale in the Units column in the table under the lab exercise title.

Step 5

Check your drawing with the key. The key name is the same as the drawing name. (Figure Step 5)



Insert

Name: AutoCAD 2D Lab 04-1

Angle: 0

Unit: Inches

Factor: 1.0000

☑ Uniform Scale

☐ Explode

☐ Specify On-screen

AutoCAD 2D Lab 03-1
AutoCAD 2D Lab 04-1
AutoCAD 2D Lab 05-1
AutoCAD 2D Lab 07-1
AutoCAD 2D Lab 08-1
AutoCAD 2D Lab 08-2
AutoCAD 2D Lab 08-3
AutoCAD 2D Lab 09-2
AutoCAD 2D Lab 10-1
AutoCAD 2D Lab 12-1
AutoCAD 2D Lab 13-1
AutoCAD 2D Lab 14-1
AutoCAD 2D Lab 15-1
AutoCAD 2D Lab 17-2
Browse...
Path:
Loca
Insertion
Spec
X: 0
Y: 0
Z: 0
creen
Rotation
Block Unit
OK
Cancel
Help

Figure Step 5

AUTHOR'S COMMENTS: If you have trouble doing this, redo Module 3.

Step 6

Your drawing should match the figure. (Figure Step 6)

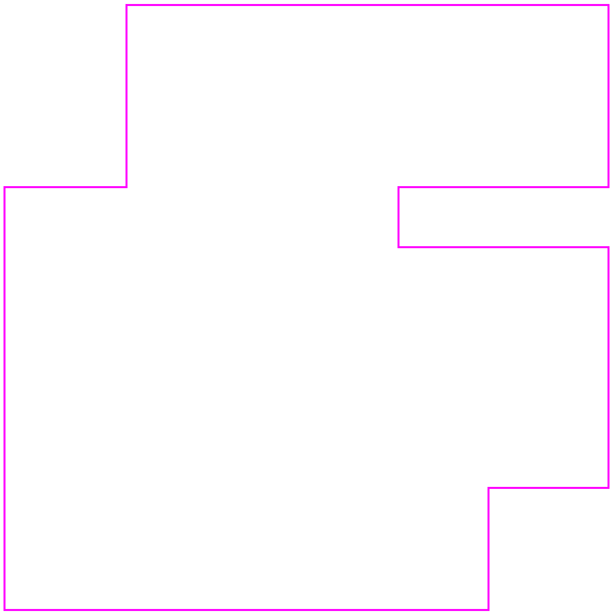


Figure Step 6

AUTHOR’S COMMENTS: Step 5 will insert a magenta colored overlay key on your drawing. If you see double objects or places where your objects and the magenta objects do not match, your drawing is inaccurate. If you only see one object, even though it may share the colors magenta and red, your drawing is accurate.

AUTHOR’S COMMENTS: If you have trouble completing this drawing the first try, do not be concerned. Start it over again, from scratch, until you can complete it. The more practice you get drawing, the easier it will get. When you get to Module 8, you will be taught how to fix a drawing so that you do not have to start over.

Step 7

Save and close the drawing.

AUTHOR’S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hint. If you get stuck and can not complete it on your own, use the following hint to help you.

Hint 1

See Figure Hint 1.

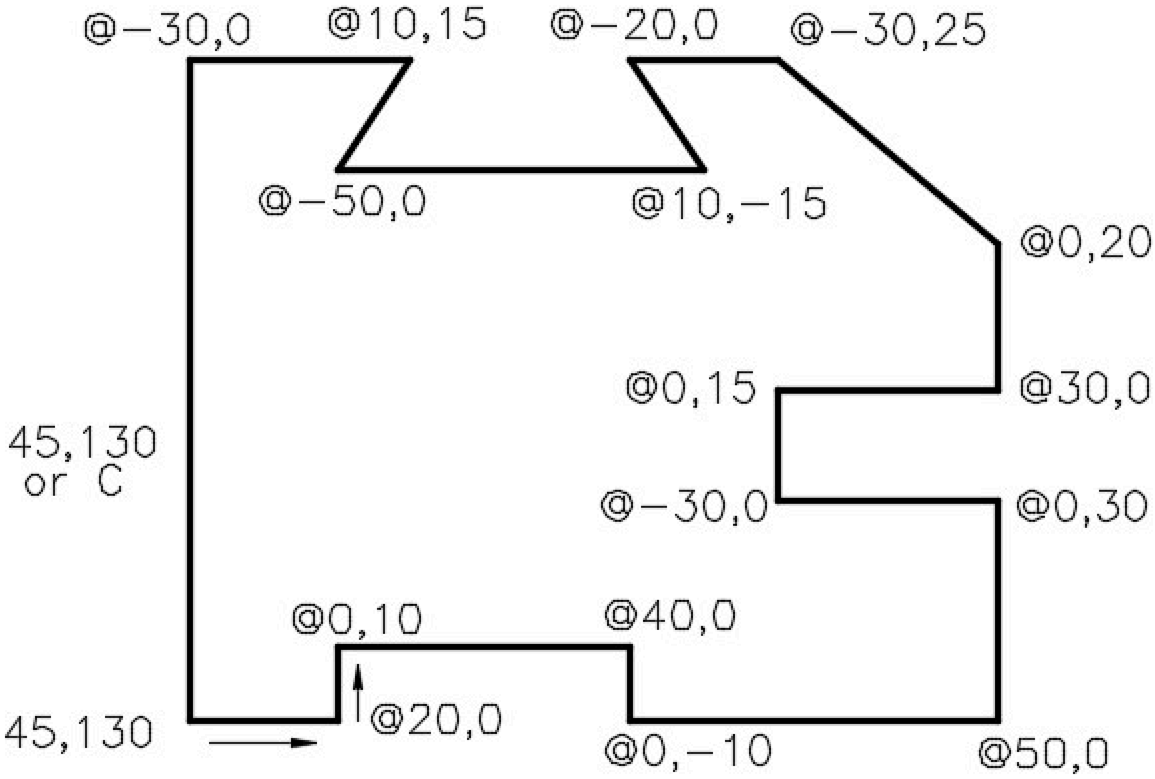

@−30,0
@10,15
@−20,0
@−30,25
@−50,0
@10,−15
@0,20
@0,15
@30,0
45,130
or C
@−30,0
@0,30
@0,10
@40,0
45,130
@20,0
@0,−10
@50,0

Figure Hint 1

Lab Exercise 4-2

Time allowed: 30 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 04-2	2D Metric	Millimeters

Step 1

Start a new drawing using the template shown above.

Step 2

Save and name the drawing: AutoCAD 2D Lab 04-2 in the folder: CAD Courses/AutoCAD 2D/Lab Exercises.

Step 3

Using the LINE command, draw the object shown in the figure. (Figure Step 3A and 3B)

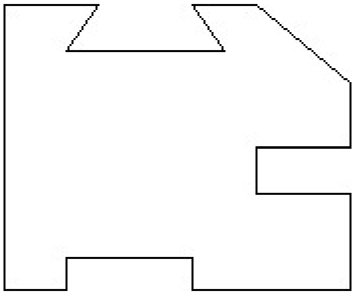


Figure Step 3A
Completed Drawing

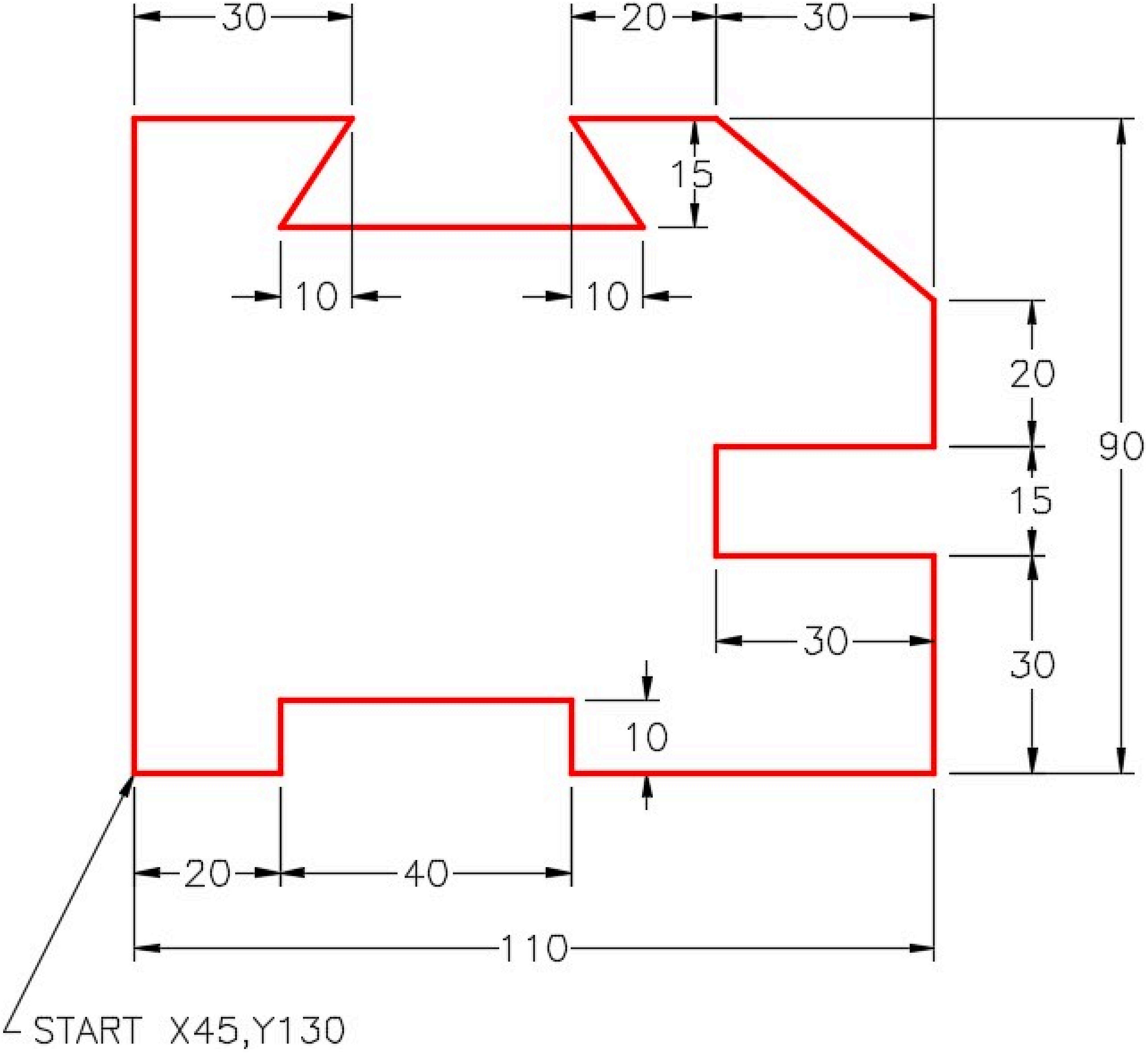
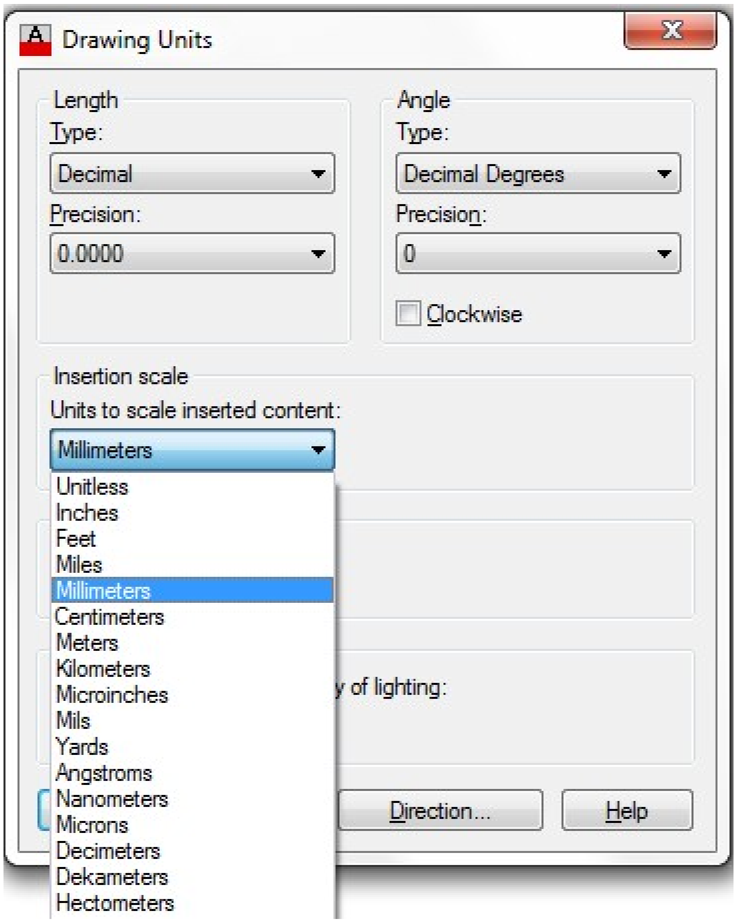

30
20
30
15
10
10
20
90
15
30
30
10
20
40
110
START X45,Y130

Figure Step 3B

Step 4

Enter the UNITS command. In the Drawing Units dialogue box, set the Insertion Units to Millimeters. (Figure Step 4)



Drawing Units

Length

Type: Decimal

Precision: 0.0000

Angle

Type: Decimal Degrees

Precision: 0

☐ Clockwise

Insertion scale

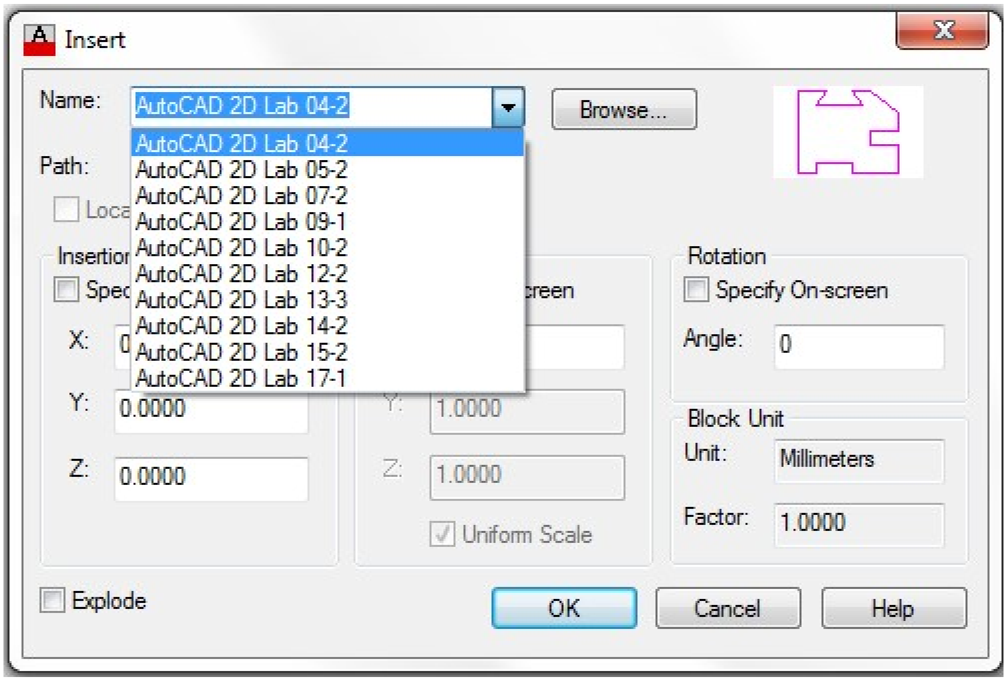
Units to scale inserted content: Millimeters

Unitless
Inches
Feet
Miles
Millimeters
Centimeters
Meters
Kilometers
Microinches
Mils
Yards
Angstroms
Nanometers
Microns
Decimeters
Dekameters
Hectometers
y of lighting:
Direction...
Help

Figure Step 4

Step 5

Check your drawing with the key. The key name is the same as the drawing name. (Figure Step 5)



Insert

Name: AutoCAD 2D Lab 04-2

AutoCAD 2D Lab 04-2
AutoCAD 2D Lab 05-2
AutoCAD 2D Lab 07-2
AutoCAD 2D Lab 09-1
AutoCAD 2D Lab 10-2
AutoCAD 2D Lab 12-2
AutoCAD 2D Lab 13-3
AutoCAD 2D Lab 14-2
AutoCAD 2D Lab 15-2
AutoCAD 2D Lab 17-1

Path:

☐ Loca

Insertion

☐ Spec

X: 0

Y: 0.0000

Z: 0.0000

reen

Y: 1.0000

Z: 1.0000

☑ Uniform Scale

Rotation

☐ Specify On-screen

Angle: 0

Block Unit

Unit: Millimeters

Factor: 1.0000

☐ Explode

Figure Step 5

AUTHOR’S COMMENTS: If you have trouble doing this, redo Module 3.

Step 6

Your drawing should match the figure. (Figure Step 6)

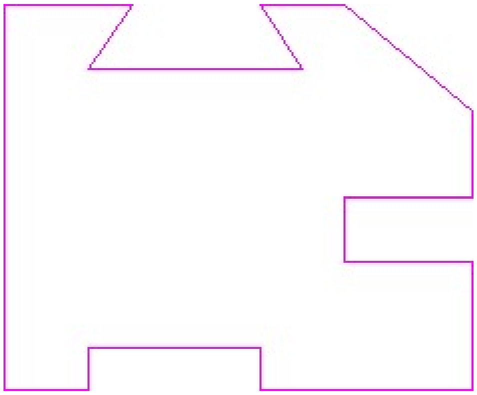


Figure Step 6

AUTHOR’S COMMENTS: Step 5 will insert a magenta colored overlay key on your drawing. If you see double objects or places where your objects and the magenta objects do not match, your drawing is inaccurate. If you only see one object, even though it may share the colors magenta and red, your drawing is accurate.

AUTHOR'S COMMENTS: If you have trouble completing this drawing the first try, do not be concerned. Start it over again, from scratch, until you can complete it. The more practice you get drawing, the easier it will get. When you get to Module 8, you will be taught how to fix a drawing so you do not have to start over.

Step 7

Save and close the drawing.

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hint. If you get stuck and can not complete it on your own, use the following hint to help you.

Hint 1

See Figure Hint 1.

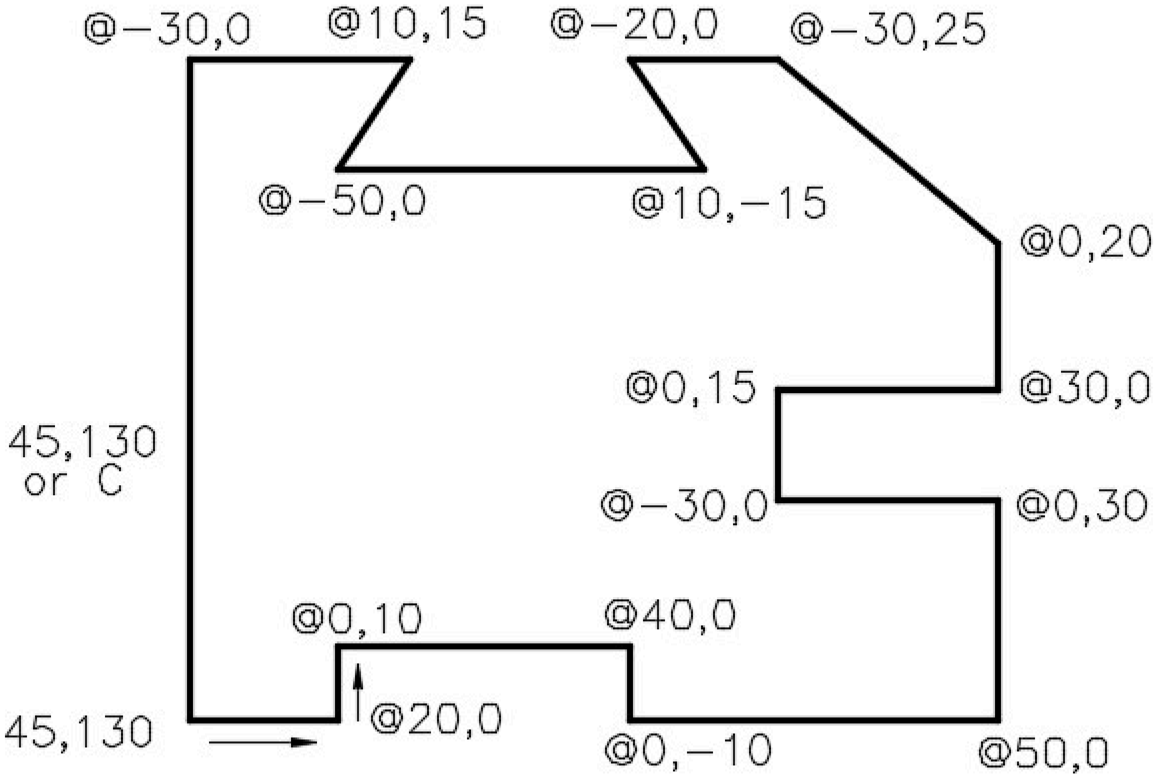

@-30,0
@10,15
@-20,0
@-30,25
@-50,0
@10,-15
@0,20
@0,15
@30,0
45,130
or C
@-30,0
@0,30
@0,10
@40,0
45,130
@20,0
@0,-10
@50,0

Figure Hint 1

Lab Exercise 4-3

Time allowed: 30 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 04-3	2D English	Inches

Step 1

Start a new drawing using the template shown above.

Step 2

Save and name the drawing: AutoCAD 2D Lab 04-3 in the folder: CAD Courses/AutoCAD 2D/Lab Exercises.

Step 3

Using the LINE command, draw the object shown in the figure. (Figure Step 3A and 3B)

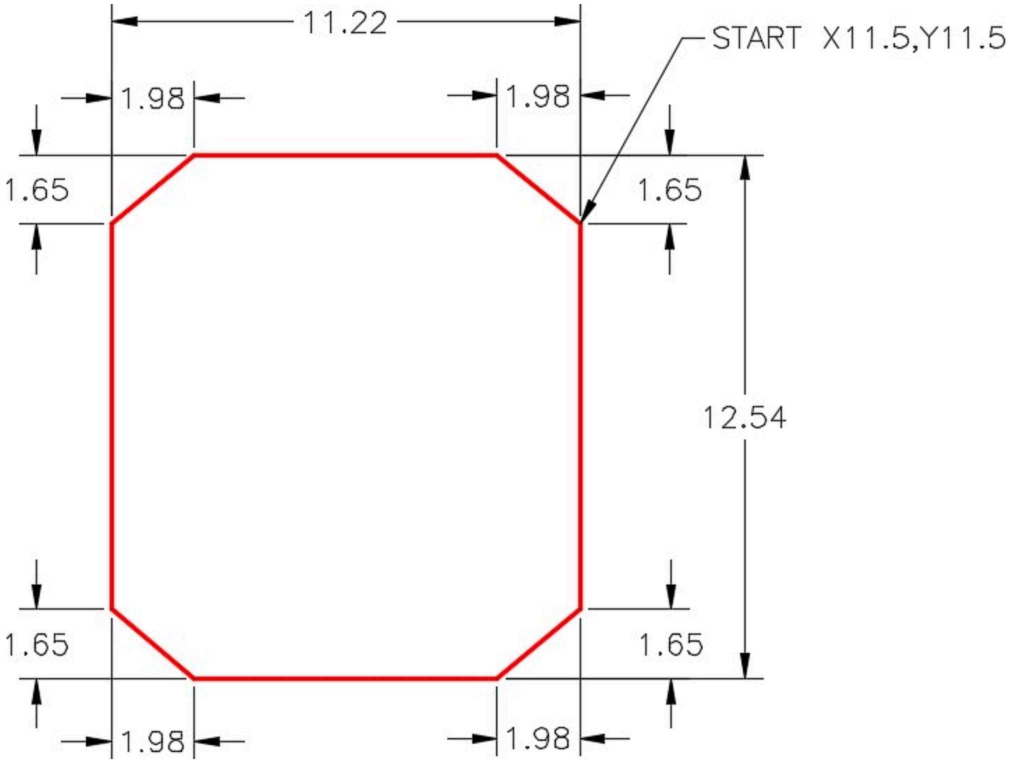

11.22
START X11.5,Y11.5
1.98
1.98
1.65
1.65
12.54
1.65
1.65
1.98
1.98

Figure Step 3A

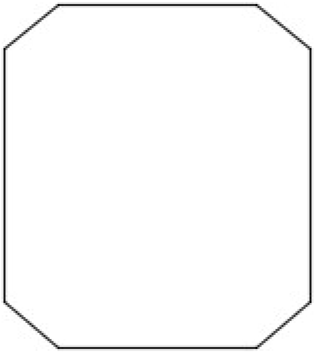


Figure Step 3B
Completed Drawing

Step 4

Enter the UNITS command. In the Drawing Units dialogue box, set the Insertion Units to Inches.

Step 5

Check your drawing with the key.

Step 6

Save and close the drawing.

Module 5: Layers

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe AutoCAD's layering system.
2. Apply the LAYER command to create, toggle on/off, thaw/freeze, lock/unlock, select color, and set the current layer in the current drawing.

Layers

AutoCAD's layering system is one of its most important features. *Layers* allow you to organize, segregate, set the drawing object's color and linetype, control the visibility, and manage plotting of the drawing.

Think of layering as being able to draw on transparent overlays with each overlay containing relevant or grouped objects as shown in Figure 5-1. When all of the overlays are stacked together as shown in Figure 5-2, you can see and work on all of them. At any time, while you are working on drawing or printing it, you can control the layer's visibly.

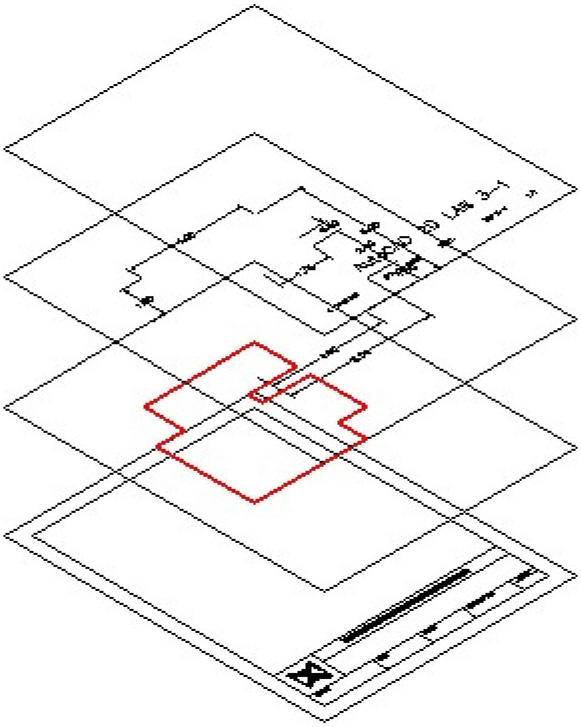


Figure 5-1 Layers Seen as Overlays

There is no limit to the number of layers that can be created in an AutoCAD drawing. Layers can be named by you or you can allow AutoCAD to automatically name them. A color, linetype, lineweight, and plot style can also be assigned. In this module, only naming and assigning a color to layers will be taught.

In addition to what was just discussed, the visibility of each layer can be controlled by toggling layers off or on and freezing or thawing them. A layer can also be locked to prevent objects on them from being modified.

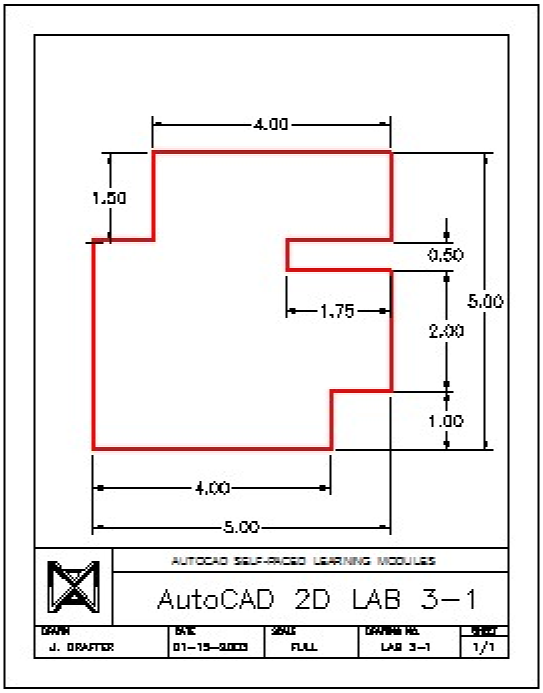

4.00
1.50
0.50
5.00
1.75
2.00
1.00
4.00
5.00
AUTOCAD SELF-PACED LEARNING MODULES
AutoCAD 2D LAB 3-1
J. DRAFTER
FULL
LAB 3-1
1/1

Figure 5-2 Stacked Layers

AutoCAD's Special Layer

AutoCAD has a *special layer*, named 0 (the number zero). Each new drawing created in AutoCAD will automatically contain this layer. It cannot be deleted or renamed. The importance of layer 0 will be discussed further in future modules.

The Current Layer

The *current layer* is the layer that AutoCAD will place all newly created drawing objects on. One layer is current at all times. The current layer can be turned off but never frozen. You can change which layer is the current layer at any time.

ByLayer vs ByObject

AutoCAD assigns the color, linetype, and lineweight properties to drawing objects either ByLayer or ByObject. In the AutoCAD 2D book, all work is done ByLayer. An easy way for you to check if you are working ByLayer is to look at the first three boxes in Properties menu as shown in Figures 5-3 and 5-4.

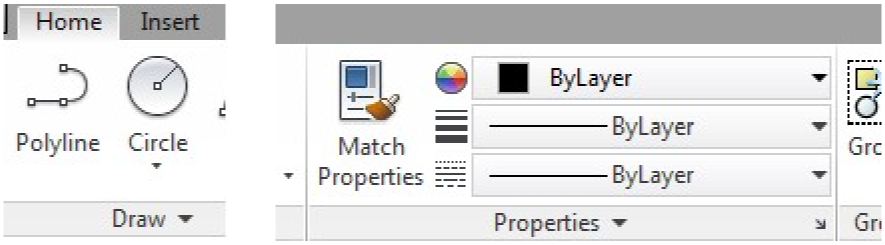


Figure Step 5-3 Home Ribbon

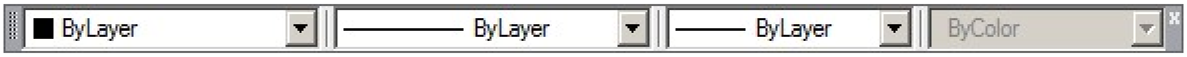


Figure 5-4 Properties Toolbar

The three properties boxes should always display the words ByLayer as shown in the figure.

Simply, *ByLayer* means that when a drawing object is inserted into the drawing or an existing drawing object is regenerated, it looks in the drawing's database to see what color, linetype, and lineweight it should display. If it is set ByLayer, it checks what layer the drawing object resides on and displays the color, linetype, and lineweight assigned to that layer. Most company standards require the color, linetype, and lineweight properties of their drawings to be drawn Bylayer.

It is important to know the different between a layer that is Off and a layer that is Frozen. A layer that is Off does NOT display but WILL regenerate. A layer that is Frozen does NOT display and will NOT regenerate. The importance of this will become apparent in future AutoCAD 2D modules.

Study the following four properties and their symbols:

On/Off:

Toggles the drawing object's display on that layer on or off. In other words, visible or not visible. 💡 💡

Freeze/Thaw:

Toggles the layer to be frozen or thawed. The drawing objects on a frozen layer are not visible and do not regenerate. The objects on a thawed layer are visible and will regenerate. ☀ ❄

Lock/Unlock:

Toggles the layer locked or unlocked. Drawing objects on a locked layer are visible but cannot be modified or deleted. Drawing objects on an unlocked layer are visible and can be modified or deleted.
🔓
🔒

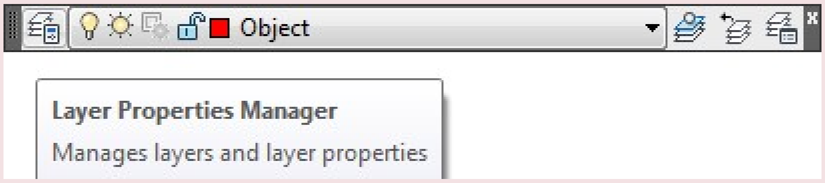
Color:

Sets the color for the layer. All drawing objects on that layer, with the property ByLayer, will display the color assigned to the layer. 🟥 red
🟦 blue

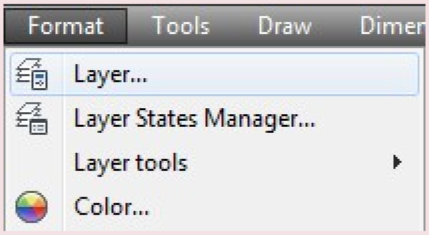
AutoCAD Command: LAYER

The LAYER command is used to open the Layer Property Manager window.

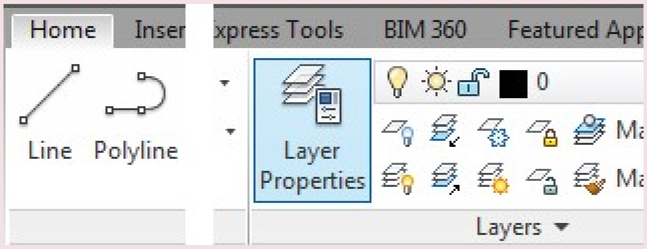
Shortcut: **LA**


Object
Layer Properties Manager
Manages layers and layer properties

Layer Toolbar


Format
Tools
Draw
Dimer
Layer...
Layer States Manager...
Layer tools
Color...

Format Pull-down


Home
Inser
xpress Tools
BIM 360
Featured App
Line
Polyline
Layer Properties
0
Ma
Ma
Layers

Home Ribbon

MUST KNOW: The current layer can be turned off but CANNOT be frozen. To freeze it, set one of the other layers as the current layer.

WORKALONG: Working with Layers

Step 1

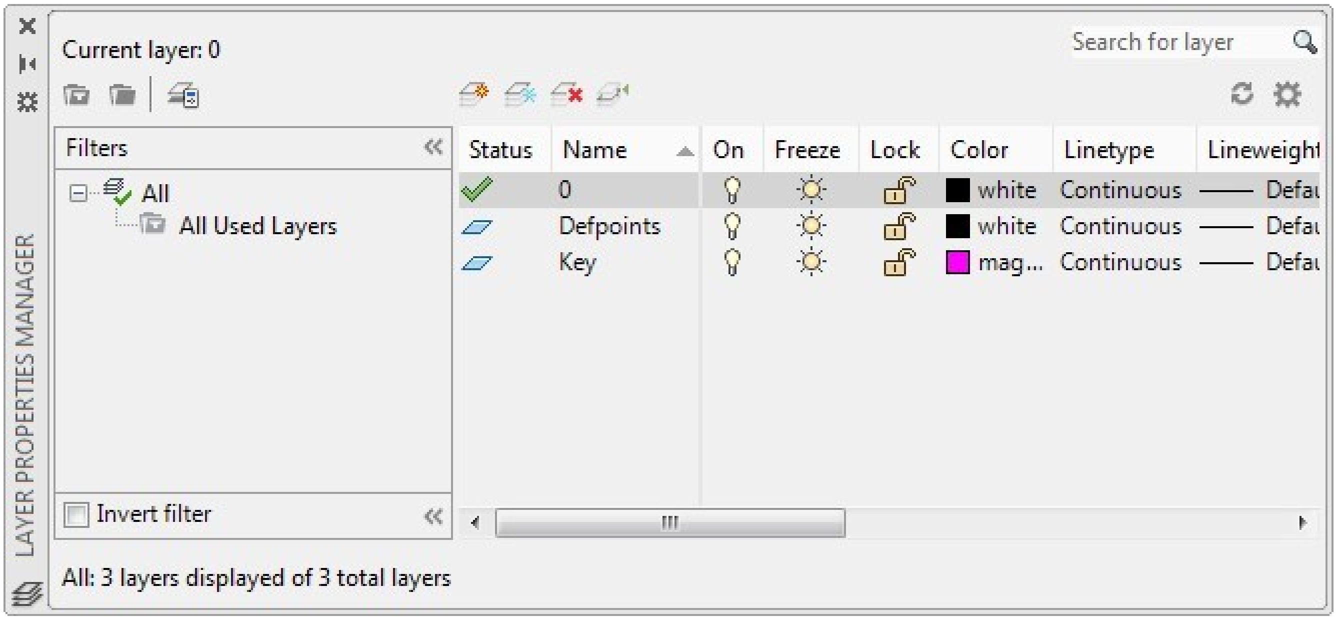
Using the NEW command, start a new drawing using the template: 2D English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 05-1. Save it in the folder: CAD Courses/AutoCAD 2D/Lab Exercises

Step 3

Enter the LAYER command to open the Layer Properties Manager window. (Figure Step 3)



Current layer: 0

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight
✓	0	💡	☀	🔓	■ white	Continuous	—— Defau
▱	Defpoints	💡	☀	🔓	■ white	Continuous	—— Defau
▱	Key	💡	☀	🔓	■ mag...	Continuous	—— Defau

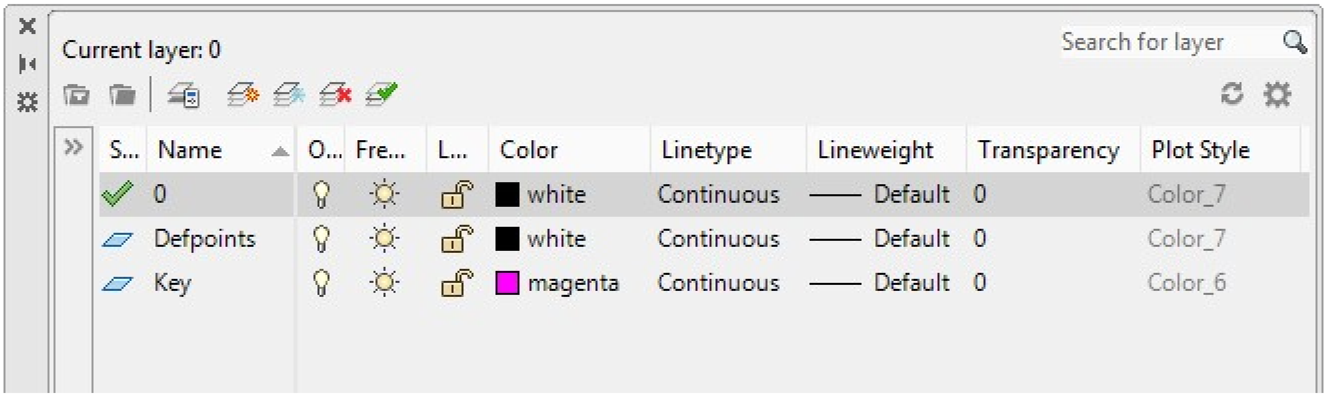
All: 3 layers displayed of 3 total layers

Figure Step 3

AUTHOR'S COMMENTS: Take note of layer 0 (zero). It exists in all AutoCAD drawings and cannot be deleted or renamed. The other two layers are part of the template drawings, ignore them for now. Note the Check Mark icon beside layer 0. This indicates it is the current layer.

Step 4

Click the small arrow in the upper right of the Filters area to close it. (Figure Step 4)



Current layer: 0

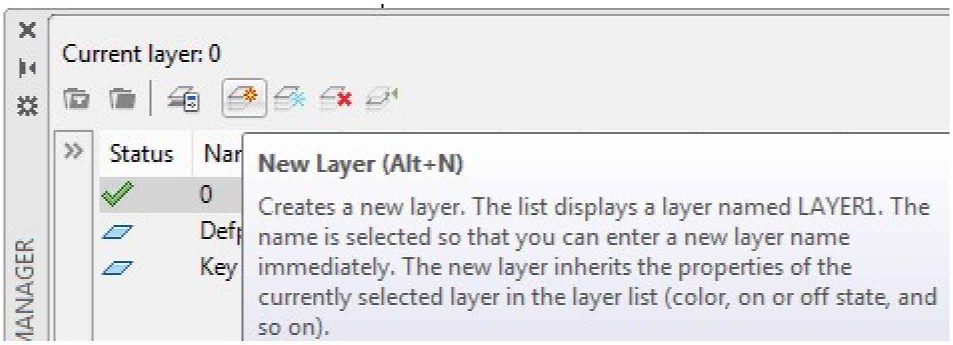
S...	Name	O...	Fre...	L...	Color	Linetype	Lineweight	Transparency	Plot Style
✓	0	💡	☀	🔓	■ white	Continuous	—— Default	0	Color_7
▱	Defpoints	💡	☀	🔓	■ white	Continuous	—— Default	0	Color_7
▱	Key	💡	☀	🔓	■ magenta	Continuous	—— Default	0	Color_6

Figure Step 4

AUTHOR'S COMMENTS: You can leave the Filter Area closed for the duration of this book. Layer filters are taught in the AutoCAD 2D Advanced book.

Step 5

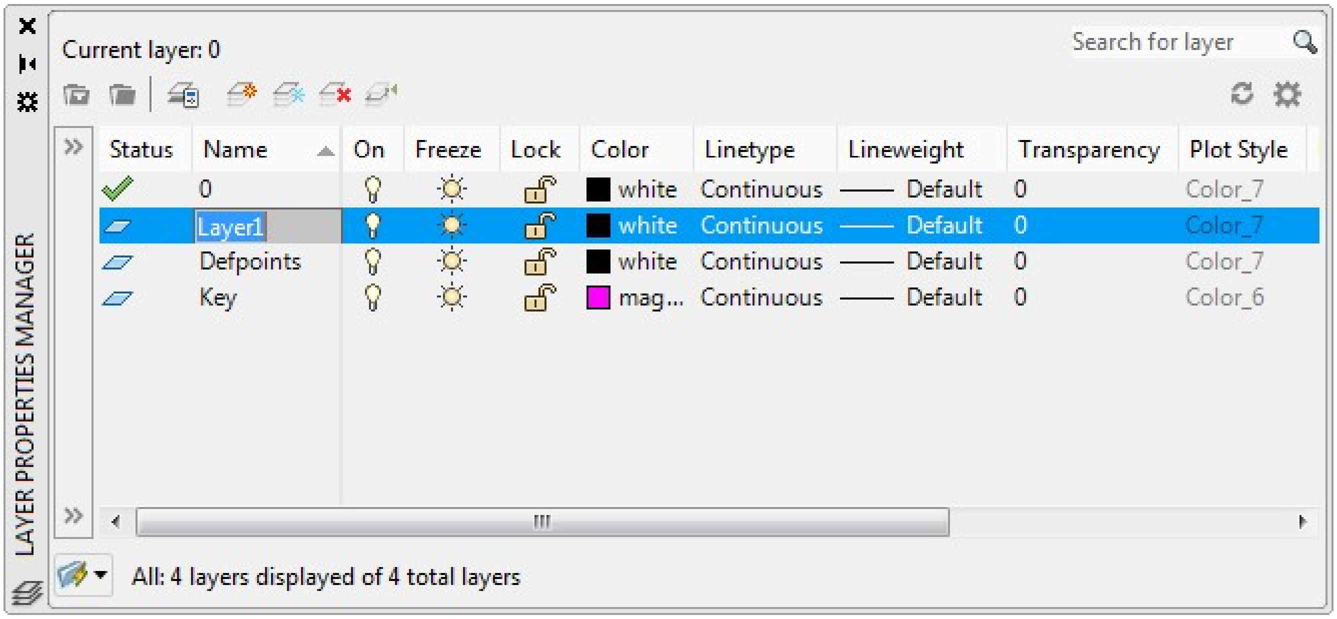
Click the New Layer icon. AutoCAD will create a new layer and automatically name it: Layer1. (Figure Step 5A and 5B)


Current layer: 0
Status
Nar
0
Defp
Key
MANAGER

New Layer (Alt+N)

Creates a new layer. The list displays a layer named LAYER1. The name is selected so that you can enter a new layer name immediately. The new layer inherits the properties of the currently selected layer in the layer list (color, on or off state, and so on).

Figure Step 5A


LAYER PROPERTIES MANAGER
Current layer: 0
Search for layer

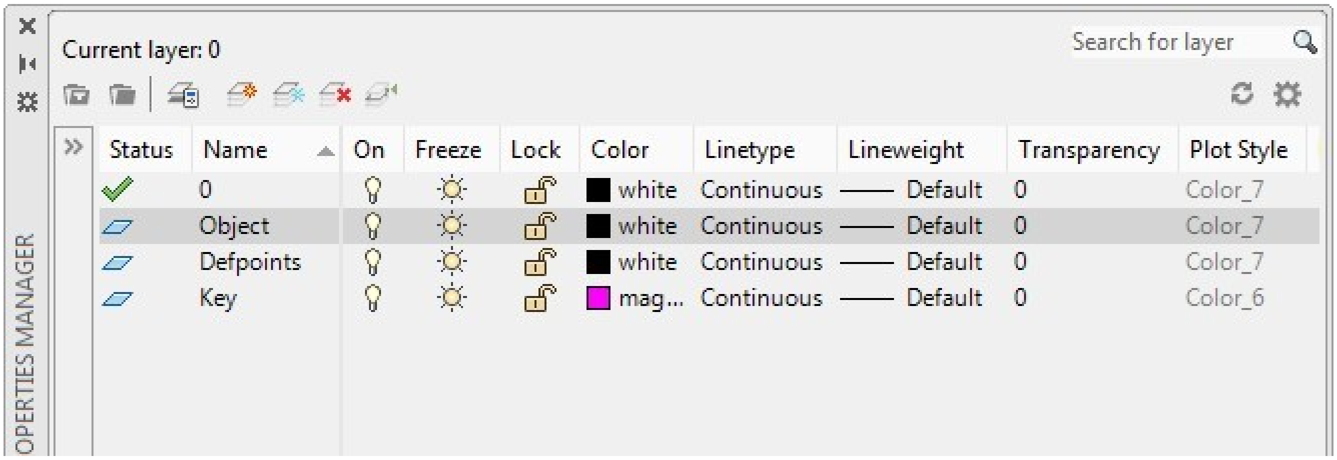
Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight	Transparency	Plot Style
✓	0				■ white	Continuous	—— Default	0	Color_7
	Layer1				■ white	Continuous	—— Default	0	Color_7
	Defpoints				■ white	Continuous	—— Default	0	Color_7
	Key				■ mag...	Continuous	—— Default	0	Color_6

All: 4 layers displayed of 4 total layers

Figure Step 5B

Step 6

Rename the layer to: Object. (Figure Step 6.)



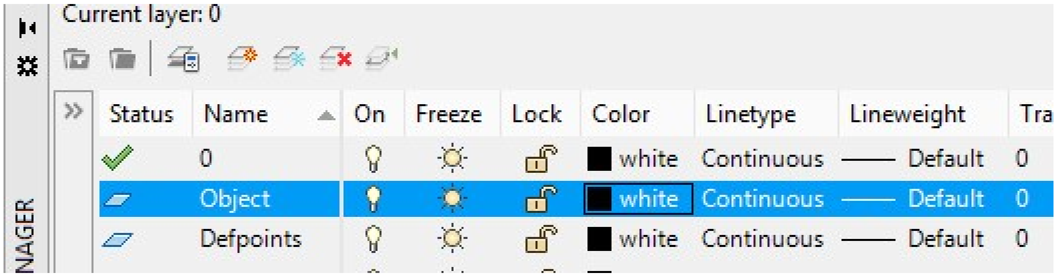
Current layer: 0

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight	Transparency	Plot Style
✓	0				■ white	Continuous	—— Default	0	Color_7
	Object				■ white	Continuous	—— Default	0	Color_7
	Defpoints				■ white	Continuous	—— Default	0	Color_7
	Key				■ mag...	Continuous	—— Default	0	Color_6

Figure Step 6

Step 7

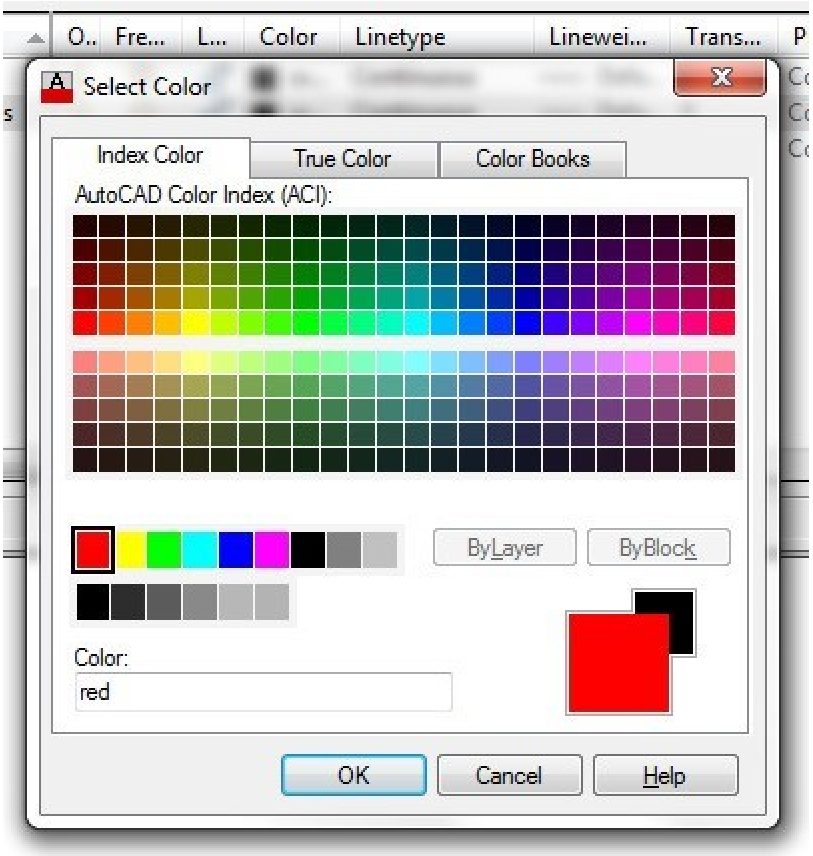
Set the color of layer: Object by clicking the color name white. This will open the Select Color dialogue box. Enable the Index Color tab. Select the color red by clicking it or enter the name in the Color box. Click OK to close the dialogue box. (Figure Step 7A and 7B)



Current layer: 0

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight	Tra
✓	0				■ white	Continuous	—— Default	0
	Object				■ white	Continuous	—— Default	0
	Defpoints				■ white	Continuous	—— Default	0

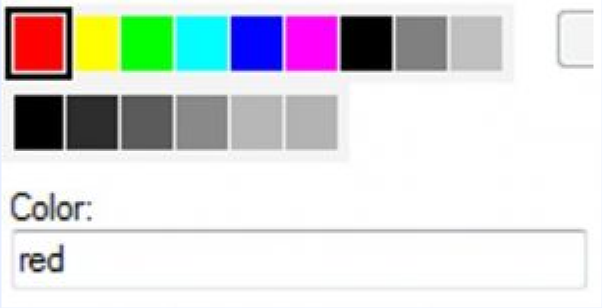
Figure Step 7A


O..
Fre...
L...
Color
Linetype
Linewei...
Trans...
Select Color
Index Color
True Color
Color Books
AutoCAD Color Index (ACI):
ByLayer
ByBlock
OK
Cancel
Help

Color:
red

Figure Step 7B

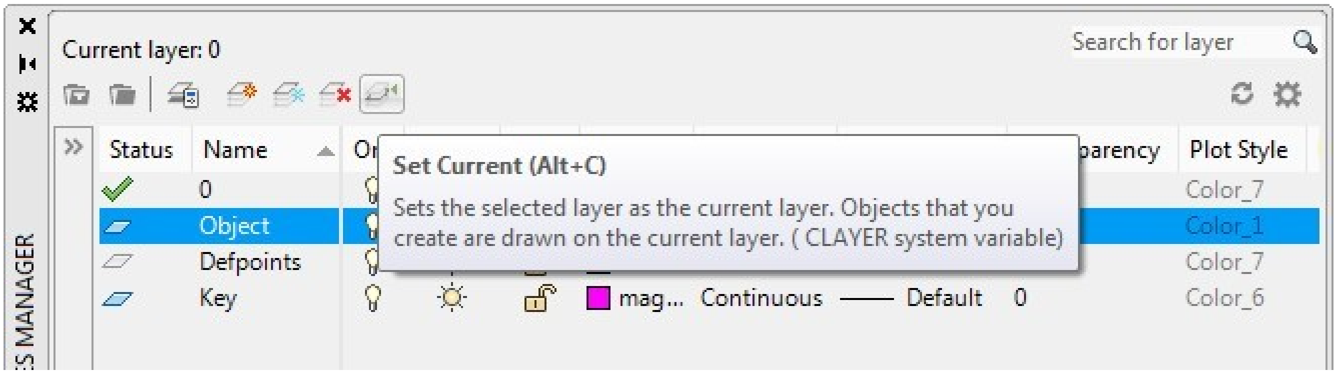
MUST KNOW: Unless otherwise instructed while working in the AutoCAD 2D book, select the colors from the Standard Color Palette in the Index Color tab as shown in the figure on the right. The colors included in this palette are the common colors and are named. White and black are the same color to AutoCAD. White will display black on a white background and black will display white on a black background.



Color:
red

Step 8

Click layer: Object to highlight it. Click the Set Current icon. (Figure Step 8A and 8B)



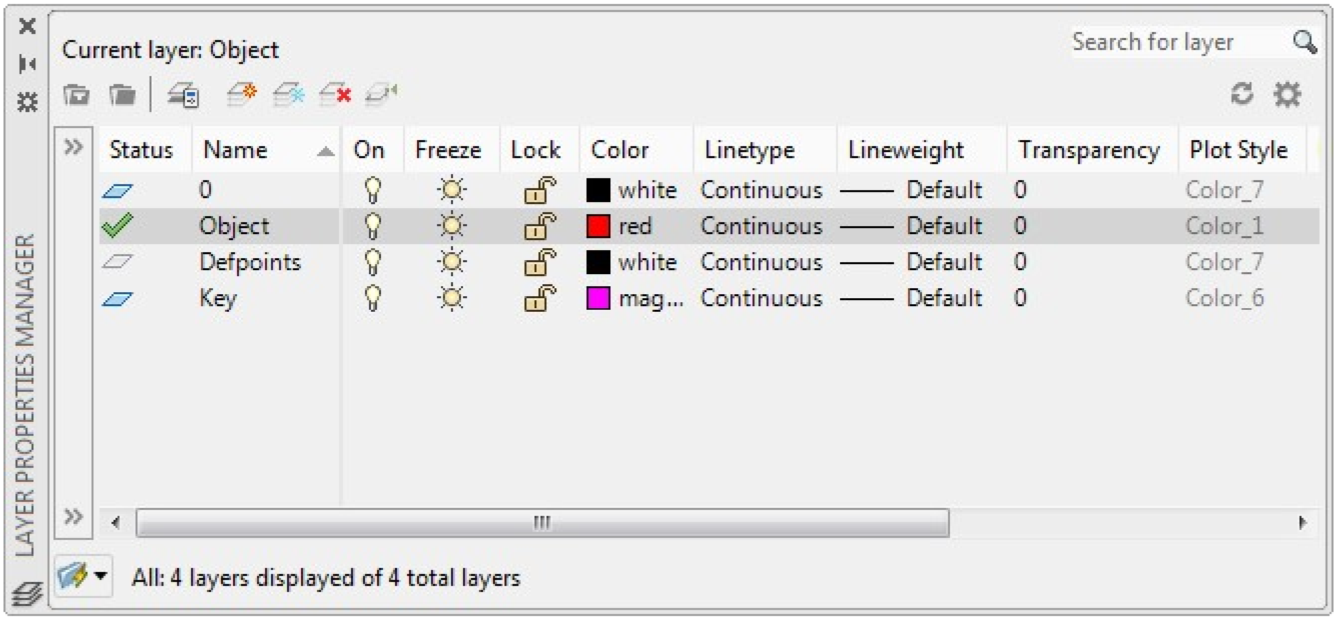
Current layer: 0

Set Current (Alt+C)

Sets the selected layer as the current layer. Objects that you create are drawn on the current layer. (CLAYER system variable)

Status	Name	Plot Style
✓	0	Color_7
▱	Object	Color_1
▱	Defpoints	Color_7
▱	Key	Color_6

Figure Step 8A



Current layer: Object

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight	Transparency	Plot Style
▱	0	○	☼	🔓	■ white	Continuous	—— Default	0	Color_7
✓	Object	○	☼	🔓	🟥 red	Continuous	—— Default	0	Color_1
▱	Defpoints	○	☼	🔓	■ white	Continuous	—— Default	0	Color_7
▱	Key	○	☼	🔓	■ mag...	Continuous	—— Default	0	Color_6

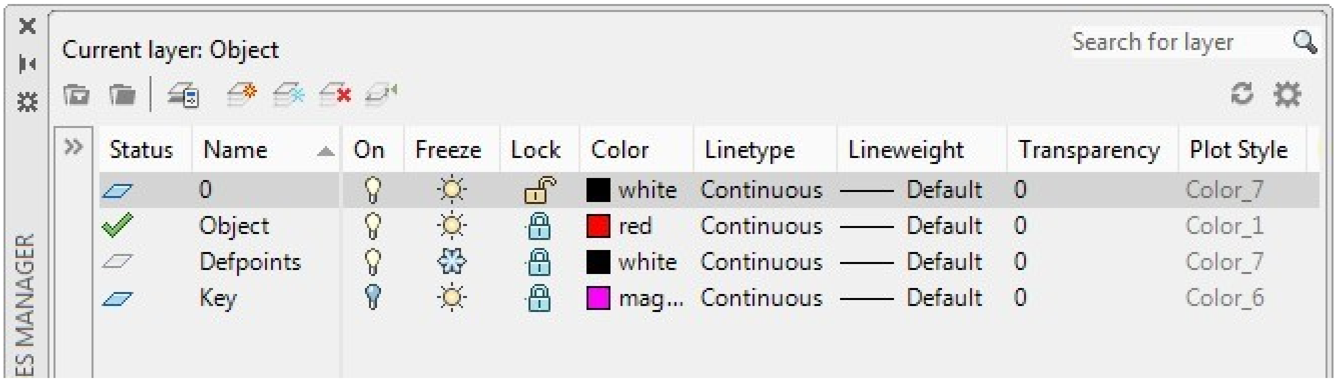
All: 4 layers displayed of 4 total layers

Figure Step 8B

AUTHOR'S COMMENTS: The Green Check Mark icon in the Status column indicates that the current layer is Object as you can see in Figure Step 8B.

Step 9

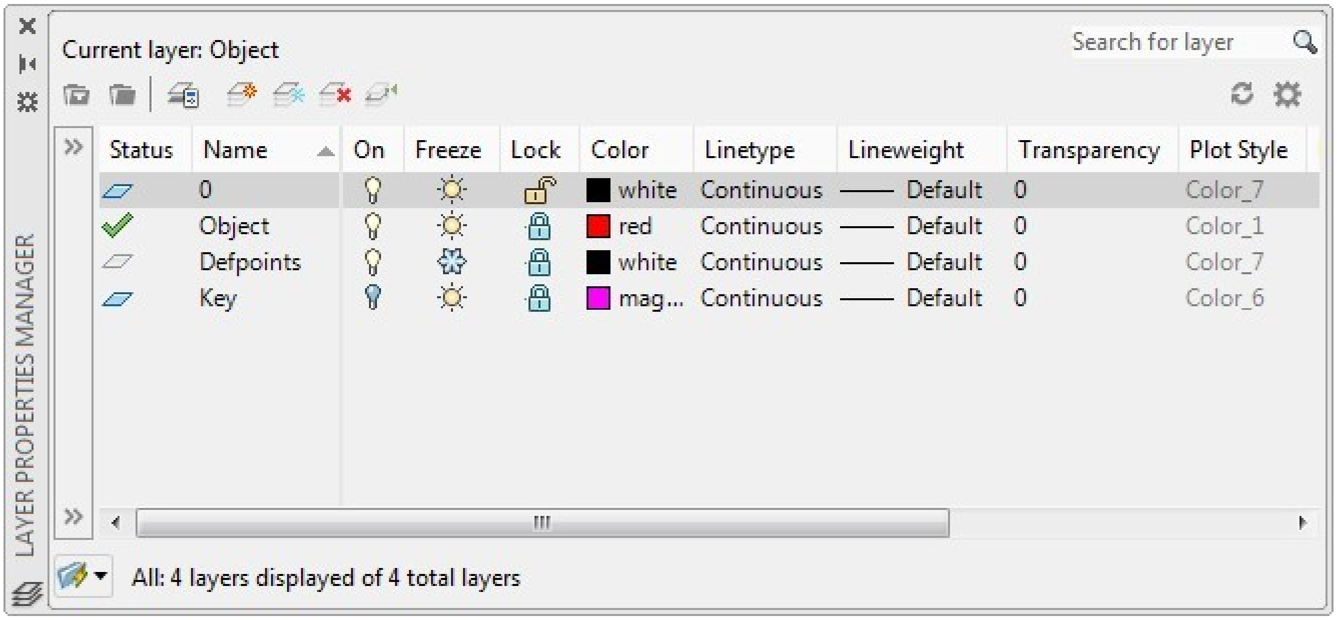
Toggle the on/off, thaw/frozen, and lock/unlock properties of the layers by clicking the icons to match the figure. Close the window by clicking the X in the top left corner. (Figure Step 9A and 9B)



Current layer: Object

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight	Transparency	Plot Style
▱	0	💡	☀	🔓	■ white	Continuous	—— Default	0	Color_7
✓	Object	💡	☀	🔒	🟥 red	Continuous	—— Default	0	Color_1
▱	Defpoints	💡	❄	🔒	■ white	Continuous	—— Default	0	Color_7
▱	Key	💡	☀	🔒	■ mag...	Continuous	—— Default	0	Color_6

Figure Step 9A



Current layer: Object

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight	Transparency	Plot Style
▱	0	💡	☀	🔓	■ white	Continuous	—— Default	0	Color_7
✓	Object	💡	☀	🔒	🟥 red	Continuous	—— Default	0	Color_1
▱	Defpoints	💡	❄	🔒	■ white	Continuous	—— Default	0	Color_7
▱	Key	💡	☀	🔒	■ mag...	Continuous	—— Default	0	Color_6

All: 4 layers displayed of 4 total layers

Figure Step 9B

AUTHOR’S COMMENTS: Once you have created the layers in the drawing, you can use the Layer toolbar menu, to change the current layer or to toggle the layer properties. This is much faster than using the Layer Properties Manager window. The Layer toolbar’s display should always be enabled and located across the top left side of the Graphic window.

Step 10

Take note how the current layer will display in the Layer toolbar. In this case, it is Object. (Figure Step 10)

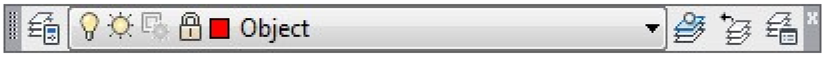


Figure Step 10

Step 11

Pull down the layer list by clicking the small triangle on the right side. Change the current layer to 0 by clicking the layer name. (Figure Step 11A and 11B)

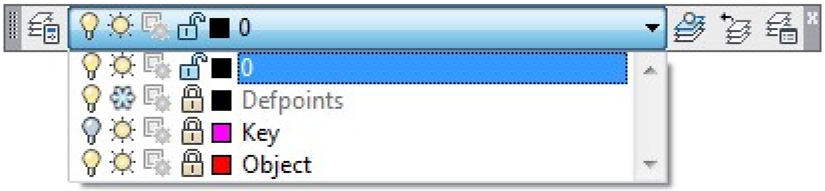

0
0
Defpoints
Key
Object

Figure Step 11A

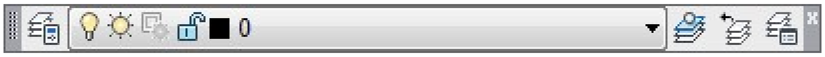


Figure Step 11B

Step 12

Pull down the layer list again and this time toggle the layer properties so that all four layer are on, thawed, and unlocked. (Figure Step 12)

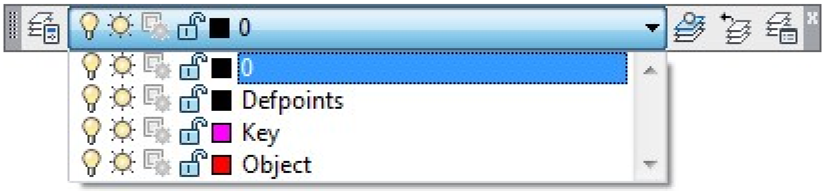

0
0
Defpoints
Key
Object

Figure Step 12

Step 13

Change the current layer to: Object. (Figure Step 13A and 13B)

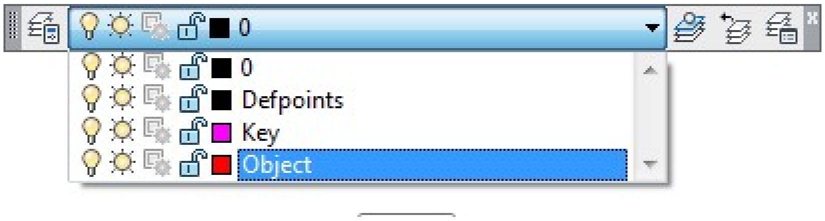

0
0
Defpoints
Key
Object

Figure Step 13A

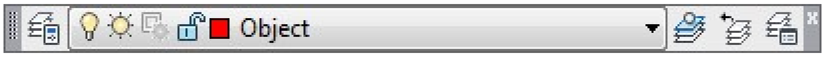


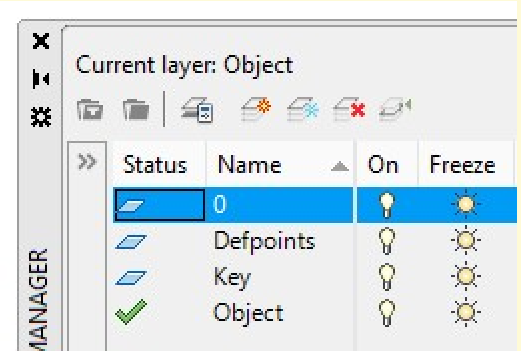
Figure Step 13B

Step 14

Save and close the drawing.

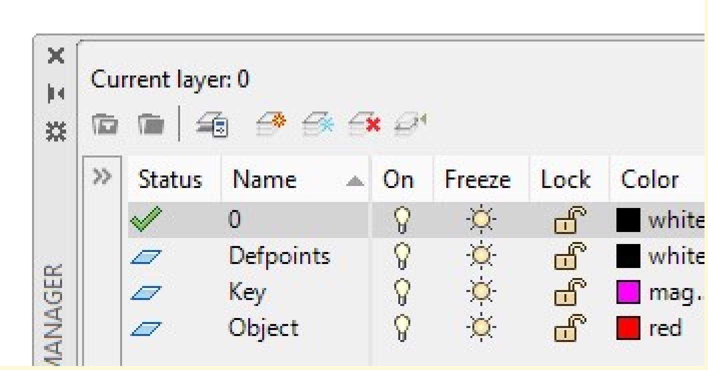
MUST KNOW: It is important to know the difference between a layer that is Off and a layer that is Frozen. A layer that is Off does NOT display but WILL regenerate. A layer that is Frozen will NOT display and will NOT regenerate. This is important as you will learn as you work your way through the book.

USER TIP: To quickly set the current layer in the Layer Property Managers window, double click the Status icon beside the layer name as shown in the figures below.



Current layer: Object

Status	Name	On	Freeze
	0		
	Defpoints		
	Key		
✓	Object		



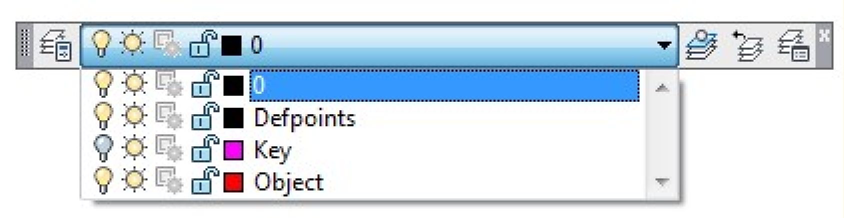
Current layer: 0

Status	Name	On	Freeze	Lock	Color
✓	0				■ white
	Defpoints				■ white
	Key				■ mag.
	Object				■ red

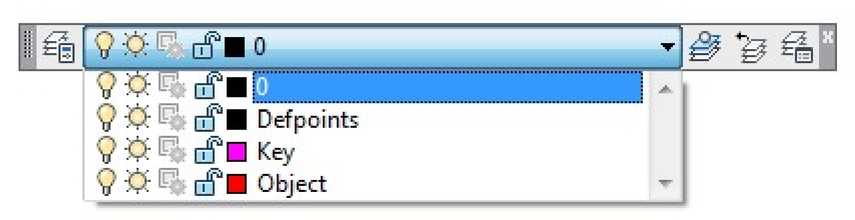
MUST KNOW: AutoCAD does not know the difference between the colors black and white. On a black background, the color black will display white and on a white background, the color white will display black.

MUST KNOW: AutoCAD has a special layer named 0 (the number zero). Each new AutoCAD drawing will contain layer 0 . It cannot be deleted or renamed. It has other special attributes as you will see in future modules.

USER TIP: In Module 3, you were shown how to insert a key onto your drawing to check it's accuracy. The key resides on the layer Key. If you are required to work on your drawing after you inserted the key, toggle layer Key off. To re-check your drawing after you finish the editing, toggle layer Key on as shown in the figures below.



Key Off



Key On

MUST KNOW: The current layer is the layer that any newly created drawing object will reside on. It is important to always be aware of what the current layer is before creating drawing objects. The current layer cannot be frozen.

Key Principles

Key Principles in Module 5

1. AutoCAD's layering system is one of the most important AutoCAD features. Layers allow you to organize, segregate, set the drawing object's color, control the visibility, and manage plotting of the drawing.
2. The current layer is the layer that any newly created drawing object will reside on.
3. AutoCAD has a special layer named 0 (the number zero). Each new drawing created in AutoCAD will automatically contain layer 0. It cannot be renamed or deleted.
4. A layer that is Off, does NOT display but WILL A layer that is frozen does NOT display and will NOT regenerate.
5. The current layer can be turned off but not frozen.

Lab Exercise 5-1

Time allowed: 30 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 05-1	2D English	Inches

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Object	All lines	Red

Step 1

Start a new drawing using the template shown above.

Step 2

Save and name the drawing: AutoCAD 2D Lab 05-1. Save it in the folder: CAD Courses/AutoCAD 2D/Lab Exercises.

Step 3

Create a new layer and name it: Object. Set its color red, as shown in the Layering Scheme above.

Step 4

Draw the object shown in the figure. All lines should be on layer: Object and display red. (Figure Step 4A and 4B)

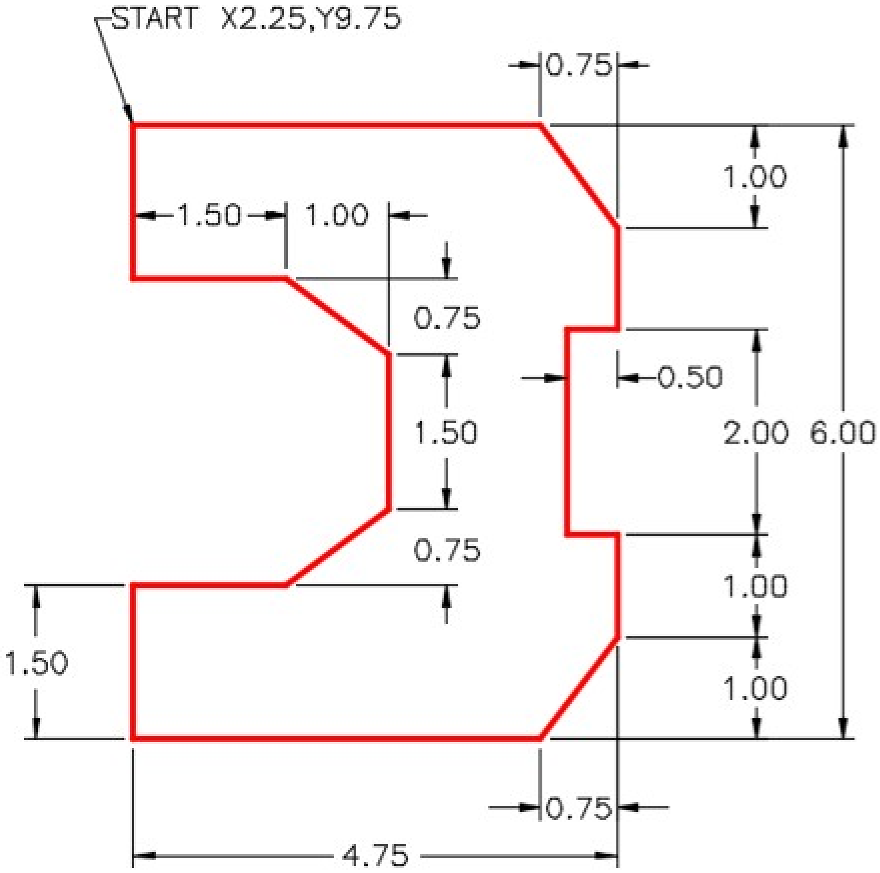

START X2.25,Y9.75
0.75
1.00
1.50
1.00
0.75
0.50
1.50
2.00
6.00
0.75
1.00
1.50
1.00
0.75
4.75

Figure Step 4A

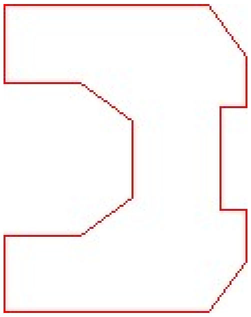
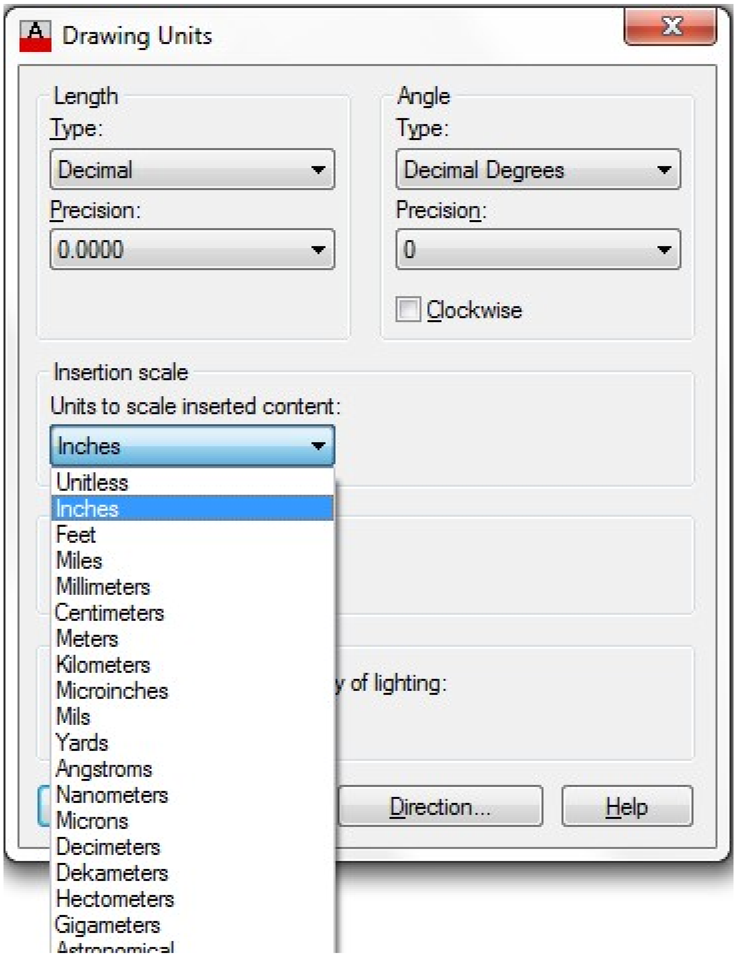


Figure Step 4B

Step 5

Enter the UNITS command. In the Drawing Units dialogue box, set the Insertion Units to Inches. (Figure Step 5)



Drawing Units

Length

Type: Decimal

Precision: 0.0000

Angle

Type: Decimal Degrees

Precision: 0

☐ Clockwise

Insertion scale

Units to scale inserted content: Inches

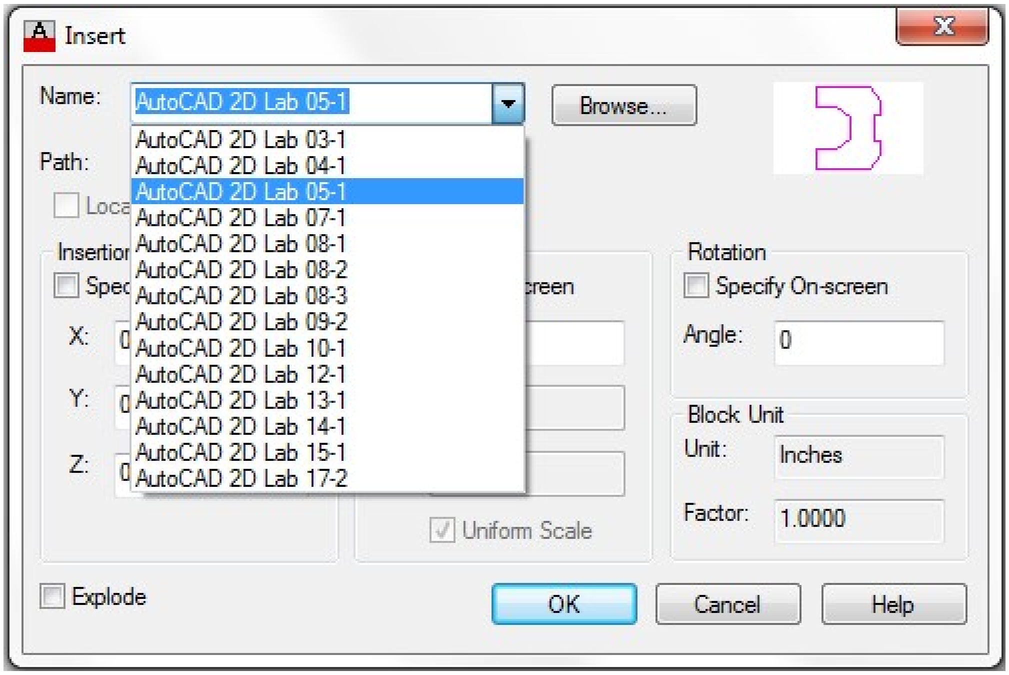
Unitless
Inches
Feet
Miles
Millimeters
Centimeters
Meters
Kilometers
Microinches
Mils
Yards
Angstroms
Nanometers
Microns
Decimeters
Dekameters
Hectometers
Gigameters
y of lighting:
Direction...
Help

Figure Step 5

AUTHOR'S COMMENTS: You can find the Insertion scale in the Units column in the lab exercise title.

Step 6

Check your drawing with the key. The key name is the same as the drawing name. (Figure Step 6)



Insert

Name: AutoCAD 2D Lab 05-1

Rotation

☐ Specify On-screen

Angle: 0

Block Unit

Unit: Inches

Factor: 1.0000

☑ Uniform Scale

☐ Explode

Browse...
Path:
Loca
Insertion
Spec
X: 0
Y: 0
Z: 0
AutoCAD 2D Lab 03-1
AutoCAD 2D Lab 04-1
AutoCAD 2D Lab 05-1
AutoCAD 2D Lab 07-1
AutoCAD 2D Lab 08-1
AutoCAD 2D Lab 08-2
AutoCAD 2D Lab 08-3
AutoCAD 2D Lab 09-2
AutoCAD 2D Lab 10-1
AutoCAD 2D Lab 12-1
AutoCAD 2D Lab 13-1
AutoCAD 2D Lab 14-1
AutoCAD 2D Lab 15-1
AutoCAD 2D Lab 17-2
creen
OK
Cancel
Help

Figure Step 6

AUTHOR'S COMMENTS: If you have trouble doing this, redo Module 3.

Step 7

The drawing should match the figure. (Figure Step 7)

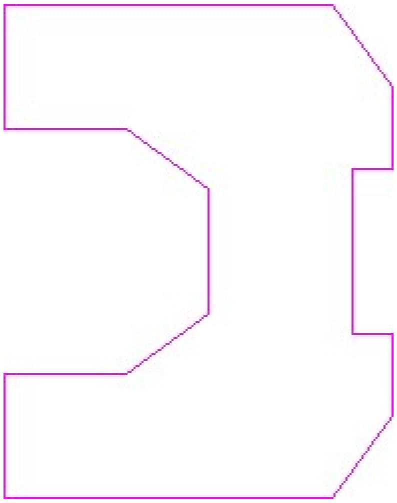


Figure Step 7

AUTHOR'S COMMENTS: Step 6 will insert a magenta colored overlay key on your drawing. If you see double objects or places where your objects and the magenta objects do not match, your drawing is inaccurate. If you only see one object, even though it may share the colors magenta and red, your drawing is accurate.

AUTHOR'S COMMENTS: If you have trouble completing this drawing the first try, do not be concerned. Start it over again, from scratch, until you can complete it. The more practice you get drawing, the easier it will get. When you get to Module 8, you will be taught how to fix a drawing so you do not have to start over.

Step 8

Pull down the layer list and turn layer: Key off. (Figure Step 8A and 8B)

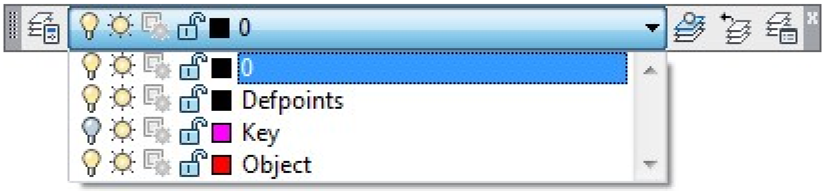


Figure Step 8A

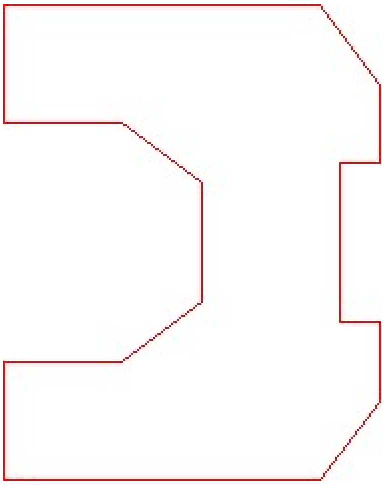


Figure Step 8B

AUTHOR'S COMMENTS: If you had made errors on the drawing you first have to turn layer Key off. Then you can make the changes to the drawing. After you make the changes, turn layer Key back on , as done in Step 9, to check the accuracy of your changes.

Step 9

Pull down the layer list and turn layer Key on. (Figure Step 9A and 9B)

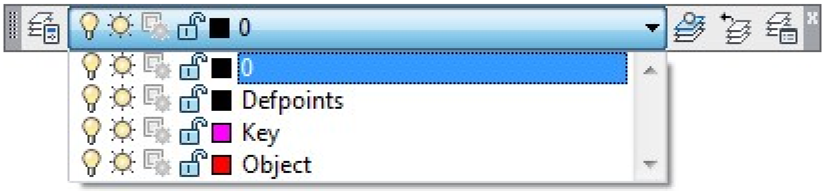


Figure Step 9A

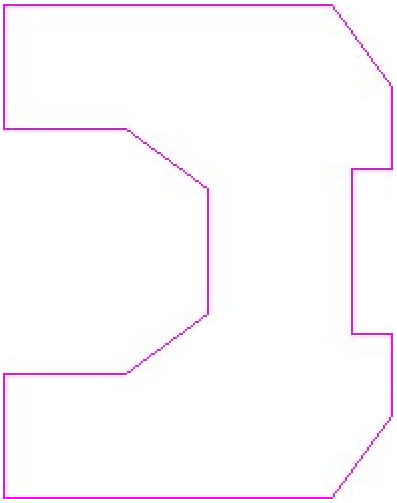


Figure Step 9B

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hint. If you get stuck and cannot complete it on your own, use the following hint to help you.

Hint 1

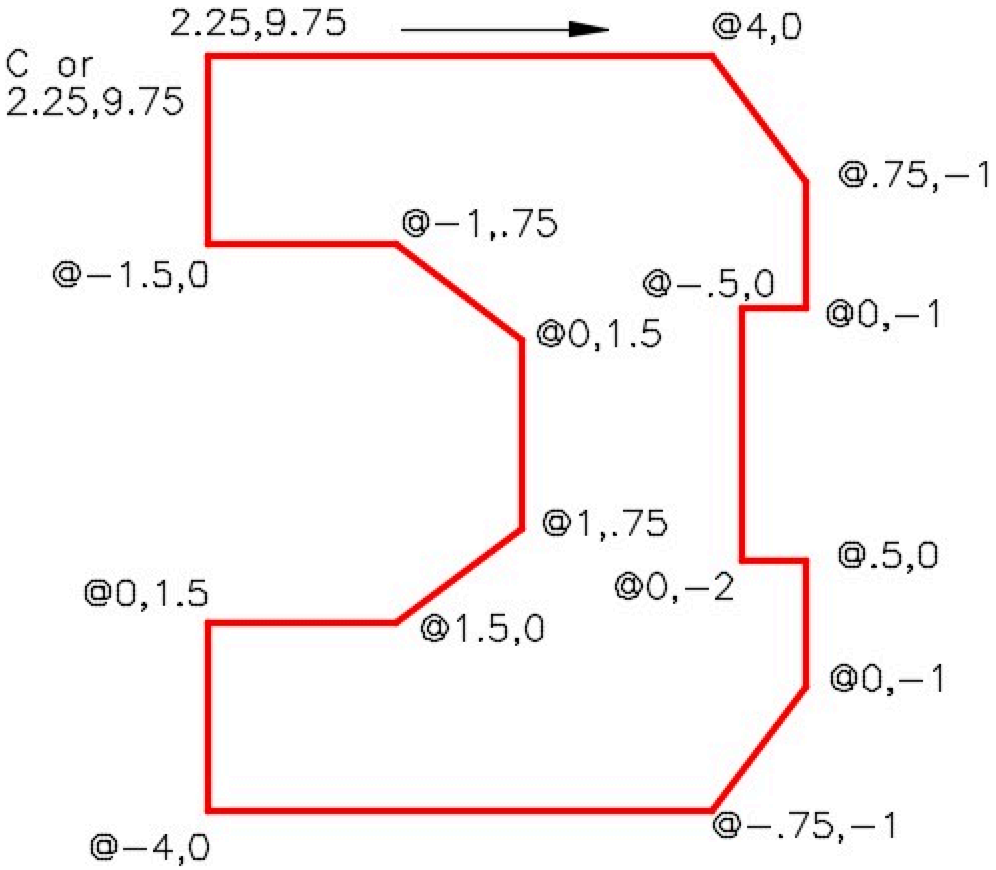

2.25,9.75
@4,0
C or
2.25,9.75
@.75,−1
@−1,.75
@−1.5,0
@−.5,0
@0,−1
@0,1.5
@1,.75
@.5,0
@0,1.5
@0,−2
@1.5,0
@0,−1
@−.75,−1
@−4,0

Figure Hint 2

Lab Exercise 5-2

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 05-2	2D Metric	Millimeters

Layering Scheme

Layer Name	Objects on Layer	Color
Object	All lines	Red

Step 1

Start a new drawing using the template shown above.

Step 2

Save and name the drawing: AutoCAD 2D Lab 05-2. Save it in the folder: CAD Courses/AutoCAD 2D/Lab Exercises.

Step 3

Create the layer: Object, color red, as shown in the Layering Scheme above.

Step 4

Draw the object shown in the figure. All lines should be on layer: Object and display red. (Figure Step 4A and 4B)

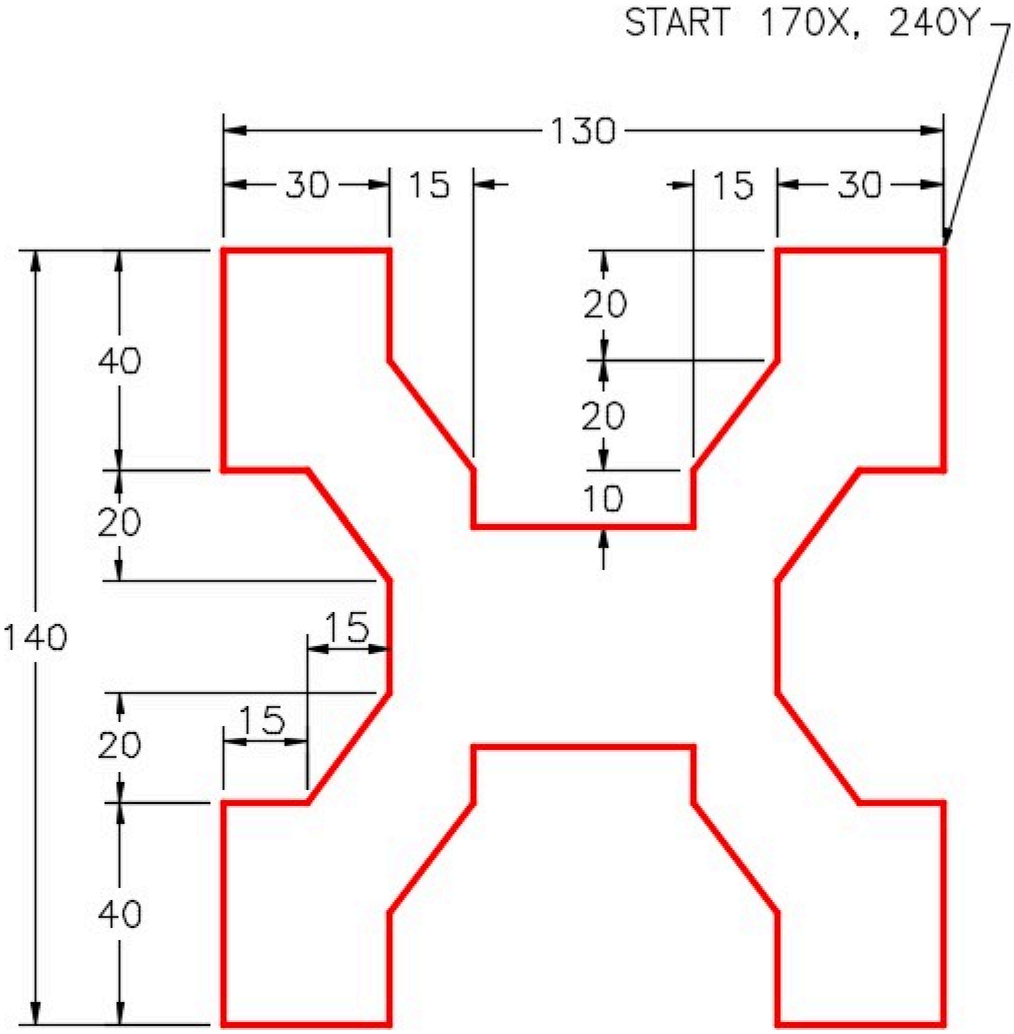

START 170X, 240Y
130
30
15
15
30
20
20
10
40
20
140
15
15
20
40

Figure Step 4A

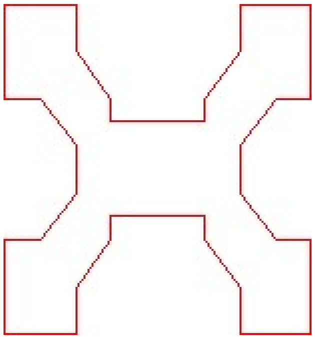
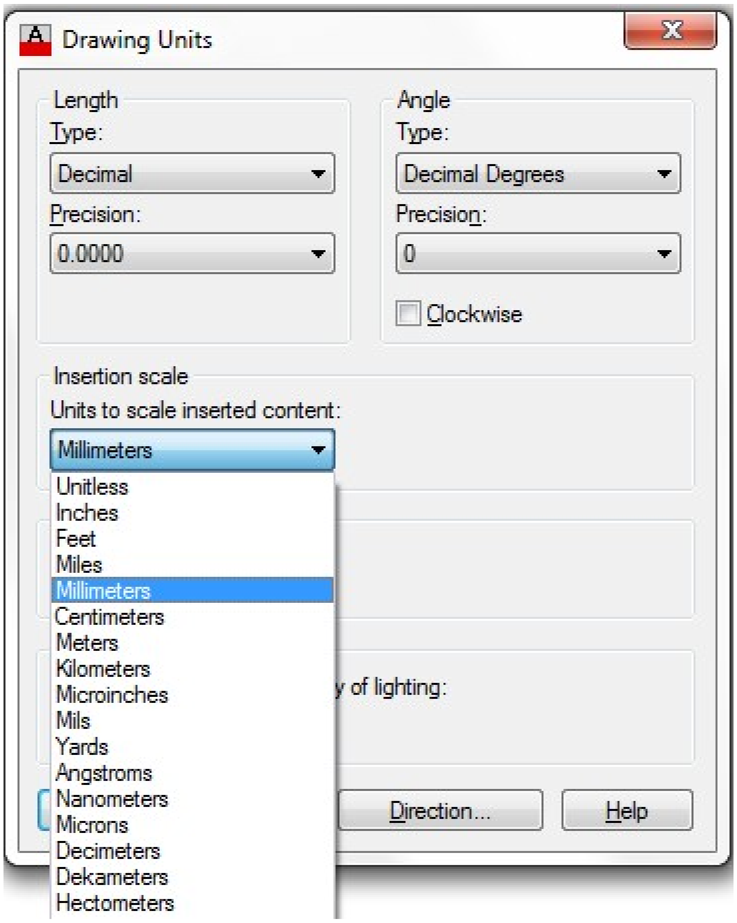


Figure Step 4B
Completed Drawing

Step 5

Enter the UNITS command. In the Drawing Units dialogue box, set the Insertion Units to Millimeters. (Figure Step 5)



Length Type: Decimal
Precision: 0.0000
Angle Type: Decimal Degrees
Precision: 0
☐ Clockwise
Insertion scale
Units to scale inserted content: Millimeters

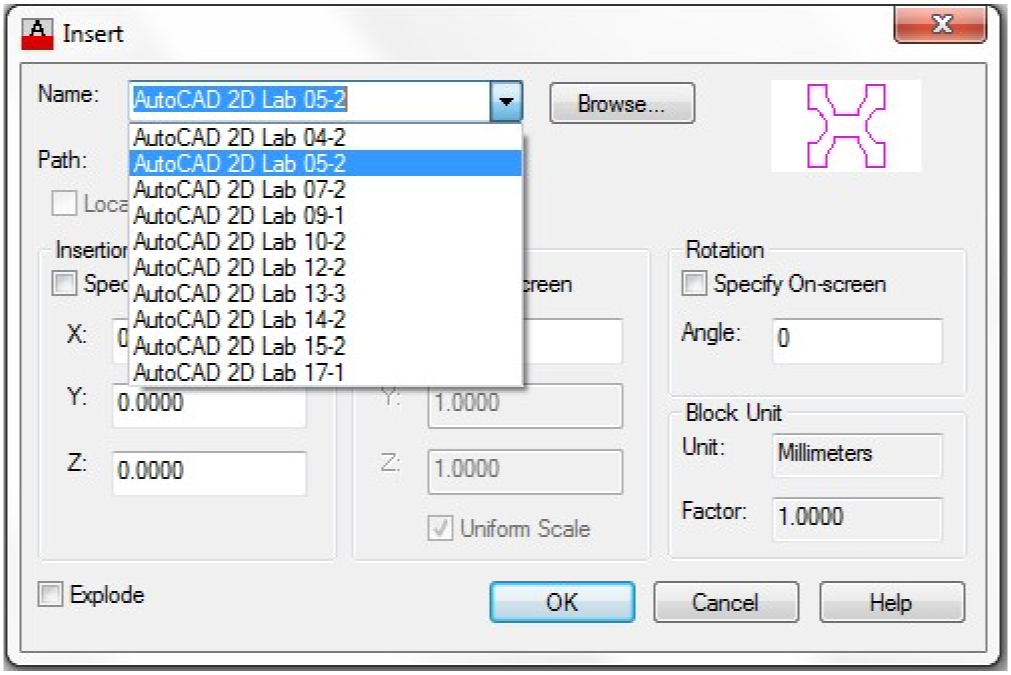
Drawing Units
Unitless
Inches
Feet
Miles
Millimeters
Centimeters
Meters
Kilometers
Microinches
Mils
Yards
Angstroms
Nanometers
Microns
Decimeters
Dekameters
Hectometers
y of lighting:
Direction...
Help

Figure Step 5

AUTHOR'S COMMENTS: You can find the Insertion scale in the Units column in the lab exercise title.

Step 6

Check your drawing with the key. The key name is the same as the drawing name. (Figure Step 6)



Insert

Name: AutoCAD 2D Lab 05-2

Path:

☐ Loca

Insertion

☐ Spec

X: 0

Y: 0.0000

Z: 0.0000

Y: 1.0000

Z: 1.0000

☑ Uniform Scale

Rotation

☐ Specify On-screen

Angle: 0

Block Unit

Unit: Millimeters

Factor: 1.0000

☐ Explode

Browse...
AutoCAD 2D Lab 04-2
AutoCAD 2D Lab 05-2
AutoCAD 2D Lab 07-2
AutoCAD 2D Lab 09-1
AutoCAD 2D Lab 10-2
AutoCAD 2D Lab 12-2
AutoCAD 2D Lab 13-3
AutoCAD 2D Lab 14-2
AutoCAD 2D Lab 15-2
AutoCAD 2D Lab 17-1
reen
OK
Cancel
Help

Figure Step 6

Step 7

Your drawing should match the figure. (Figure Step 7)

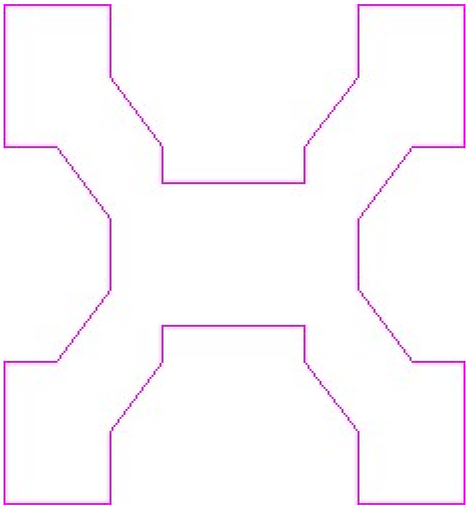


Figure Step 7

AUTHOR'S COMMENTS: Step 6 will insert a magenta colored overlay key on your drawing. If you see double objects or places where your objects and the magenta objects do not match, your drawing is inaccurate. If you only see one object, even though it may share the colors magenta and red, your drawing is accurate.

AUTHOR'S COMMENTS: If you have trouble completing this drawing the first try, do not be concerned. Start it over again, from scratch, until you can complete it. The more practice you get drawing, the easier it will get. When you get to Module 8, you will be taught how to fix a drawing so you do not have to start over.

Step 8

Save and close the drawing.

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hint. If you get stuck and cannot complete it on your own, use the following hint to help you.

Hint 1

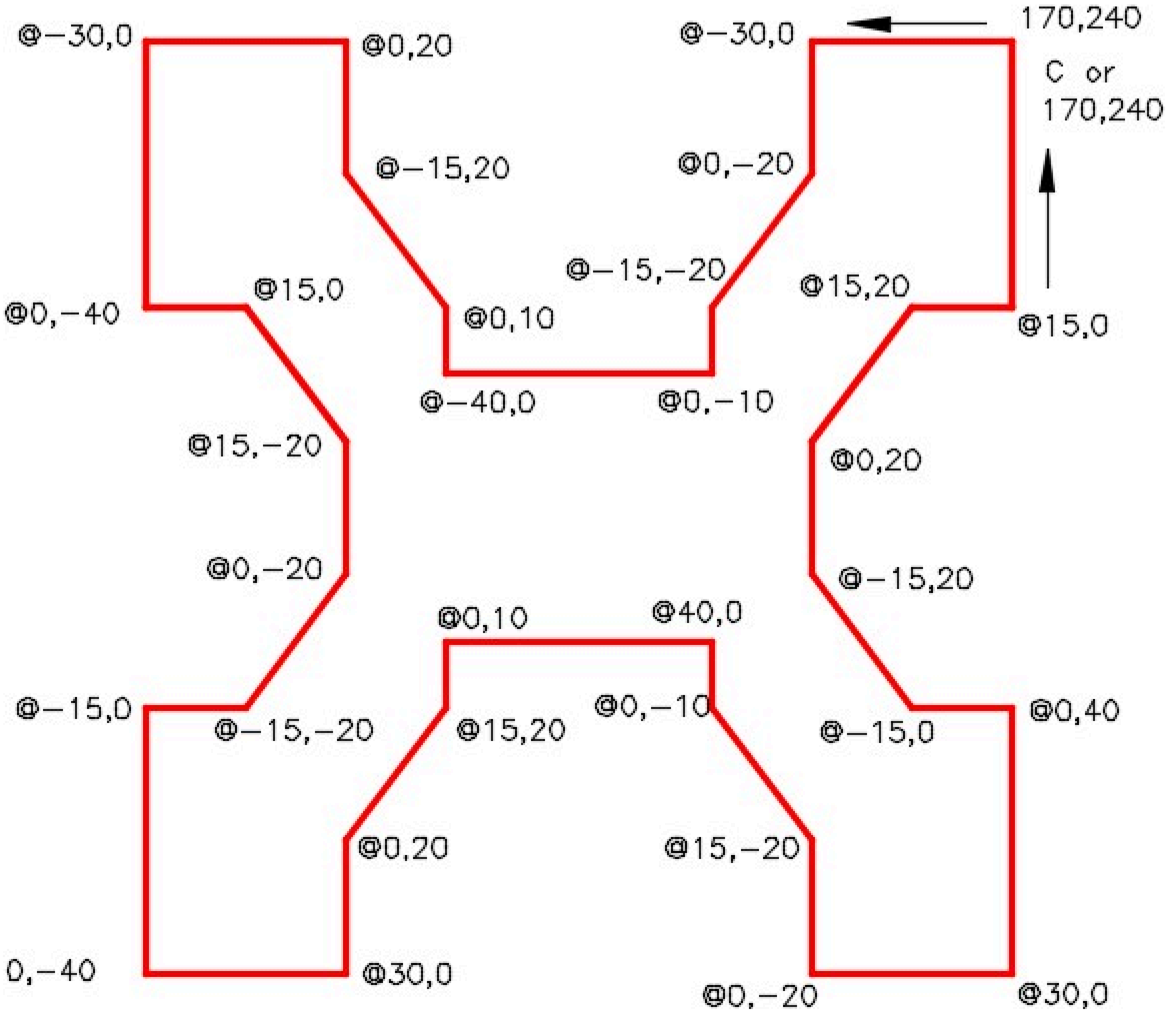

@-30,0
@0,20
@-30,0
170,240
C or
170,240
@-15,20
@0,-20
@-15,-20
@15,20
@0,-40
@15,0
@0,10
@15,0
@-40,0
@0,-10
@15,-20
@0,20
@0,-20
@-15,20
@0,10
@40,0
@-15,0
@-15,-20
@15,20
@0,-10
@-15,0
@0,40
@0,20
@15,-20
0,-40
@30,0
@0,-20
@30,0

Figure Step Hint 1

Module 6: Competency Test 1 Open Book

Learning Outcomes

When you have completed this module, you will be able to:

1. Within a one-hour time limit, complete a written exam and a lab exercise without the aid of a key.

The AutoCAD 2D book was written with competency based modules. What that means is that you have not completed each module until you have mastered it. The Competency Test module contains multiple choice questions and a comprehensive lab exercise to test your mastery of the set of modules that you completed. There are no answers or keys supplied in a Competency Test module since it is meant to be checked by your instructor. If there are any parts of this module that you have trouble completing, you should go back and reread the module or modules containing the information that you are having trouble with. If necessary, redo as many lab exercises required until you fully understand the material.

If you are completing this book:

- Without the aid of an instructor, complete the written test and the lab exercise.
- In a classroom with an instructor, the instructor will give instructions on what to do after this module has been completed.

Multiple Choice Questions

Select the BEST answer.

1. What two keyboard keys, when pressed, execute a command or accept an AutoCAD default?
    - A. Esc and Space Bar
    - B. Enter and Esc
    - C. Enter and Space Bar
    - D. F2 and Enter
    - E. Space Bar and F6
2. To AutoCAD, what does the @ symbol mean?
    - A. A default
    - B. An option

    C. Abort
    D. Execute
    E. The last absolute coordinate

3. What keyboard key opens and closes the Text window?
    A. F1
    B. F2
    C. F3
    D. F4
    E. F6
4. What is the main purpose of the left mouse button?
    A. To execute a command
    B. The pick button
    C. The abort button
    D. Repeats the last command
    E. The menu button
5. What is the name of AutoCAD's special layer that exists in all new drawings and cannot be deleted?
    A. 0 (zero)
    B. Object
    C. 2
    D. Special
    E. Layer 1
6. What keyboard key aborts an AutoCAD command?
    A. F2
    B. Ctrl
    C. Esc
    D. F6
    E. Shift
7. When a drawing object is inserted in a drawing, what layer will AutoCAD place it on?
    A. Layer 0 (zero)
    B. The last layer used
    C. The current layer
    D. The object layer

    E. The displayed layer

8. Which layer cannot be frozen?
    A. Layer 0 (zero)
    B. The last layer used
    C. Any layer that is currently displayed
    D. The object layer
    E. The current layer
9. Which one of the following inputs the relative coordinate location X2Y4?
    A. 2,4
    B. 4,2
    C. <2,4>
    D. @2,4
    E. #2,4
10. What is the name of the file that contains the AutoCAD defaults and standards set by the creator?
    A. The template file
    B. The object file
    C. The backup file
    D. The text file
    E. The readme file

Lab Exercise 6-1 OPEN BOOK

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 06-1	2D English	Inches

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Object	All lines	Red

Step 1

Start a new drawing using the template shown above.

Step 2

Save and name the drawing: AutoCAD 2D Lab 06-1.

Step 3

Create layer: Object, color red, as shown in the Layering Scheme above.

Step 4

Draw the object shown in the figure. All lines should be on layer: Object and display red. (Figure Step 4A and 4B)

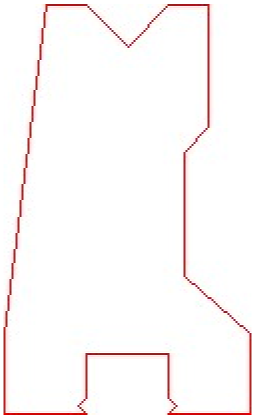


Figure Step 4A
Completed Drawing

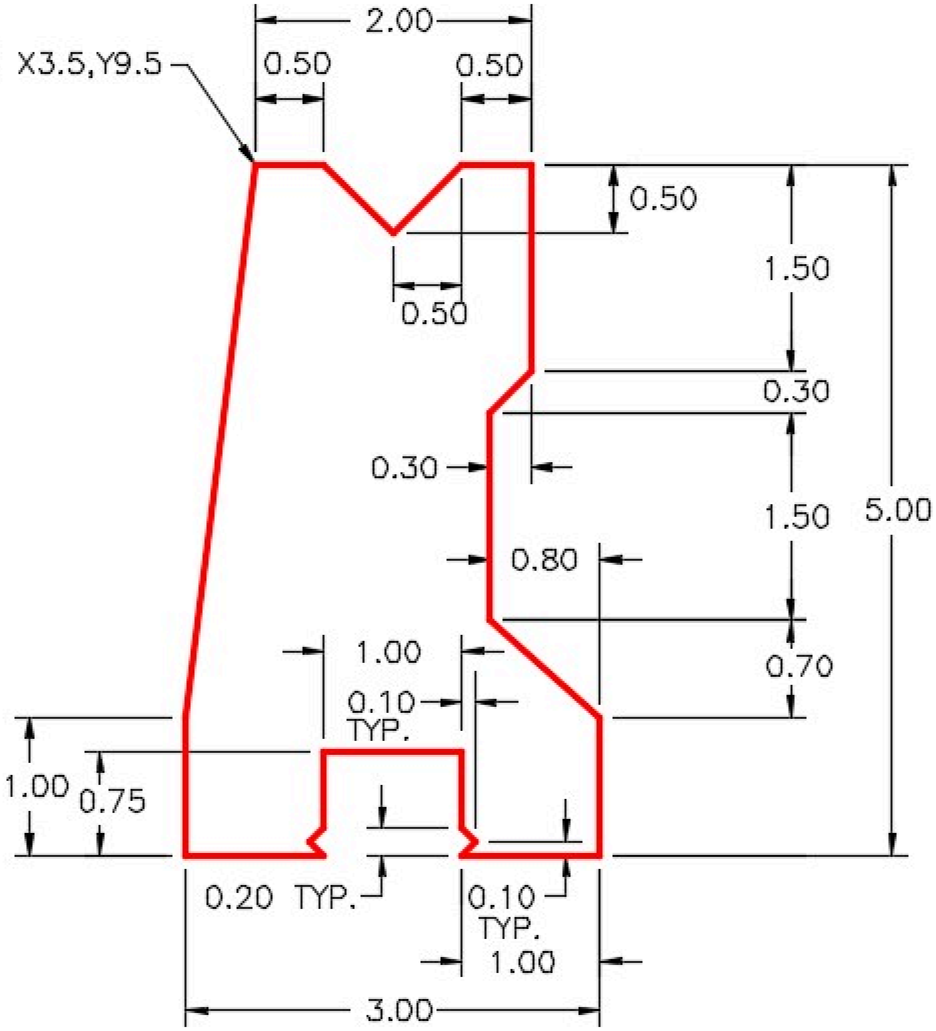

START X3.5,Y9.5
2.00
0.50
0.50
0.50
0.50
1.50
0.30
0.30
1.50
5.00
0.80
1.00
0.70
0.10
TYP.
1.00
0.75
0.20 TYP.
0.10
TYP.
1.00
3.00

Figure Step 4B

AUTHOR'S COMMENTS: TYP means typical. This simply means that there is at least one other dimension that is the same. In this drawing, the opposite side of the slot on the bottom of the object is the same size as the dimensioned one.

Step 5

Save and close the drawing.

AUTHOR'S COMMENTS: There is no key for this lab.

Part 2

Module 7: Circles and Arcs

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe the geometry of circles and arcs.
2. Apply the CIRCLE and ARC commands to draw circles and arcs.
3. Describe how to undo a command or redo an undo.

Geometry Lesson: Circles

A *circle* is defined as a closed curve in which all points are the same distance from its center point. The center point is a single XY coordinate.

A circle is 360 degrees and can be divided into four *quadrants*. All points on a circle are at a given distance from is center point. The distance between any of the points and the center is called the *radius*.

Study the drawings in Figure 7-1, 7-2, 7-3, and 7-4 for a description of the geometry of a circle.

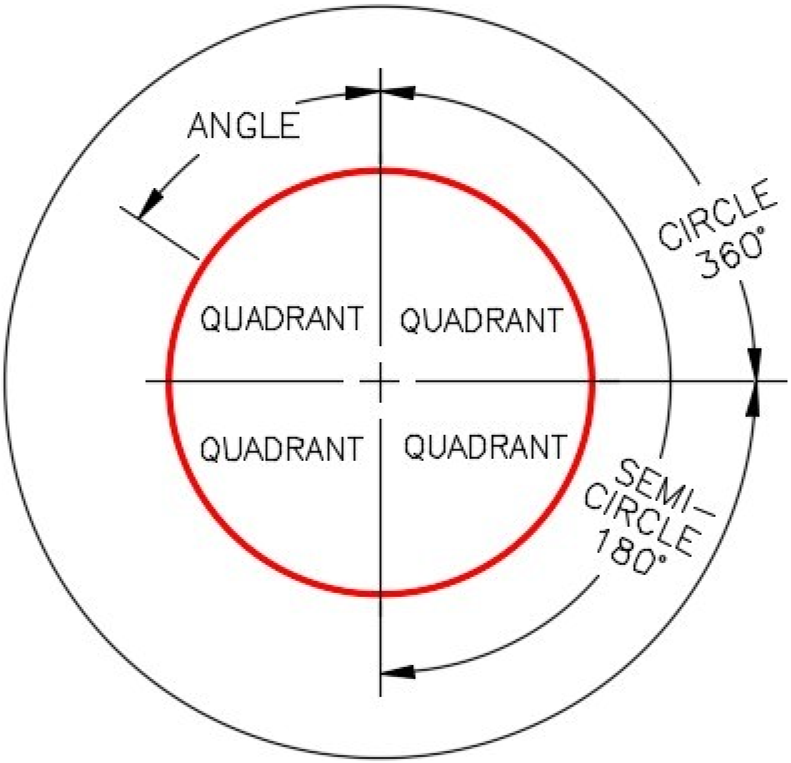

ANGLE
CIRCLE 360°
QUADRANT
QUADRANT
QUADRANT
QUADRANT
SEMI-CIRCLE 180°

Figure 7-1 Geometry of a Circle – Part 1

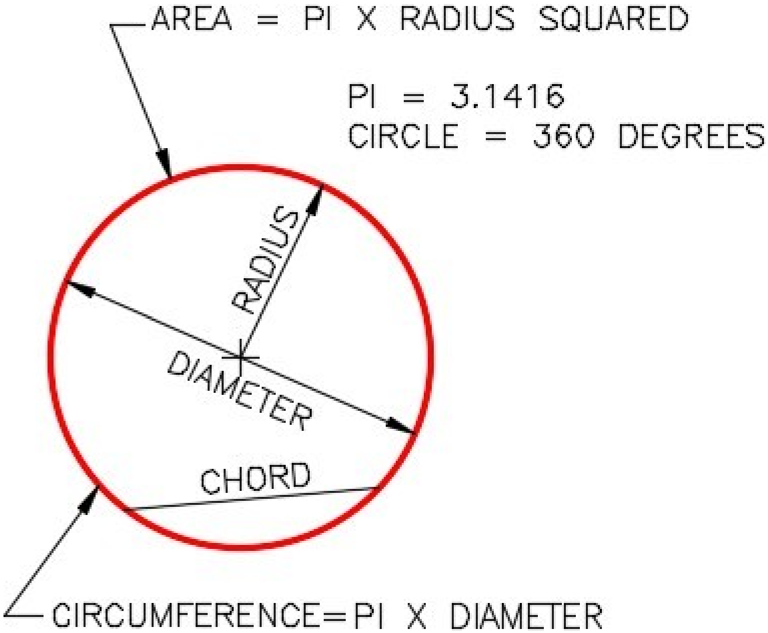

AREA = PI X RADIUS SQUARED
PI = 3.1416
CIRCLE = 360 DEGREES
RADIUS
DIAMETER
CHORD
CIRCUMFERENCE=PI X DIAMETER

Figure 7-2 Geometry of a Circle – Part 2

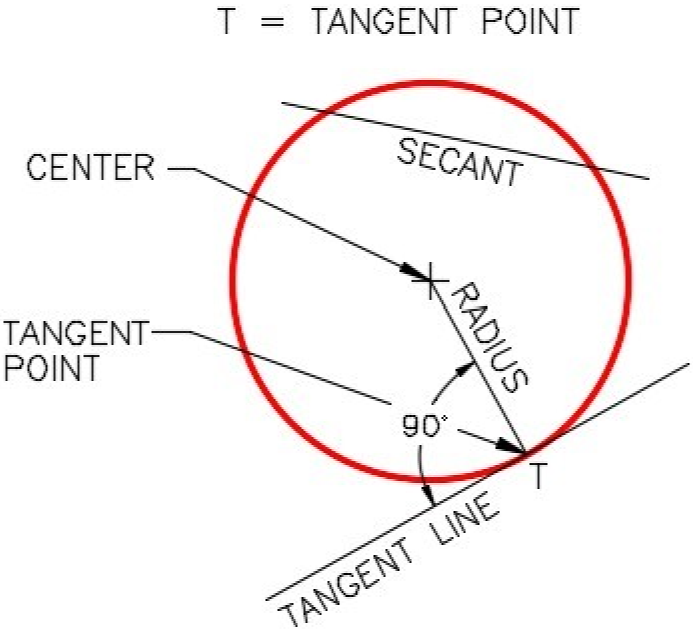

T = TANGENT POINT
SECANT
CENTER
RADIUS
TANGENT POINT
90°
T
TANGENT LINE

Figure 7-3 Geometry of a Circle – Part 3

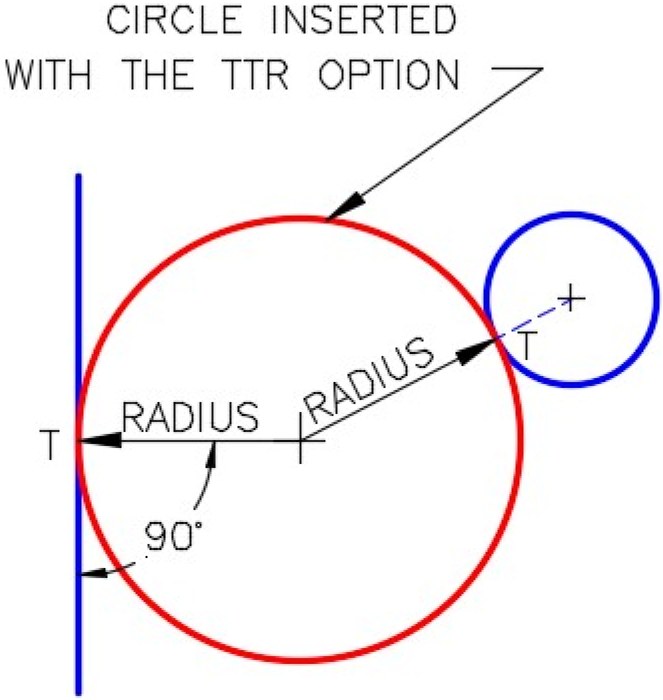

CIRCLE INSERTED WITH THE TTR OPTION
RADIUS
T
T
RADIUS
90°

Figure 7-4 Geometry of a Circle – Part 4

Geometry Lesson: Arcs

An *arc* is defined as an open curve in which all points are the same distance from its center point. Study the drawings in Figure 7-5 and 7-6.for a description of the geometry of an arc.

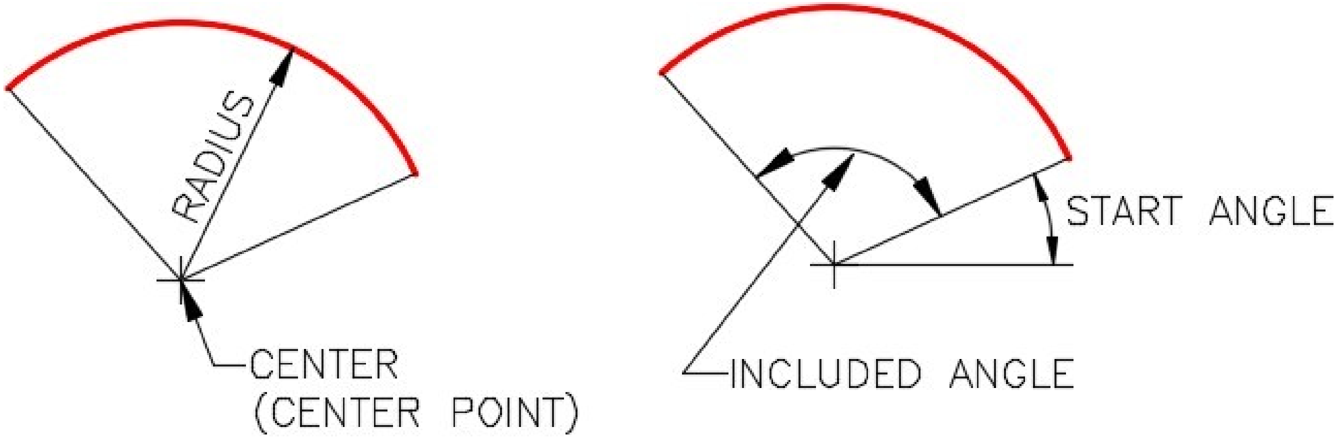

RADIUS
CENTER
(CENTER POINT)
START ANGLE
INCLUDED ANGLE

Figure 7-5 Geometry of an Arc – Part 1

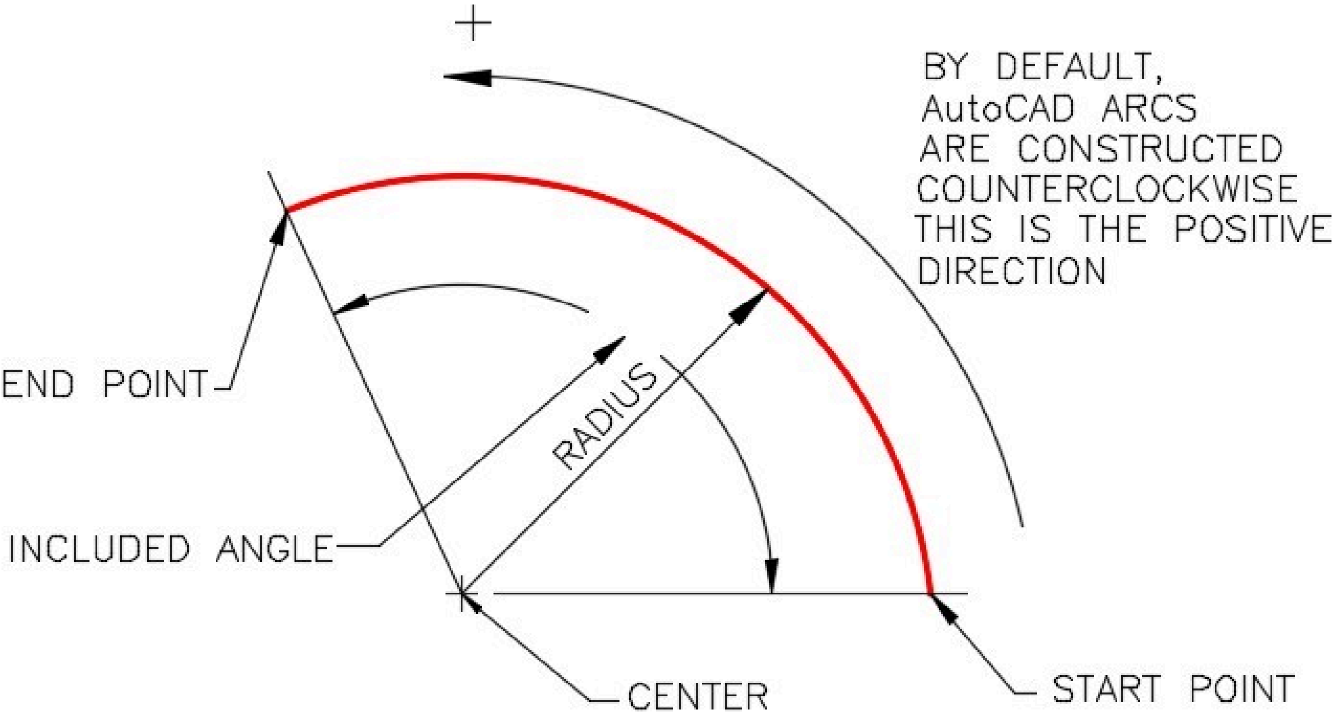
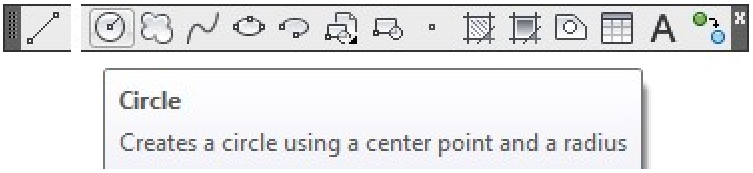

BY DEFAULT,
AutoCAD ARCS
ARE CONSTRUCTED
COUNTERCLOCKWISE
THIS IS THE POSITIVE
DIRECTION
END POINT
RADIUS
INCLUDED ANGLE
CENTER
START POINT

Figure 7-6 Geometry of an Arc – Part 2

AutoCAD Command: CIRCLE

The CIRCLE command is used to draw circles.

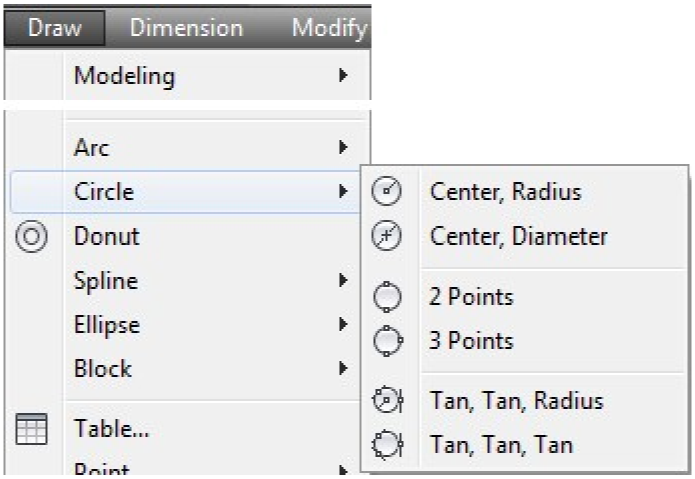
Shortcut: **C**



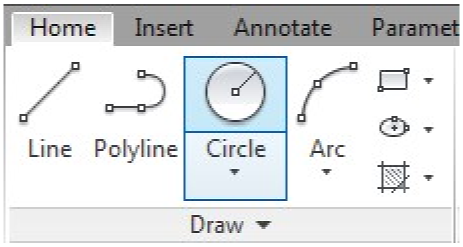
Circle

Creates a circle using a center point and a radius

Draw Toolbar


Draw
Dimension
Modify
Modeling
Arc
Circle
Donut
Spline
Ellipse
Block
Table...
Point
Center, Radius
Center, Diameter
2 Points
3 Points
Tan, Tan, Radius
Tan, Tan, Tan

Draw Pull-down

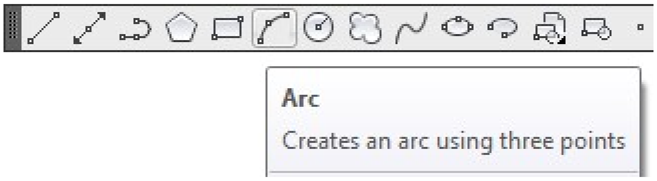

Home
Insert
Annotate
Paramet
Line
Polyline
Circle
Arc
Draw

Home Ribbon

AutoCAD Command: ARC

The ARC command is used to draw arcs.

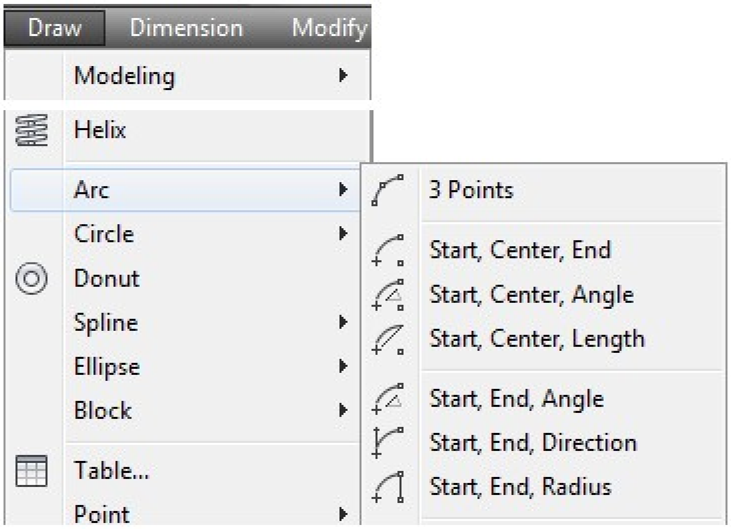
Shortcut: **A**



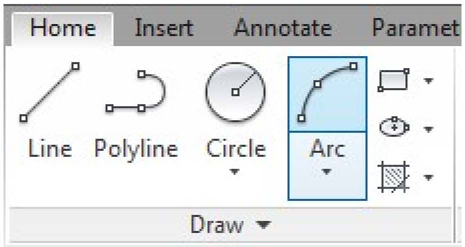
Arc

Creates an arc using three points

Draw Toolbar


Draw
Dimension
Modify
Modeling
Helix
Arc
Circle
Donut
Spline
Ellipse
Block
Table...
Point
3 Points
Start, Center, End
Start, Center, Angle
Start, Center, Length
Start, End, Angle
Start, End, Direction
Start, End, Radius

Draw Pull-down


Home
Insert
Annotate
Paramet
Line
Polyline
Circle
Arc
Draw

Home Ribbon

MUST KNOW: When you are drawing arcs, it is best to specify the center point first and the start point second. The radius and the direction of the arc can be entered at the same time. Enter the angle or the end point to complete the arc.

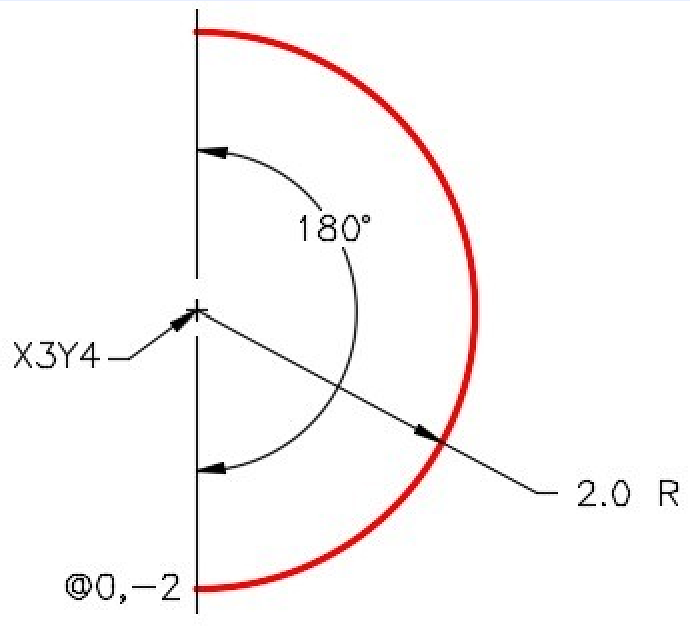

180°
X3Y4
2.0 R
@0,−2

Figure 7-7 Constructing an Arc

```
Command: ARC
Specify start point of arc or [Center]: C
Specify center point of arc: 3,4
Specify start point of arc: @0,-2
(Specifying the radius and the direction for the start point.)
Specify end point of arc or [Angle/chord Length]: A
Specify included angle: 180
Command:
```

WORKALONG: Drawing Circles and Arcs

Step 1

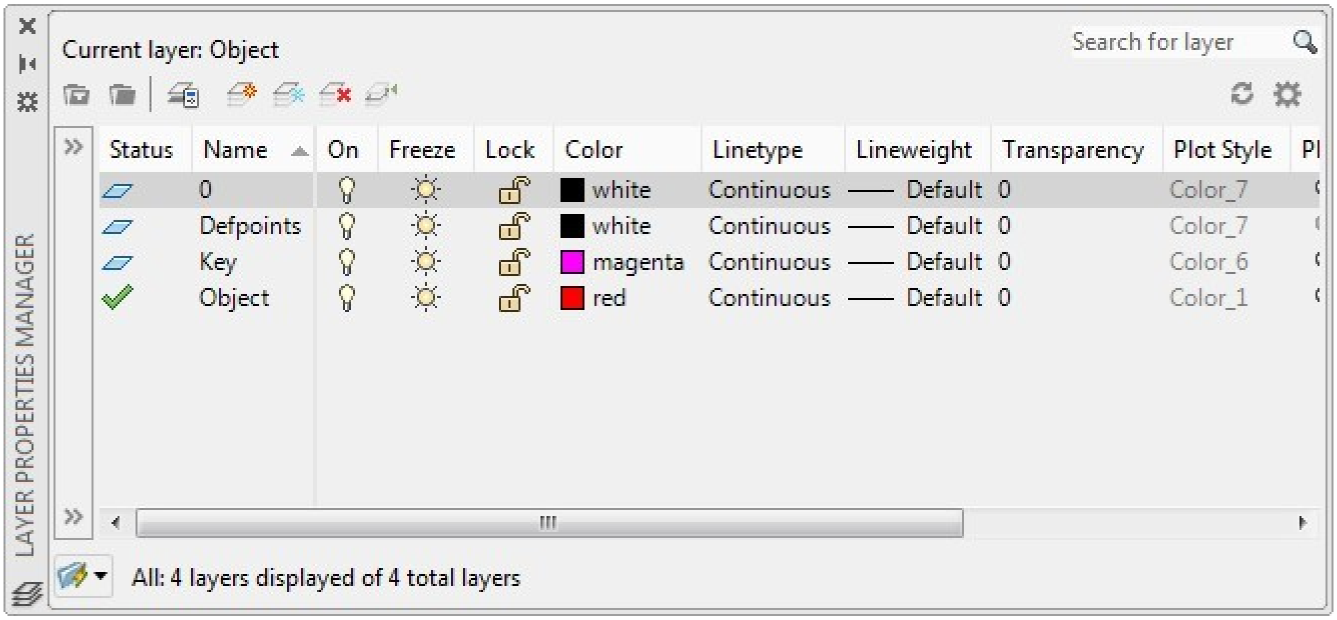
Start a new drawing using the template: 2D English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 07-1.

Step 3

Create layer: Object, color red. Set it as the current layer. (Figure Step 3)



Current layer: Object

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight	Transparency	Plot Style	Pl
▱	0	💡	☼	🔓	■ white	Continuous	—— Default	0	Color_7	
▱	Defpoints	💡	☼	🔓	■ white	Continuous	—— Default	0	Color_7	
▱	Key	💡	☼	🔓	■ magenta	Continuous	—— Default	0	Color_6	
✓	Object	💡	☼	🔓	■ red	Continuous	—— Default	0	Color_1	

All: 4 layers displayed of 4 total layers

Figure Step 3

Step 4

Using the figures and what you already leaned, draw the two lines shown in Figure Step 4B. (Figure Step 4A and 4B)

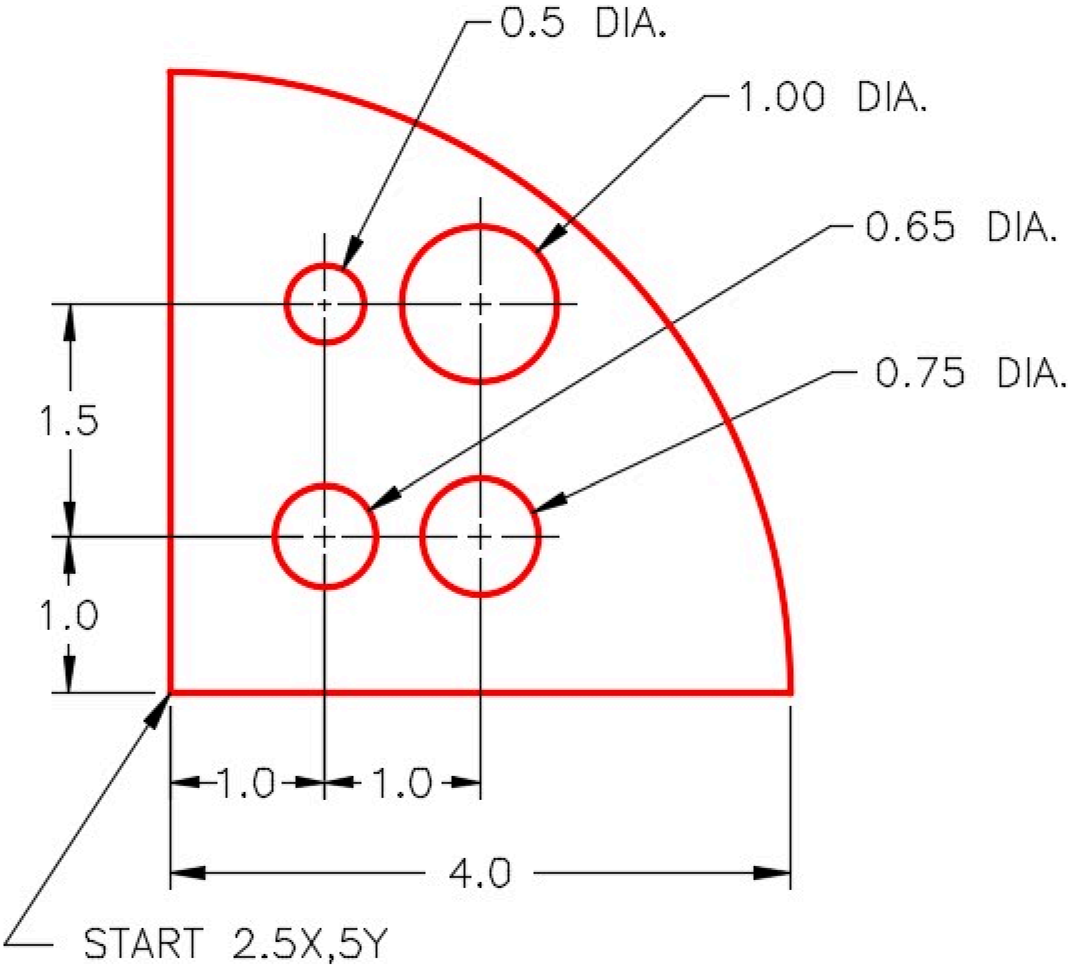

0.5 DIA.
1.00 DIA.
0.65 DIA.
0.75 DIA.
1.5
1.0
1.0
1.0
4.0
START 2.5X,5Y

Figure Step 4A

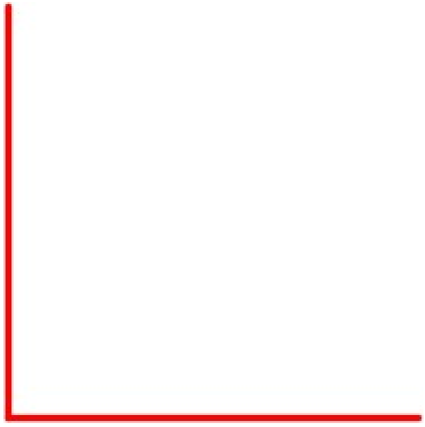


Figure Step 4B

AUTHOR’S COMMENTS: Draw the two lines using the LINE command twice.

USER TIP: By default, AutoCAD draw arcs counterclockwise which is the positive arc angle. If you use the Angle option, AutoCAD allows you to draw an arc by entering a negative angle.

Step 5

Enter the ARC command, as shown below, to draw the arc. (Figure Step 5)

Command: **ARC**

Specify start point of arc or [Center]: **C**

(If possible, always draw arcs starting with the center point first.)

Specify center point of arc: **2.5, 5**

(Enter the center point.)

Specify start point of arc: **@4,0**

(Enter the start point. Here the trick is to give AutoCad the start point and the radius of the arc at the same time)

Specify end point of arc or [Angle/chord Length]: **A**

Specify included angle: **90**

(Enter the angle of the arc. In this case, it is 90 degrees.)

Command:

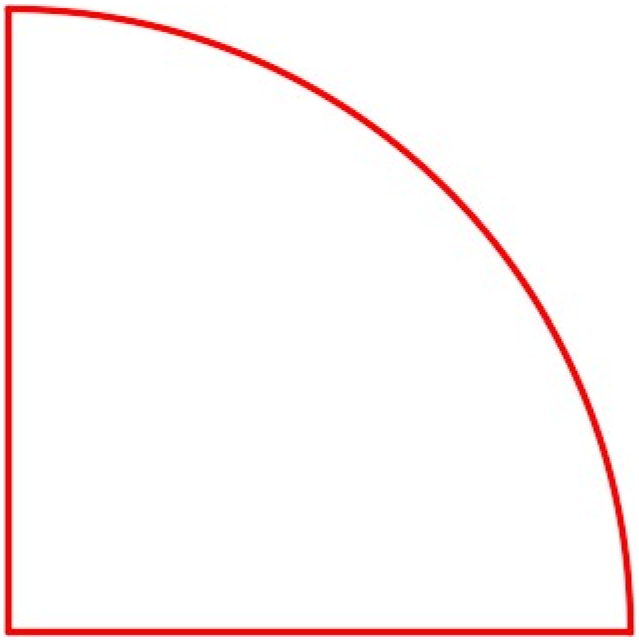


Figure Step 5

USER TIP: Arcs can be drawn in either counterclockwise or clockwise direction by holding down the CTRL key, while in the ARC command, to reverse the direction.

Step 6

Enter the CIRCLE command, as shown below, to draw the four circles to complete the drawing. (Figure Step 6A and 6B)

Command: **CIRCLE**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **3.5,6**

(Since this is the first circle, enter an absolute coordinate for the center point.)

Specify radius of circle or [Diameter]: **D**

(Circles are usually dimensioned with diameters rather then a radius, enter D)

Specify diameter of circle: **0.65**

Command: **CIRCLE**

(Press Enter or Space bar as that will repeat the last command.)

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **@1,0**

(Notice here, since AutoCAD always remembers the lastpoint, use an @. The lastpoint was the center of the first circle placed.)

Specify radius of circle or [Diameter] <0.3250>: **D**

Specify diameter of circle <0.6500>: **0.75**

Command: **CIRCLE** (Press Enter or Space bar as that will repeat the last command.

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **@0,1.5**

(Use an @ from the center of the last circle drawn.)

Specify radius of circle or [Diameter] <0.3750>: **D**

Specify diameter of circle <0.7500>: **1**

Command: **CIRCLE**

(Press Enter or Space bar. That will repeat the last command and save you entering it again.)

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **@-1,0**

Specify radius of circle or [Diameter] <0.5000>: **D**

Specify diameter of circle <1.0000>: **0.5**

Command:

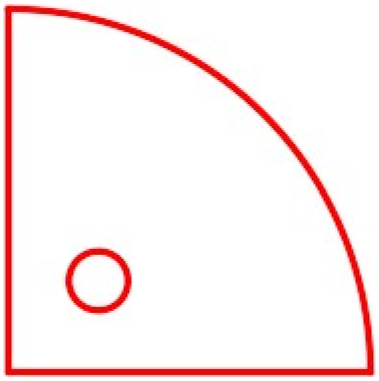


Figure Step 6A

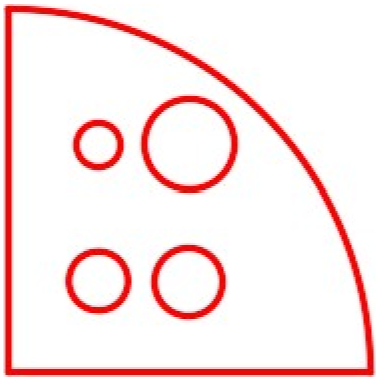


Figure Step 6B

Step 7

Save and close the drawing.

WORKALONG: Drawing Circles Using the TTR Option

Step 1

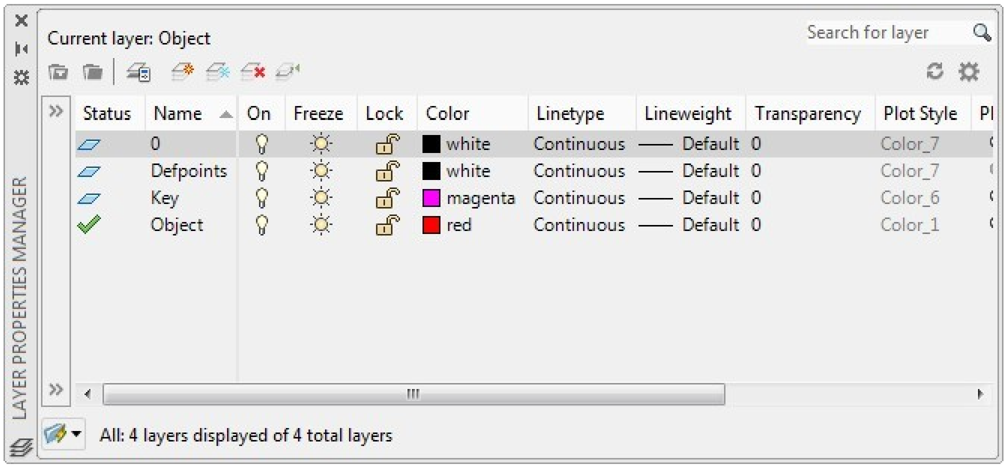
Start a new drawing using the template: 2D English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 07-2.

Step 3

Create layer: Object, color red. Set it as the current layer. (Figure Step 3)



Current layer: Object

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight	Transparency	Plot Style	Pl
▱	0	💡	☼	🔓	■ white	Continuous	—— Default	0	Color_7	
▱	Defpoints	💡	☼	🔓	■ white	Continuous	—— Default	0	Color_7	
▱	Key	💡	☼	🔓	🟪 magenta	Continuous	—— Default	0	Color_6	
✓	Object	💡	☼	🔓	🟥 red	Continuous	—— Default	0	Color_1	

All: 4 layers displayed of 4 total layers

Figure Step 3

Step 4

Using the dimensioned drawing as a guide and what you already learned, use the LINE command to draw the four outside lines. (Figure Step 4A and 4B)

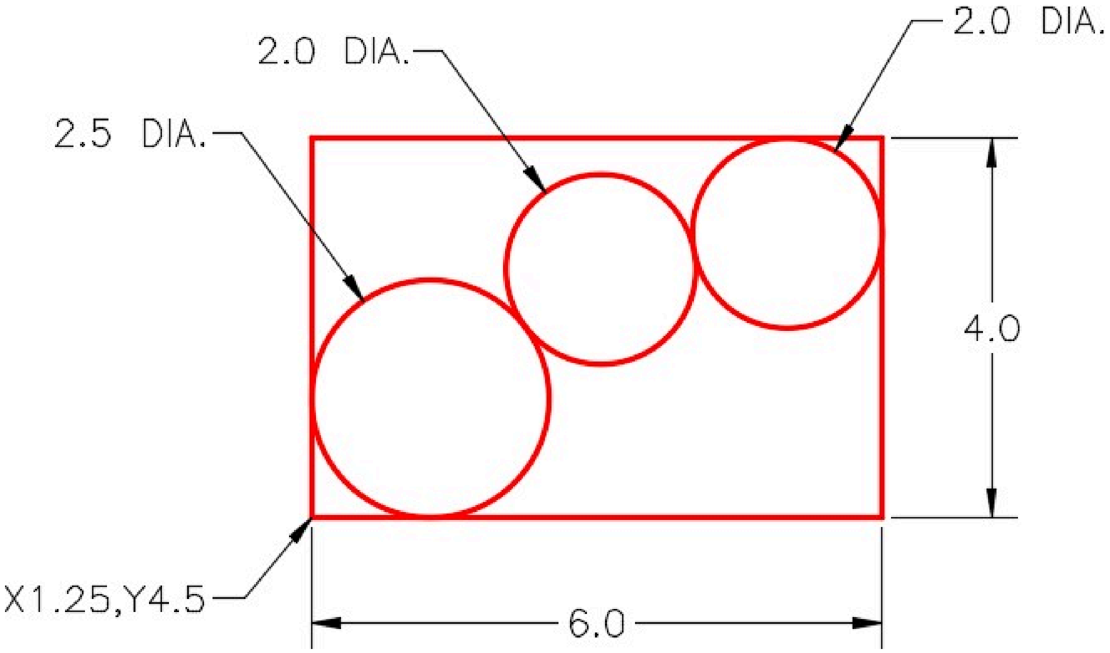

2.0 DIA.
2.0 DIA.
2.5 DIA.
4.0
X1.25,Y4.5
6.0

Figure Step 4A

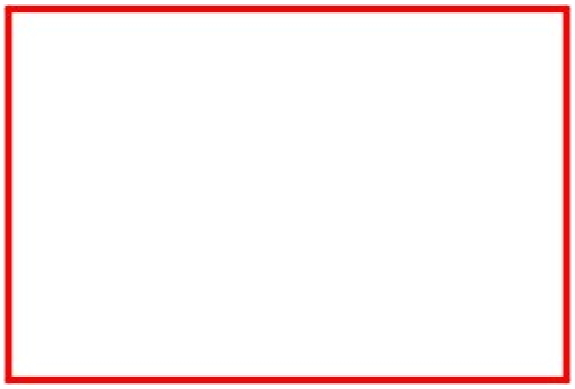


Figure Step 4B

Step 5

Enter the CIRCLE command and the TTR option, as shown below, to insert the 2 inch diameter circle. (Figure Step 5A and 5B)

Command: **CIRCLE**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **TTR**

Specify point on object for first tangent of circle: **P1**

(Using the graphic cursor, move it on the top horizontal line. When the tangent snap icon appears, press the left mouse button.)

Specify point on object for second tangent of circle: **P2**

(Move the cursor on the vertical line. When the tangent icon appears, pick it.)

Specify radius of circle <0.7000>: **1**

(Enter the radius of the circle. Be careful, since circles are usually dimensioned in diameters you will have to divide it in half.)

Command:

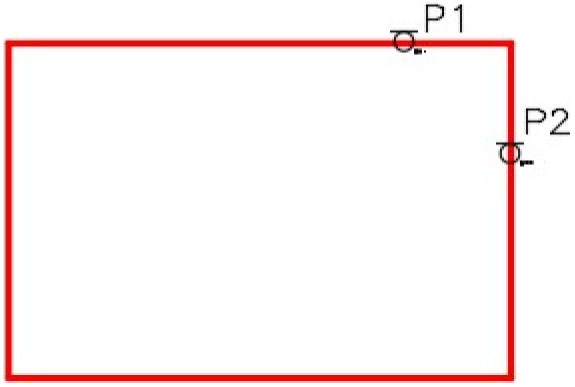


Figure Step 5A

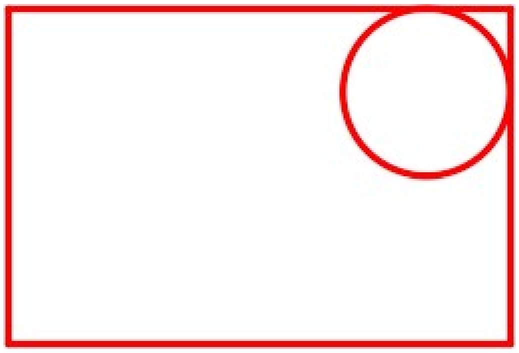


Figure Step 5B

AUTHOR'S COMMENTS: TTR means ' Tangent Tangent Radius '. Do not enter the P1 or P2 on the keyboard. They simply mean Pick 1 and Pick 2. To pick, move the Graphic cursor to the desired location, wait for the tangent Autosnap marker to display and when it dose, press the left mouse button.

Step 6

Enter the CIRCLE command and the TTR option, as shown below, to insert the 2.5 inch diameter circle. (Figure Step 6A and 6B)

Command: **CIRCLE**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **TTR**

Specify point on object for first tangent of circle: **P3** Specify point on object for second tangent of circle: **P4** Specify radius of circle <1.0000>: **1.25**

Command:

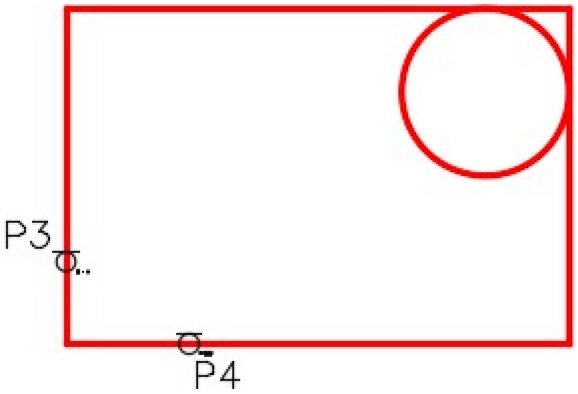


Figure Step 6A

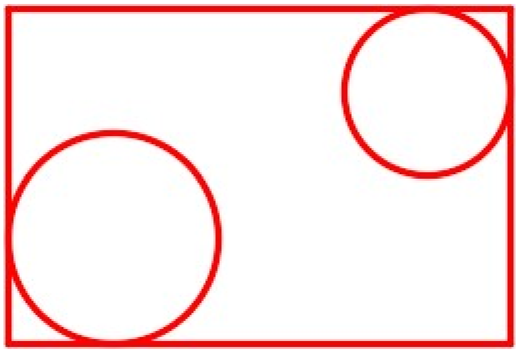


Figure Step 6B

AUTHOR'S COMMENTS: Keep in mind that you do not enter the P3 or P4 on the keyboard. They simply mean Pick 3 and Pick 4 of this workalong.

Step 7

Enter the CIRCLE command and the TTR option, as shown below, to insert the middle 2 diameter circle. (Figure Step 7A and 7B)

Command: **CIRCLE**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **TTR**

Specify point on object for first tangent of circle: **P5**

Specify point on object for second tangent of circle: **P6**

Specify radius of circle <1.2500>: **1**

Command:

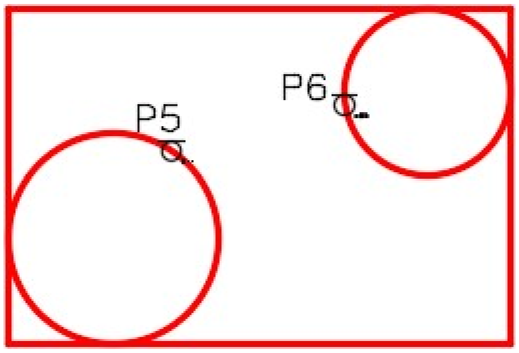


Figure Step 7A

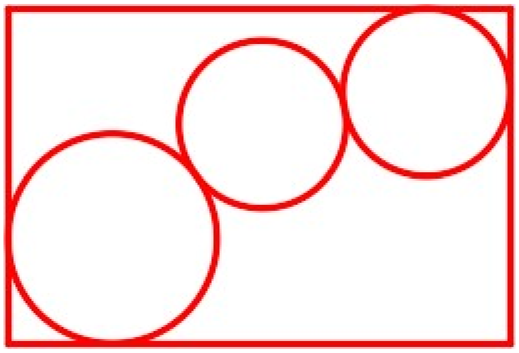


Figure Step 7B

Step 8

Save and close the drawing.

MUST KNOW: A very useful feature in AutoCAD is being able to undo a command or redo an undo. All of the commands in the current drawing session can be undone, one at a time. The last undo only can be reversed with a redo.To undo the last command, enter the U command, press CTRL+Z, or click the icon as shown in the figure on the right.

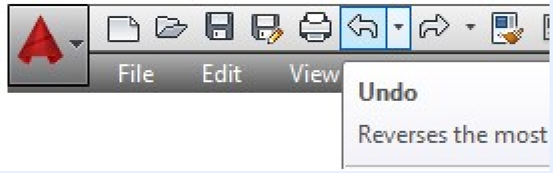
Command: **U**

If more then one U is entered, it will step back one command at time. This is useful if you made an error several commands earlier.

Command: **U**

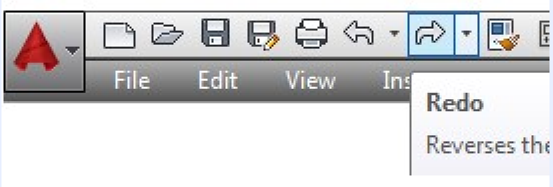
Command: **U**

Command: **U**



Enter the REDO command, press CTRL+Y, or click the icon as shown in the figure on the right to redo the undo. The redo will undo the last undo only.

Command: **REDO**



Drafting Lesson: Reading Dimensions for Circles and Arcs

Circles and arcs are dimensioned as follows:

Circles are dimensioned with their diameter. For example: 2.0 DIA.

Arcs are dimensioned with their radius.

For example: 1.5 R

When there is more than one circle of the same diameter, it is only dimensioned once. For example: 0.5 DIA., 4 PLACES

Sometimes multiple arcs are dimensioned as *typical* (TYP.). For example: 2.0 R TYP. Typical simply means that there is at least one more arc of the same size. See Figure 7-8

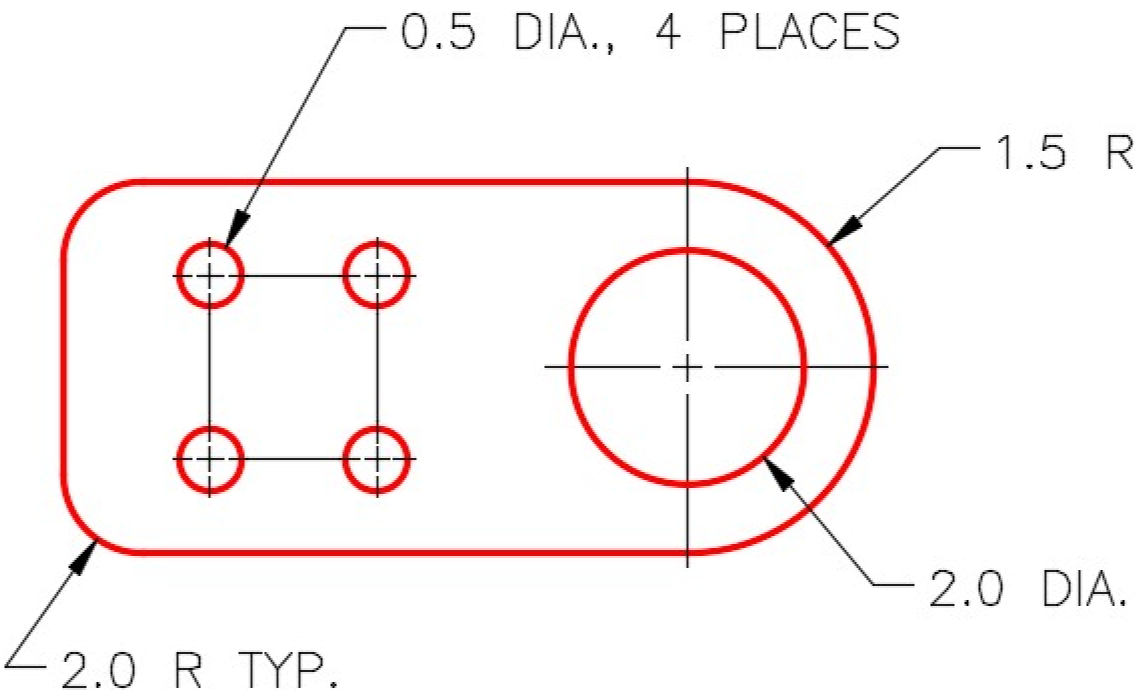


Figure 7-8 Reading Dimensions

Key Principles

Key Principles in Module 7

1. A circle is 360 degrees and can be divided into four quadrants.
2. All points on a circle are at a given distance from is center point.
3. An arc is defined as an open curve in which all points are the same distance from its center point.
4. By default, AutoCAD draws arcs counterclockwise.
5. When drawing arcs it is best to specify the center point first, the start point that sets the radius second, and an angle to locate the endpoint third.
6. A very useful feature in AutoCAD is being able to undo a command or redo an undo.

Lab Exercise 7-1

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 07-1	2D English	Inches

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Object	All lines, circles and arcs	Red

Step 1

Start a new drawing using the template shown above.

Step 2

Save and name the drawing with the name shown above.

Step 3

Setup the layers using the Layering Scheme shown above.

Step 4

Draw the object shown in the figure. In this drawing, all objects should be on layer: Object and display red. (Figure Step 4A and 4B)

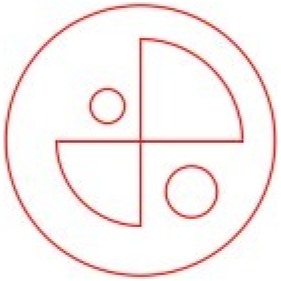


Figure Step 4A
Completed Drawing

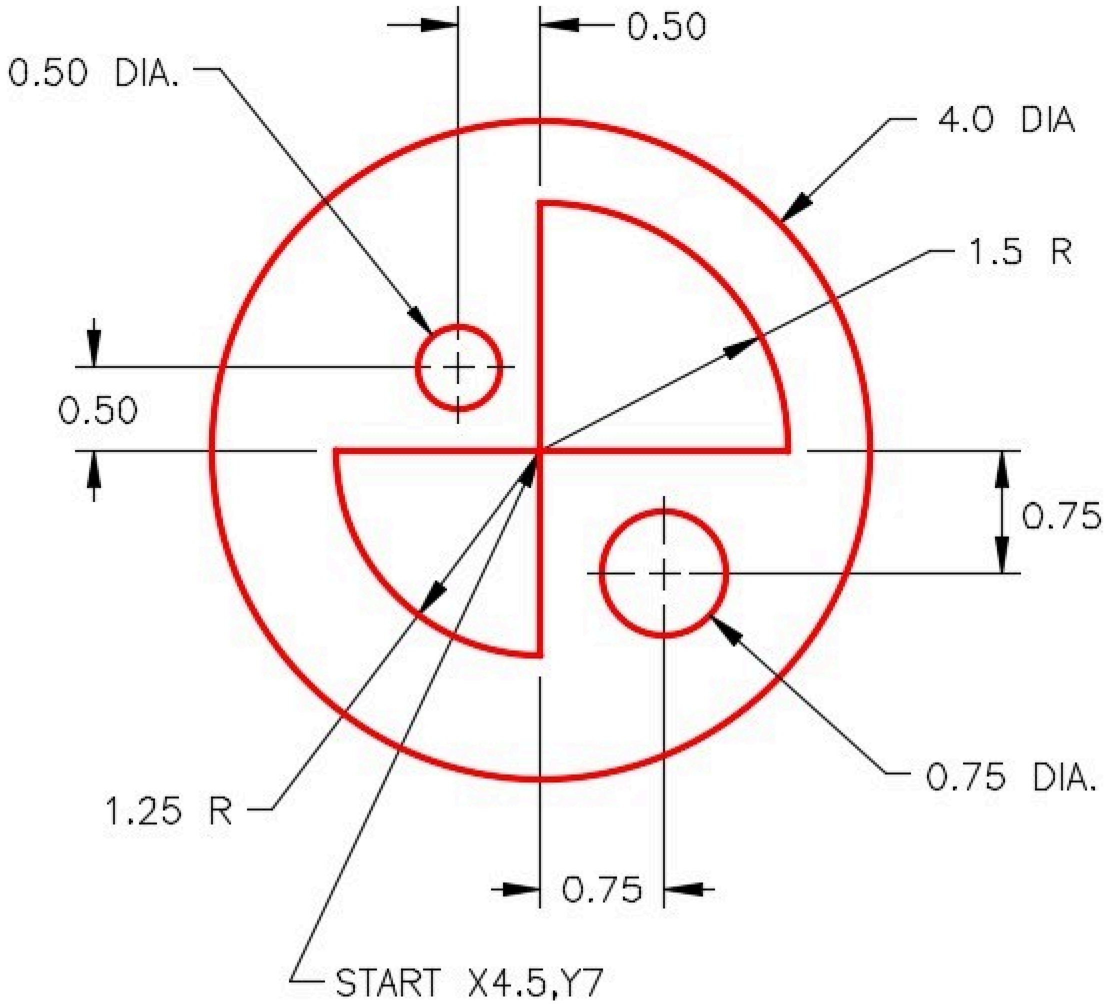

0.50
0.50 DIA.
4.0 DIA
1.5 R
0.50
0.75
0.75 DIA.
1.25 R
0.75
START X4.5,Y7

Figure Step 4B

Step 5

Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Inches.

Step 6

Check the drawing's accuracy with the key.

Step 7

If there are any errors, turn layer: Key off and correct the drawing. If necessary, start a new drawing and draw it again.

Step 8

Turn layer Key on. If the drawing still inaccurate, go back to Step 7.

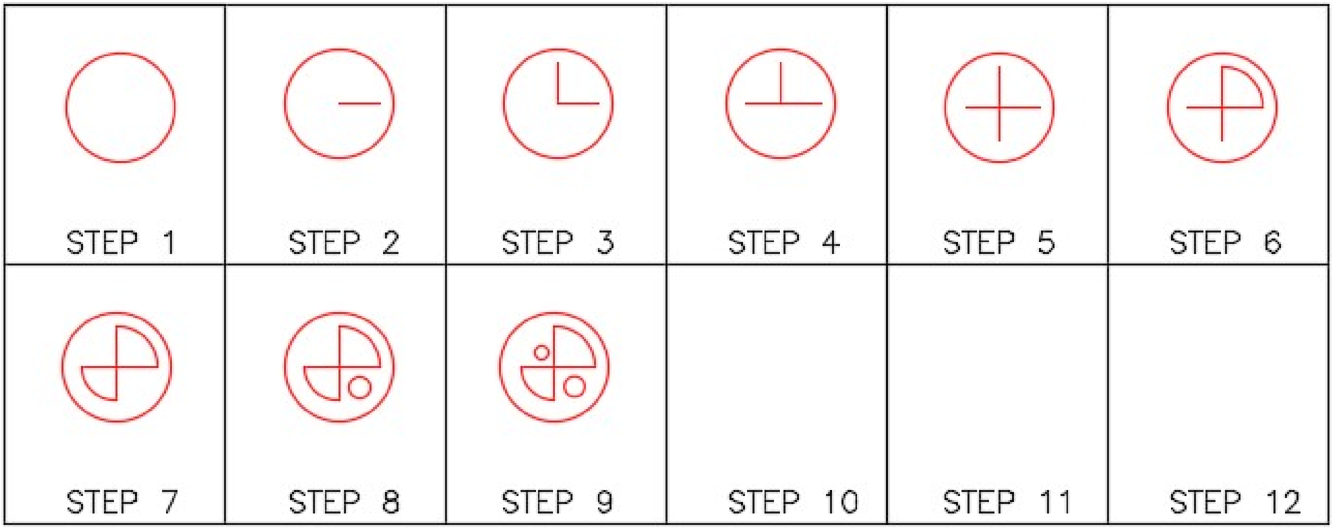
Step 9

Turn layer Key off.

Step 10

Save and close the drawing.

AUTHOR’S CONSTRUCTION TECHNIQUES: The following steps are the construction technique suggested by the author to help you learn how to construct objects using AutoCAD. It is only the suggested method and if you can complete the drawing accurately using a different construction technique, that is what is important. You may want to compare your construction technique with the authors.


STEP 1
STEP 2
STEP 3
STEP 4
STEP 5
STEP 6
STEP 7
STEP 8
STEP 9
STEP 10
STEP 11
STEP 12

AUTHOR’S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hint. If you get stuck and cannot complete it on your own, use the following hint to help you.

Hint 1

(Figure Hint 1)

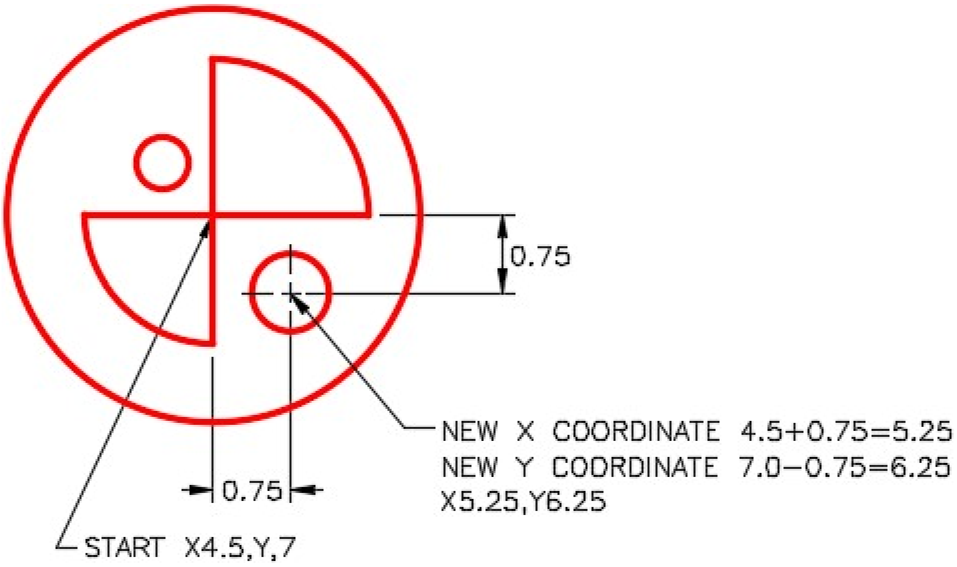

0.75
0.75
NEW X COORDINATE 4.5+0.75=5.25
NEW Y COORDINATE 7.0−0.75=6.25
X5.25,Y6.25
START X4.5,Y,7

Figure Hint 1

Lab Exercise 7-2

Time allowed: 40 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 07-2	2D Metric	Millimeters

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Object	All lines, circles and arcs	Red

Step 1

Start a new drawing using the template shown above.

Step 2

Save and name the drawing with the name shown above.

Step 3

Setup the layers using the Layering Scheme shown above.

Step 4

Draw the object shown in the figure. In this drawing, all objects should be on layer: Object and display red. (Figure Step 4A and 4B)

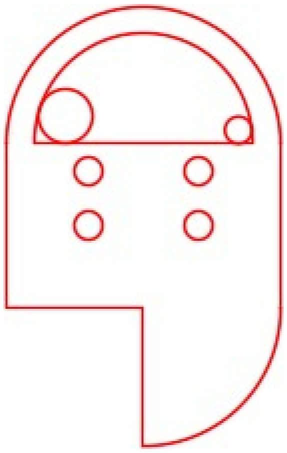


Figure Step 4A
Completed Drawing

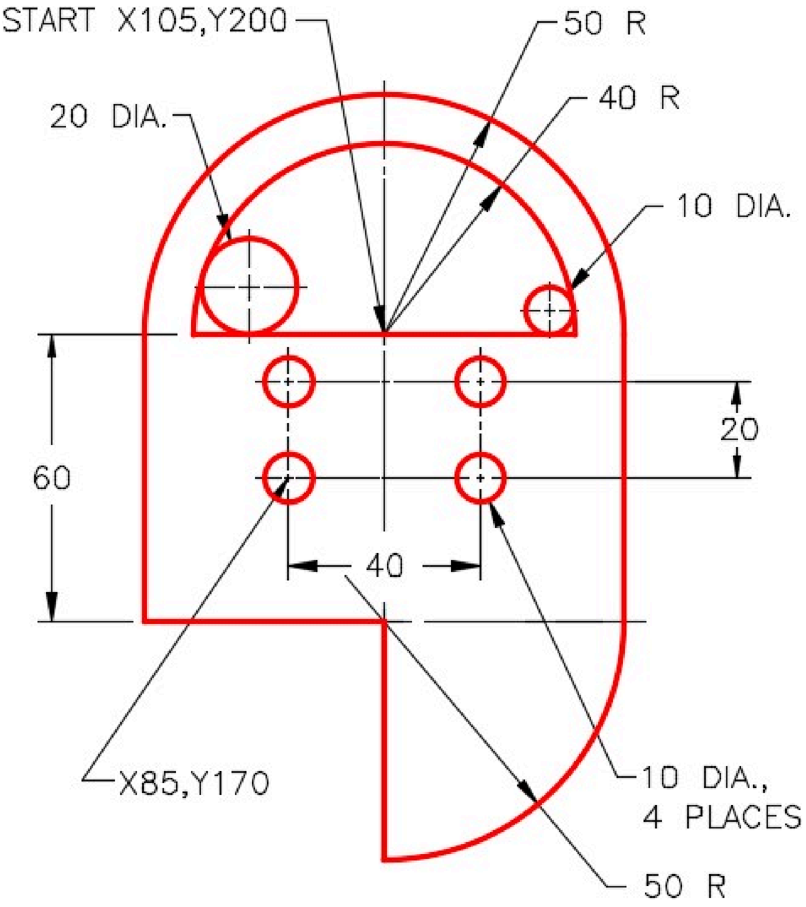

START X105,Y200
50 R
20 DIA.
40 R
10 DIA.
60
20
40
X85,Y170
10 DIA.,
4 PLACES
50 R

Figure Step 4B

Step 5

Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Millimeters.

Step 6

Check the drawing's accuracy with the key.

Step 7

If there are any errors, turn layer Key off and correct the drawing. If necessary, start a new drawing and draw it again.

Step 8

Turn layer: Key on. If the drawing still inaccurate, go back to Step 7.

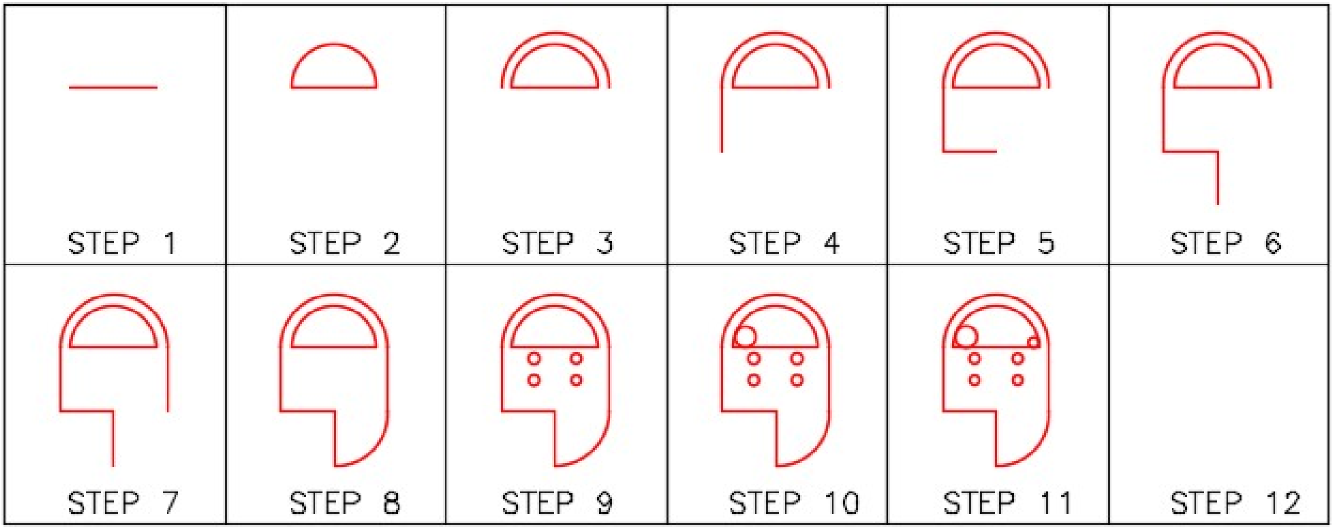
Step 9

Turn layer: Key off.

Step 10

Save and close the drawing.

AUTHOR'S CONSTRUCTION TECHNIQUES: The following steps are the construction technique suggested by the author to help you learn how to construct objects using AutoCAD. It is only the suggested method and if you can complete the drawing accurately using a different construction technique, that is what is important. You may want to compare your construction technique with the authors.



AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

Draw the two circles using the TTR option. (Figure Hint 1)

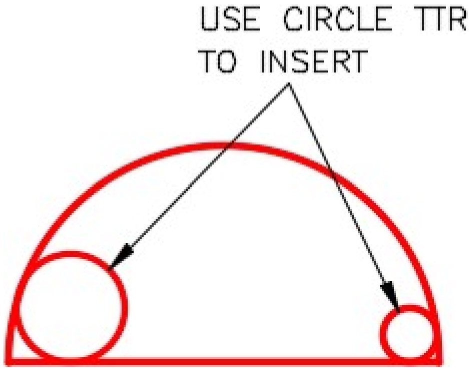


Figure Hint 1

Hint 2

Draw the four smaller circles using the @ as shown below. (Figure Hint 2)

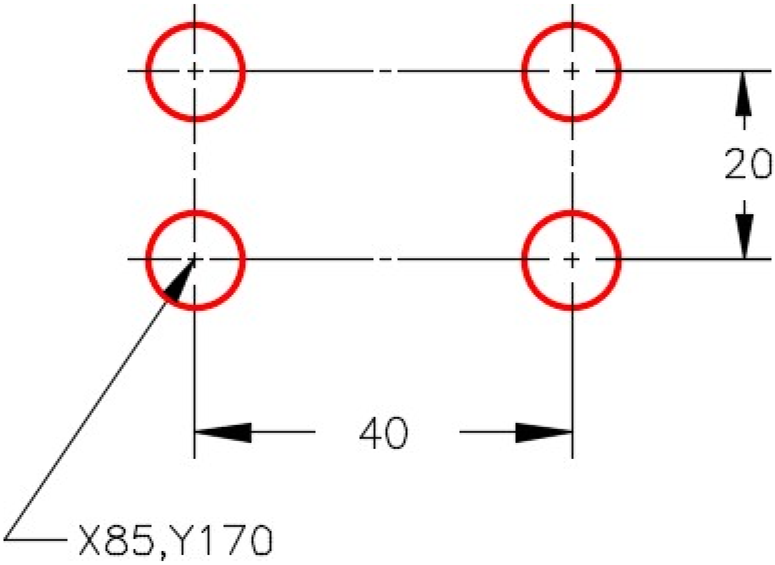


Figure Hint 2

Command: **C**

CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **85,170**

Specify radius of circle or [Diameter]: **D**

Specify diameter of circle: **10**

```
Command: C
CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: @40,0
Specify radius of circle or [Diameter] <5.0000>:
Command: C
CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: @0,20
Specify radius of circle or [Diameter] <5.0000>:
Command: C
CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: @-40,0
Specify radius of circle or [Diameter] <5.0000>:
Command:
```

Module 8: Object Snap

Learning Outcomes

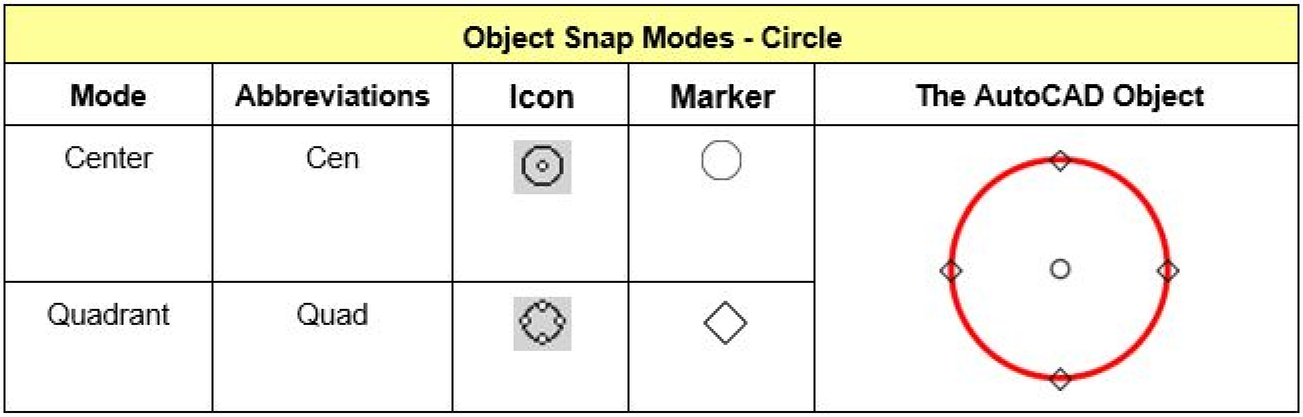
1. Describe object snap, AutoSnap, and manual object snap modes.
2. List the object snap modes for a line, a circle, and an arc.
3. Apply the OSNAP command to set the object snap modes for AutoSnap.
4. Describe how to enable and disable object snap and its importance when creating drawings.
5. Use object snap to complete drawings.

Object Snap

Object snap allows you to immediately locate and attach to predefined object snap modes (locations) on existing drawing objects. All drawing objects have at least one object snap mode. In this module, the object snap modes for the drawing objects line, circle, and arc are taught. Many more will be taught throughout the AutoCAD 2D book.

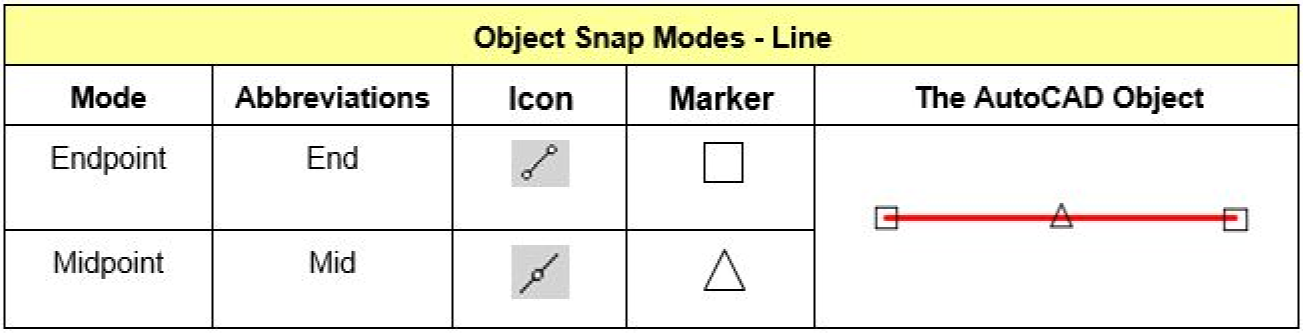
Using object snap allows you to draw quickly and accurately using the existing geometry in the drawing. This is done without having to know the coordinate locations of those objects. Object snap is one of the most important features in any CAD system.

Study Figure 8-1, 8-2, and 8-3. They show the *object snap modes* and their locations for a line, a circle, and an arc. A line has 3 snap mode locations that can be snapped to while circles and arcs have 5 each. When requested, AutoCAD can find and snap to these exact locations.



Object Snap Modes - Circle				
Mode	**Abbreviations**	**Icon**	**Marker**	**The AutoCAD Object**
Center	Cen	⊙	○	
Quadrant	Quad		◇	

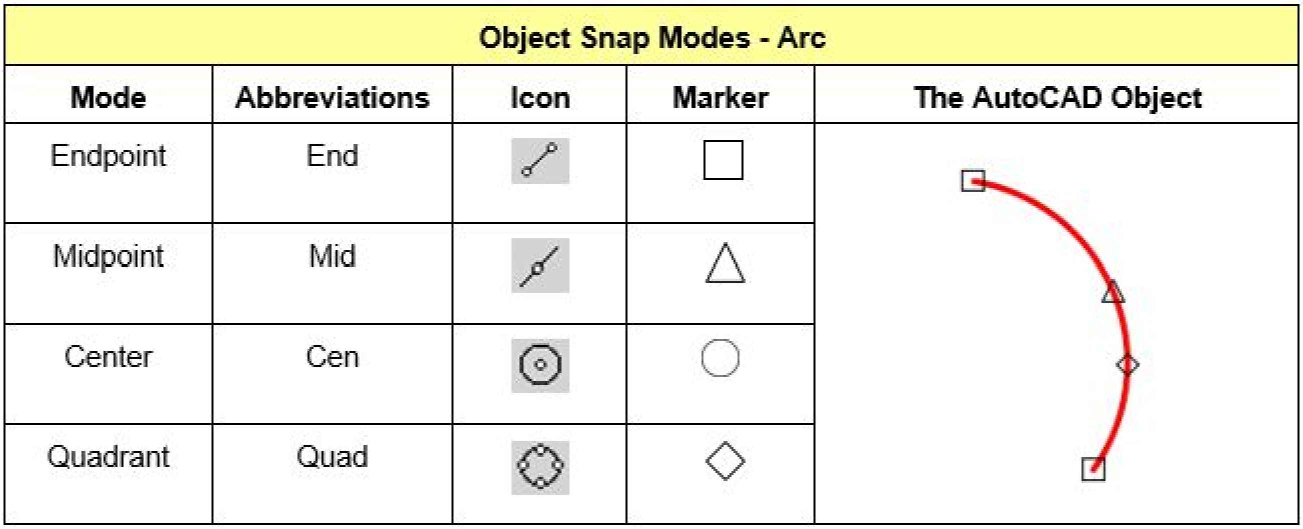
Figure 8-1 Object Snap Modes for a Circle



Object Snap Modes - Line

Mode	Abbreviations	Icon	Marker	The AutoCAD Object
Endpoint	End		□	
Midpoint	Mid		△	

Figure 8-2 Object Snap Modes for a Line



Object Snap Modes - Arc

Mode	Abbreviations	Icon	Marker	The AutoCAD Object
Endpoint	End		□	
Midpoint	Mid		△	
Center	Cen		○	
Quadrant	Quad		◇	

Figure 8-3 Object Snap Modes for an Arc

AutoSnap

AutoSnap is a visual aid to help you see and use object snap modes (locations) more efficiently. It allows you to set the object snap modes that AutoSnap will find automatically when object snap is enabled. Object snap can be enabled or disabled anytime by pressing F3. While a command is active, AutoSnap displays a *marker* and a *tool tip* when the cursor is moved over an object snap mode location that it has been set to find. In Figure 8-4, AutoSnap is indicating that it found the endpoint of the line. Note that the Graphic cursor does not have to be located exactly at the endpoint of the line. It only has to be close.

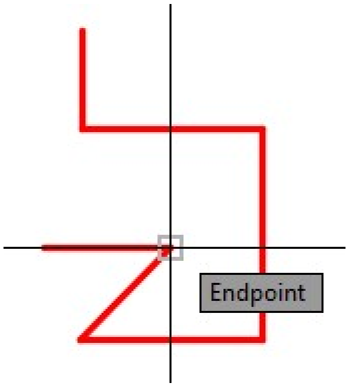


Figure 8-4

Geometry Lesson: Finding the Center of a Square or Rectangle

Using geometry, the center of a square or a rectangle can easily and accurately be located without knowing its size or doing any math. To do this, either one of the following methods can be used.

Method 1

Step 1

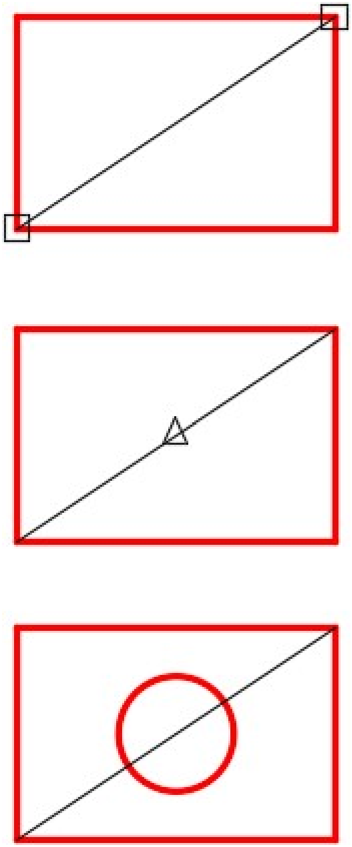
Draw a construction line between any two diagonal corners by snapping to the endpoints.

Step 2

The midpoint of the construction line that you inserted in Step 1 is the center of the square or rectangle.

Step 3

In this step, a circle is inserted at the midpoint of the line. The circle is located exactly in the center of the square or rectangle.



Method 1

Method 2

Step 1

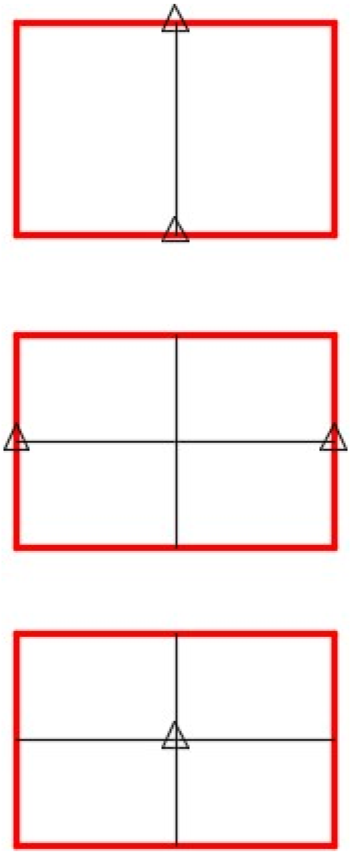
Draw a line between the midpoints of two opposite sides of the rectangle or square.

Step 2

Draw another line between the midpoint of the other opposite sides of the rectangle or square.

Step 3

The midpoint of either line and the intersection of the two lines is the center of the rectangle or square. Actually, one line would have been enough to find the center.



Method 2

Geometry Lesson: Using Construction Objects

The best way to construct most drawings is to draw construction objects as an aid. There will be more and more drawings like this in future lab exercises. Construction objects are objects that are drawn by you to be used for construction only and will not be part of the finished drawing. It is important to save those objects for later use or to see how the construction was preformed.

From this point forward, in all workalongs and lab exercises, create a layer and name it Construction, color 253. Draw all construction objects on this layer and do not delete them. When you complete the drawing, freeze layer Construction. When required, simply thaw layer Construction to display the construction objects. See Figure 8-5. The drawing on the left has layer Construction thawed while the drawing on the right has layer Construction frozen.

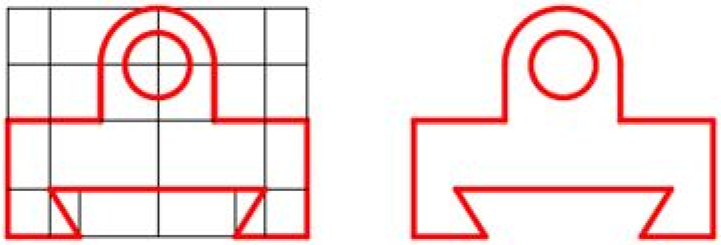


Figure 8-5
Construction Objects

AUTHOR'S COMMENTS: Some drawings do not require any construction objects to be drawn while others have a lot.

Using Construction Objects

AUTHOR'S COMMENTS: There are many ways to draw using AutoCAD and it seems that each CAD operator has his/her own drawing method. There is no right or wrong way to draw as long as the drawing is complete and accurate. Layers are used extensively by companies to keep the drawings standardized and consistent within the company. Layers are also used when plotting the drawings to achieve lineweight as you will see later in the AutoCAD 2D book.

I recommend using a construction layer. The construction layer contains drawing objects that are used to construct the object being drawn but in the end are not part of the actual drawing itself.

After you complete the drawing, freeze the construction layer so it cannot be seen by others. If in the future, you have to make revisions to the drawing simply thaw the construction layer and use the construction objects. That means you do not have to redraw them, which will save you time. Some drawings do not have any construction objects while other have a lot of them.

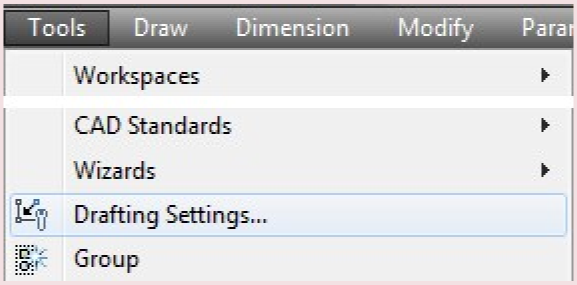
As an AutoCAD instructor, it also help me to see how student's are constructing drawings. This is especially true for online students. When I cannot observe a student drawing, I use the construction layer to help me see the student's drawing techniques.

When I draw in AutoCAD, I draw all objects on the layer Construction and complete the drawing. After I complete the drawing, I change the layers of the objects I want to be part of the finished drawing. The objects that remain on layer Construction are the construction objects. There are many ways to draw and each person has to choose his/her method that works the best for them. As you work your way through the book, you will learn many techniques and drawing methods. Over time, you will develop your own method of drawing that suits you best

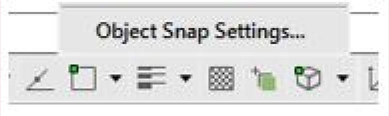
AutoCAD Command: OSNAP

The OSNAP command is used to open the Drafting Settings dialogue box which allows you to set object snap mode defaults for Autosnap. The function key F3 toggles osnap off and on.

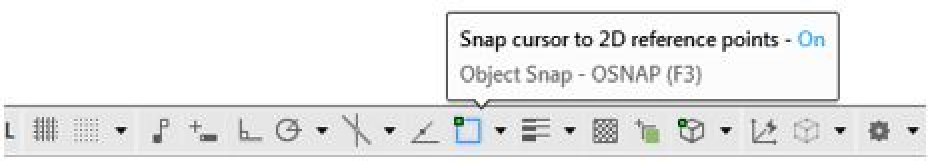
Shortcut: **OS**


Tools
Draw
Dimension
Modify
Para
Workspaces
CAD Standards
Wizards
Drafting Settings...
Group

Tools Pull-down


Object Snap Settings...

Right click Status Bar



Snap cursor to 2D reference points - On
Object Snap - OSNAP (F3)

Left click Status Bar

WORKALONG: Using Object Snap

Step 1

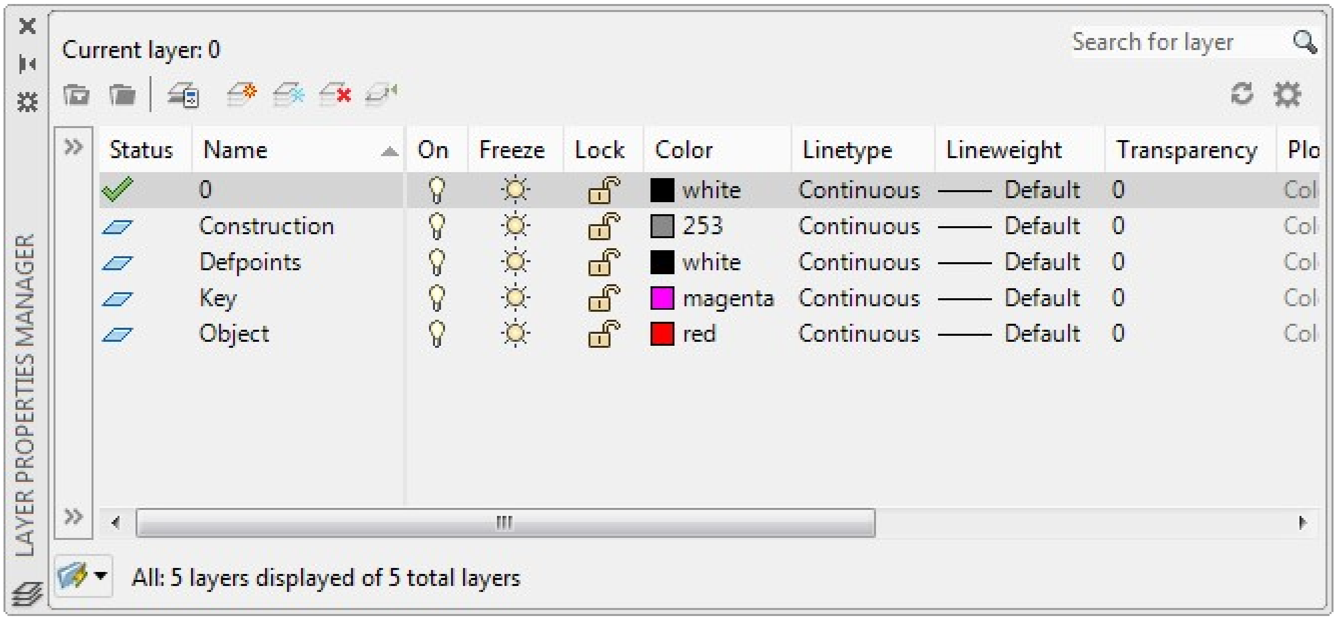
Start a new drawing using the template: 2D English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 08-1.

Step 3

Create layer: Object, color red and layer: Construction, color 253. (Figure Step 3)


LAYER PROPERTIES MANAGER
Current layer: 0
Search for layer

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight	Transparency	Plo
✓	0	💡	☼	🔓	■ white	Continuous	—— Default	0	Col
▱	Construction	💡	☼	🔓	■ 253	Continuous	—— Default	0	Col
▱	Defpoints	💡	☼	🔓	■ white	Continuous	—— Default	0	Col
▱	Key	💡	☼	🔓	🟪 magenta	Continuous	—— Default	0	Col
▱	Object	💡	☼	🔓	🟥 red	Continuous	—— Default	0	Col

All: 5 layers displayed of 5 total layers

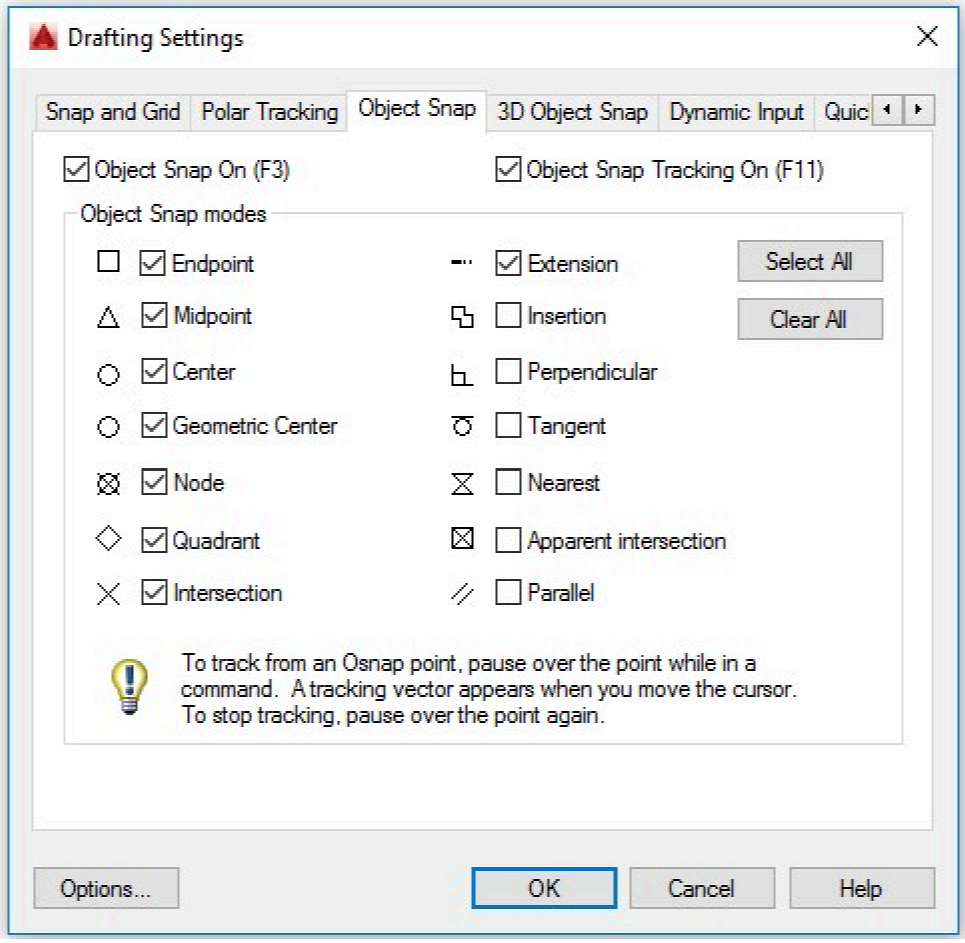
Figure Step 3

Step 4

Enter the OSNAP command to open Drafting Settings dialogue box.

Step 5

Step 4 will open the Drafting Settings dialogue box. Enable the Object Snap tab. In the Object Snap modes area, enable the modes to match the figure. Click OK to close the dialogue box. (Figure Step 5)


Drafting Settings
Snap and Grid
Polar Tracking
Object Snap
3D Object Snap
Dynamic Input
Quic
Select All
Clear All
Options...
OK
Cancel
Help

☑ Object Snap On (F3)

☑ Object Snap Tracking On (F11)

Object Snap modes

- ☑ Endpoint
- ☑ Midpoint
- ☑ Center
- ☑ Geometric Center
- ☑ Node
- ☑ Quadrant
- ☑ Intersection
- ☑ Extension
- ☐ Insertion
- ☐ Perpendicular
- ☐ Tangent
- ☐ Nearest
- ☐ Apparent intersection
- ☐ Parallel

To track from an Osnap point, pause over the point while in a command. A tracking vector appears when you move the cursor. To stop tracking, pause over the point again.

Figure Step 5

AUTHOR'S COMMENTS: Autosnap is now set to look for the object snap modes Endpoint, Midpoint, Center, Quadrant, Intersection, and Extension. These settings can be changed at anytime. I do not suggest enabling too many or all of the snap modes at one time as this may cause confusion when drawing.

Step 6

Enable object snap by pressing the function key F3. Notice how the Object Snap icon in the Status bar will display enabled as shown in Figure Step 6A. Press F3 again and note how the Object Snap icon displays disabled as shown in Figure Step 6B.

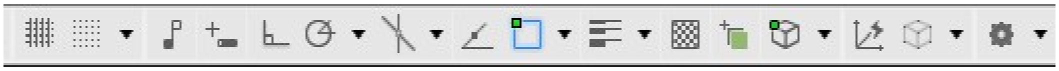


Figure Step 6A

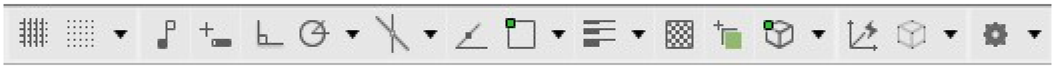
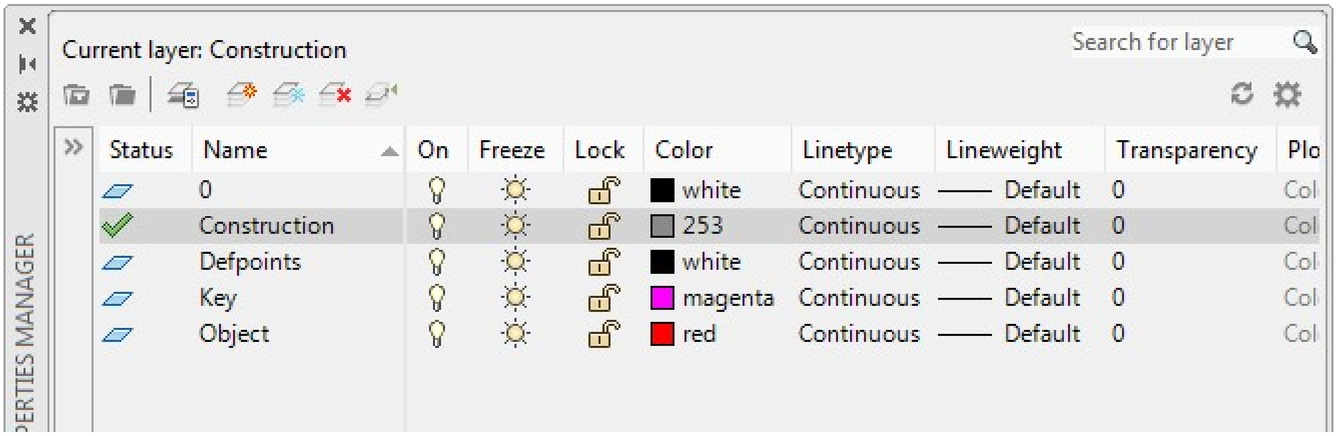


Figure Step 6B

Step 7

Set layer: Construction as the current layer. (Figure Step 7)



Current layer: Construction

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight	Transparency	Plo
	0				■ white	Continuous	—— Default	0	Col
✓	Construction				■ 253	Continuous	—— Default	0	Col
	Defpoints				■ white	Continuous	—— Default	0	Col
	Key				🟪 magenta	Continuous	—— Default	0	Col
	Object				🟥 red	Continuous	—— Default	0	Col

Figure Step 7

Step 8

Using the dimensioned drawing as a reference, enter the LINE command and draw the 4 lines to form the outer rectangle. (Figure Step 8A and 8B)

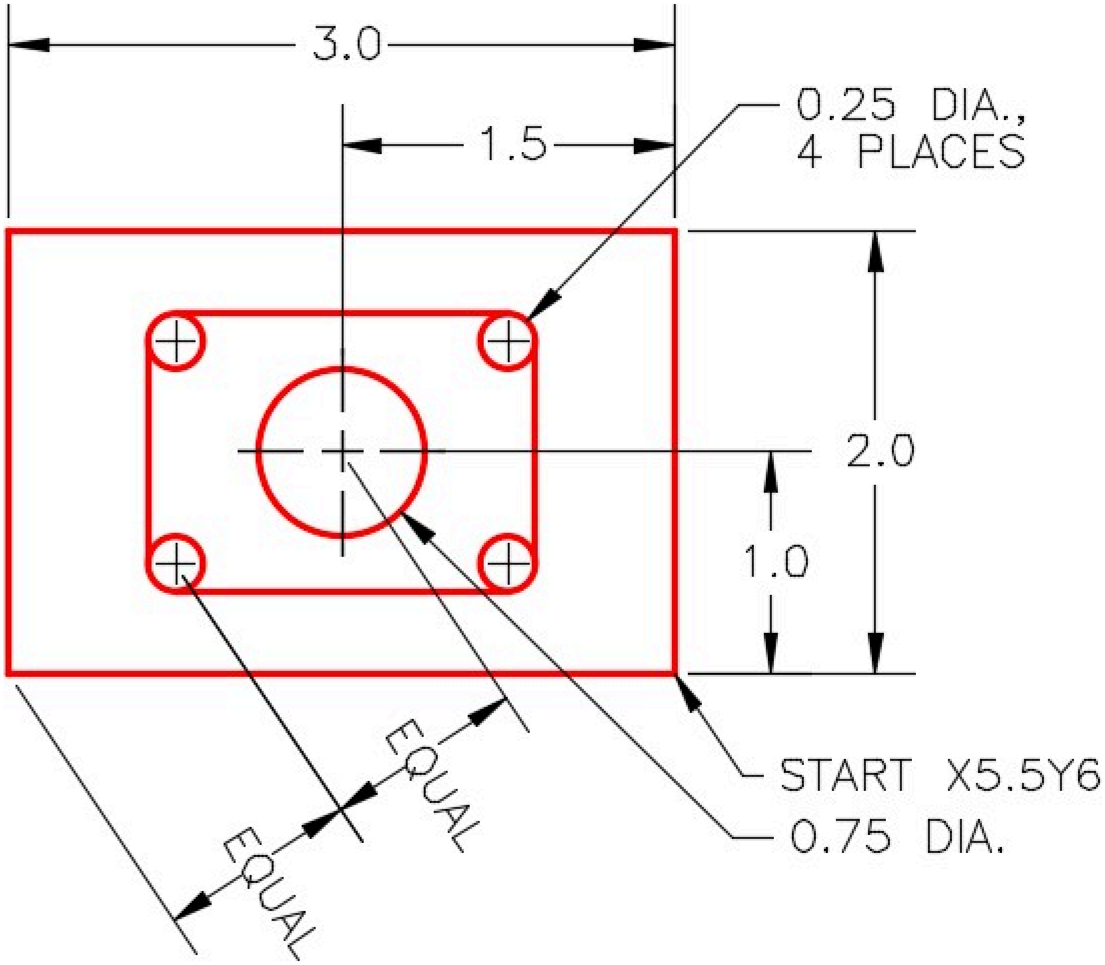

3.0
1.5
0.25 DIA.,
4 PLACES
2.0
1.0
START X5.5Y6
0.75 DIA.
EQUAL
EQUAL

Figure Step 8A

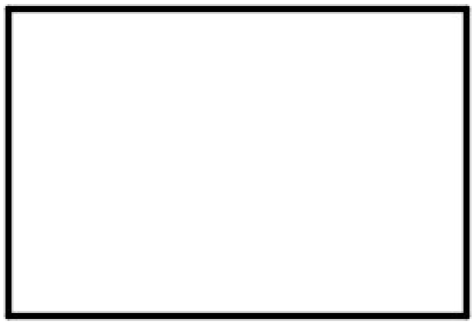


Figure Step 8B

AUTHOR'S COMMENTS: Since layer Construction is the current layer, the lines in your drawing should be colored gray.

Step 9

Press F3 to enable object snap. Check to ensure that it is enabled by checking its status in the Status bar.

Step 10

Enter the LINE command and move the cursor on the top horizontal line. Move it approximately to the midpoint of the line. When the AutoSnap marker displays the midpoint icon and the tool tip, pick it. Figure Step 10)

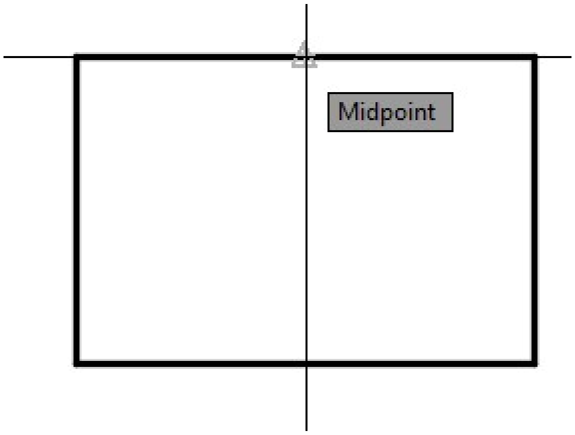


Figure Step 10

AUTHOR'S COMMENTS: Note that each object snap mode marker displays a different icon. For midpoint, it is a triangle.

Step 11

Move the cursor to the bottom horizontal line and do the same as Step 10. Press Enter to exit the command. You now have a vertical line from the midpoint of the top line to midpoint of the bottom line. (Figure Step 11A and 11B)

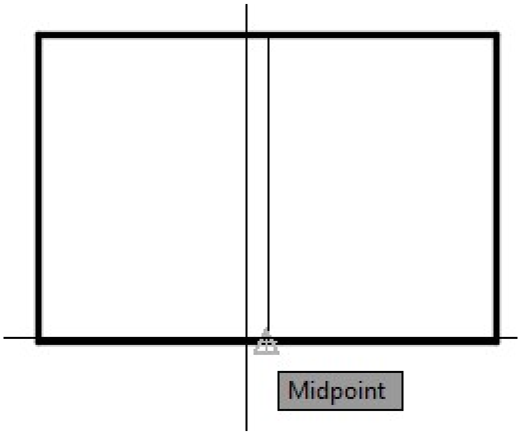


Figure Step 11A

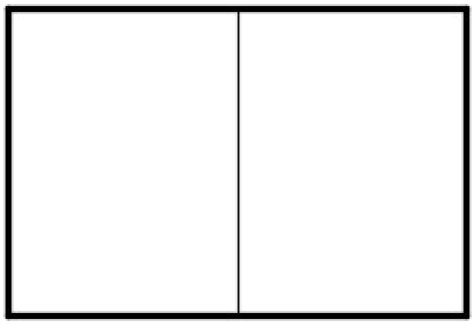


Figure Step 11B

AUTHOR'S COMMENTS: Note that you do not have to have Graphic cursor exactly at the object's snap mode location. As long as the AutoSnap marker is displayed when you pick it, AutoCAD will snap exactly to that location. It takes a little practice to using object snap.

Step 12

Enter the CIRCLE command. To locate the center of the circle, move the cursor to the midpoint of the vertical line that you drew in Step 11. When the midpoint icon and the tool tip displays, pick it. Enter a D, for diameter, and then 0.75 to complete the circle. (Figure Step 12A and 12B)

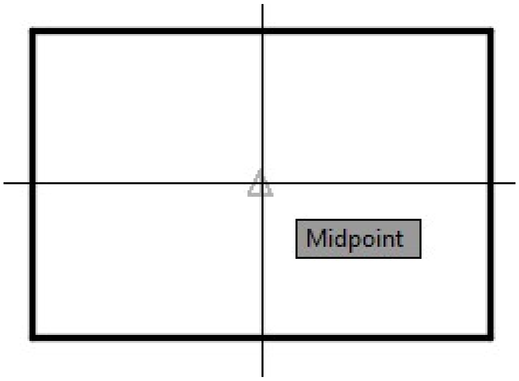


Figure Step 12A

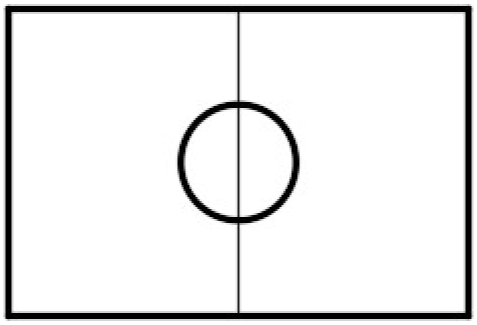


Figure Step 12B

AUTHOR'S COMMENTS: The circle is located exactly in the center of the rectangle.

Step 13

Enter the LINE command and for the first point snap to the center of the circle. For the second point, snap to the endpoint in the top right corner. (Figure Step 13A, 13B, and 13C)

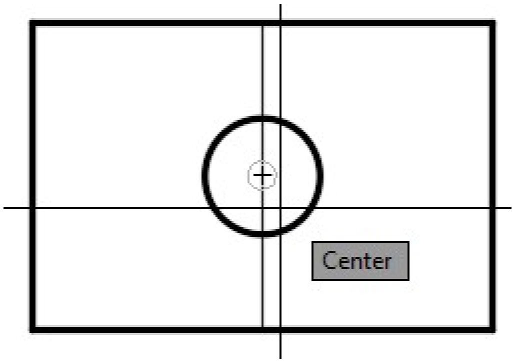


Figure Step 13A

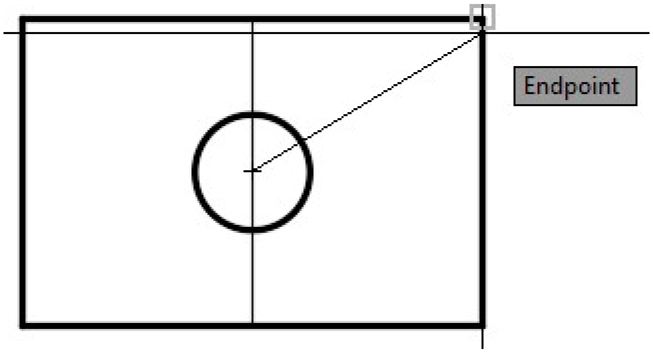


Figure Step 13B

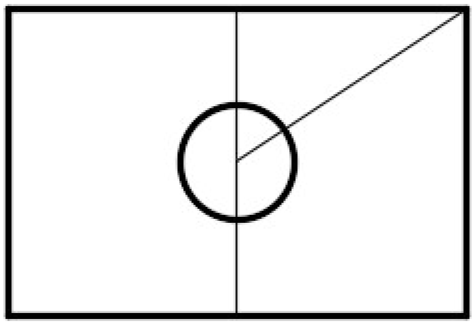


Figure Step 13C

USER TIP: The function key F3 toggles object snap enabled, or disabled. Although there are other ways to do this, try to get into the habit of using F3 as it will greatly improve your drawing speed.

Step 14

Repeat Step 13 three additional times to draw 3 lines from the center of the rectangle to each of the corners of the rectangle. (Figure Step 14)

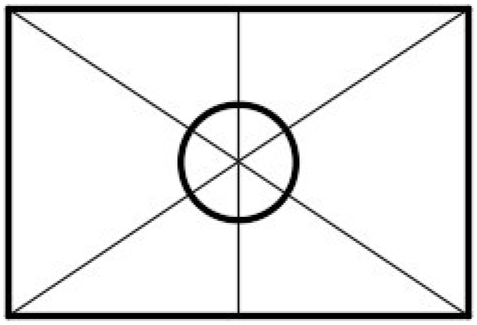


Figure Step 14

Step 15

Enter the CIRCLE command. For the center location of the circle, snap to the midpoint of one of the diagonal lines. Enter the radius of 0.125 or the diameter of 0.25. (Figure Step 15A and 15B)

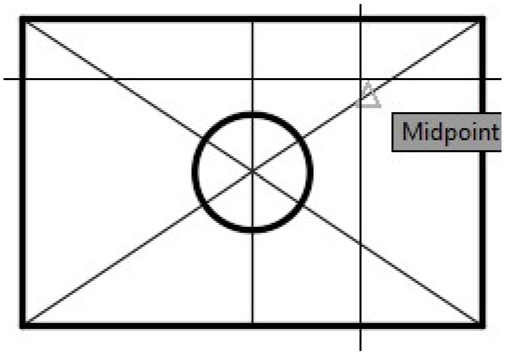


Figure Step 15A

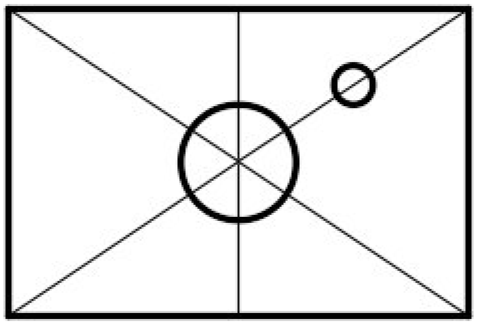


Figure Step 15B

Step 16

Repeat Step 15 three times to draw a circle at the midpoint of each inclined line. (Figure Step 16)

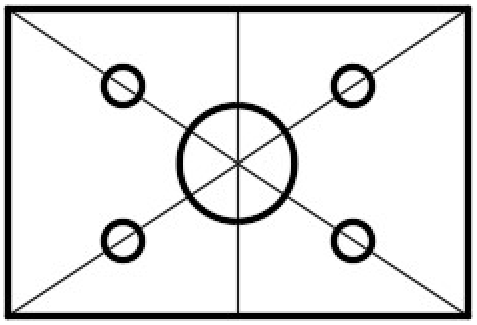


Figure Step 16

Step 17

Enter the LINE command and for the first point, snap to the top quadrant of the top left circle. For the second point, snap to the top quadrant of top right circle. (Figure Step 17A, 17B, and 17C)

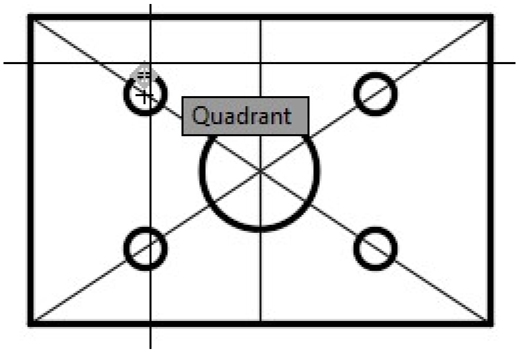


Figure Step 17A

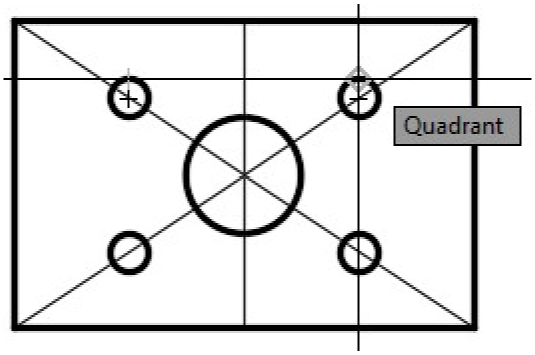


Figure Step 17B

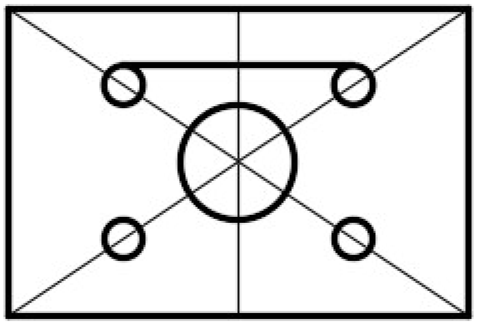


Figure Step 17C

AUTHOR'S COMMENTS: Zoom in on the drawing if you have to snap to the quadrant.

Step 18

Repeat Step 17 three additional times to draw 3 more lines. Ensure that you snap to the quadrant of the circle to locate the end of each line. (Figure Step 18)

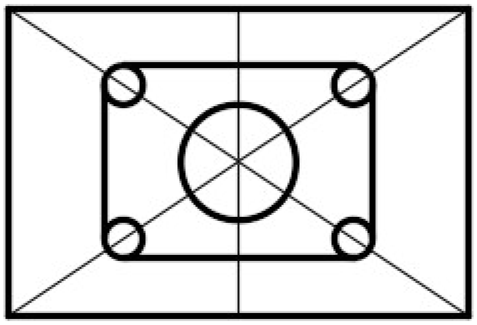


Figure Step 18

Step 19

Without entering a command, individually select the objects that you want to reside on layer Object. While the objects are selected, pull down the Layer toolbar list and click layer: Object. After you select the name, click inside the Graphic window and then press Esc to clear the selected objects. (Figure Step 19A and 19B)

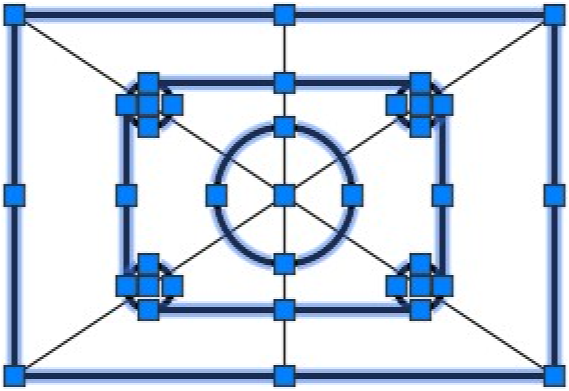


Figure Step 19A

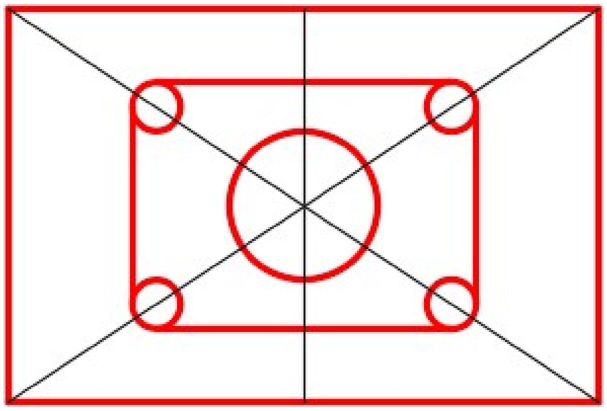


Figure Step 19B

AUTHOR'S COMMENTS: The objects you selected in Step 19 should display red. If they do not display red, redo Step 19. This process is a little tricky so it may have to practice it a few times to complete it successfully.

Step 20

Pull down the layer list again and freeze layer: Construction. Your drawing is complete and should appear as shown in the figure. (Figure Step 20A and 20B)

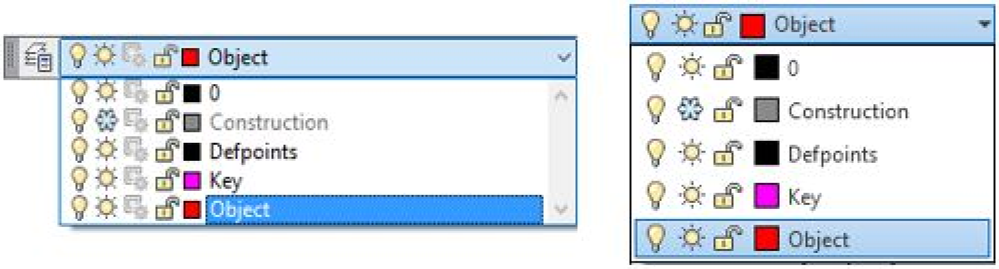

Object
0
Construction
Defpoints
Key
Object
Object
0
Construction
Defpoints
Key
Object

Figure Step 20A

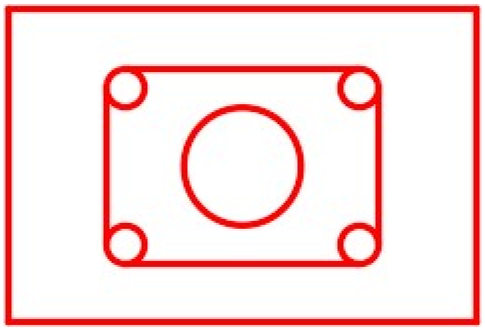


Figure Step 20B

Step 21

Save and close the drawing.

USER TIP: Whenever possible, disable object snap. When it is enabled at the wrong time during the drawing process, it can cause drawing problems. It is important to use object snap whenever necessary, but it is also important to disable it when you do not require it. It can be toggled enabled or disabled with the F3 key.

Manual Object Snap Modes

Object snap modes can be entered manually inside any command that uses them. This can be a handy feature for modes that are not used very often and are not enabled in AutoSnap. It also overrides the current settings to force AutoCAD to find the snap mode you entered. i.e.

Command: **LINE**

Specify first point: **end** of

(Inside the LINE command, you must start the next line at the end of an existing Line. Instead of using AutoSnap, enter the mode end on the keyboard to AutoCAD's prompt. When a line is picked, AutoCAD will snap to the closest endpoint on that line.)

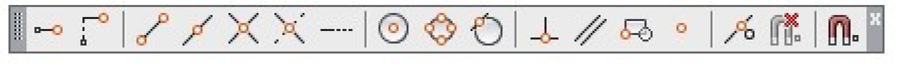
or

Command: **CIRCLE**

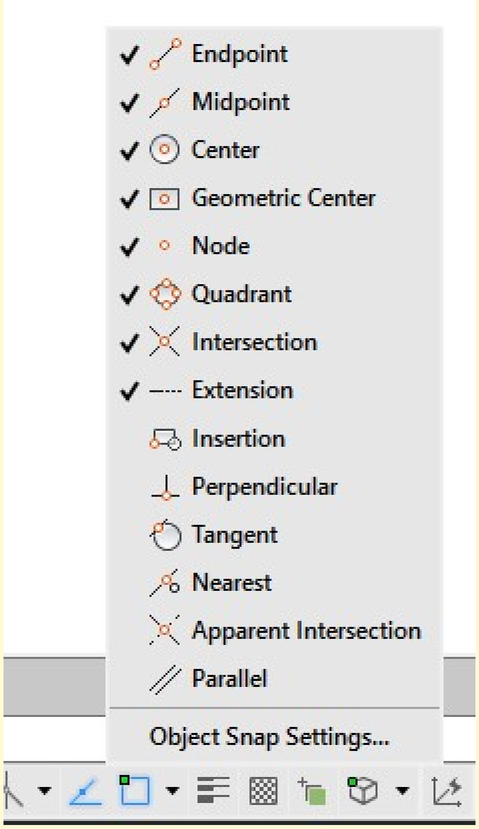
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **mid** of

(In the CIRCLE command, the snap mode mid is entered and AutoCAD responds of. Pick an existing object and AutoCAD will snap to its midpoint.)

USER TIP: When manually entering object snap modes, the Object Snap toolbar can be used rather than typing the object snap modes. A tool tip will open as you pass your cursor over each icon to see which mode each one represents.



Object Snap Toolbar


✓ Endpoint
✓ Midpoint
✓ Center
✓ Geometric Center
✓ Node
✓ Quadrant
✓ Intersection
✓ Extension
Insertion
Perpendicular
Tangent
Nearest
Apparent Intersection
Parallel
Object Snap Settings...

Right Click Status Bar

Key Principles

Key Principles in Module 8

1. Object snap is one of the most important features in any CAD system. Use object snaps to quickly and accurately draw using existing geometry in the drawing.
2. The function key F3 toggles object snap between enabled or disabled.
3. Use geometry and object snap rather than math to find the centers of squares or rectangles.
4. Disable object snap when you are not using it.
5. Draw construction objects on layer Construction and freeze that layer after you complete the drawing.
6. You can override or force AutoCAD to snap to an object snap location on an object by manually entering the osnap mode in the command line.

Lab Exercise 8-1

Time allowed: 30 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 08-1	2D English	Inches

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Construction	Construction objects	253
Object 1	All lines and arcs	Red
Object 2	All circles	Blue

Step 1

Start a new drawing using the template shown above.

Step 2

Setup the layers using the Layering Scheme shown above.

Step 3

Draw the object shown in the figure using the layering scheme. In this drawing, place the lines and arcs on layer: Object 1 and the circles on layer: Object 2. Lines and arcs should display red and the circles should display blue. (Figure Step 3A and 3B)

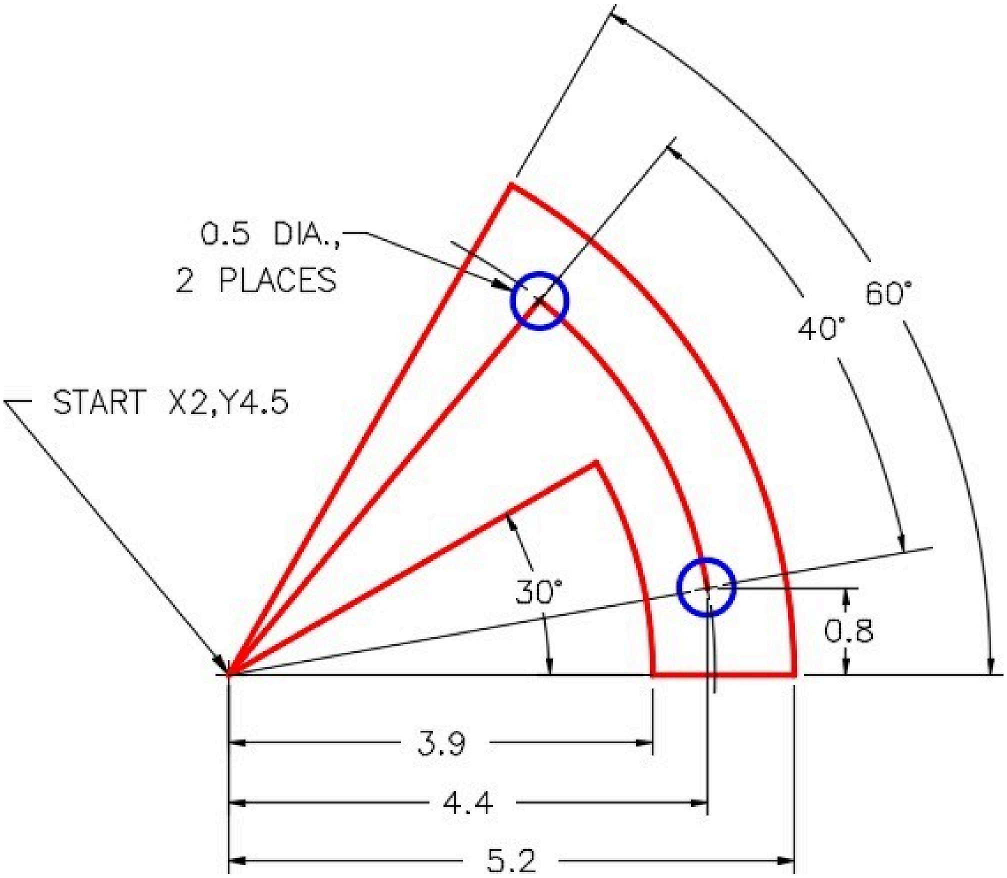

0.5 DIA.,
2 PLACES
START X2,Y4.5
60°
40°
30°
0.8
3.9
4.4
5.2

Figure Step 3A

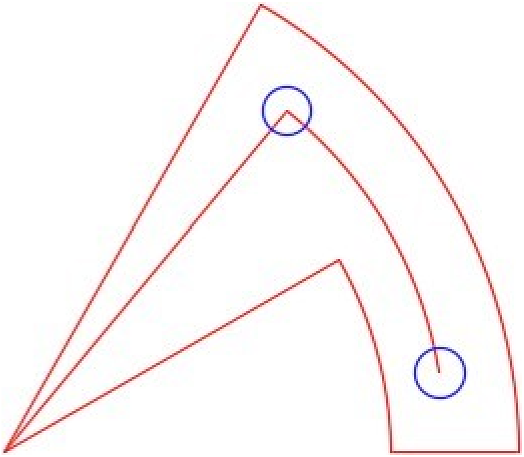


Figure Step 3B
Completed Drawing

Step 4

Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Inches.

Step 5

Check the drawing's accuracy with the key.

Step 6

If there are any errors, turn layer Key off and correct the drawing. If necessary, start a new drawing and draw it again.

Step 7

Turn layer Key on. If the drawing still inaccurate, go back to Step 6.

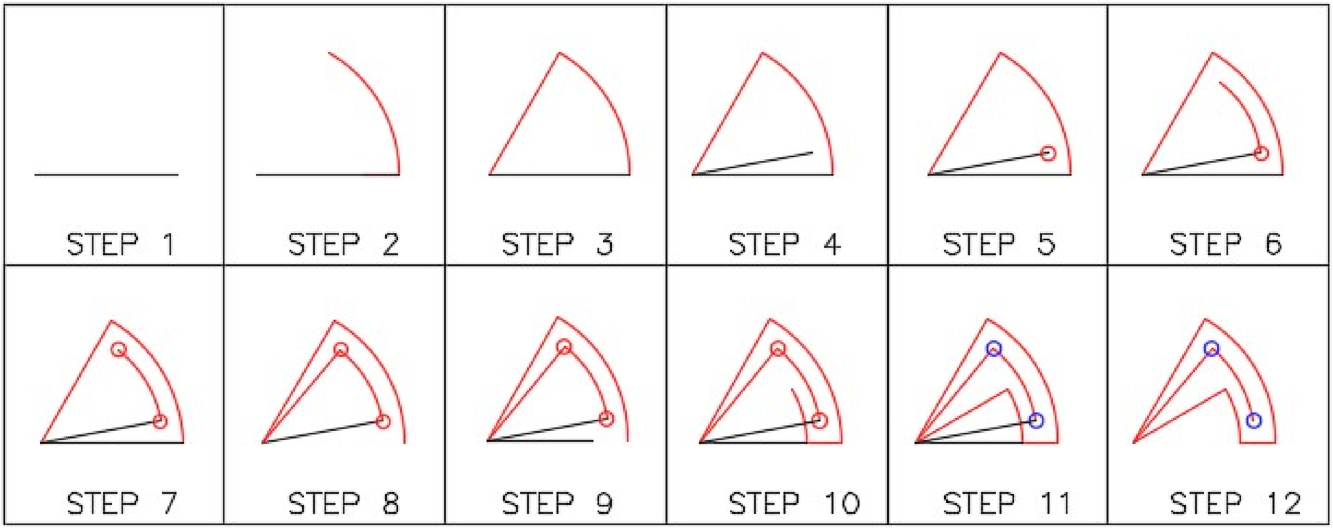
Step 8

Turn layer Key off.

Step 9

Save and close the drawing.

AUTHOR'S CONSTRUCTION TECHNIQUES: The following steps are the construction technique suggested by the author to help you learn how to construct objects using AutoCAD. It is only the suggested method and if you can complete the drawing accurately using a different construction technique, that is what is important. You may want to compare your construction technique with the authors.



AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

Use object snap modes and snap to as many locations as possible rather then entering XY coordinates on the keyboard. (Figure Hint 1)

Command: **ARC**

Specify start point of arc or [Center]: **C**

Specify center point of arc: *(end)* **P1**

Specify start point of arc: *(end)* **P2**

Specify end point of arc or [Angle/chord Length]: **A**

Specify included angle: **60**

Command:

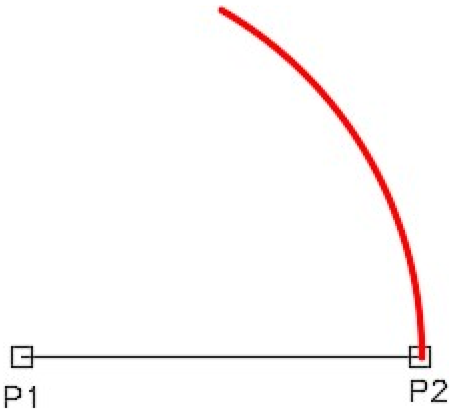


Figure Hint 1

Hint 2

Use object snap modes and snap to as many locations as possible rather then entering XY coordinates on the keyboard. (Figure Hint 2)

Command: **LINE**

Specify first point: *(end)* **P3**

Specify next point or [Undo]: *(end)* **P4**

Specify next point or [Undo]:

Command:

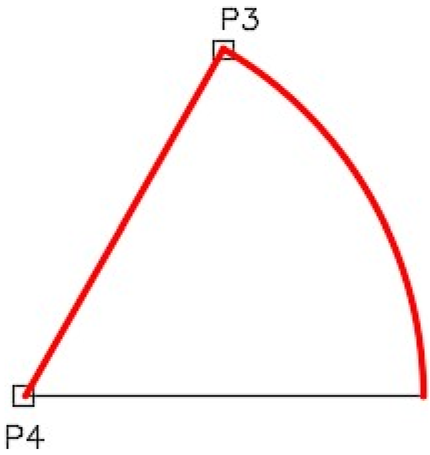


Figure Hint 2

Hint 3

There should be two construction objects in this drawing. (Figure Hint 3)

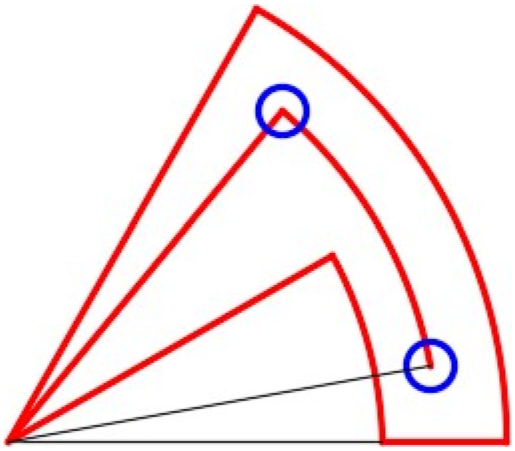


Figure Hint 3

Lab Exercise 8-2

Time allowed: 40 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 08-2	2D English	Inches

Layering Scheme

Layer Name	Objects on Layer	Color
Construction	Construction objects	253
Object 1	All lines and arcs	Red
Object 2	All circles	Blue

Step 1

Start a new drawing using the template shown above.

Step 2

Setup the layers using the Layering Scheme shown above.

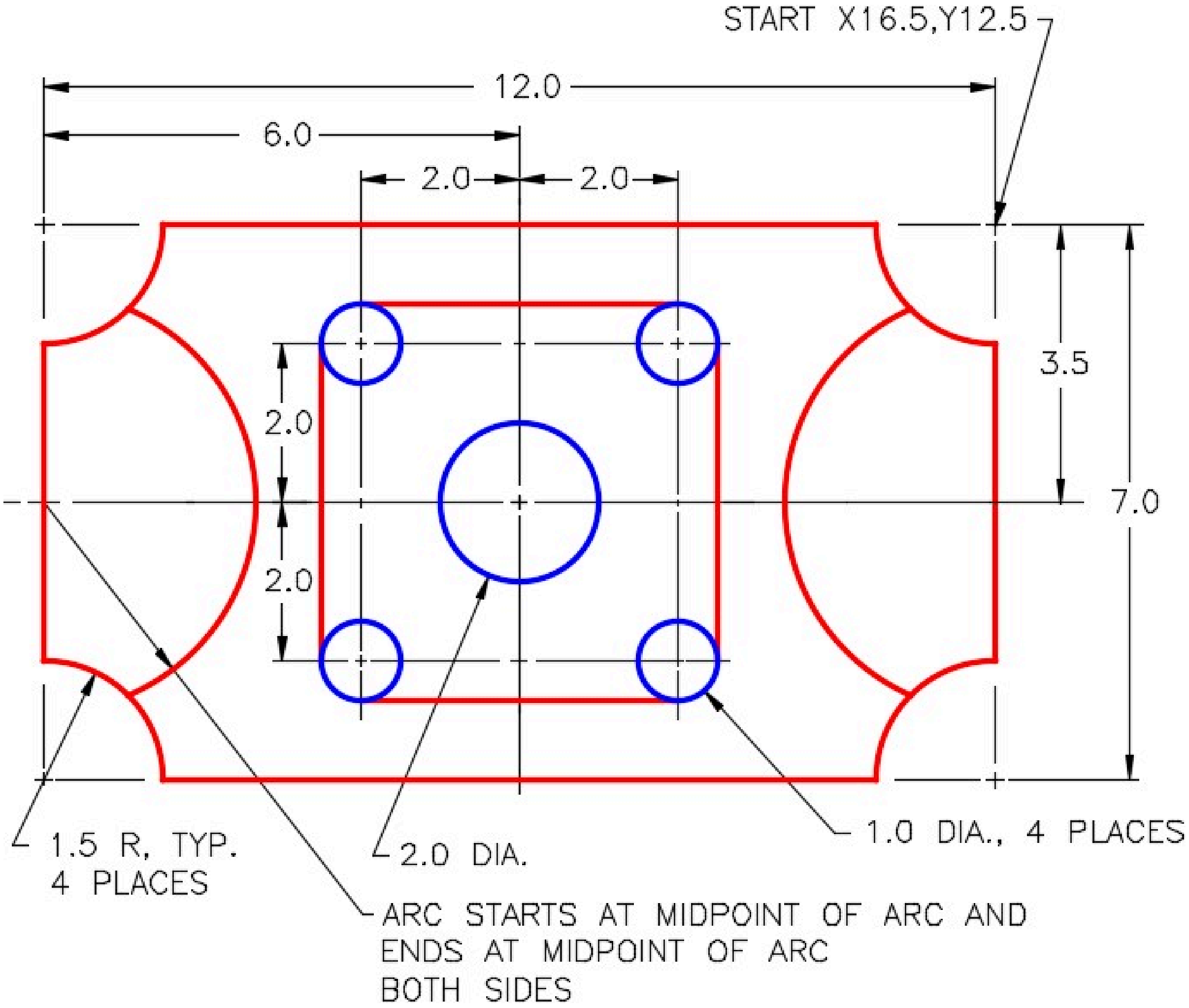

START X16.5,Y12.5
12.0
6.0
2.0
2.0
3.5
7.0
2.0
2.0
1.5 R, TYP.
4 PLACES
2.0 DIA.
1.0 DIA., 4 PLACES
ARC STARTS AT MIDPOINT OF ARC AND
ENDS AT MIDPOINT OF ARC
BOTH SIDES

Figure Step 3A

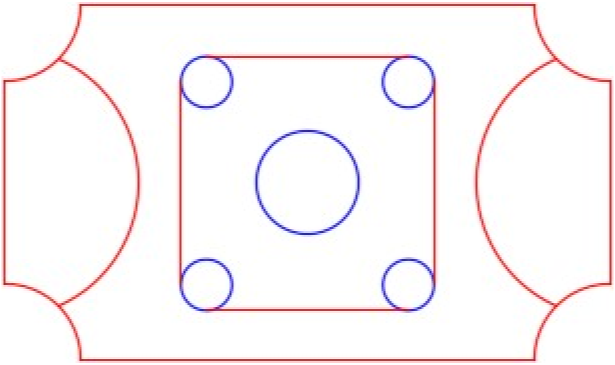


Figure Step 3B
Completed Drawing

Step 3

Draw the object shown below using the layering scheme. In this drawing, place the lines and arcs on layer: Object 1 and the circles on layer: Object 2. The lines and arcs should display red and the circles blue. (Figure Step 3A and 3B)

Step 4

Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Inches.

Step 5

Check your drawing with the key. The key name is the same as the drawing name.

Step 6

If you have any errors, turn layer Key off and correct your drawing. If you cannot correct it, start it over from the beginning.

Step 7

Turn layer Key on to check your drawing. If it is still inaccurate, go back to Step 6.

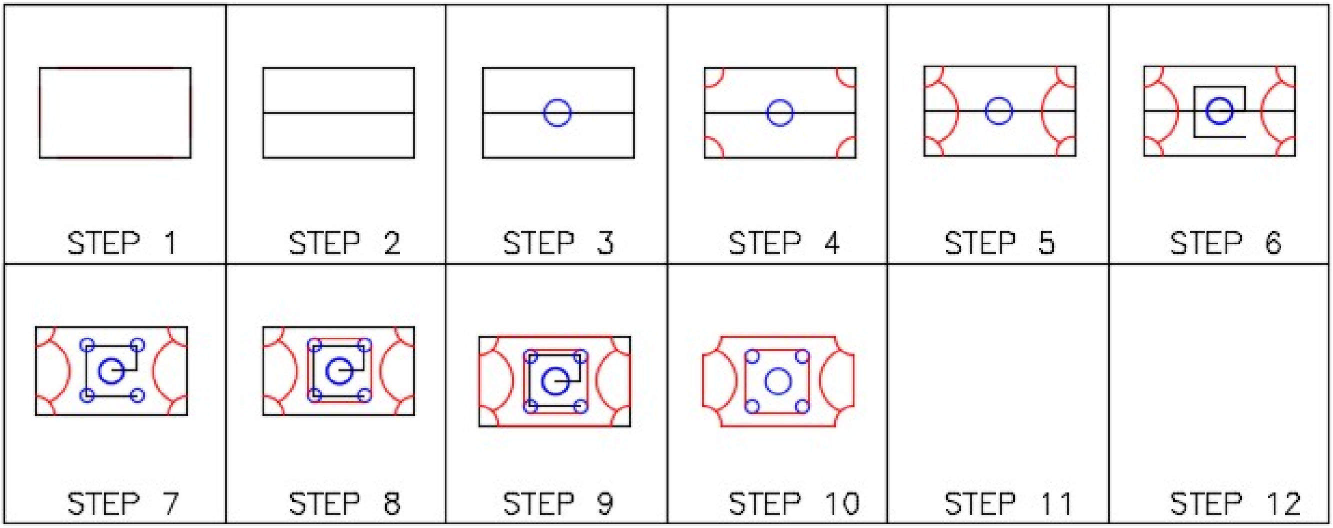
Step 8

Turn layer Key off and freeze layer: Construction

Step 9

Save and close the drawing.

AUTHOR'S CONSTRUCTION TECHNIQUES: The following steps are the construction technique suggested by the author to help you learn how to construct objects using AutoCAD. It is only the suggested method and if you can complete the drawing accurately using a different construction technique, that is what is important. You may want to compare your construction technique with the authors.


STEP 1
STEP 2
STEP 3
STEP 4
STEP 5
STEP 6
STEP 7
STEP 8
STEP 9
STEP 10
STEP 11
STEP 12

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

The arcs on the corners can be drawn using the command shown below. (Figure Hint 1)

```
Command: ARC

Specify start point of arc or [Center]: C

Specify center point of arc: (end) P1

Specify start point of arc: @-1.5,0

(This sets the radius and the direction of the start point.)

Specify end point of arc or [Angle/chord Length]: (end) P2

Command:
```

(Repeat on the other three corners. Keep in mind that arcs are constructed counterclockwise in AutoCAD.)

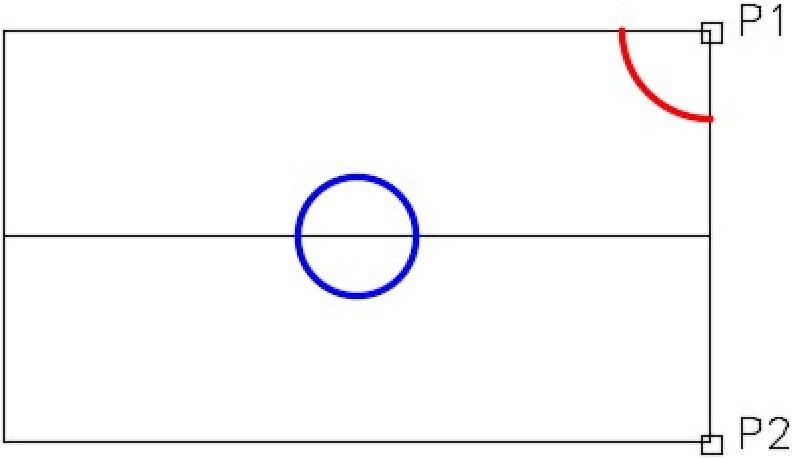


Figure Hint 1

Hint 2

The arcs on the sides can be drawn using the command shown below. (Figure Hint 2)

Command: **ARC**

Specify start point of arc or [Center]: **C**

Specify center point of arc: *(mid)* **P3**

Specify start point of arc: *(mid)* **P4**

Specify end point of arc or [Angle/chord Length]: (mid*)* **P5**

(Do the same on the other side. Ensure that you construct the arc counterclockwise.)

Command:

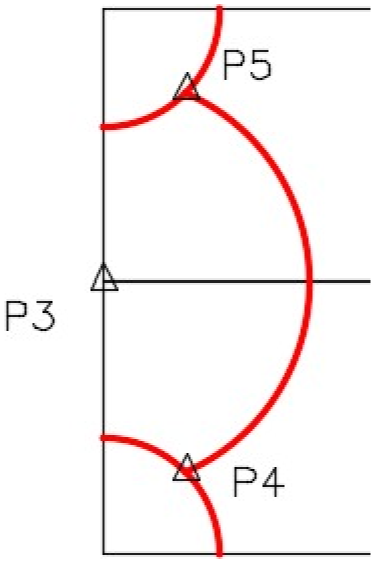


Figure Hint 2

Hint 3

The figure shows the construction objects in this drawing. (Figure Hint 3)

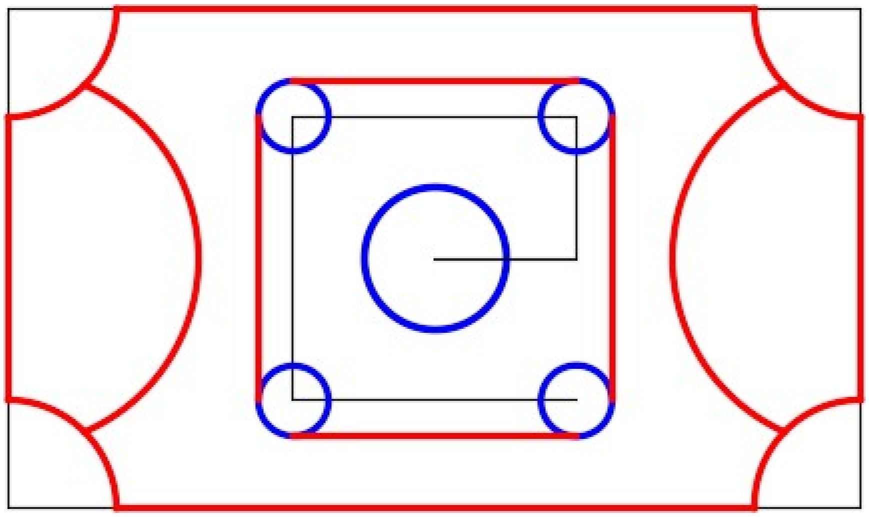


Figure Hint 3

Lab Exercise 8-3

Time allowed: 40 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 08-3	N/A	Inches

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Construction	Construction objects	253
Object 1	All lines and arcs	Red
Object 2	All circles	Blue

Step 1

Open the drawing: AutoCAD 2D Lab 08-2.

Step 2

Using the SAVEAS command, save it with the name: AutoCAD 2D Lab 08-3.

Step 3

Turn off all the layers except for layer: Key. If the magenta key from Lab 08-2 displays, select it and press the delete key. Turn all of the other layers back on.

Step 4

Complete the object by adding and erasing objects to match the figure. (Figure Step 4A and 4B)

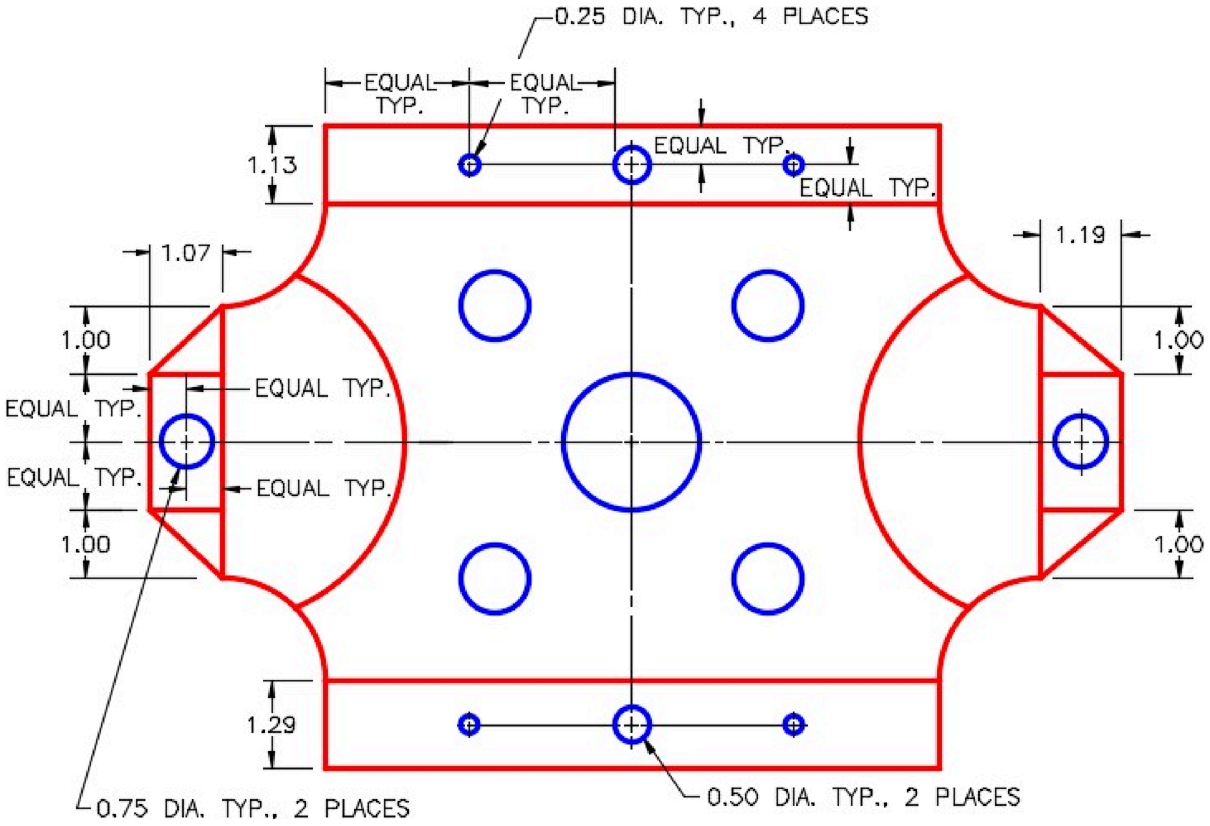

0.25 DIA. TYP., 4 PLACES
EQUAL TYP.
EQUAL TYP.
1.13
EQUAL TYP.
EQUAL TYP.
1.07
1.19
1.00
1.00
EQUAL TYP.
EQUAL TYP.
EQUAL TYP.
EQUAL TYP.
1.00
1.00
1.29
0.75 DIA. TYP., 2 PLACES
0.50 DIA. TYP., 2 PLACES

Figure Step 4A

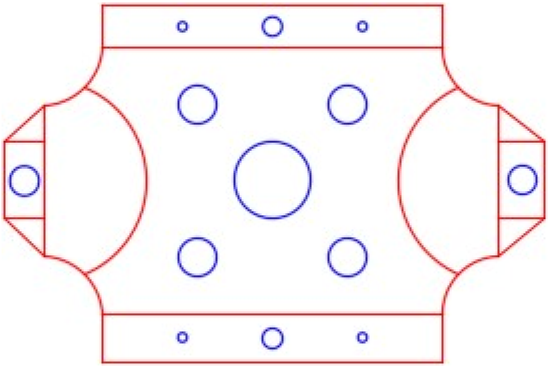


Figure Step 4B
Completed Drawing

Step 5

Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Inches.

Step 6

Check your drawing with the key. The key name is the same as the drawing name.

Step 7

If you have any errors, turn layer Key off and correct them.

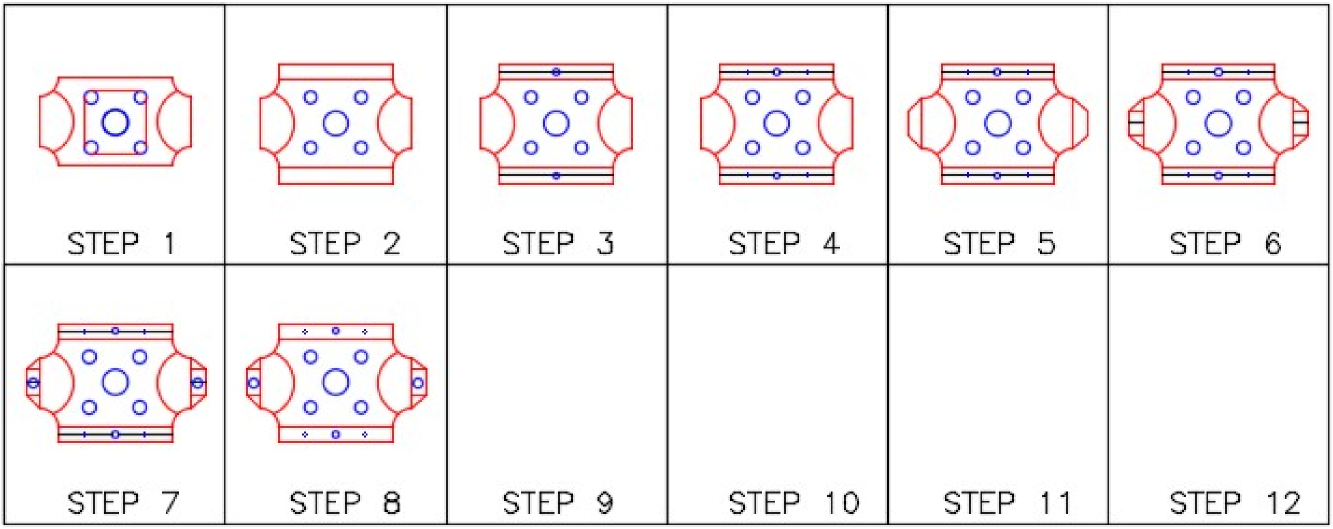
Step 8

Turn layer Key off and freeze layer Construction

Step 9

Save and close the drawing.

AUTHOR'S CONSTRUCTION TECHNIQUES: The following steps are the construction technique suggested by the author to help you learn how to construct objects using AutoCAD. It is only the suggested method and if you can complete the drawing accurately using a different construction technique, that is what is important. You may want to compare your construction technique with the authors.



AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

Locate the larger circle as shown in the figure. (Figure Hint 1)

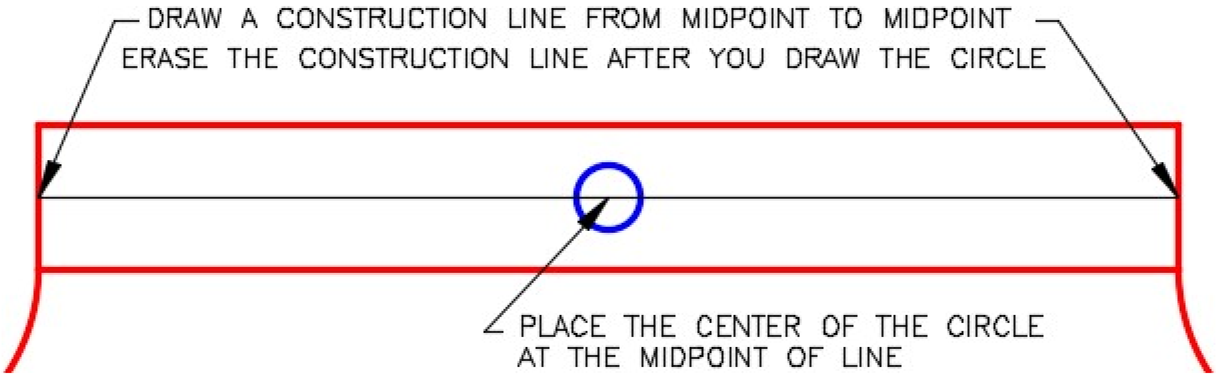

DRAW A CONSTRUCTION LINE FROM MIDPOINT TO MIDPOINT
ERASE THE CONSTRUCTION LINE AFTER YOU DRAW THE CIRCLE
PLACE THE CENTER OF THE CIRCLE
AT THE MIDPOINT OF LINE

Figure Hint 1

Hint 2

Locate the smaller circle as shown in the figure. (Figure Hint 2)

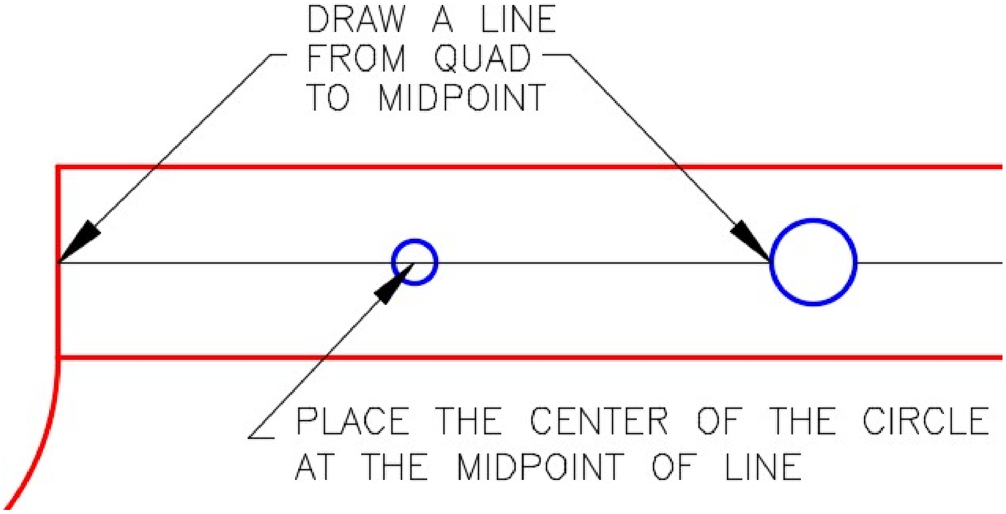

DRAW A LINE
FROM QUAD
TO MIDPOINT
PLACE THE CENTER OF THE CIRCLE
AT THE MIDPOINT OF LINE

Figure Hint 2

Hint 3

The figure shows the construction objects in this drawing. (Figure Hint 3)

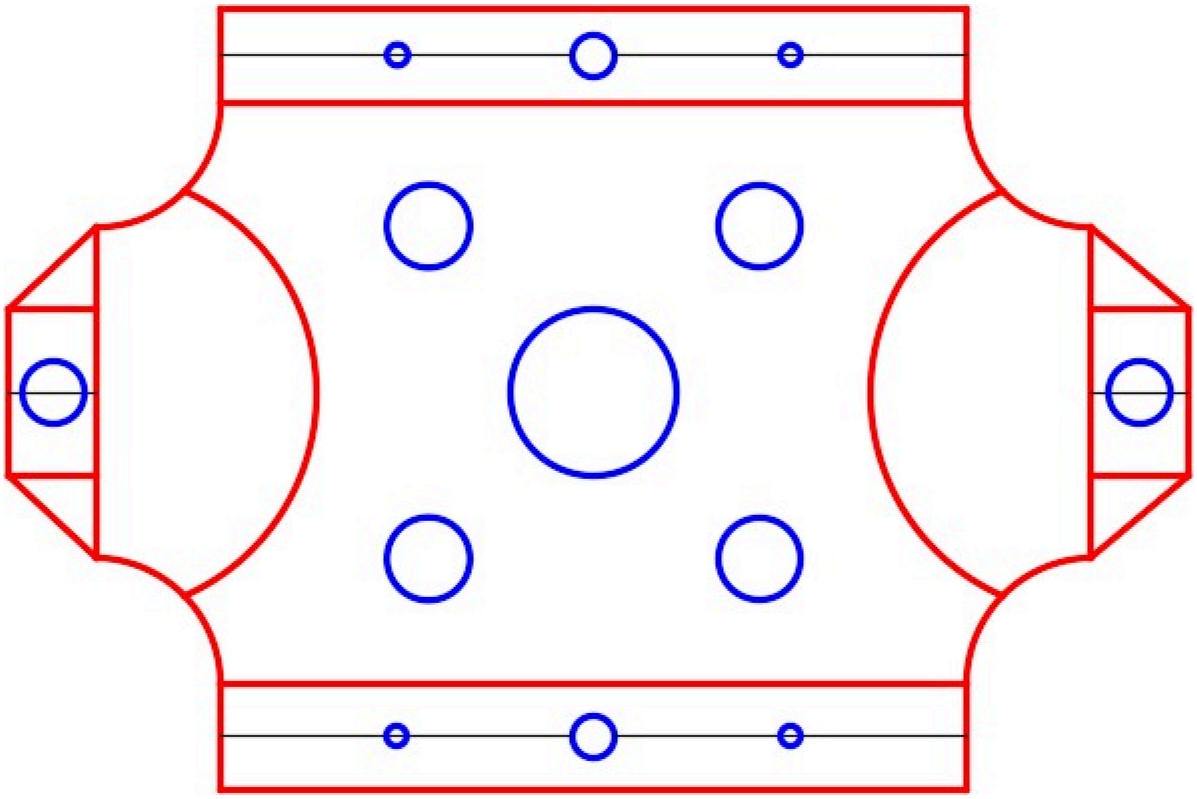


Figure Hint 3

Lab Exercise 8-4

Time allowed: 40 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 08-4	2D English	Inches

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Gasket	All drawing objects	Blue

Step 1

Draw the top view of the simple gasket shown in the dimensioned drawing using the layering scheme. (Figure Step 1A, 1B, and 1C)

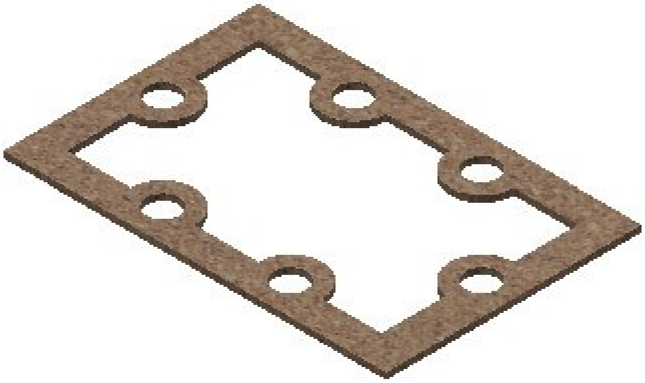


Figure Step 1A
Simple Gasket

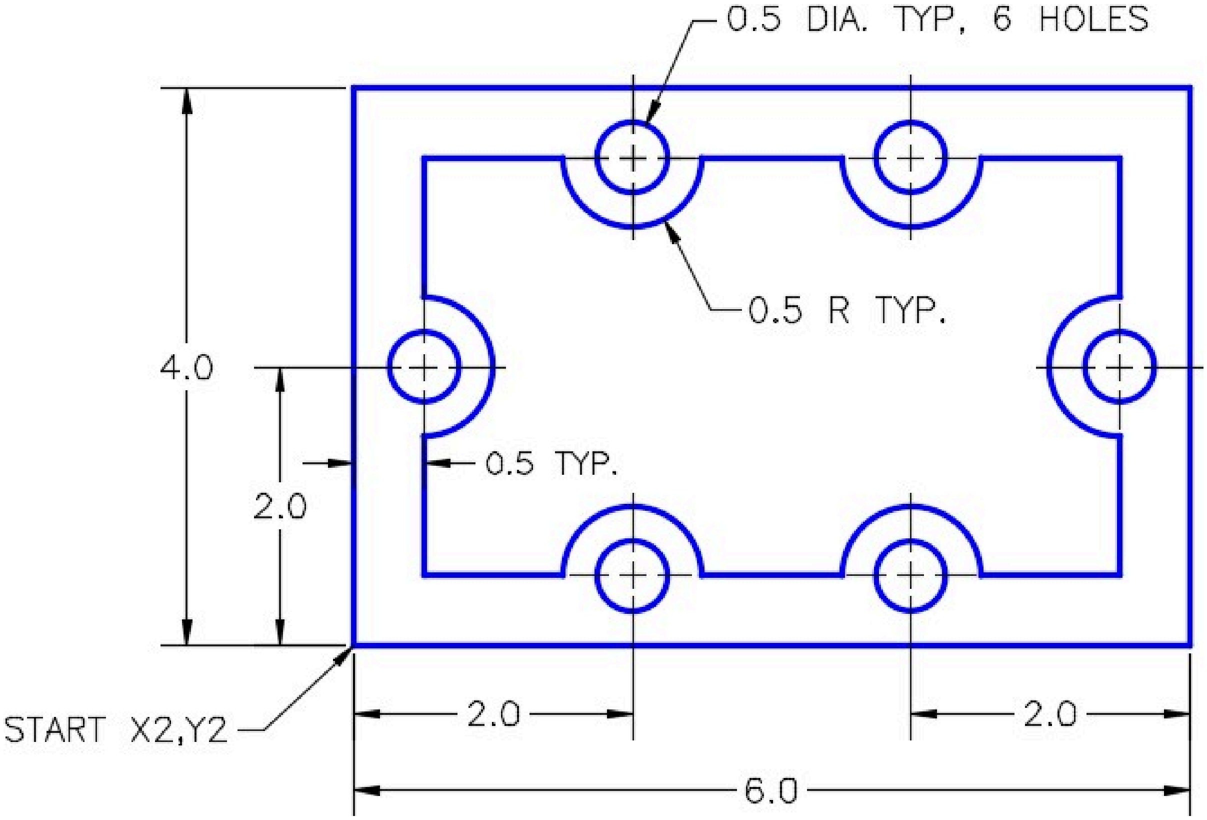

0.5 DIA. TYP, 6 HOLES
0.5 R TYP.
4.0
2.0
0.5 TYP.
START X2,Y2
2.0
2.0
6.0

Figure Step 1B

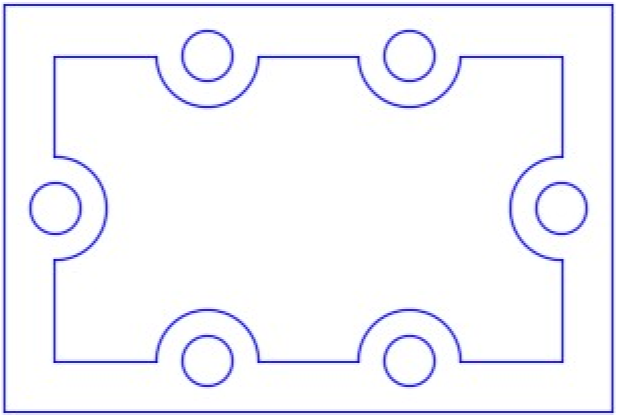


Figure Step 1C
Completed Drawing

Step 3

Set the insertion units and check your drawing with the key. Turn layer Key off and freeze layer Construction.

Step 4

Save and close the drawing.

Module 9: Controlling the Drawing

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe the drawing terms; limits, extents, scale, units, and regeneration.
2. Apply the ZOOM, PAN, REGEN, and VIEWRES commands and explain how you use them to control the drawing.

Controlling the Drawing

To control the drawing and improve your drawing speed, it is important for you to understand the drawing terms; limits, extents, scale, units, and regeneration and how they relate to the current drawing.

Drawing Limits

An AutoCAD drawing has an unlimited amount of model space available to draw in. The *drawing limits* are a user defined rectangular or square shape of 2D model space. The size of the drawing limits can be changed, as required, at anytime.

Drawing limits were important before layouts were added to AutoCAD in AutoCAD 2000. They are not that important to AutoCAD 2000 and newer users but are still part of the software.

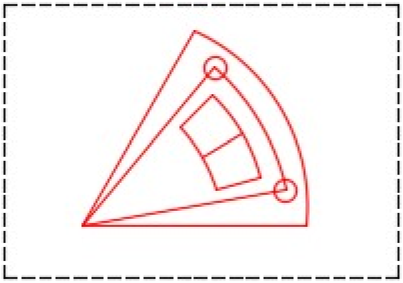


Figure 9-1 The Drawing Limits

Drawing Extents

The *drawing extents* is defined as the smallest rectangle that will fit around all existing drawing objects in the current drawing that are visible (on a thawed layer). It is possible to draw outside the drawing limits and therefore objects that are outside the limits may not be visible in the normal drawing display.

By being able to display the drawing extents, all visible (thawed) drawing objects that exist in the current drawing will display in the Graphic window. Being able to display the drawing extents is very important.

Since Model space is very large, objects are sometimes placed, by accident, somewhere out in space. If this happens in a drawing, it can severely derogate the performance of the drawing. Sometimes the only way to work on a drawing that is derogated is to find the erroneous objects and delete them. They can easily be found by displaying the drawing extents.

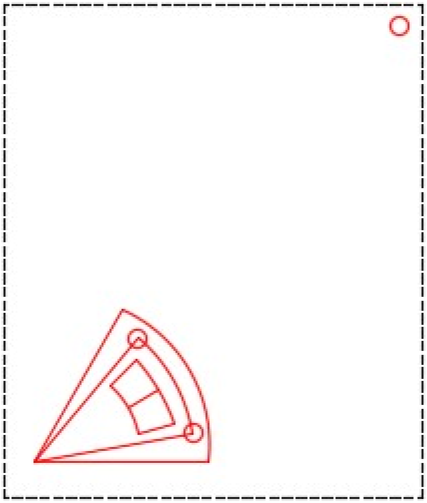


Figure 9-2 The Drawing Extents

Drawing Scale

The general rule for all AutoCAD drawings is to always draw the object at full scale or full size. Scaling is done in the viewports in the layouts. This is taught in Module 18.

Drawing Units

AutoCAD drawings are unitless. If a unit of 1 is entered into a drawing and the units selected by the creator for that drawing are inches, then 1 means 1 inch. From that point forward all units entered into that drawing must be entered as inches. In another drawing, the operator creating the drawing picks the units to be millimeters. All numbers entered in that drawing must be entered in millimeters.

To sum up, when a drawing is first started, the creator must make a decision as to what units will be used for that drawing. From that point on, all numbers entered into that drawing must be entered in those units.

Zooming

Zooming is the process of moving the drawing objects closer or further away from your eyes without actually changing the size of the objects or their location in model or paper space. It is an important tool for you and is used extensively in the drawing process. Drawings can be very large and therefore you must be able to zoom in to be able to see in greater detail and zoom out again to work on the overall drawing.

The most efficient method to zoom a drawing is to locate the Graphic cursor in the center of the area to be zoom and then rotate the wheel on the mouse forward or backward. While the AutoCAD command ZOOM can be also used to zoom the drawing, using the wheel to zoom will greatly increase your drawing speed.

Panning

Panning is the process of moving the existing drawing objects around the Graphic window without actually physically moving any of the drawing object's current location in model or paper space.

The most efficient method to pan a drawing is with the wheel on the mouse. Press and hold the wheel down and them move the mouse to pan the drawing. A Pan Hand cursor will replace the Graphic cursor when you hold the mouse wheel down. While the AutoCAD command PAN can be used to pan the drawing, using the wheel will greatly increase your drawing speed.

Regenerating the Drawing

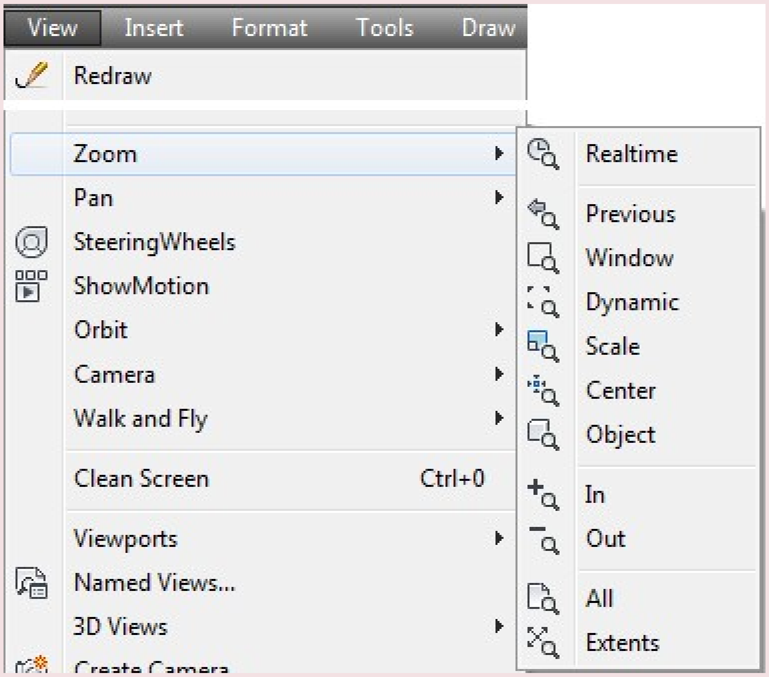
AutoCAD stores all existing drawing objects and their properties in a database format in the .dwg file. When a drawing is opened, AutoCAD reads the .dwg file and constructs the drawing objects, one at time, displaying them in the Graphic window.

There are times when working on a drawing that it must be reconstructed from the .dwg file. This is called *regeneration*. The command used to do this is REGEN. There are many reasons for regeneration and they will become clear in future modules. Since some commands do not automatically regenerate, it is up to you to manually execute the REGEN command to force the display of any changes that happened when those commands was executed.

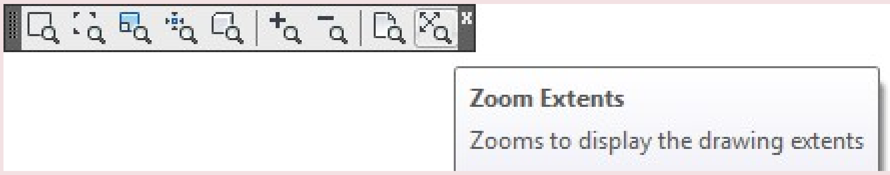
AutoCAD Command: ZOOM

The ZOOM command is used to move the existing drawing objects closer or farther away from the user's eyes without changing the drawing object's physical size.

Shortcut: **Z**


View
Insert
Format
Tools
Draw
Redraw
Zoom
Pan
SteeringWheels
ShowMotion
Orbit
Camera
Walk and Fly
Clean Screen Ctrl+0
Viewports
Named Views...
3D Views
Realtime
Previous
Window
Dynamic
Scale
Center
Object
In
Out
All
Extents

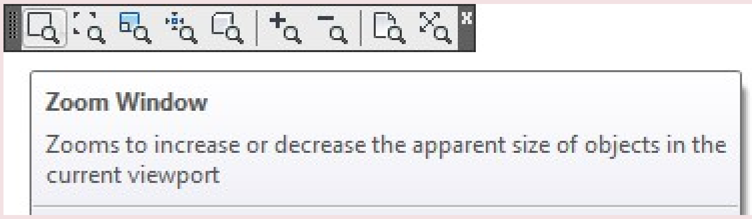
View Pull-down



Zoom Extents

Zooms to display the drawing extents

Zoom Toolbar



Zoom Window

Zooms to increase or decrease the apparent size of objects in the current viewport

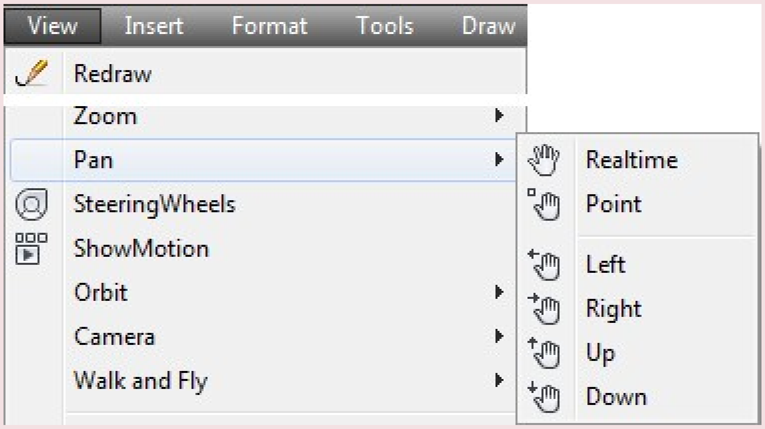
Zoom Toolbar

AUTHOR'S COMMENTS: If you have a mouse with a wheel as the center button, you can use it to zoom in and out of your drawing by first locating the Graphic cursor in the center of the area to be zoomed and then rotating the wheel forward or backward. While the AutoCAD command ZOOM can be used to zoom the drawing using the wheel will greatly increase your drawing speed. If you don't have a wheel mouse, you must use the ZOOM command and its options to accomplish the same thing.

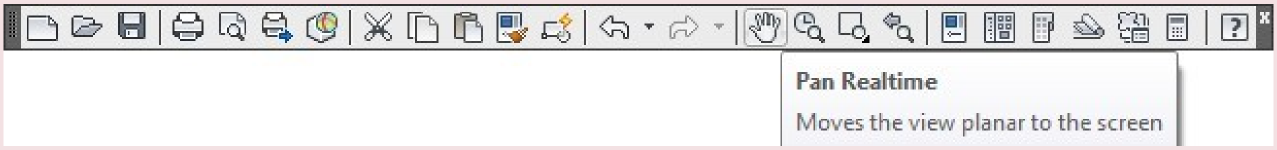
AutoCAD Command: PAN

The PAN command is used to move the drawing objects around the Graphic window without physically moving them.

Shortcut: **P**


View
Insert
Format
Tools
Draw
Redraw
Zoom
Pan
SteeringWheels
ShowMotion
Orbit
Camera
Walk and Fly
Realtime
Point
Left
Right
Up
Down

View Pull-down


Pan Realtime
Moves the view planar to the screen

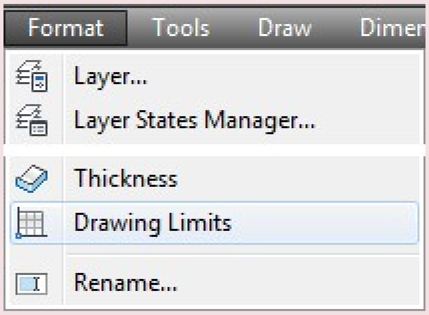
Standard Toolbar

AUTHOR'S COMMENTS: If you have a mouse with a wheel as the center button, you can use it to pan your drawing by holding wheel down and moving the mouse. While the AutoCAD command PAN can be used pan the drawing using the wheel will greatly increase your drawing speed. If you don't have a wheel mouse, you must use the PAN command and its options to accomplish the same thing.

AutoCAD Command: LIMITS

The LIMITS command is used to set the drawing limits.

Shortcut: **none**

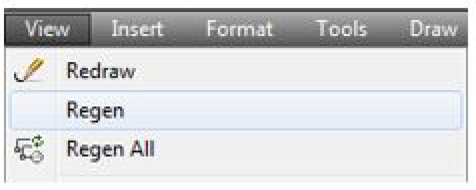

Format
Tools
Draw
Dimen
Layer...
Layer States Manager...
Thickness
Drawing Limits
Rename...

Format Pull-down

AutoCAD Command: REGEN

The REGEN command is used to reconstruct the drawing objects from the .dwg file and display them in the Graphic window.

Shortcut: **RE**


View
Insert
Format
Tools
Draw
Redraw
Regen
Regen All

View Pull-down

AutoCAD Command: VIEWRES

The VIEWRES command sets the number of vectors that AutoCAD uses when it displays circles and arcs in the Graphic window.

Shortcut: **none**

MUST KNOW: AutoCAD drawings are unitless. The creator of the drawing makes the decision of what units will be used for that drawing. From that point forward, all numbers entered in that drawing must be in those units.

MUST KNOW: The general rule for all AutoCAD drawings is to draw the object at full scale or full size. Scaling is done in the viewports in the layouts.

AUTHOR'S COMMENTS: The only objects AutoCAD can construct in any drawing are lines and points. Circles and arcs are constructed using many lines connecting two XY coordinate points that are calculated each time the drawing is regenerated. The more lines AutoCAD uses to construct the circles and arcs, the smoother or more circular they will display. The VIEWRES command sets the zoom percentage of the lines that AutoCAD uses when it constructs or regenerates the circles and arcs in the drawing. When VIEWRES is set to 20000, the maximum, the circles and arcs will appear very smooth. On the other hand, it will take AutoCAD much longer to calculate that many points for each circle and arc in the drawing. You have to take all of this into account when you are setting the viewres for the drawing.

WORKALONG: Using the LIMITS, ZOOM, PAN, REGEN, and VIEWRES Commands

Step 1

Open the drawing: AutoCAD 2D Workalong 08-1.

Step 2

Using the SAVEAS command, save the drawing with the name: AutoCAD 2D Workalong 09-1.

Step 3

Enter the ZOOM Extents command, as shown below. The drawing should appear as shown in the figure. (Figure Step 3)

Command: **ZOOM**

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: **E**

Command:

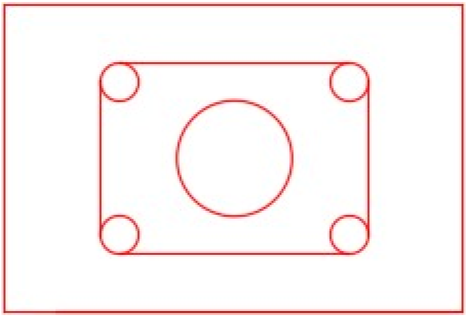


Figure Step 3

AUTHOR'S COMMENTS: I find it easier to enter the ZOOM Extents command on the keyboard. It is fast if you use the shortcuts Z and E.

Step 4

Practice zooming the drawing in and out by rotating the wheel on the mouse.

Step 5

Move the crosshairs to approximately the center of the object and push the middle wheel down and hold it. A Hand icon will replace the crosshairs. While holding the wheel down, pan the drawing around.

Step 6

Enter the ZOOM Extents command to zoom the drawing to its extents. Step 7 Enter the LIMITS command, as shown below, to set the drawing limits.

```
Command: LIMITS

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000,0.0000>:

Specify upper right corner <12.0000,9.0000>: 8.5,11

Command:
```

AUTHOR’S COMMENTS: It is always best to set the lower left corner of the drawing limits to 0,0. Doing that, help keeps the drawing objects in the positive quadrant. In other words, both X and Y are positive.

Step 8

Use the CIRCLE command, as shown below, to draw a 1 diameter circle with its center at X20,Y22.

```
Command: C

CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: 20,22

Specify radius of circle or [Diameter]: D

Specify diameter of circle: 1

Command:
```

Step 9

Enter the ZOOM Extents command. Your drawing should appear as shown in the figure. (Figure Step 9)

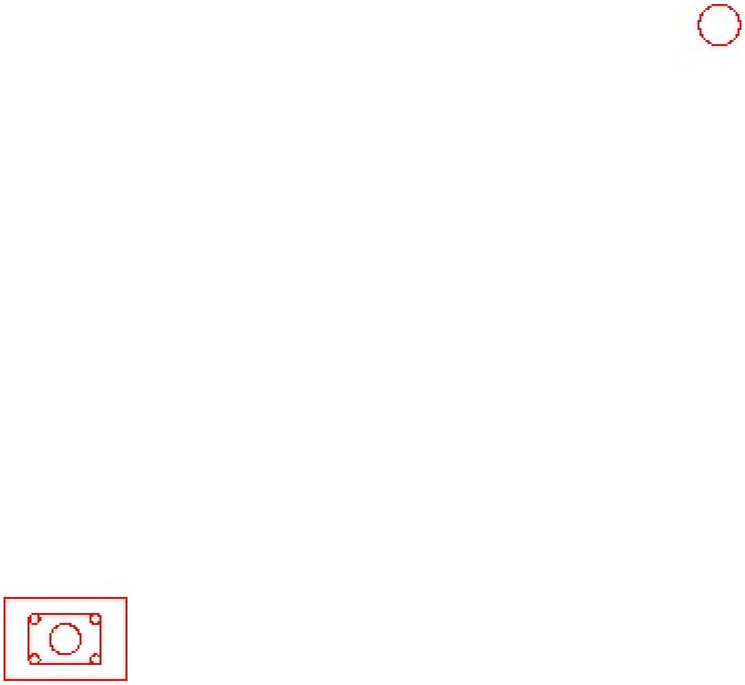


Figure Step 9

Step 10

Use the CIRCLE command, as shown below, to draw a 1 diameter circle with its center at X-35,Y-35.

```
Command: C

CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: -35,-35

Specify radius of circle or [Diameter]: D

Specify diameter of circle: 1

Command:
```

Step 11

Enter the ZOOM Extents command. Your drawing should appear as shown in the figure. (Figure Step 11)

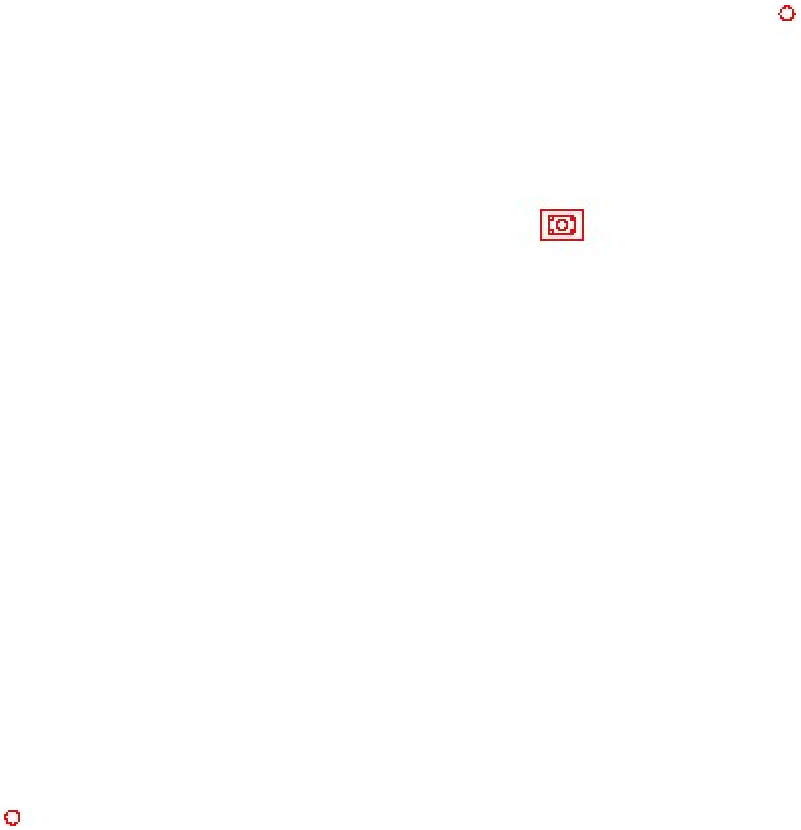


Figure Step 11

AUTHOR'S COMMENTS: Notice how the ZOOM Extents command zooms the drawing so that it displays all of the existing objects in the drawing.

Step 12

Delete the two circles that you inserted in Steps 8 and 10. Enter the ZOOM Extents command again. The drawing should match the figure. (Figure Step 12)

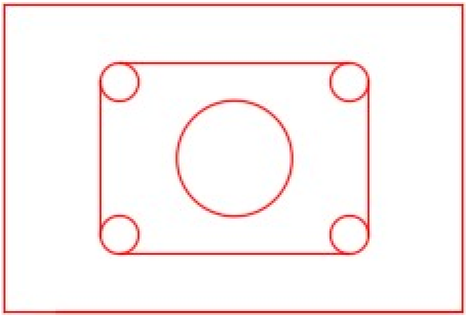


Figure Step 12

Step 13

Enter the VIEWRES and REGEN commands, as shown below. (Figure Step 13)

Command: **VIEWRES**

Do you want fast zooms? [Yes/No] <Y>: **Y**

Enter circle zoom percent (1-20000) <2000>: **20000**

Command: **REGEN**

Regenerating model

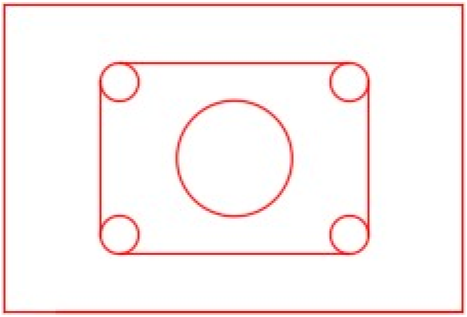


Figure Step 13

AUTHOR'S COMMENTS: Always answer Yes to fast zooms in the VIEWRES command. The circles will now display using 20,000 vectors. Since the VIEWRES command does not automatically regenerate the drawing, the REGEN must be executed after it to reconstruct the drawing objects to display the new resolution.

Step 14

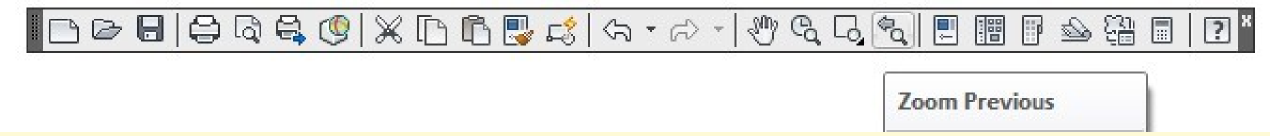
Save and close the drawing.

USER TIP: Since the ZOOM Extents command is used a lot during the drawing process, there is a shortcut to execute this command. Move the cursor to somewhere on the Graphic window and double click the wheel or middle button on the mouse.When attempting to find all the existing objects by displaying the drawing extents, ensure that all layers are thawed and on.

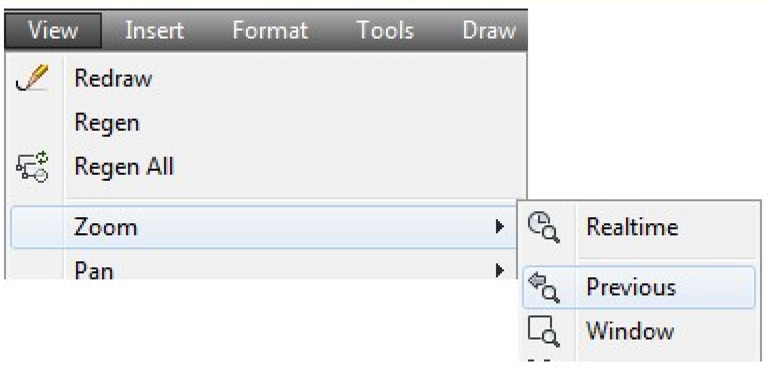
USER TIP: If you have a mouse with a wheel as the center button, use it to zoom in and out of the drawing by rotating the wheel forward or backward. Using the wheel to zoom, rather than commands, will increase your drawing speed. If your computer is not equipt with a wheel mouse, you must use the ZOOM command.The wheel can also be used to pan the drawing. Move the crosshairs to the part of the drawing to be panned and press down on the wheel. A Pan Hand icon will display as the Graphic cursor. While holding down the wheel, move the mouse to pan the drawing. Using the wheel to pan will also increase your drawing speed. If your computer does not have mouse with a center wheel, you must use the PAN command.

MUST KNOW: At times, the current drawing must be regenerated from the .dwg file. Drawing regeneration is accomplished with the REGEN command. If you are in doubt about the current appearance of the drawing, you should regenerate it.

USER TIP: The ZOOM Previous command allows you to step back through the previous zoomed states. This allows you to quickly zoom in on an object, draw or edit, and then return back to the previous zoomed state.



Zoom Previous



View Pull-down

USER TIP: The ZOOM Extents command is a very important command and if used when required, can prove to save you a lot of time and effort. When you are opening a drawing for the first time, especially one that was drawn by someone other than you, the first thing you should do is to ensure that all layers are on and thawed. Then execute the ZOOM Extents command. Doing this will display all existing drawing objects in the Graphic window which will show you the size and scope of the drawing before you begin working on it.

AUTHOR'S COMMENTS: Do not use a calculator to complete any lab exercise in the AutoCAD 2D book. By using geometry principles, drawing techniques, object snapping, and construction objects, AutoCAD will do all the math for you.

Key Principles

Key Principles in Module 9

1. The drawing extents is the smallest rectangle that will fit around all the existing drawing objects in the drawing.
2. Always draw all objects in Model space at full size or full scale.
3. AutoCAD drawings are unitless. The operator who creates the drawing makes the decision what units will be used for that drawing.
4. When a drawing is regenerated, AutoCAD reconstructs the drawing from the .dwg file. Drawings regenerate when they are opened but you can force an open drawing to regenerate with the REGEN command.

Lab Exercise 9-1

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 09-1	2D Metric	Millimeters

Layering Scheme

Layer Name	Objects on Layer	Color
Construction	Construction objects	253
Object	All objects	Red

Step 1

Setup the layers using the Layering Scheme shown above.

Step 2

Draw the object shown in the figure using the layering scheme. (Figure Step 2A, 2B, and 2C)

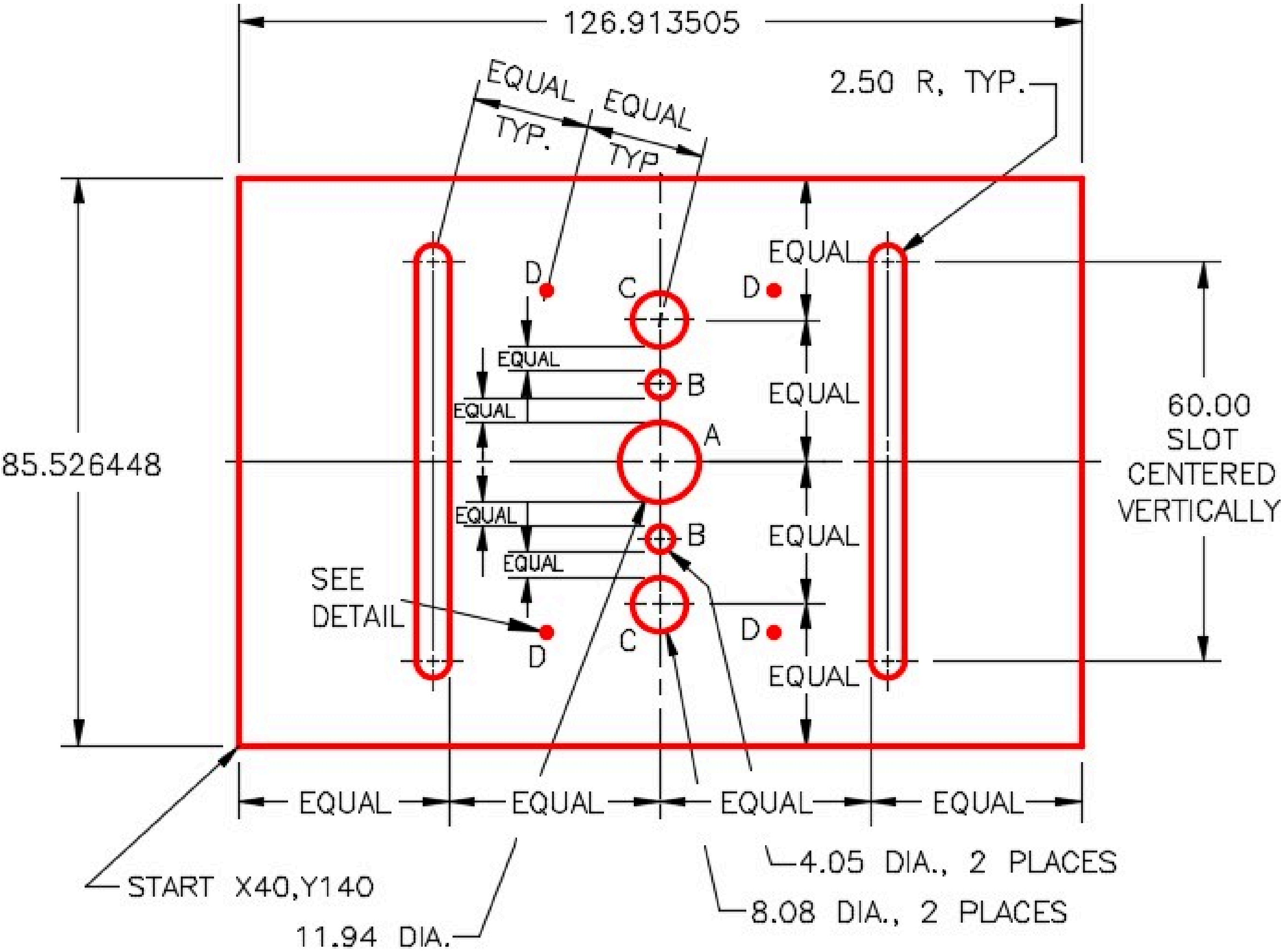

126.913505
EQUAL TYP.
EQUAL TYP.
2.50 R, TYP.
85.526448
EQUAL
EQUAL
EQUAL
EQUAL
EQUAL
EQUAL
EQUAL
EQUAL
60.00 SLOT CENTERED VERTICALLY
A
B
B
C
C
D
D
D
D
SEE DETAIL
EQUAL
EQUAL
EQUAL
EQUAL
START X40,Y140
11.94 DIA.
4.05 DIA., 2 PLACES
8.08 DIA., 2 PLACES

Figure Step 2A

AUTHOR’S COMMENTS: Do not use a calculator to complete this lab exercise or any other lab exercise in the AutoCAD 2D book. By using geometry principles, drawing techniques, object snapping, and construction objects, AutoCAD will do all the math for you.

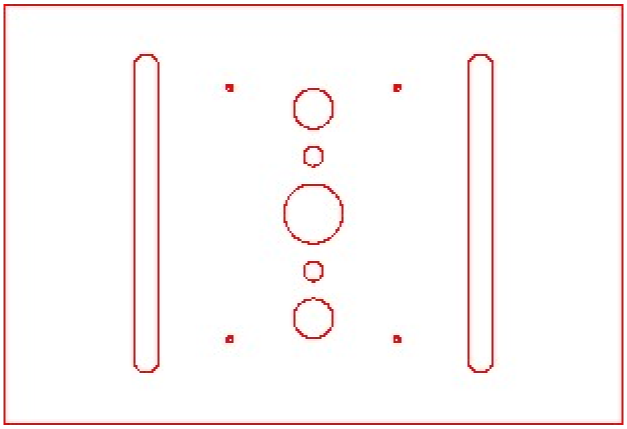


Figure Step 2B
Completed Drawing

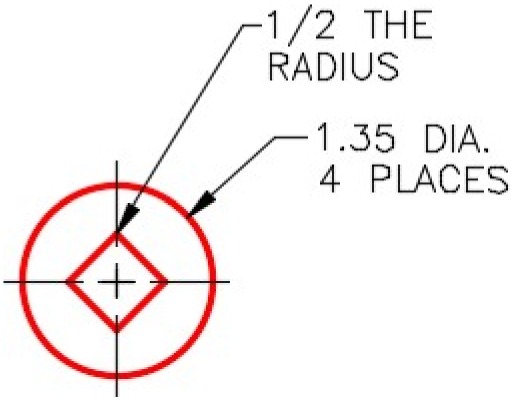

1/2 THE
RADIUS
1.35 DIA.
4 PLACES

Figure Step 2C
Detail

Step 3

Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Millimeters.

Step 4

Check the drawing's accuracy with the key.

Step 5

If there are any errors, turn layer Key off and correct the drawing.

Step 6

Turn layer: Key on. If the drawing still inaccurate, go back to Step 7.

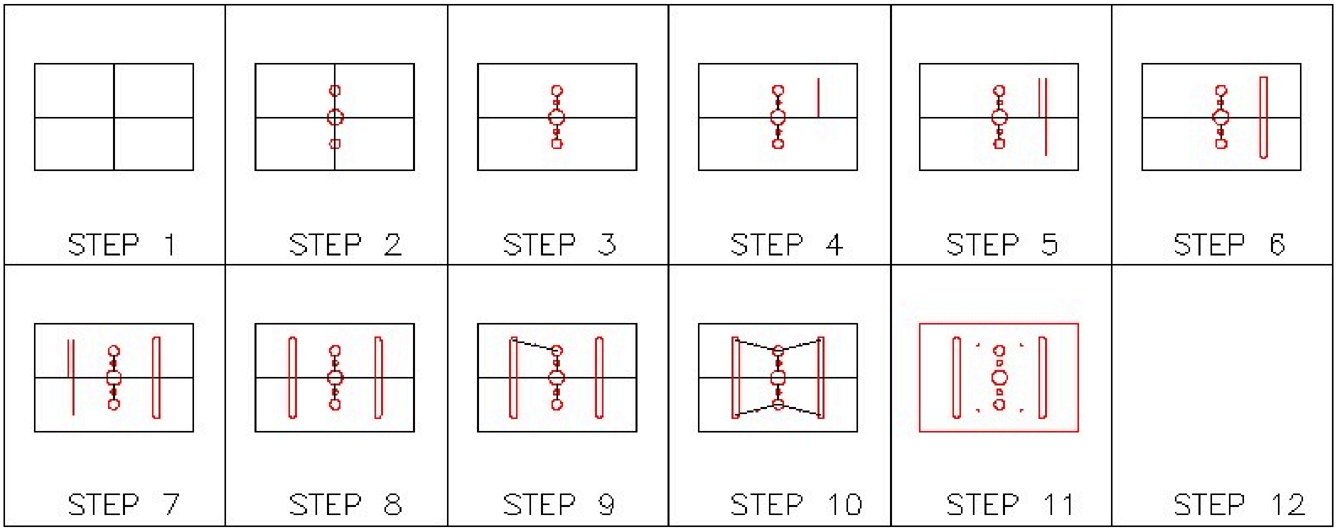
Step 7

Turn layer: Key off.

Step 8

Save and close the drawing.

AUTHOR'S CONSTRUCTION TECHNIQUES: The following steps are the construction technique suggested by the author to help you learn how to construct objects using AutoCAD. It is only the suggested method and if you can complete the drawing accurately using a different construction technique, that is what is important. You may want to compare your construction technique with the authors.



AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

See the figures. (Figure Hint 1A and 1B)

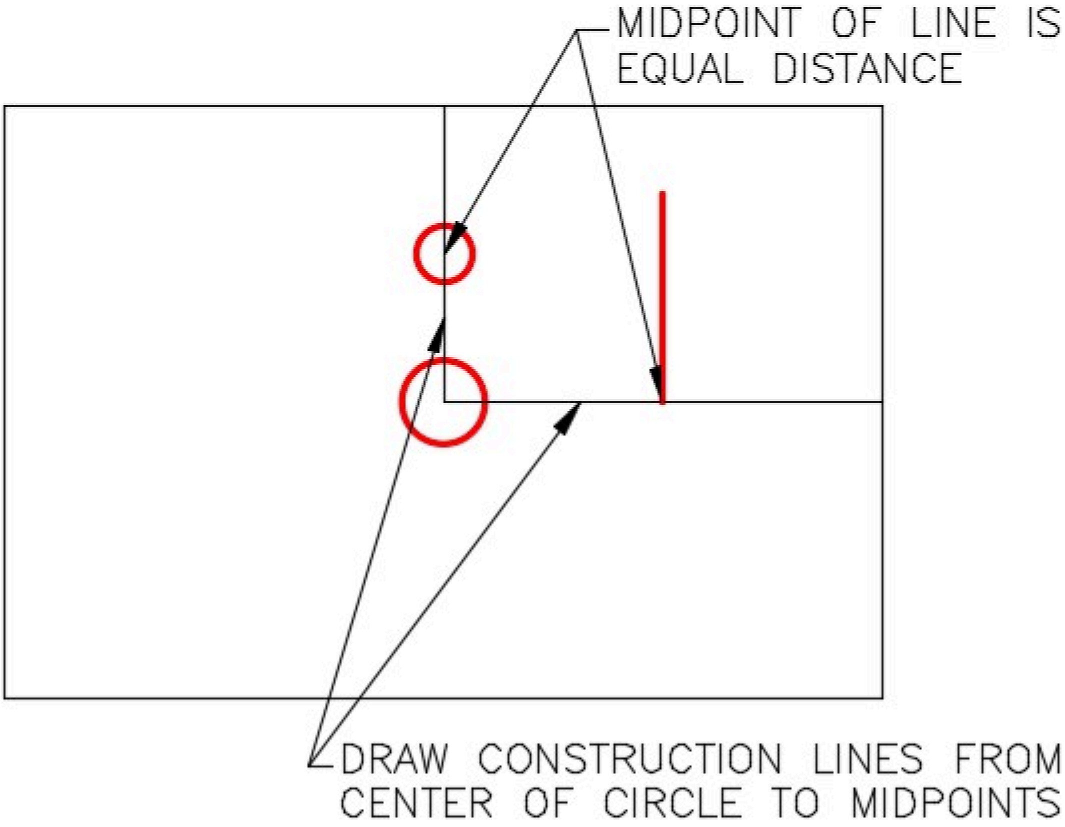

MIDPOINT OF LINE IS
EQUAL DISTANCE
DRAW CONSTRUCTION LINES FROM
CENTER OF CIRCLE TO MIDPOINTS

Figure Hint 1A

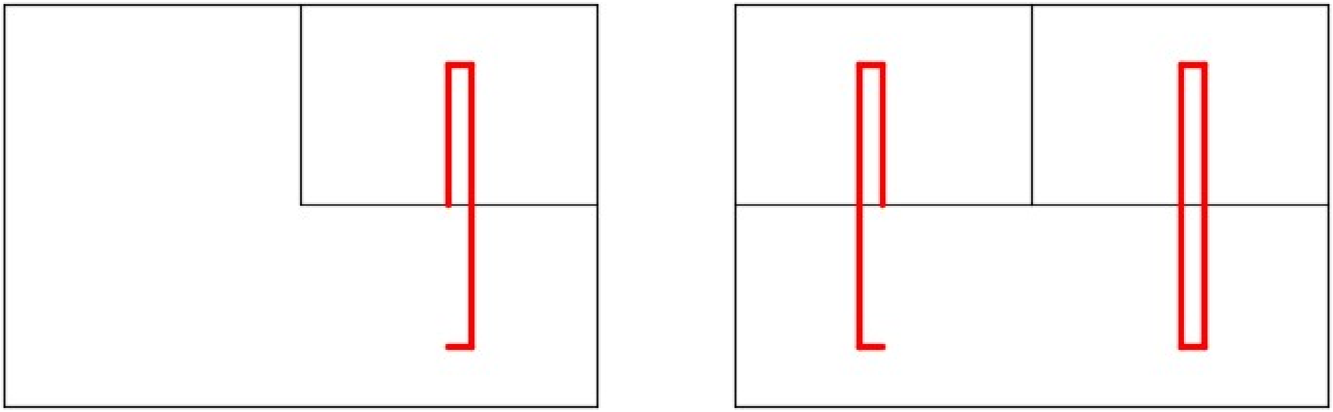


Figure Hint 1B

Hint 2

Construction lines for circle B. (Figure Hint 2)

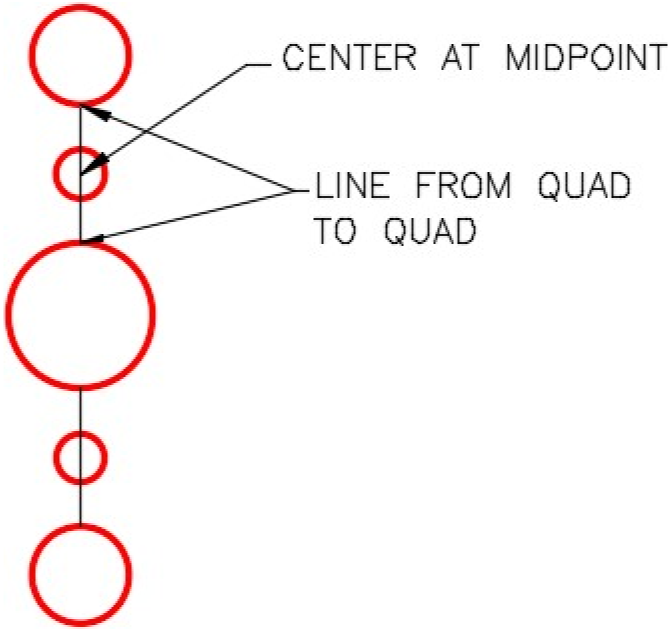

CENTER AT MIDPOINT
LINE FROM QUAD TO QUAD

Figure Hint 2

Hint 3

Construction to draw circle D. (Figure Hint 3)

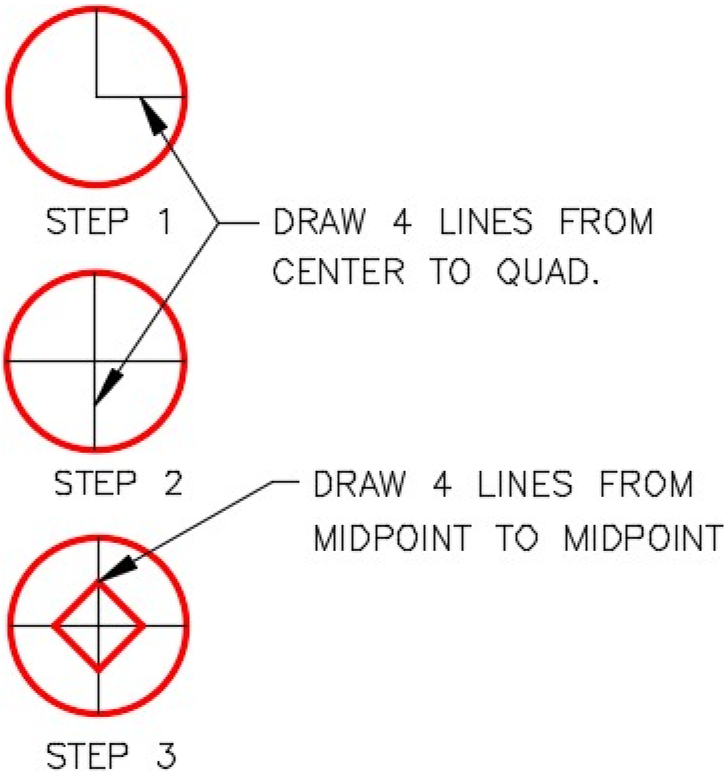

STEP 1
DRAW 4 LINES FROM
CENTER TO QUAD.
STEP 2
DRAW 4 LINES FROM
MIDPOINT TO MIDPOINT
STEP 3

Figure Hint 3

Hint 4

Construction objects in the drawing. (Figure Hint 4)

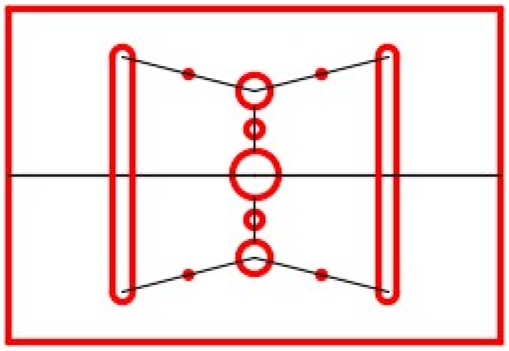


Figure Hint 4
Construction Lines

Lab Exercise 9-2

Time allowed: 30 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 09-2	2D English	Inches

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Construction	Construction objects	253
Object	All objects	Red

Step 1

Setup the layers using the Layering Scheme shown above.

Step 2

Draw the object shown in the figure using the layering scheme. (Figure Step 2A and 2B)

AUTHOR'S COMMENTS: This is a very small object. You will have to zoom in to work on it.

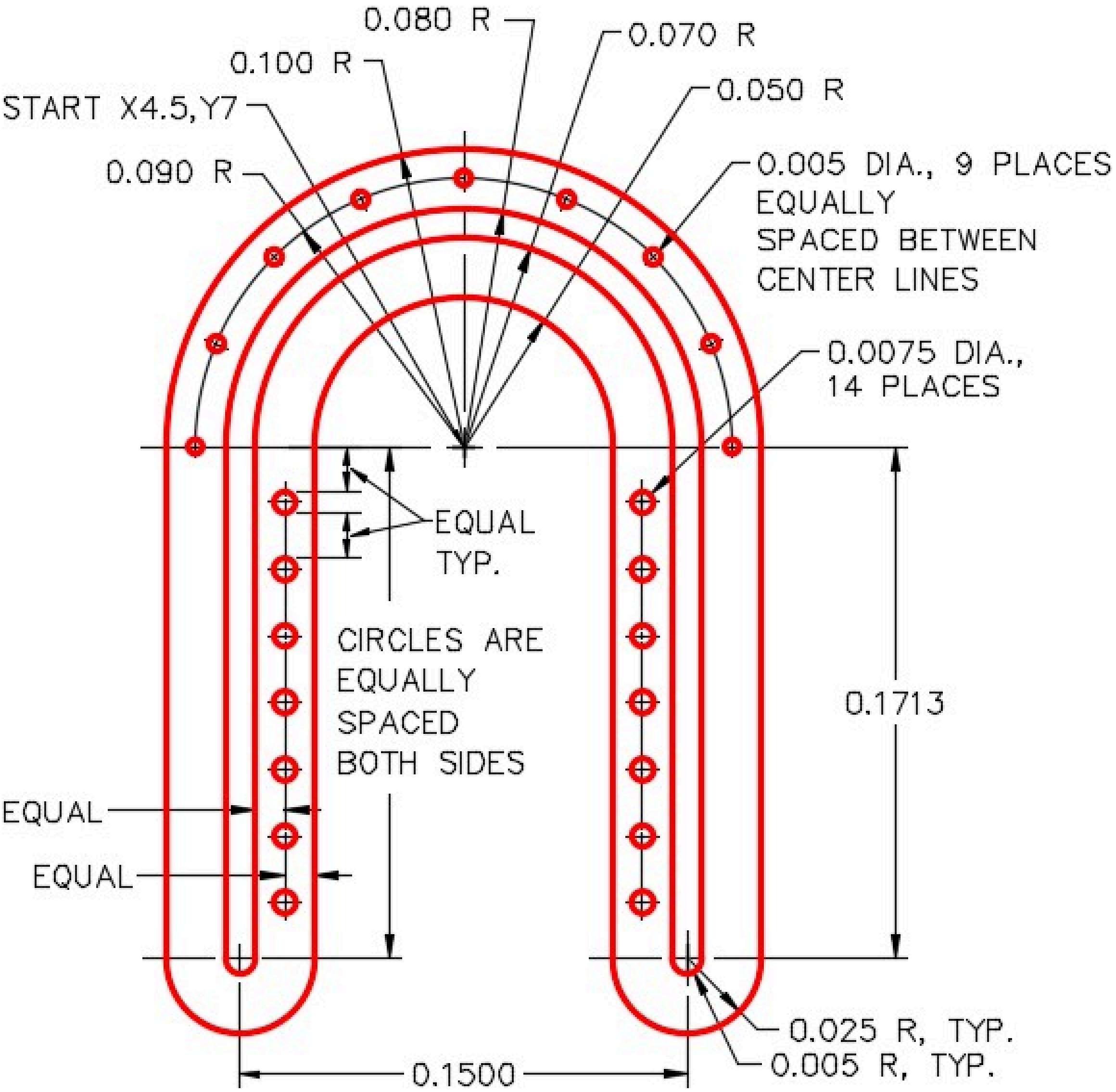

0.080 R
0.070 R
0.100 R
0.050 R
START X4.5,Y7
0.090 R
0.005 DIA., 9 PLACES EQUALLY SPACED BETWEEN CENTER LINES
0.0075 DIA., 14 PLACES
EQUAL TYP.
CIRCLES ARE EQUALLY SPACED BOTH SIDES
0.1713
EQUAL
EQUAL
0.025 R, TYP.
0.005 R, TYP.
0.1500

Figure Step 2A

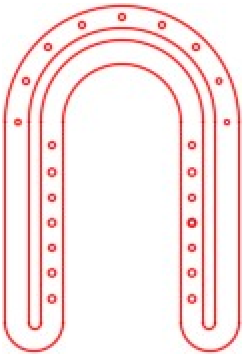


Figure Step 2B
Completed Drawing

AUTHOR'S COMMENTS: Do not use a calculator to complete this lab exercise or any other lab exercise in the AutoCAD 2D book. By using geometry principles, drawing techniques, object snapping, and construction objects, AutoCAD will do all the math for you.

Step 3

Set the viewres to 10,000 and regenerate your drawing.

Step 4

Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Inches.

Step 5

Check your drawing with the key. The key name is the same as the drawing name.

Step 6

If you have any errors, turn layer: Key off and correct your drawing.

Step 7

Turn layer: Key on to check your drawing. If it is still inaccurate, go back to Step 7.

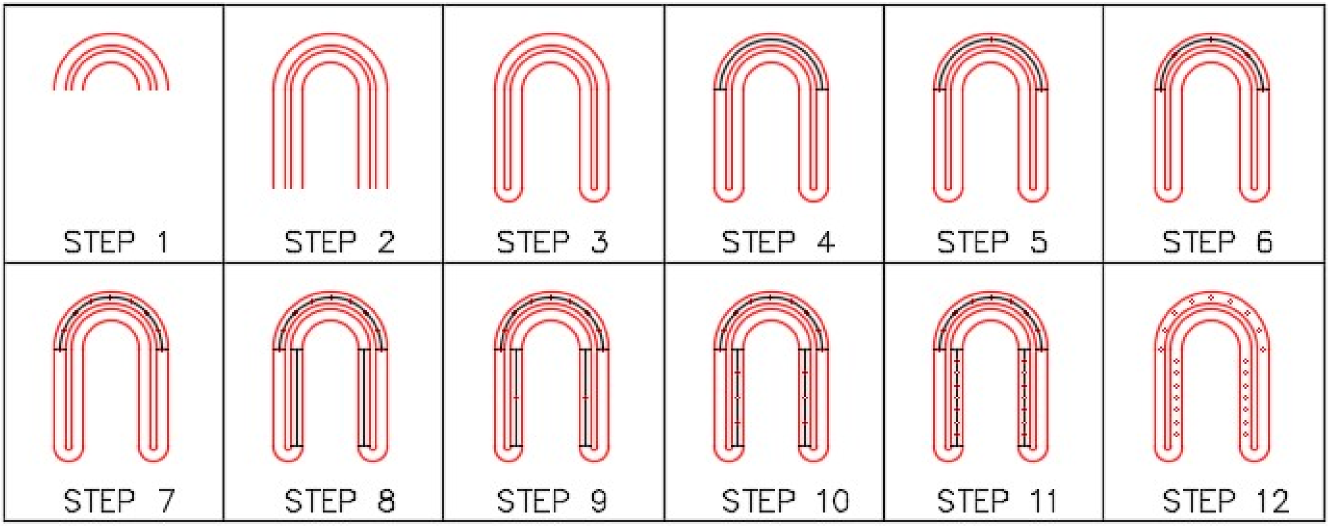
Step 8

Turn layer: Key off and freeze layer: Construction.

Step 9

Save and close the drawing.

AUTHOR’S CONSTRUCTION TECHNIQUES: The following steps are the construction technique suggested by the author to help you learn how to construct objects using AutoCAD. It is only the suggested method and if you can complete the drawing accurately using a different construction technique, that is what is important. You may want to compare your construction technique with the authors.


STEP 1
STEP 2
STEP 3
STEP 4
STEP 5
STEP 6
STEP 7
STEP 8
STEP 9
STEP 10
STEP 11
STEP 12

AUTHOR’S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

Draw two construction lines from end of arc to end of arc as shown in the figure. (Figure Hint 1)

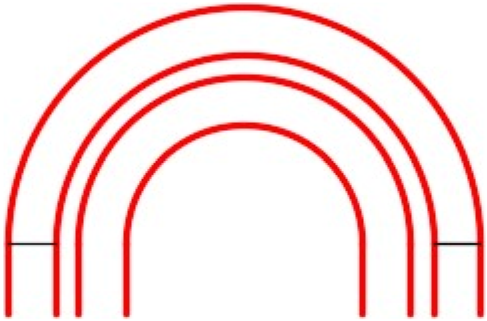


Figure Hint 1

Hint 2

Draw two circles locating their centers at the midpoint of the construction lines that you drew in Hint Step 1. Construction an arc from center of the circle on the right side to the center of the circle on the left side. Draw a circle at the midpoint of the arc. (Figure Step Hint 2)

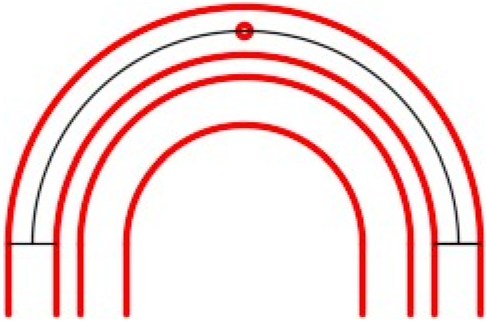


Figure Hint 2

Hint 3

Erase the construction arc you drew in Hint Step 2. Draw a construction arc from the center of the circle to the center of the circle. Draw a circle at the midpoint of the arc. You will have to repeat this three more times. (Figure Hint 3A and 3B)

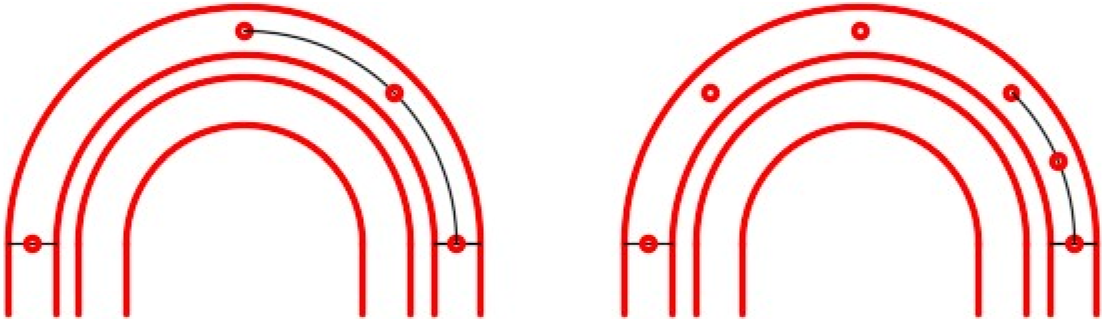


Figure Hint 3A

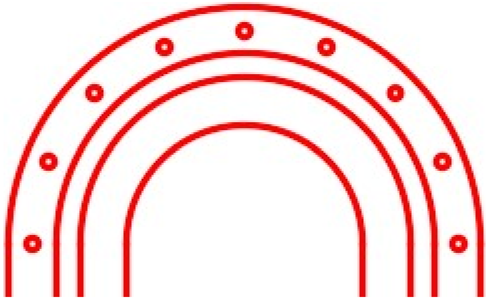


Figure Hint 3B

Hint 4

Insert four construction lines from the end of arc to the end of arc. Draw a construction line from the midpoint of line to midpoint of line on both sides. (Figure Hint 4)

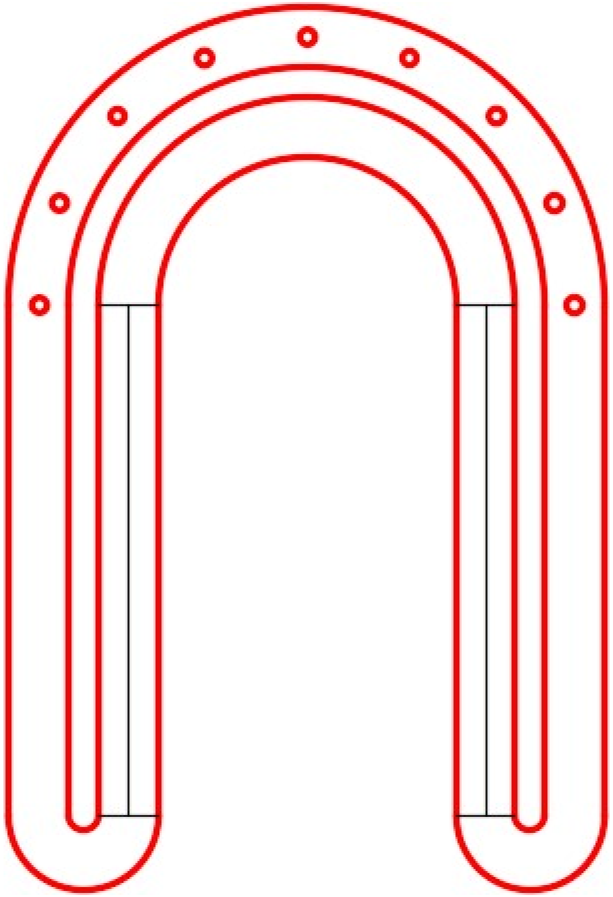


Figure Hint 4

Hint 5

Draw a circle at the midpoint of the line on both sides. Erase the construction line and draw a new construction line from the quad of the circle to the midpoint of the line. It is important to draw from the quad to the midpoint. Insert a circle at the midpoint of the line. Erase the construction line and repeat as shown in the figures. (Figure Hint 5A and 5B)

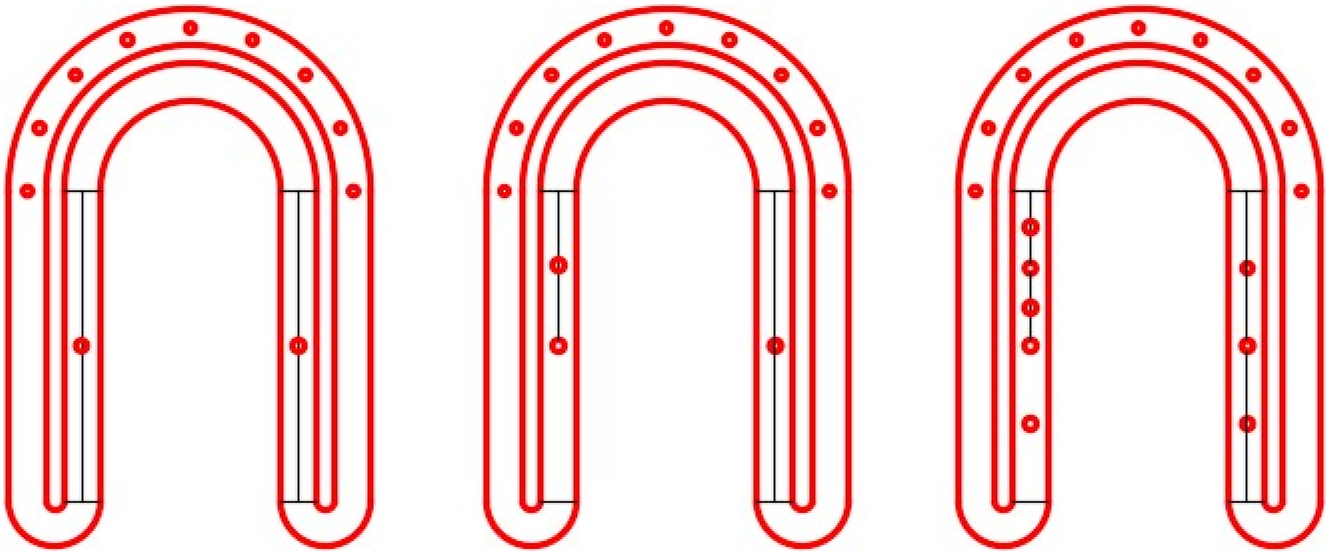


Figure Hint 5A

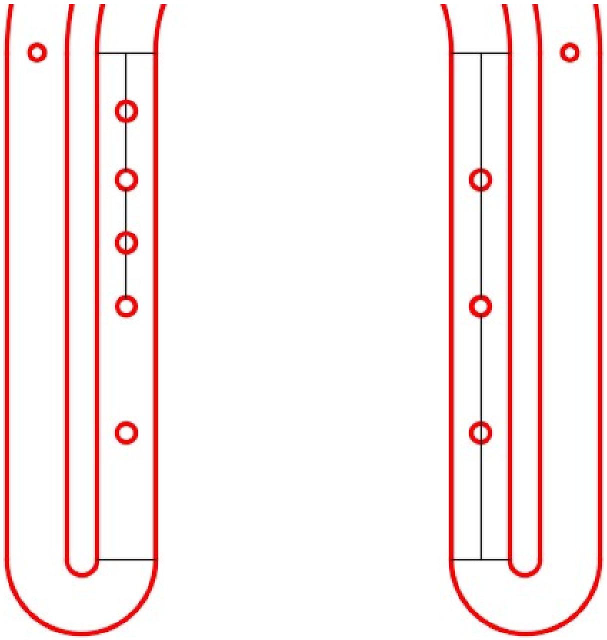


Figure Hint 5B

Module 10: Drawing Lines Using Polar Coordinates

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe the geometry of angles.
2. Describe the Polar Coordinate System.
3. Apply the LINE command to draw lines using the Polar Coordinate System.

Geometry Lesson: Angles

An *angle* is formed by two intersecting lines. When measuring an angle in a drawing, AutoCAD defines zero degrees at 3:00 o'clock with the positive angle in the counterclockwise direction and a negative angle in the clockwise direction. Study angle geometry in Figure 10-1, 10-2, and 10-3. These principles are used extensively to construct drawings in AutoCAD.

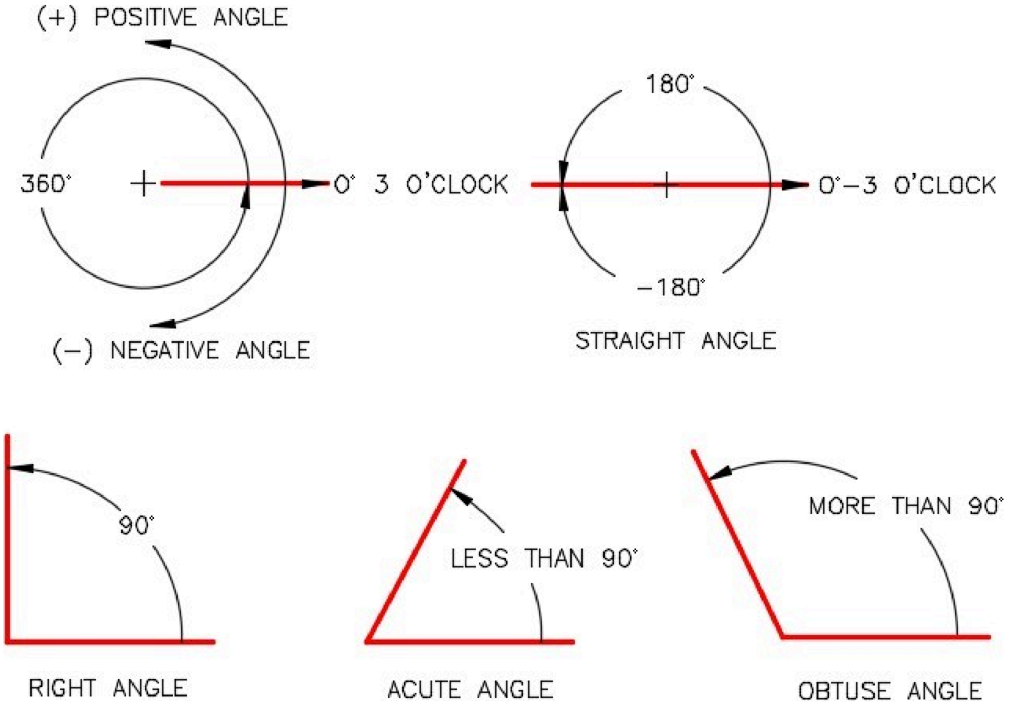
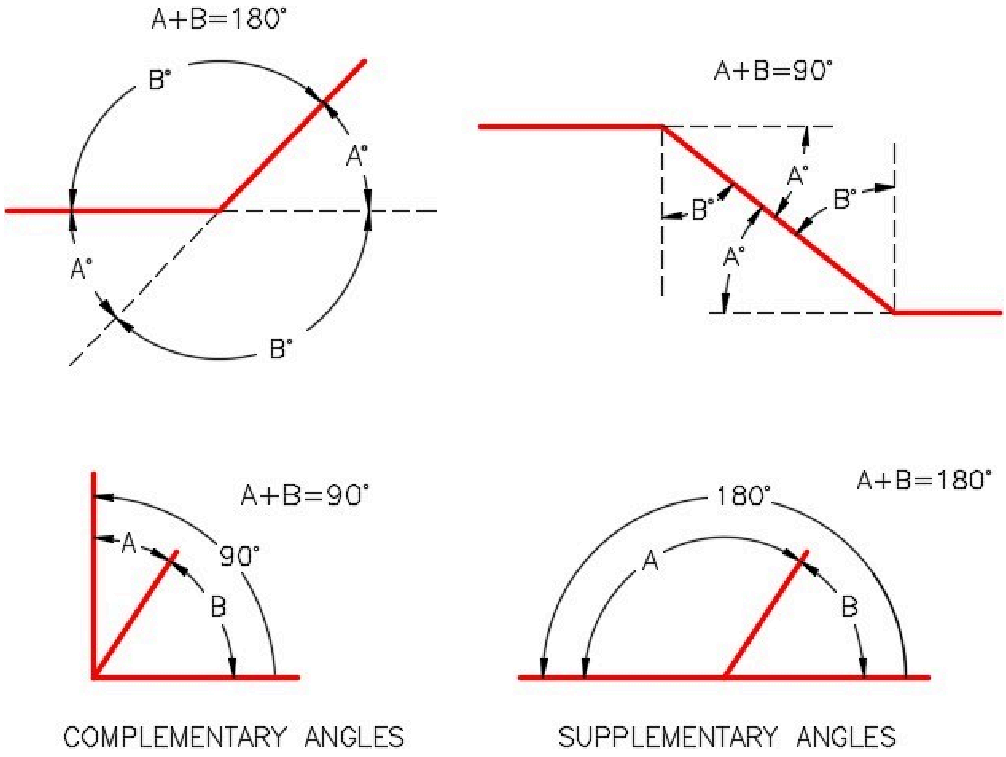

(+) POSITIVE ANGLE
360°
0° 3 O'CLOCK
(−) NEGATIVE ANGLE
180°
0°−3 O'CLOCK
−180°
STRAIGHT ANGLE
90°
RIGHT ANGLE
LESS THAN 90°
ACUTE ANGLE
MORE THAN 90°
OBTUSE ANGLE

Figure 10-1 Angles – Part 1


A+B=180°
B°
A°
A°
B°
A+B=90°
A°
B°
B°
A°
A+B=90°
A
90°
B
180°
A+B=180°
A
B

COMPLEMENTARY ANGLES SUPPLEMENTARY ANGLES

Figure 10-1 Angles – Part 2

All angles are express in either positive or negative degrees. AutoCAD will accept either. Since positive is the default, the positive sign does not have to precede the number but a negative angle must be preceded by the negative sign.

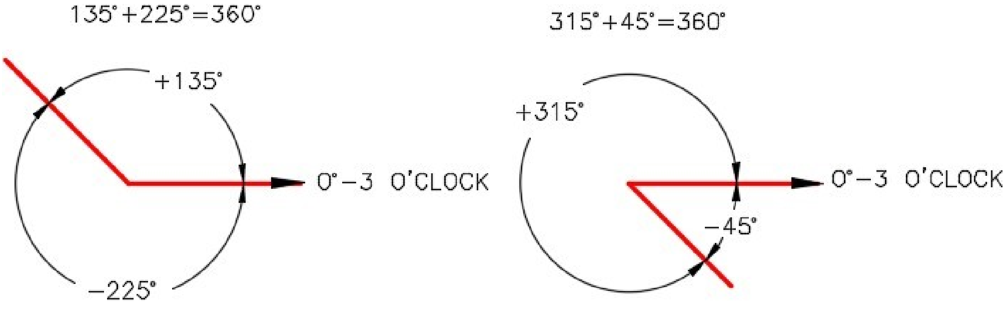

135°+225°=360°
+135°
0°–3 O'CLOCK
–225°
315°+45°=360°
+315°
0°–3 O'CLOCK
–45°

Figure 10-3 Angles – Part 3

WORKALONG: Working With Angles

Step 1

Study Figure 10-1, 10-2, and 10-3. Using what you learned and without drawing in AutoCAD, find the answers for the angles A to M. After you find a solution to all of the angles, check your answers with the solutions at the end of the module.

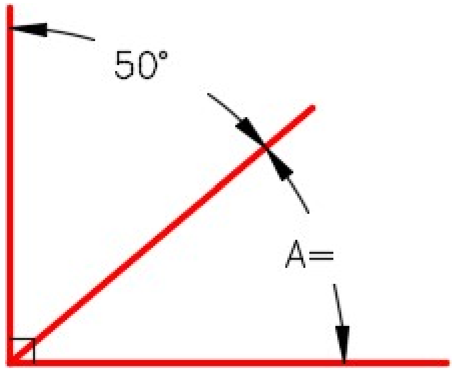


Figure Step 1A

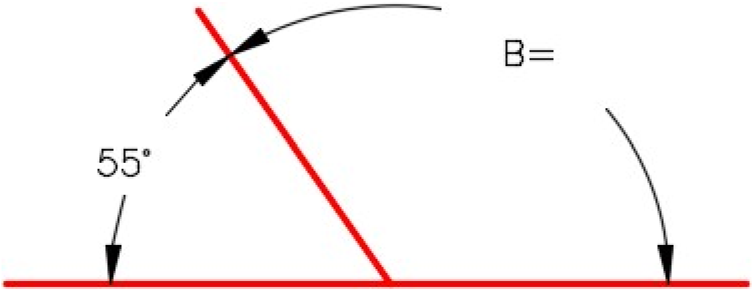


Figure Step 1B

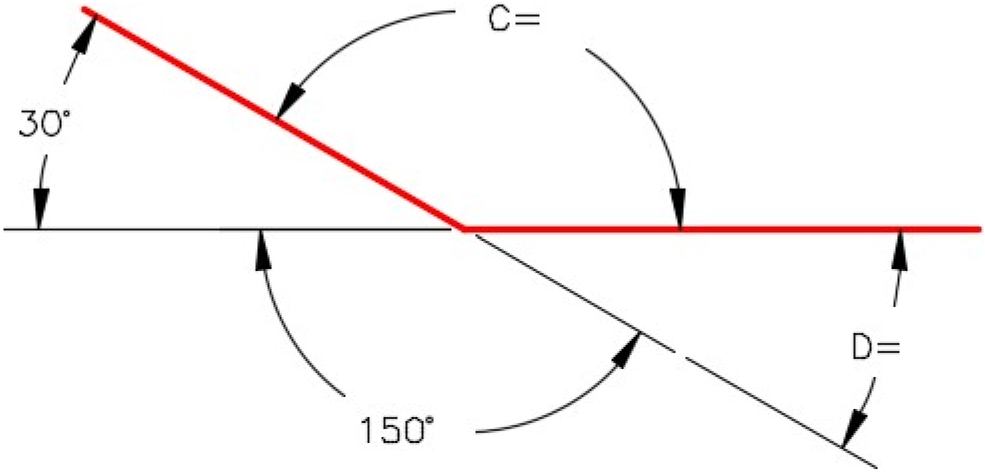


Figure Step 1C

AUTHOR'S COMMENTS: It is sometime easier to sketch the figure and calculate the angle using paper and pencil.

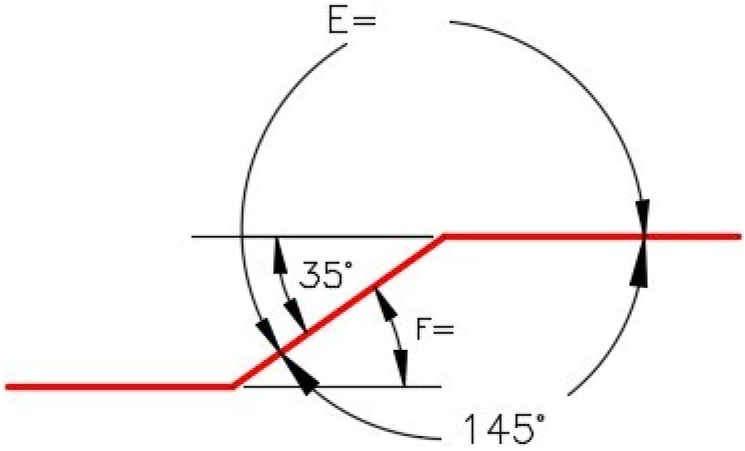


Figure Step 1D

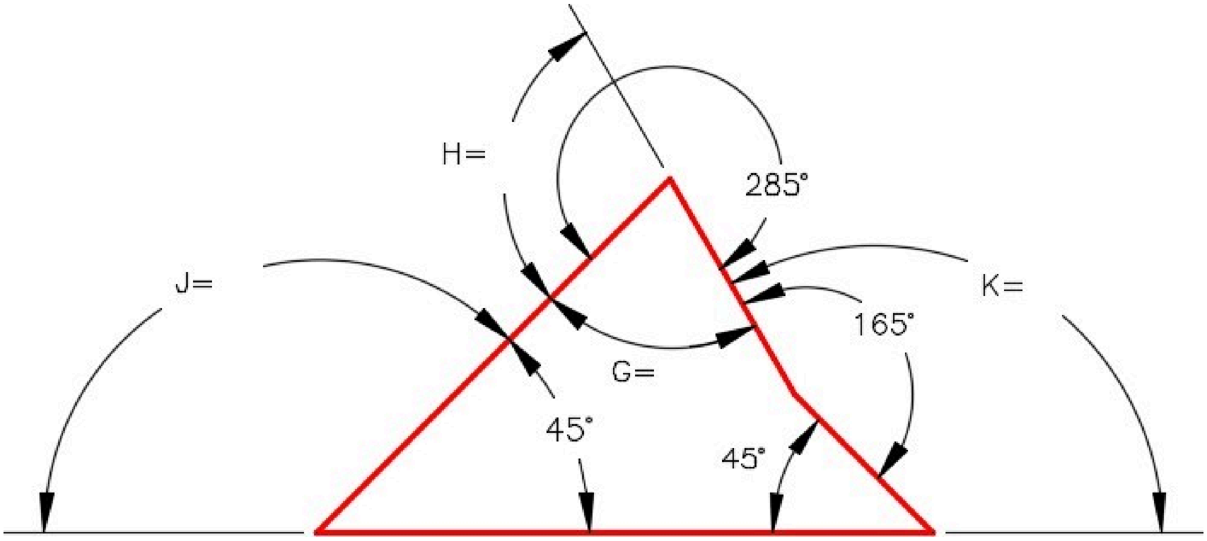

H=
285°
J=
K=
165°
G=
45°
45°

Figure Step 1E

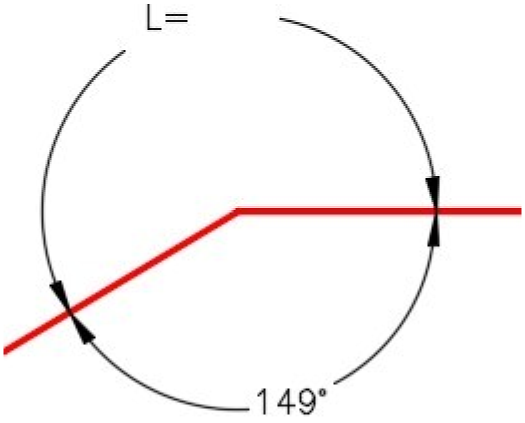


Figure Step 1F

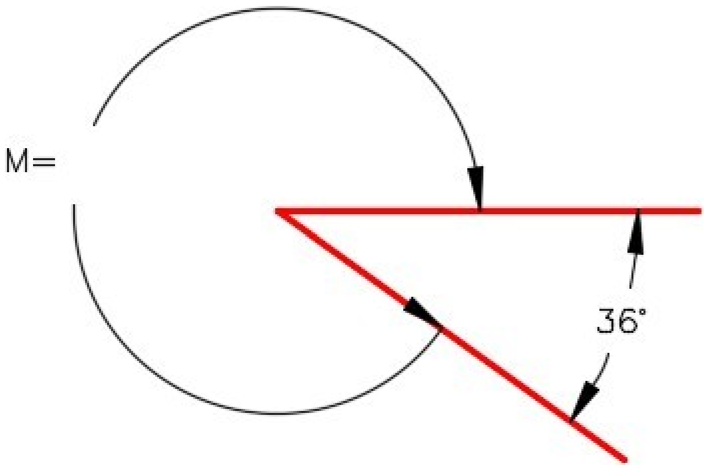


Figure Step 1G

The Polar Coordinate System

The *Polar Coordinate System* is a two-dimensional coordinate system in which each point on a plane is determined by a distance from a reference point and an angle from a reference direction. In AutoCAD, you use it when you are given a distance and an angle from a known coordinate location. All angles are measured with 0 degrees at 3:00 o'clock. The angles are positive in the counterclockwise direction and negative in the clockwise direction. See Figure 10-4.

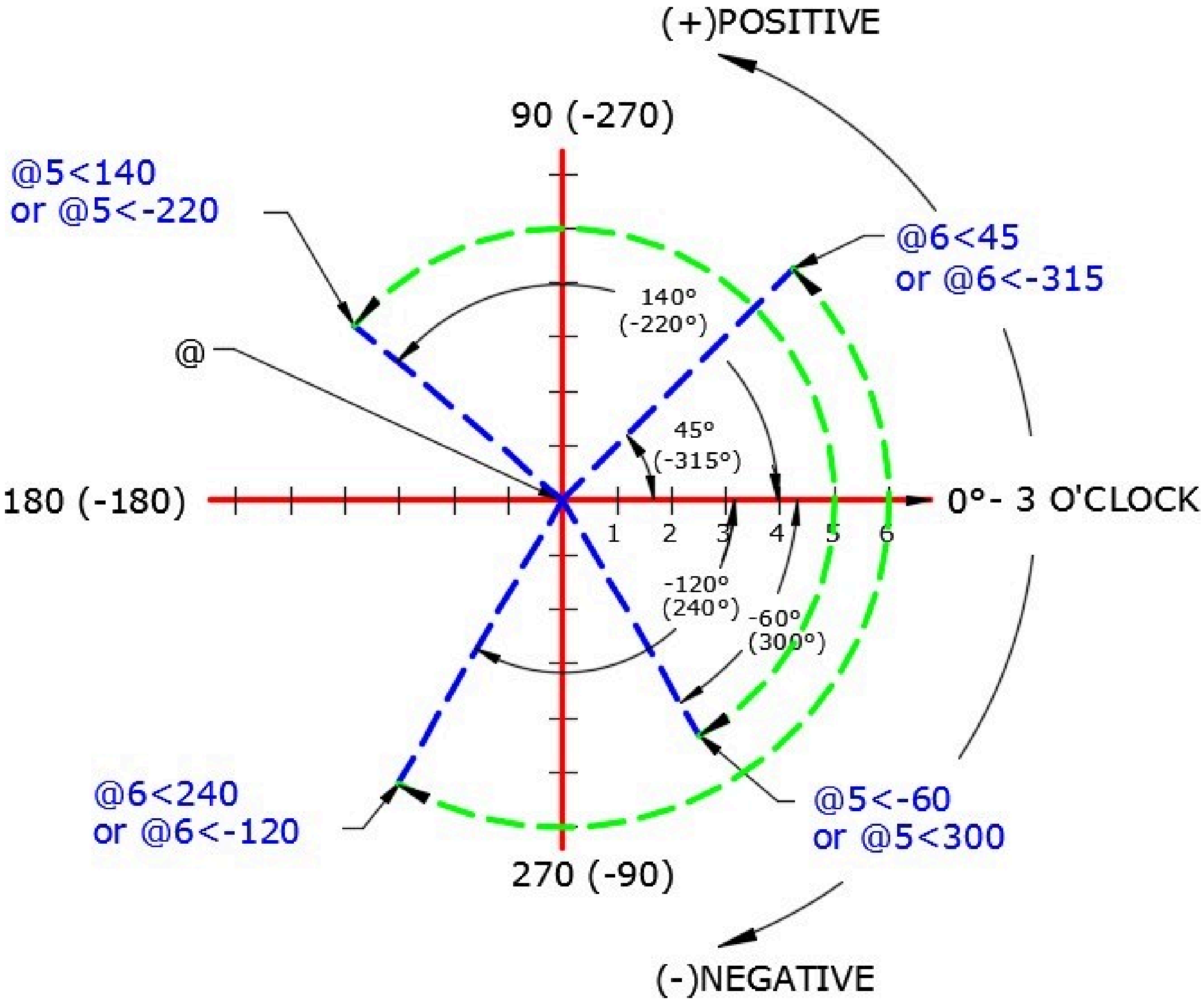

(+)POSITIVE
90 (-270)
@5<140
or @5<-220
@6<45
or @6<-315
140°
(-220°)
@
45°
(-315°)
180 (-180)
0°- 3 O'CLOCK
1
2
3
4
5
6
-120°
(240°)
-60°
(300°)
@6<240
or @6<-120
@5<-60
or @5<300
270 (-90)
(-)NEGATIVE

Figure 10-4 The Polar Coordinate System

A polar coordinate is entered into AutoCAD as a distance, a less than symbol, and an angle. In all cases, the @ symbol must precede the distance. i.e.

@6<45

means ' from the last point, go 6 units at an angle of 45 degrees, positive, starting at 3 o'clock '. AutoCAD always assumes positive as default, therefore, the positive sign can be omitted. i.e.

@5<-60

WORKALONG: Using the Polar Coordinate System – Part 1

Workalong and complete the following steps to solve the angles and polar coordinates. If necessary, use paper and pencil. Do not use AutoCAD.

Step 1

Study the figure and solutions for numbers 1 to 6. Solve the answers for numbers 7 to 11. (Figure Step 1)

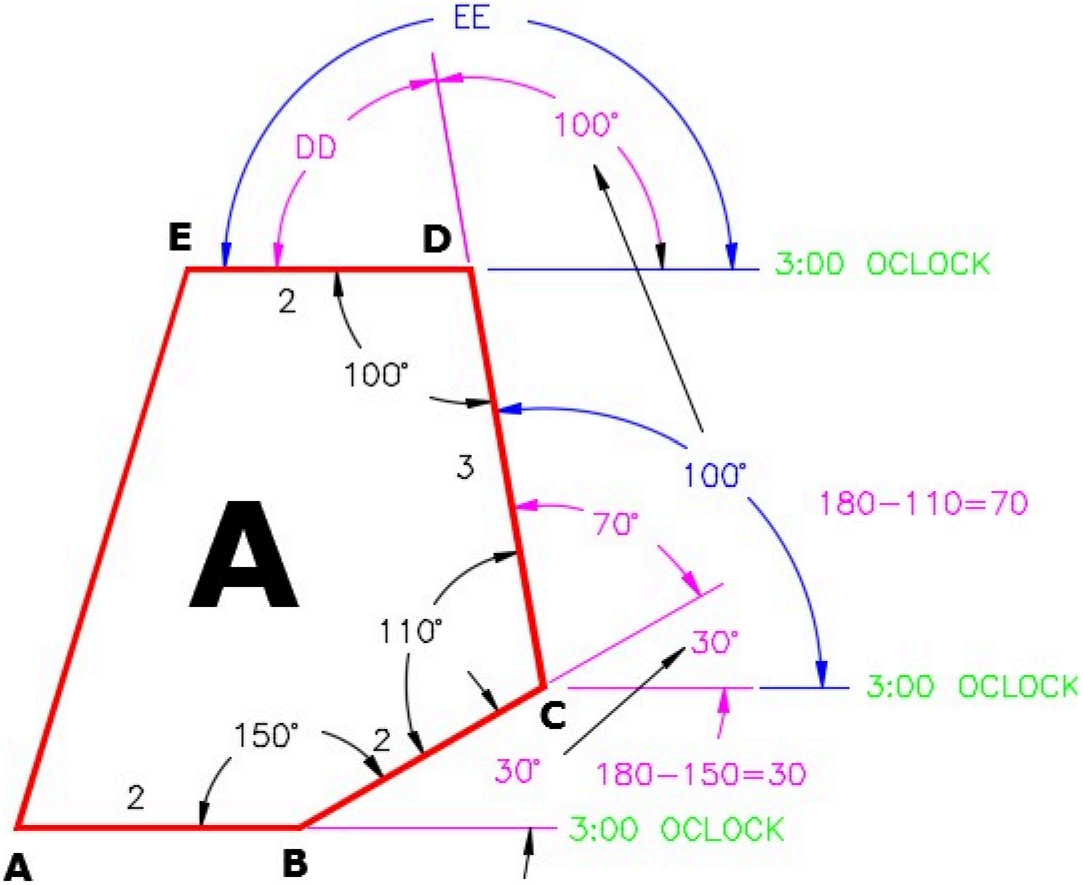

EE
DD
100°
E
D
3:00 OCLOCK
2
100°
3
100°
180−110=70
A
70°
110°
30°
3:00 OCLOCK
150°
2
C
30°
180−150=30
2
3:00 OCLOCK
A
B

Figure Step 1

AUTHOR'S COMMENTS: The answers are at the end of the module. Do not look at the answers until you think you have correctly solved each one.

Object A Solution:

1. Find the polar coordinate A TO B:	@2<0
2. Calculate the first angle:	180 – 150 = 30 degrees
3. Find the polar coordinate B TO C:	@2<30
4. Copy the 30 degree angle to the next corner.	
5. Calculate the second angle:	180 – 110 = 70 degrees
6. Add the two angles:	30 + 70 = 100
7. Find the polar coordinate C TO D:	@ <
8. Copy the angle 100 to the next corner.	
9. Calculate angle DD:	DD =
10. Find angle EE:	EE =
11. Find the polar coordinate D TO E:	@ <

Step 2

Study the figure and find the solutions for numbers 1 to 11. (Figure Step 3)

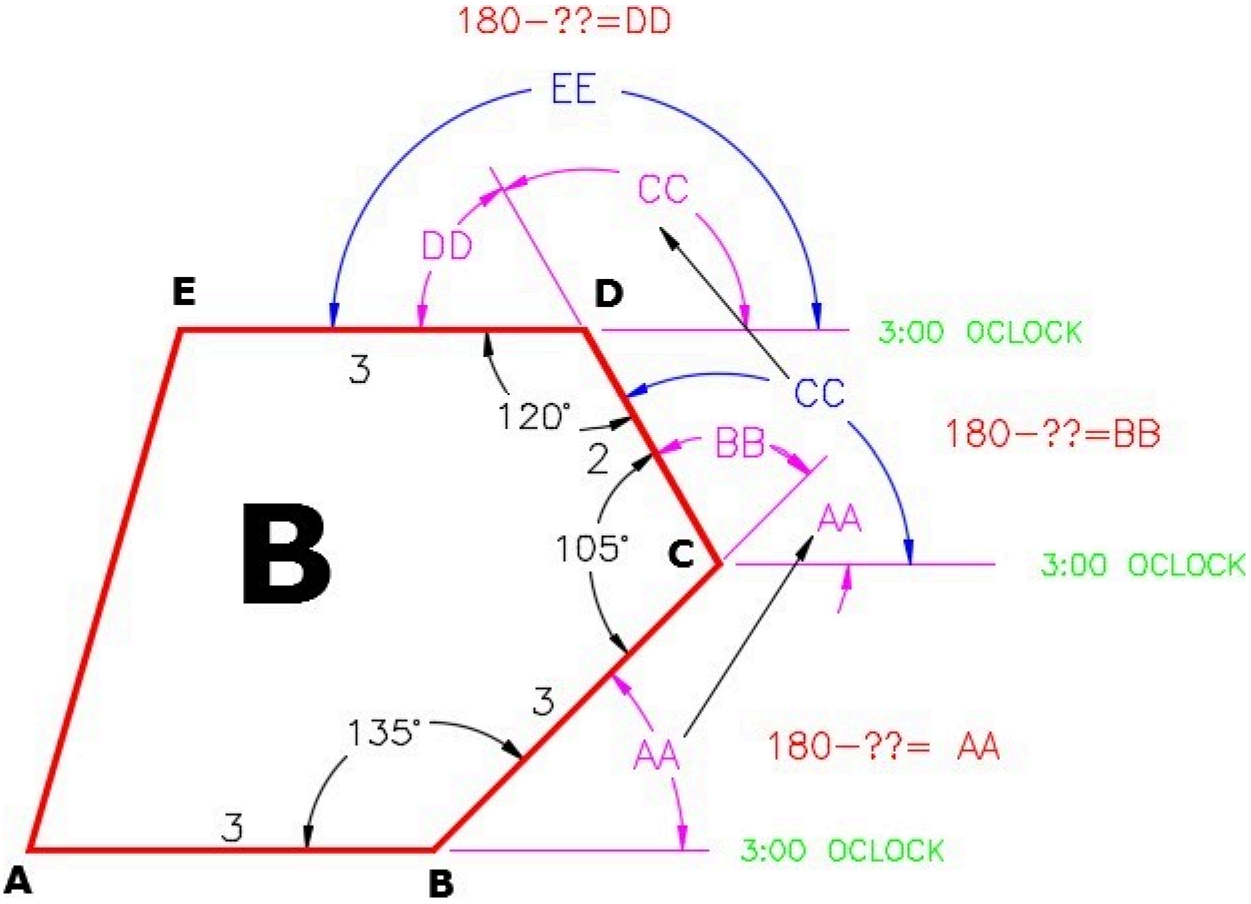

180−??=DD
EE
CC
DD
E
D
3:00 OCLOCK
3
120°
CC
180−??=BB
2
BB
AA
B
105°
C
3:00 OCLOCK
3
135°
AA
180−??= AA
3
A
B
3:00 OCLOCK

Figure Step 2

AUTHOR'S COMMENTS: The answers are at the end of the module. Do not look at the answers until you think you have correctly solved each number.

Object B Solution:

1. Find polar coordinate A TO B	@ <
2. Calculate angle AA:	
3. Find polar coordinate B TO C	@ <
4. Copy angle AA to the next corner	
5. Calculate angle BB	
6. Find angle CC	
7. Find polar coordinate C TO D	@ <
8. Copy angle CC to the next corner	
9. Calculate angle DD	
10. Find angle EE	
11. Find polar coordinate D TO E	@ <

WORKALONG: Using the Polar Coordinate System – Part 2

Step 1

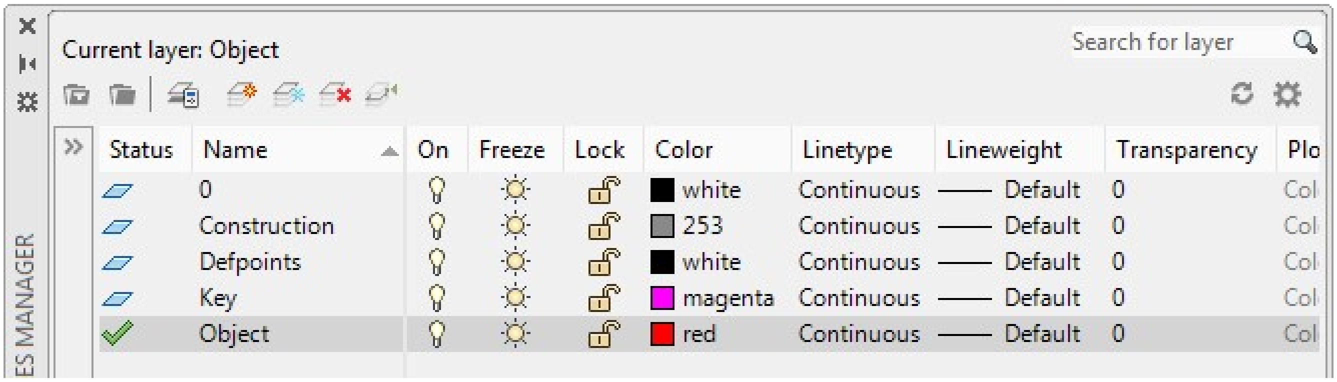
Start a new drawing using the template: 2D English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 10-1.

Step 3

Create layers: Object and Construction. (Figure Step 3)



Current layer: Object

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight	Transparency	Plo
	0				■ white	Continuous	—— Default	0	Col
	Construction				■ 253	Continuous	—— Default	0	Col
	Defpoints				■ white	Continuous	—— Default	0	Col
	Key				■ magenta	Continuous	—— Default	0	Col
✓	Object				■ red	Continuous	—— Default	0	Col

Figure Step 3

Step 4

Set layer: Object as the current layer and enter the LINE command, as shown below, to draw the object. (Figure Step 4A)

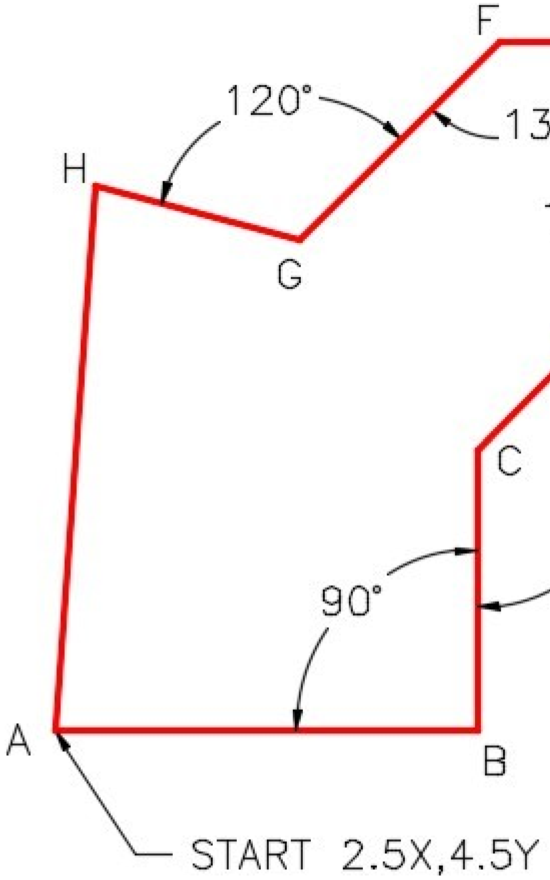

F
E
120°
135°
H
125°
D
G
C
135°
90°
A
B
START 2.5X,4.5Y

Figure Step 4A

LINE	DISTANCE
A - B	3
B - C	2
C - D	2
D - E	1.5
E - F	1
F - G	2
G - H	1.5

Command: **L**

Specify first point: **2.5,4.5**

(Start with an absolute Cartesian Coordinate.)

Specify next point or [Undo]: **@3<0**

(A to B is a horizontal line which is zero degrees.)

Specify next point or [Undo]: **@2<90**

(B to C is a vertical line at 90 degrees.)

(Figure Step 4B)

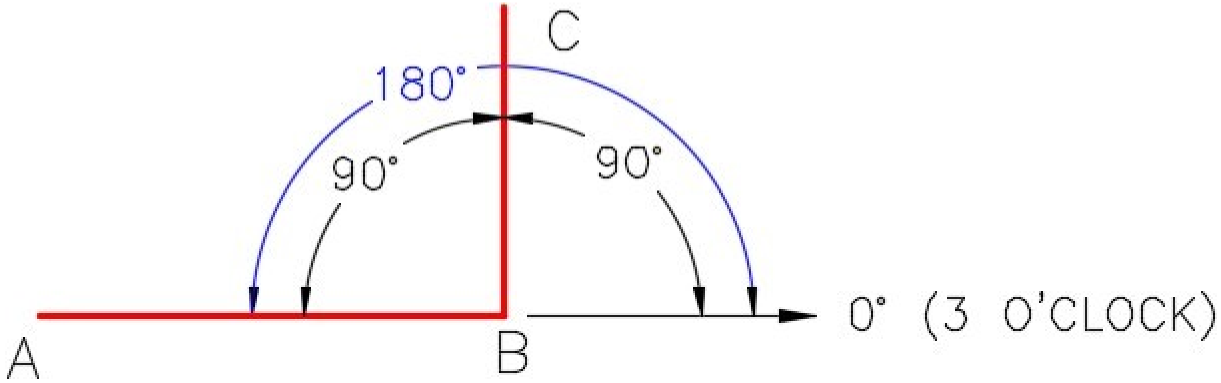


Figure Step 4B

AUTHOR'S COMMENTS: Refer to the dimensioned drawing in Figure Step 4 when you construct this drawing and try to understand the polar coordinates that you are entering.

Specify next point or [Close/Undo]: **@2<45**

(C to D is 45 degrees positive.)

(Figure Step 4C)

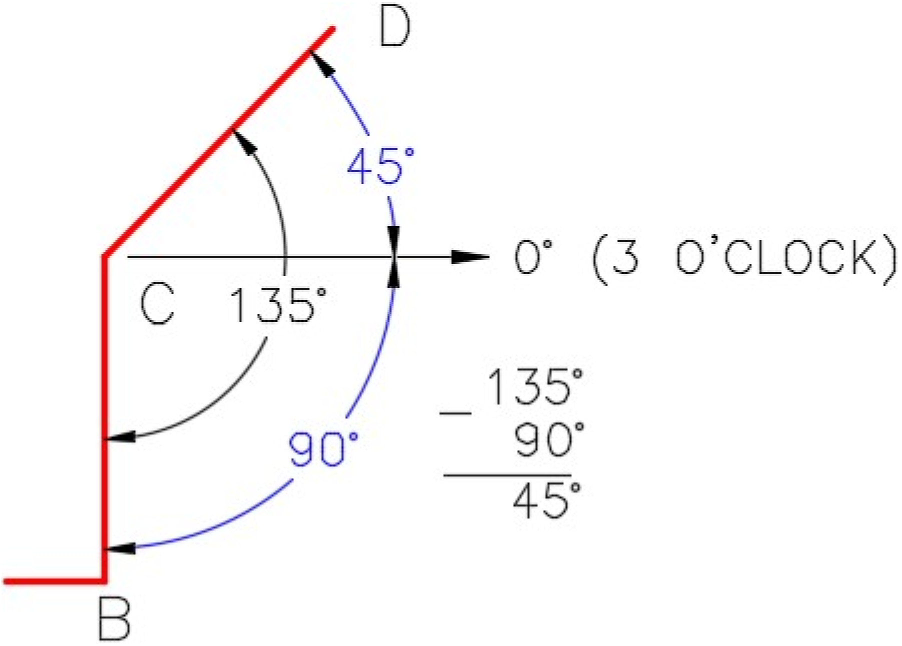

D
45°
0° (3 O'CLOCK)
C
135°
– 135°
90°
45°
90°
B

Figure Step 4C

Specify next point or [Close/Undo]: @**1.5<100**

(D to E)

(Figure Step 4D)

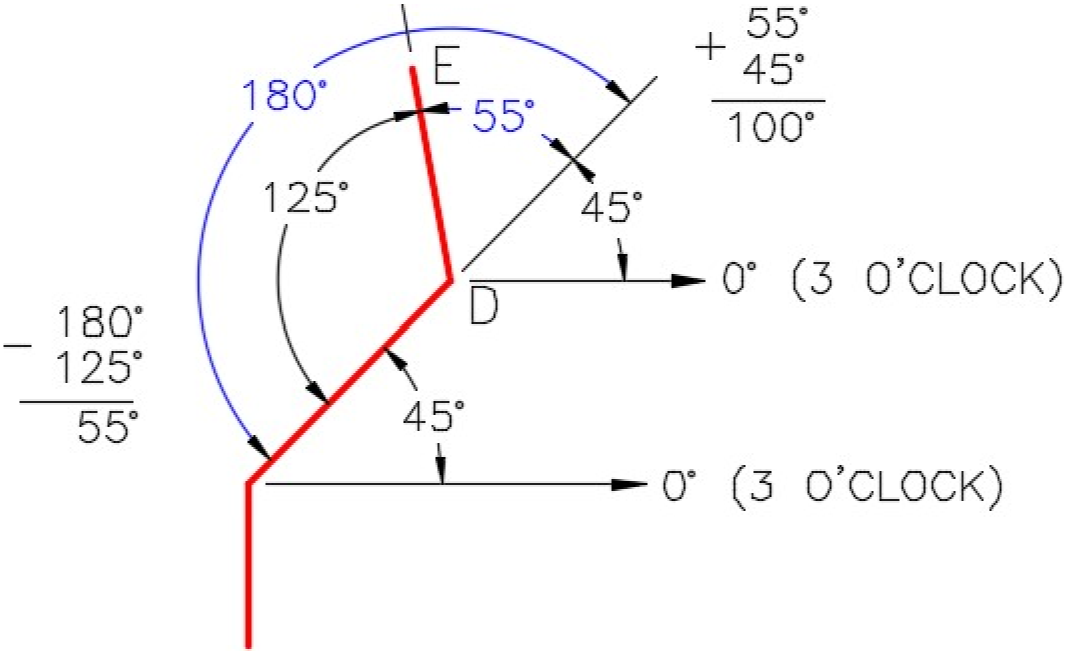

+ 55°
45°
100°
180°
E
55°
125°
45°
0° (3 O'CLOCK)
D
– 180°
125°
55°
45°
0° (3 O'CLOCK)

Figure Step 4D

Specify next point or

[Close/Undo]: **@1<180**

(E to F is a horizontal line at 180 degrees.)

(Figure Step 4E)

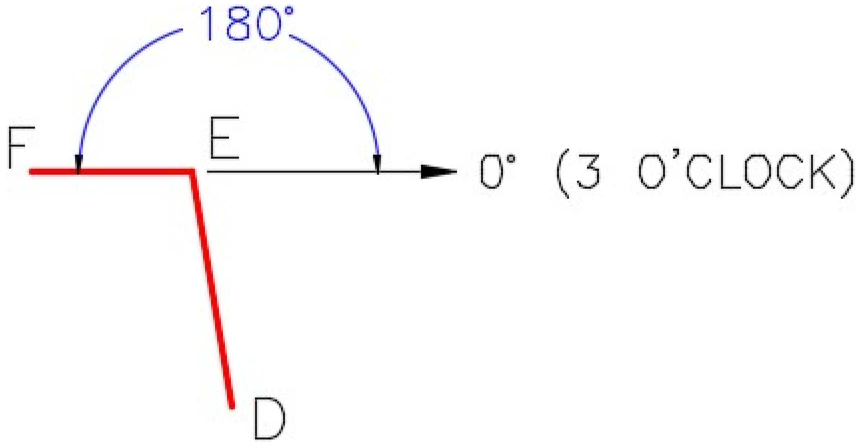
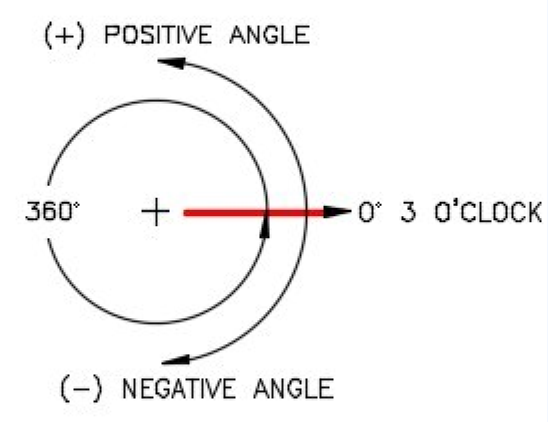

180°
F
E
0° (3 O'CLOCK)
D

Figure Step 4E

MUST KNOW: AutoCAD measures angles starting with 0 degrees at 3:00 o'clock. Positive angles are measured in the counterclockwise direction and negative angles are measured in the clockwise direction.


(+) POSITIVE ANGLE
360°
0° 3 O'CLOCK
(−) NEGATIVE ANGLE

Specify next point or [Close/Undo]: **@2<225**

(F to G)

(Figure Step 4F)

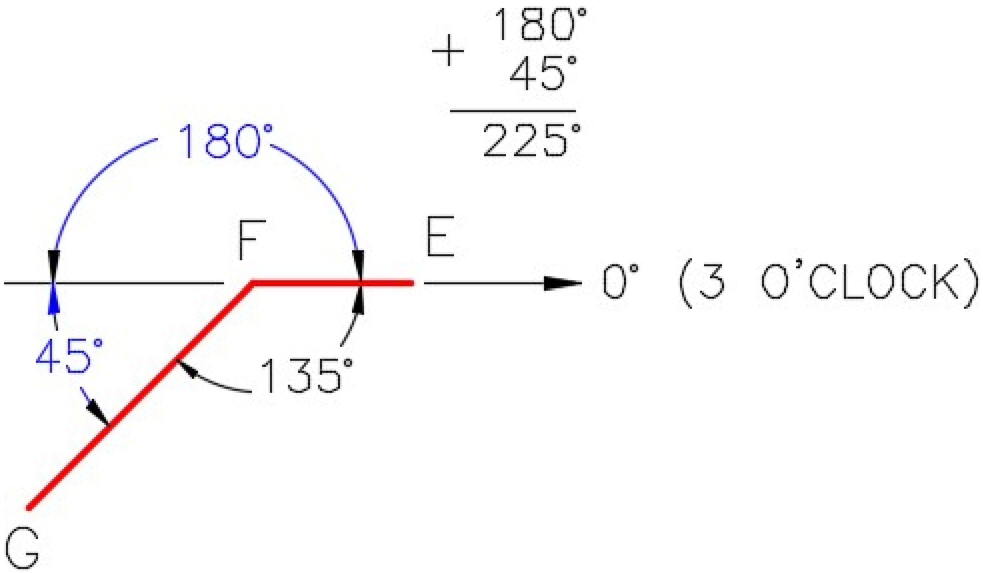

+ 180°
45°
225°
180°
F
E
0° (3 O'CLOCK)
45°
135°
G

Figure Step 4F

Specify next point or [Close/Undo]: **@1.5<165**

(G to H)

Specify next point or [Close/Undo]: **C**

(Use Close for H to A)

Specify next point or [Close/Undo]:

Command:

(Figure Step 4G)

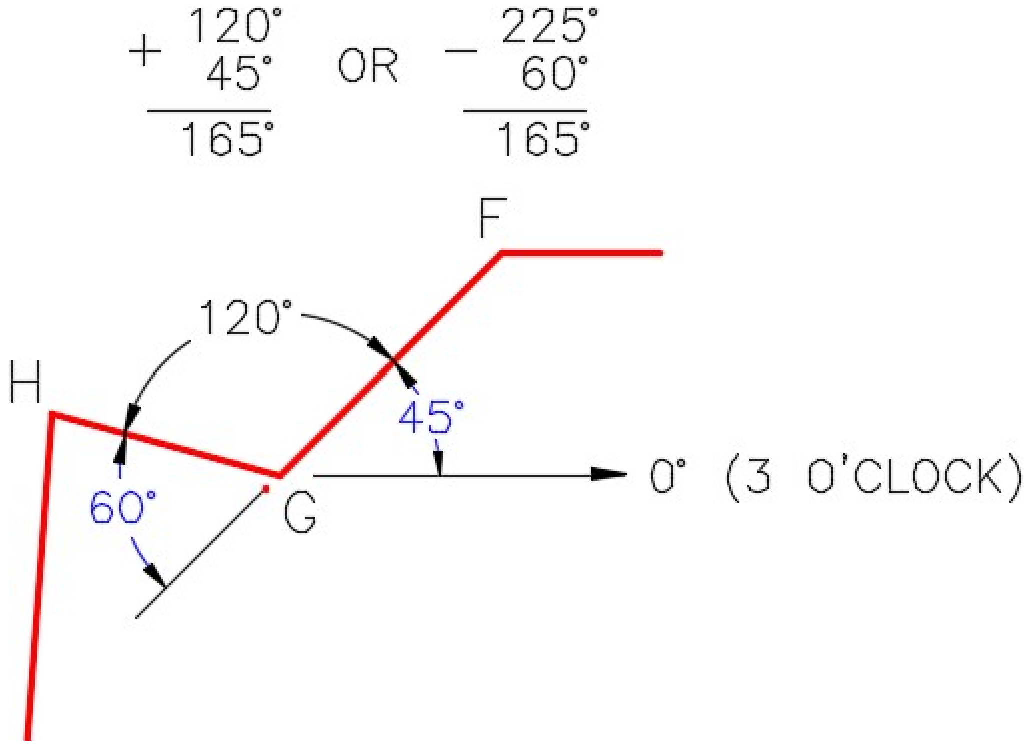

+ 120°
45°
165°
OR
− 225°
60°
165°
F
120°
H
45°
60°
G
0° (3 O'CLOCK)

Figure Step 4G

Step 5

Your completed drawing should match the figure. (Figure Step 5)

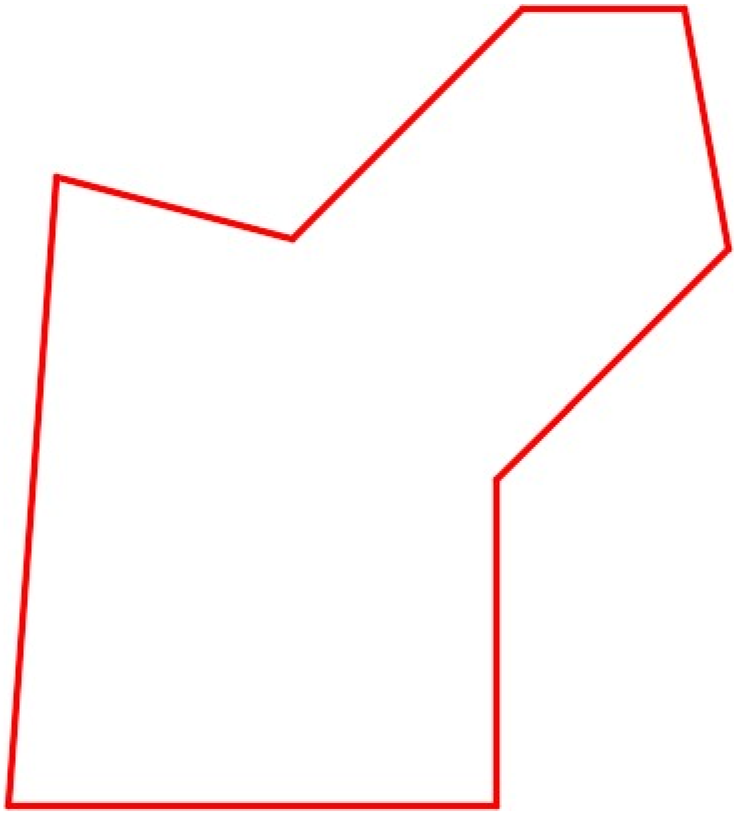


Figure Step 5

Step 6

Save and close the drawing.

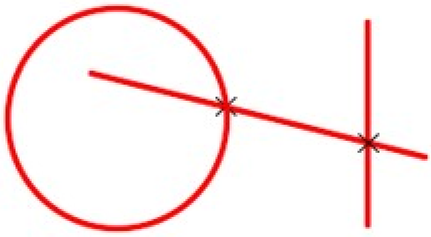
Object Snap Mode - Intersection				
Modes	**Abbreviations**	**Icon**	**Marker**	**The AutoCAD Object**
Intersection	Int	✕	—	

Figure 10-5 Object Snap Mode – Intersection

WORKALONG: Using the Object Snap Mode – Intersection

Step 1

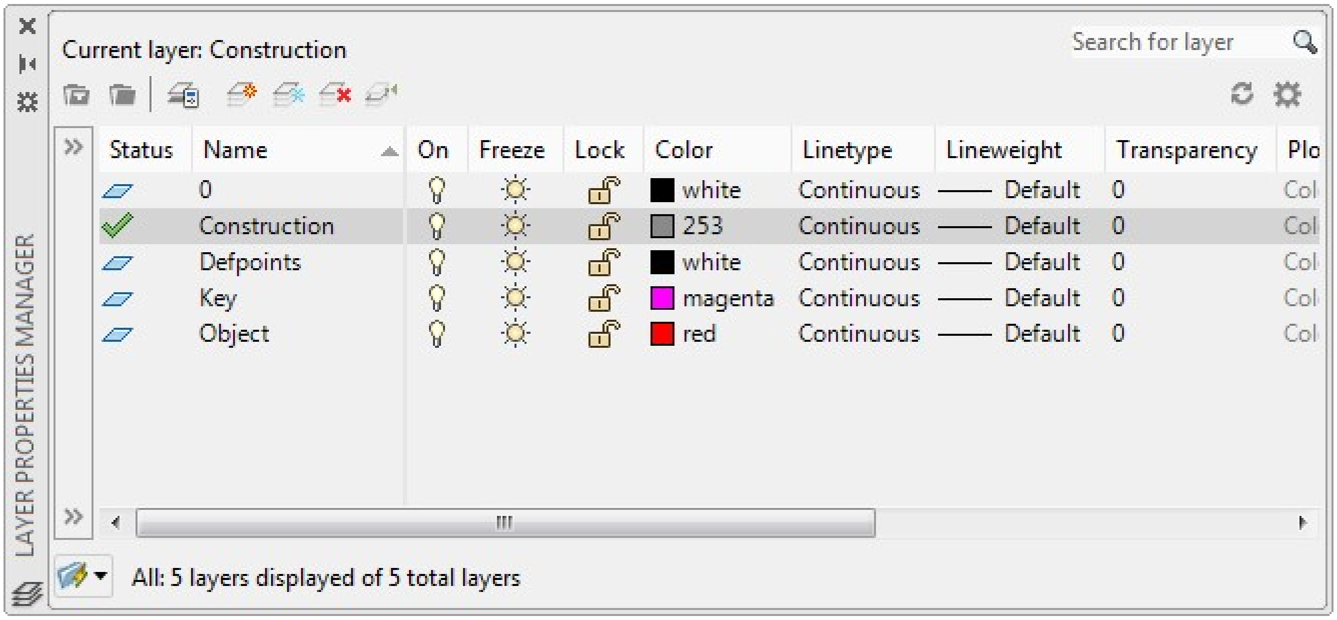
Start a new drawing using the template: 2D English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 10-2.

Step 3

Create the layers: Object and Construction. Set layer: Construction as the current layer. (Figure Step 3)



Current layer: Construction

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight	Transparency	Plo
	0				■ white	Continuous	—— Default	0	Col
✓	Construction				■ 253	Continuous	—— Default	0	Col
	Defpoints				■ white	Continuous	—— Default	0	Col
	Key				■ magenta	Continuous	—— Default	0	Col
	Object				■ red	Continuous	—— Default	0	Col

All: 5 layers displayed of 5 total layers

Figure Step 3

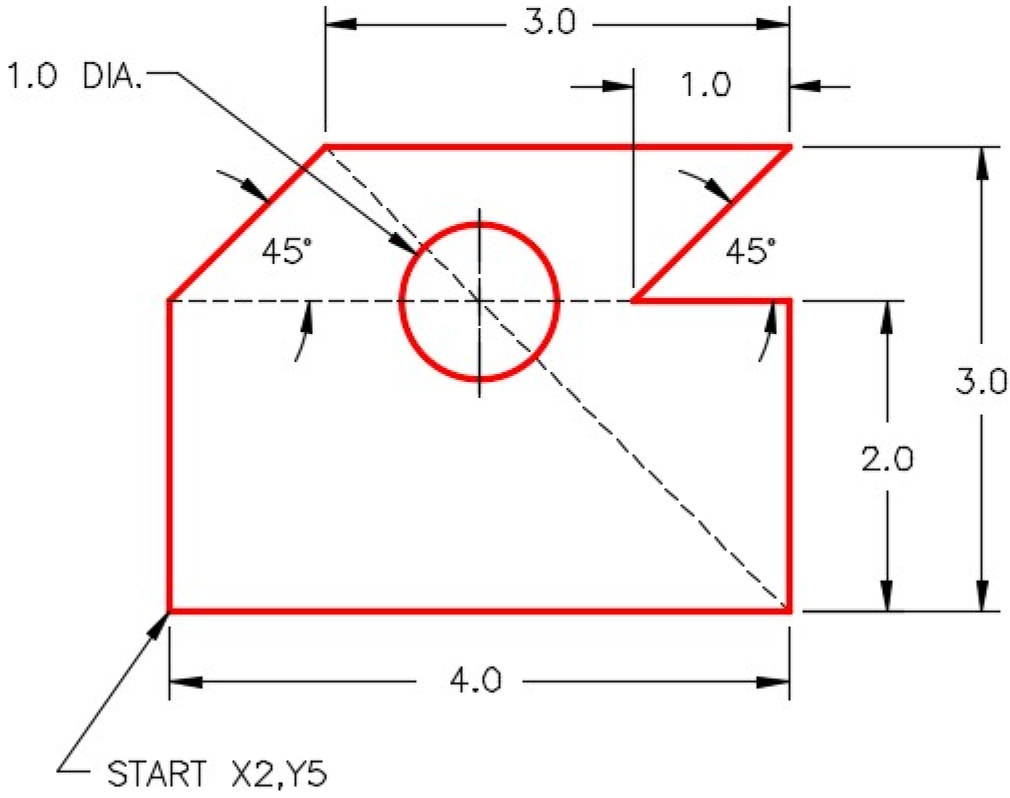


Figure Step 4A

Step 4

Refer to the figures and enter the LINE command, as shown below, to draw four construction lines. (Figure Step 4A and 4B)

Command: **L**

LINE Specify first point: **2,5**

Specify next point or [Undo]: **@4<0**

Specify next point or [Undo]: **@2<90**

Specify next point or [Close/Undo]: **@1<90**

Specify next point or [Close/Undo]: **@3<180**

Specify next point or [Close/Undo]: **@1<180**

Specify next point or [Close/Undo]: **@1<270**

Specify next point or [Close/Undo]: **C**

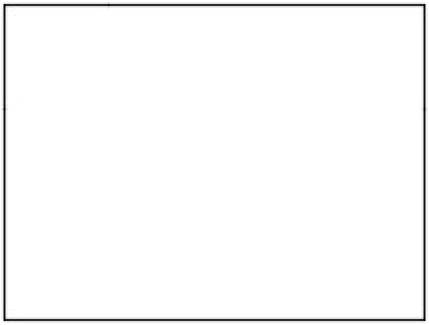


Figure Step 4B

AUTHOR'S COMMENTS: Note how I drew the rectangle by drawing lines from endpoint to endpoint of the object's geometry. This workalong is teaching you a drawing technique that will help you improve your drawing speed.

Step 5

Ensure that osnap is enable and then draw an additional construction line by snapping to the endpoint of the existing line and 1 unit in the -X direction. (Figure Step 5A and 5B)

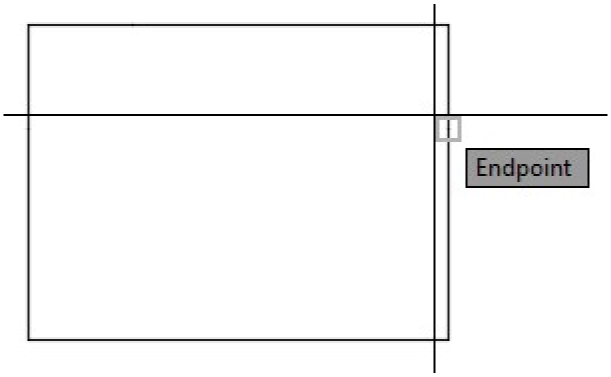


Figure Step 5B

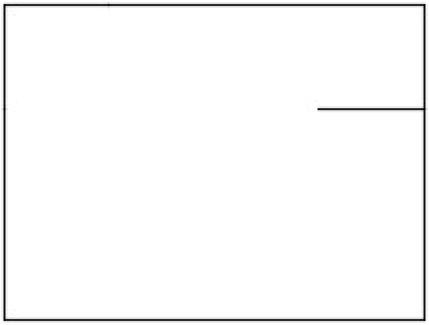


Figure Step 5B

AUTHOR'S COMMENTS: As I commented in Step 4, I drew the rectangle by drawing lines from endpoint to endpoint of the object's geometry. In this step, I can now snap to endpoint of line that is 2 inches long. Even though you cannot see the endpoint of the line, osnap will find it easily by move the graphic cursor over the line.

Step 6

Set layer: Object as the current layer. Enter the LINE command and enable osnap. Draw the perimeter lines of the object by snapping to the endpoints of the construction lines. (Figure Step 6A, 6B, 6C, 6D, 6E, 6F, 6G, 6H, and 6J)

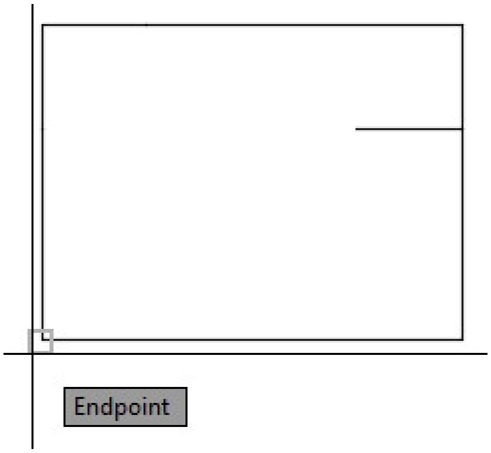


Figure Step 6A

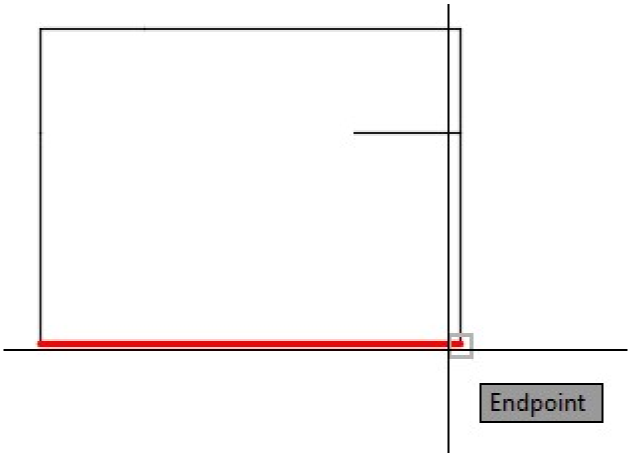


Figure Step 6B

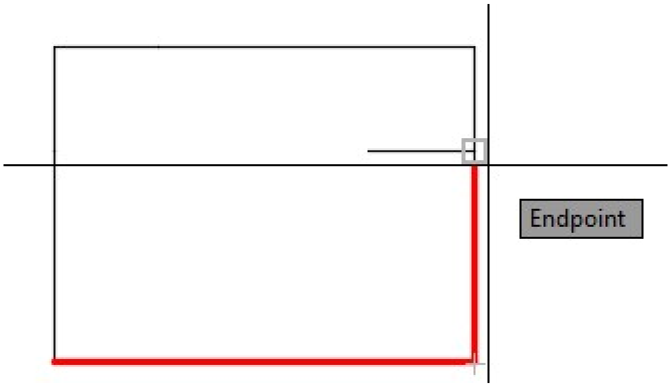


Figure Step 6C

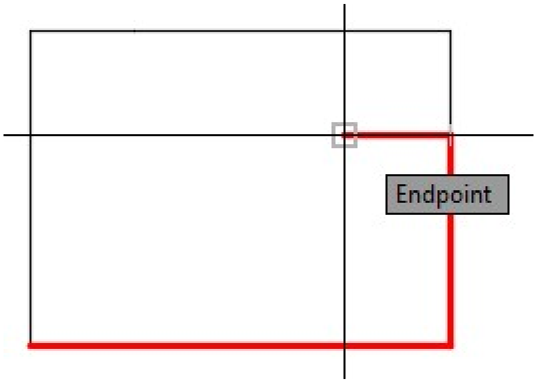


Figure Step 6D

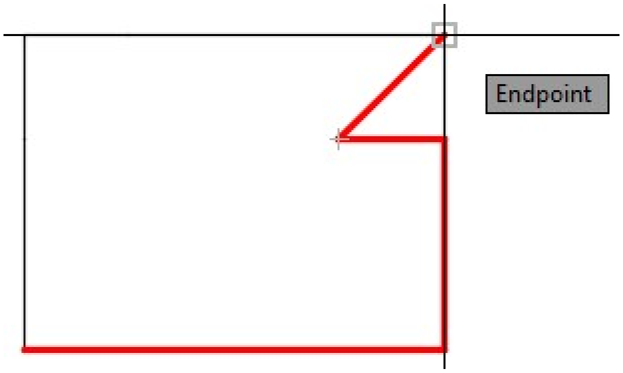


Figure Step 6E

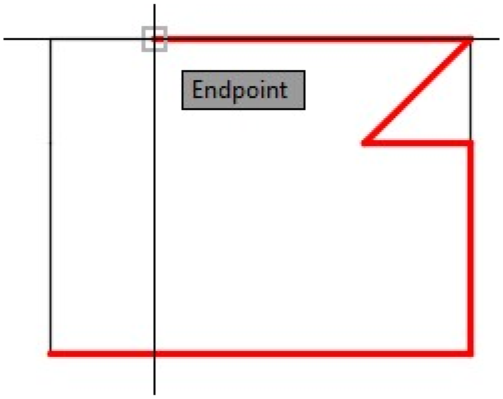


Figure Step 6F

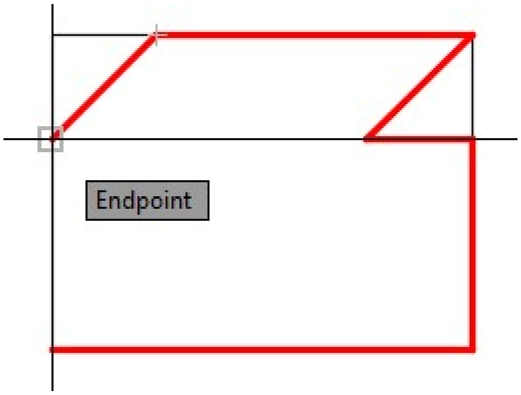


Figure Step 6G

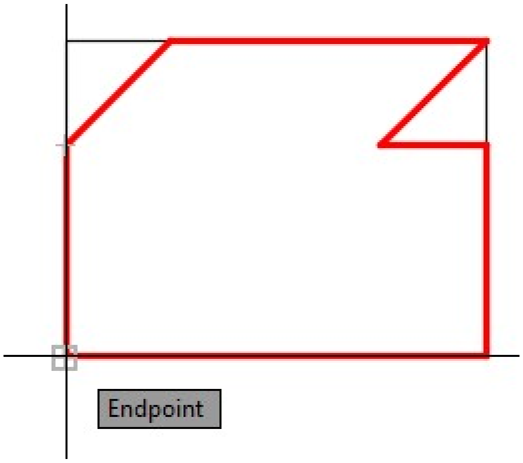


Figure Step 6H

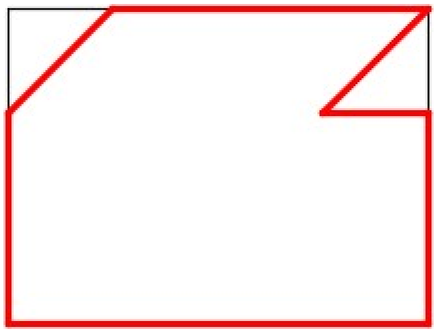


Figure Step 6J

Step 7

On layer: Construction , draw two construction lines by snapping to the endpoint of the existing lines as shown in the figure. (Figure Step 7)

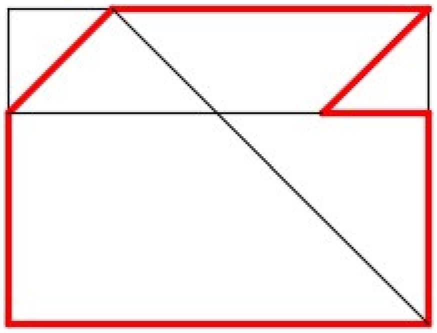


Figure Step 7

Step 8

Enter the CIRCLE command and draw a 1 diameter circle locating its center point by

snapping to intersection of the two construction lines you drew in Step 7. (Figure Step 8A and 8B)

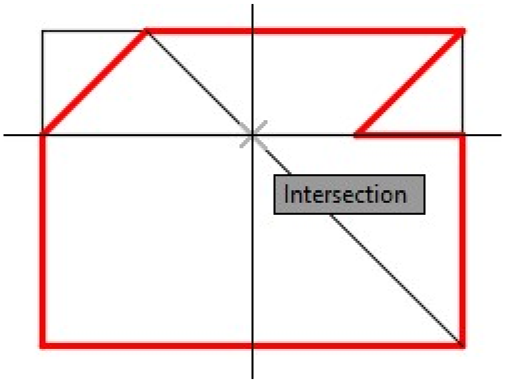


Figure Step 8A

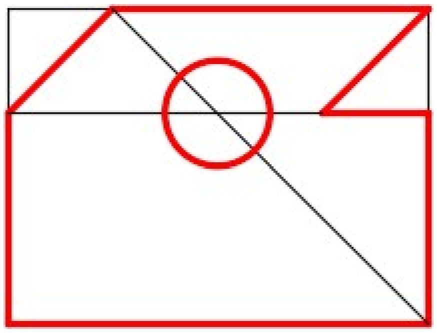


Figure Step 8B

Step 9

Freeze layer: Construction. (Figure Step 9)

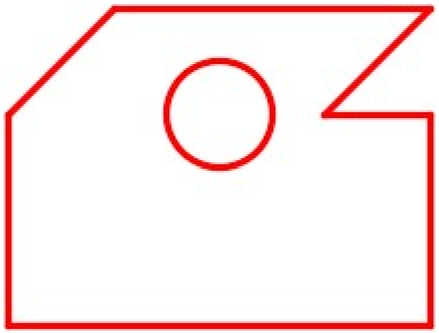


Figure Step 9

Step 10

Save and close the drawing.

Key Principles

Key Principles in Module 10

1. The Polar Coordinate System is used to draw lines using a distance and an angle from a known coordinate location.
2. By default, an AutoCAD polar coordinate measures angles starting 0 degrees at 3:00 o'clock.
3. In a polar coordinate, positive angles are measured counterclockwise and negative angles are measured clockwise.
4. A polar coordinate is entered into AutoCAD as a distance, a less than symbol, and an angle. In all cases, the @ symbol must precede the distance.

Lab Exercise 10-1

Time allowed: 30 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 10-1	2D English	Inches

Layering Scheme

Layer Name	Objects on Layer	Color
Construction	Construction objects	253
Object	Lines	Red

Step 1

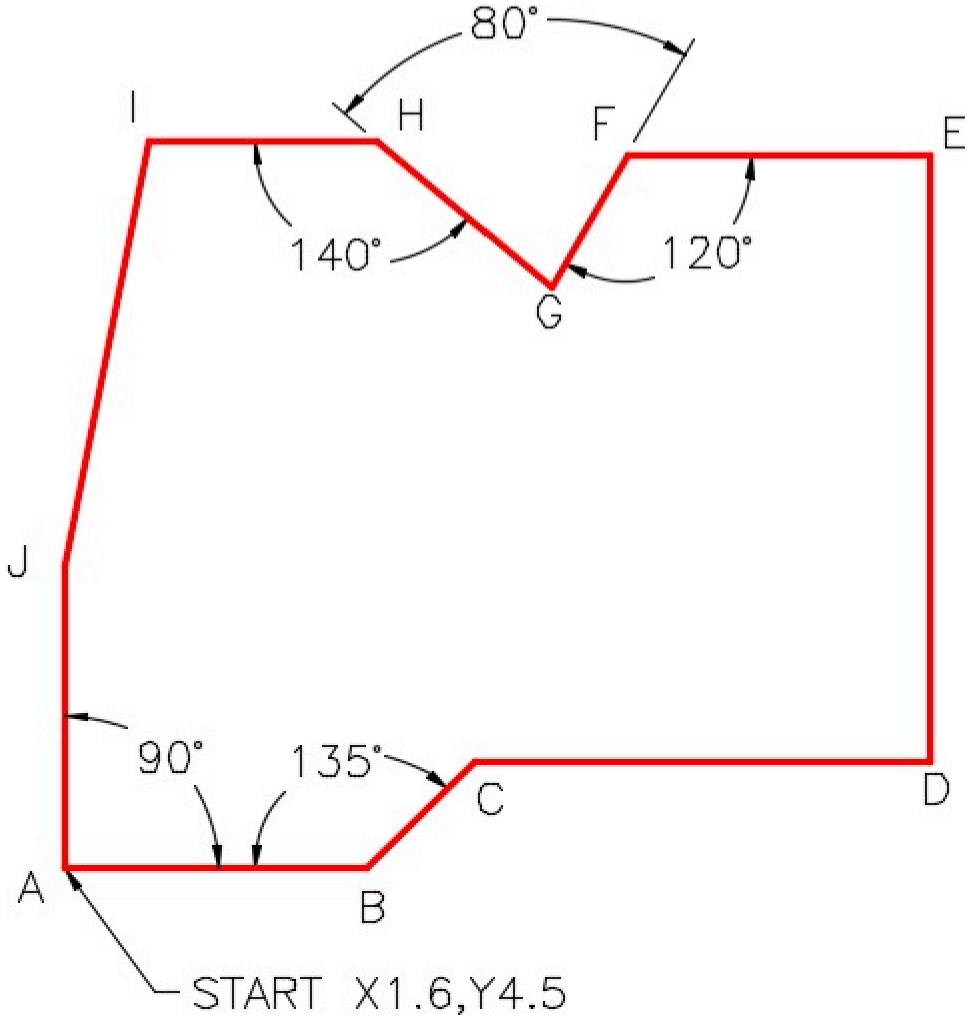
Start a new drawing using the template shown above.

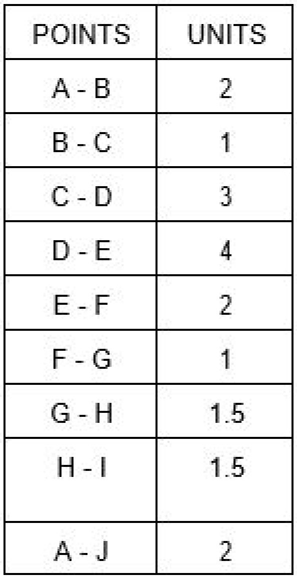
Step 2

Setup the layers using the Layering Scheme.

Step 3

Draw the object shown in the figures using the layering scheme. (Figure Step 3A and 3B)


80°
I
H
F
E
140°
120°
G
J
90°
135°
C
D
A
B
START X1.6,Y4.5



POINTS	UNITS
A - B	2
B - C	1
C - D	3
D - E	4
E - F	2
F - G	1
G - H	1.5
H - I	1.5
A - J	2

Figure Step 3A

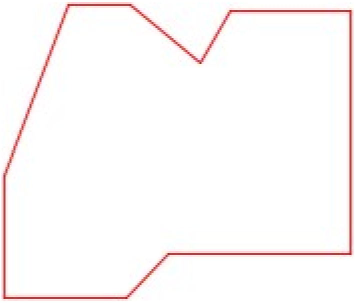


Fig Step 3B
Completed Drawing

Step 4

Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Inches.

Step 5

Check your drawing with the key. The key name is the same as the drawing name.

Step 6

If you have any errors, turn layer: Key off and correct your drawing. If you cannot correct it, start it over from the beginning.

Step 7

Turn layer: Key on to check your drawing. If it is still inaccurate, go back to Step 6.

Step 8

Turn layer: Key off.

Step 9

Save and close the drawing.

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and can not complete it on your own, use the following hints to help you.

Hint1

See the figure for the polar coordinates. (Figure Hint 1)

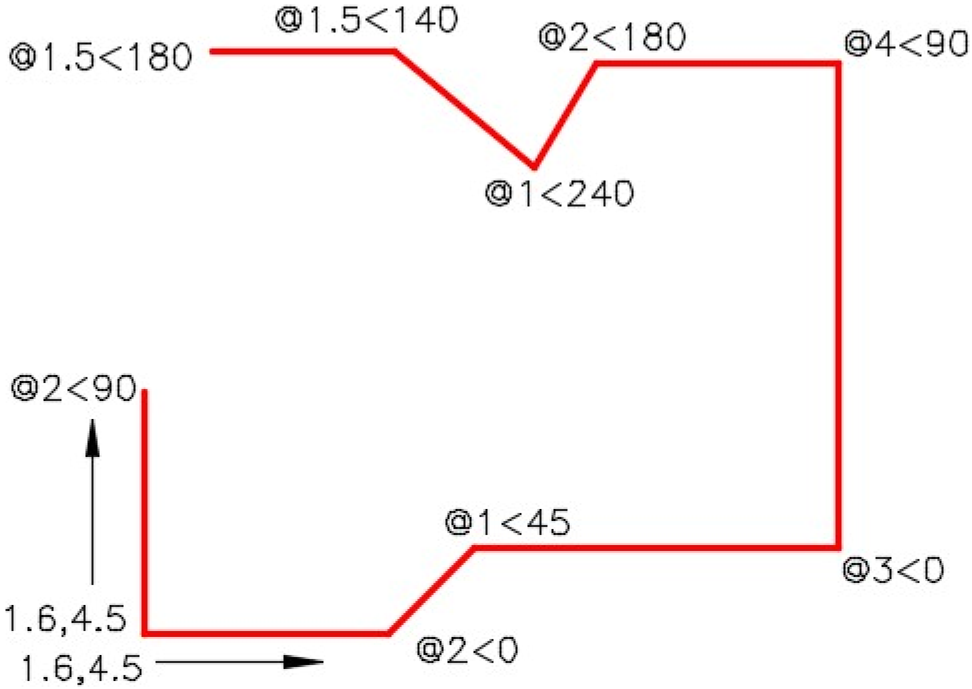

@1.5<140
@1.5<180
@2<180
@4<90
@1<240
@2<90
@1<45
@3<0
1.6,4.5
1.6,4.5
@2<0

Figure Hint 1

Hint 2

Draw the line from J to I by snapping to the endpoints. (Figure Hint 2)

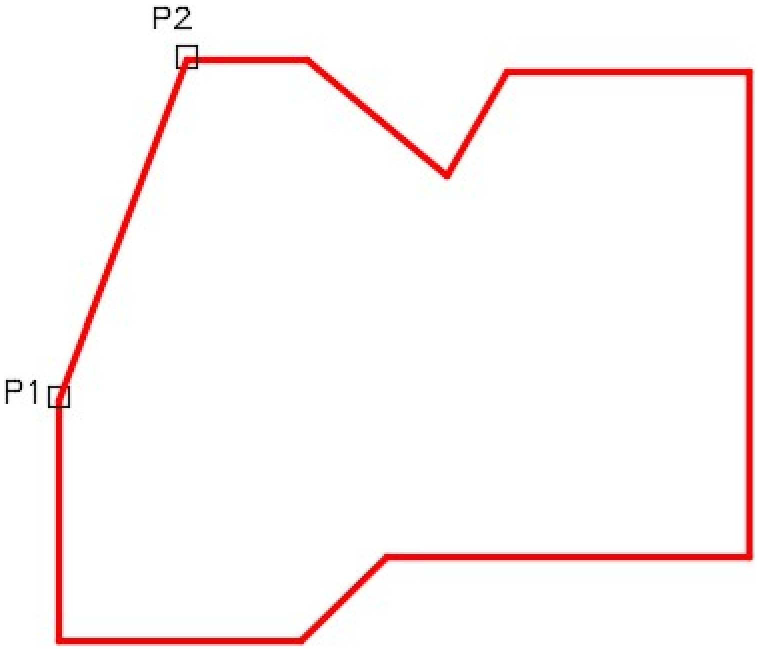


Figure Hint 2

Hint 3

The solutions to the angles at G and F are shown in the figures. (Figure Hint 3A and 3B)

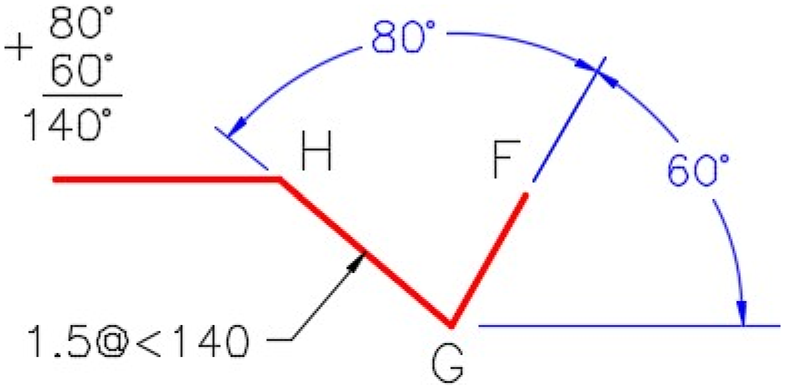

80°
+ 60°
140°
80°
H
F
60°
1.5@<140
G

Figure Hint 3A

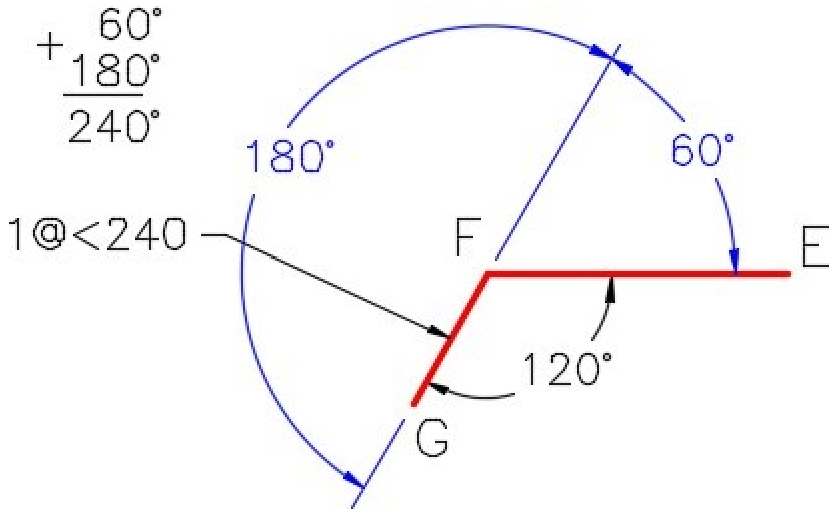


Figure Hint 3B

Lab Exercise 10-2

Time allowed: 40 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 10-2	2D Metric	Millimeters

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Construction	Construction objects	253
Object	Lines	Red

Step 1

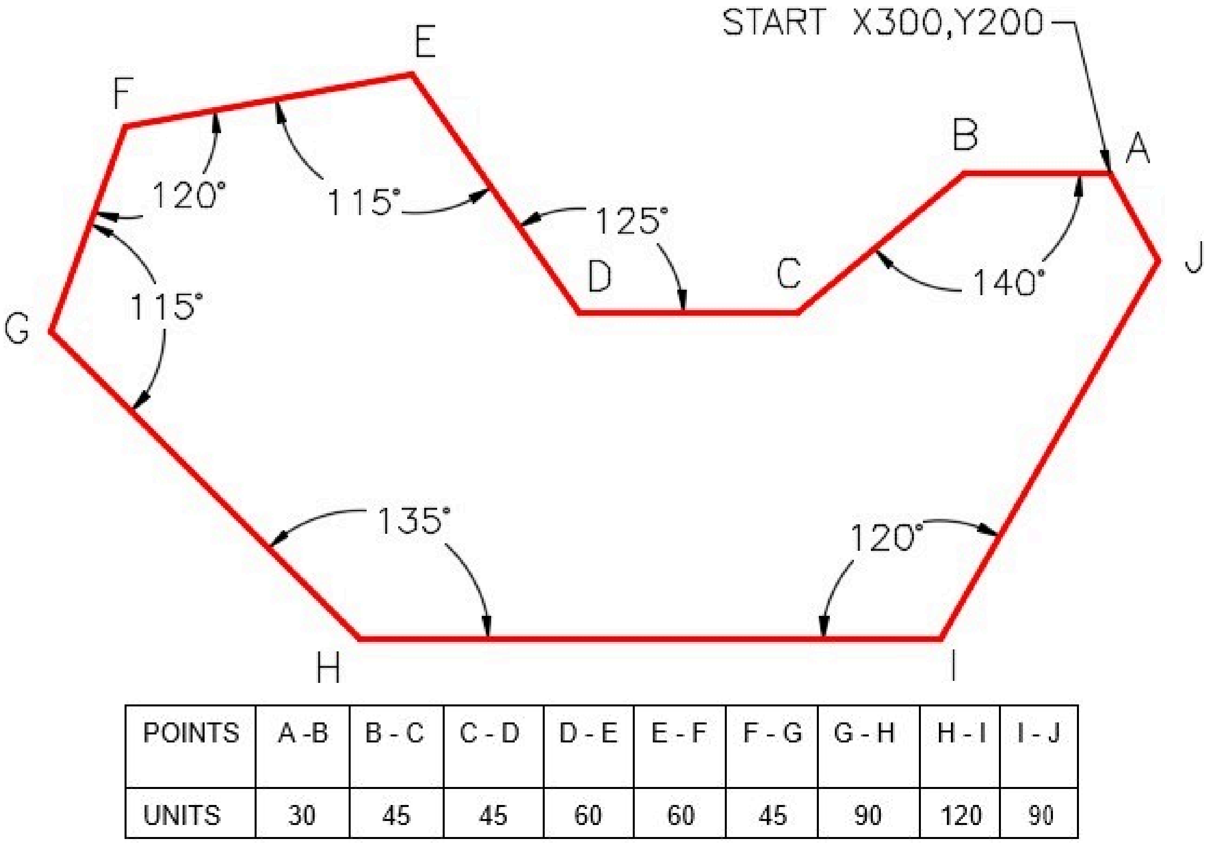
Start a new drawing using the template shown above.

Step 2

Setup the layers using the Layering Scheme.

Step 3

Draw the object shown in the figures using the layering scheme. (Figure Step 3A and 3B)



POINTS	A -B	B - C	C - D	D - E	E - F	F - G	G - H	H - I	I - J
UNITS	30	45	45	60	60	45	90	120	90

Figure Step 3A

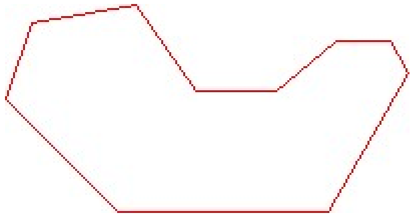


Figure Step 3B
Completed Drawing

Step 4

Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Millimeters.

Step 5

Check your drawing with the key. The key name is the same as the drawing name.

Step 6

If you have any errors, turn layer: Key off and correct your drawing. If you cannot correct it, start it over from the beginning.

Step 9

Save and close the drawing.

AUTHOR’S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hint. If you get stuck and cannot complete it on your own, use the following hint to help you.

Hint 1

See the figure for the polar coordinates. (Figure Hint 1)

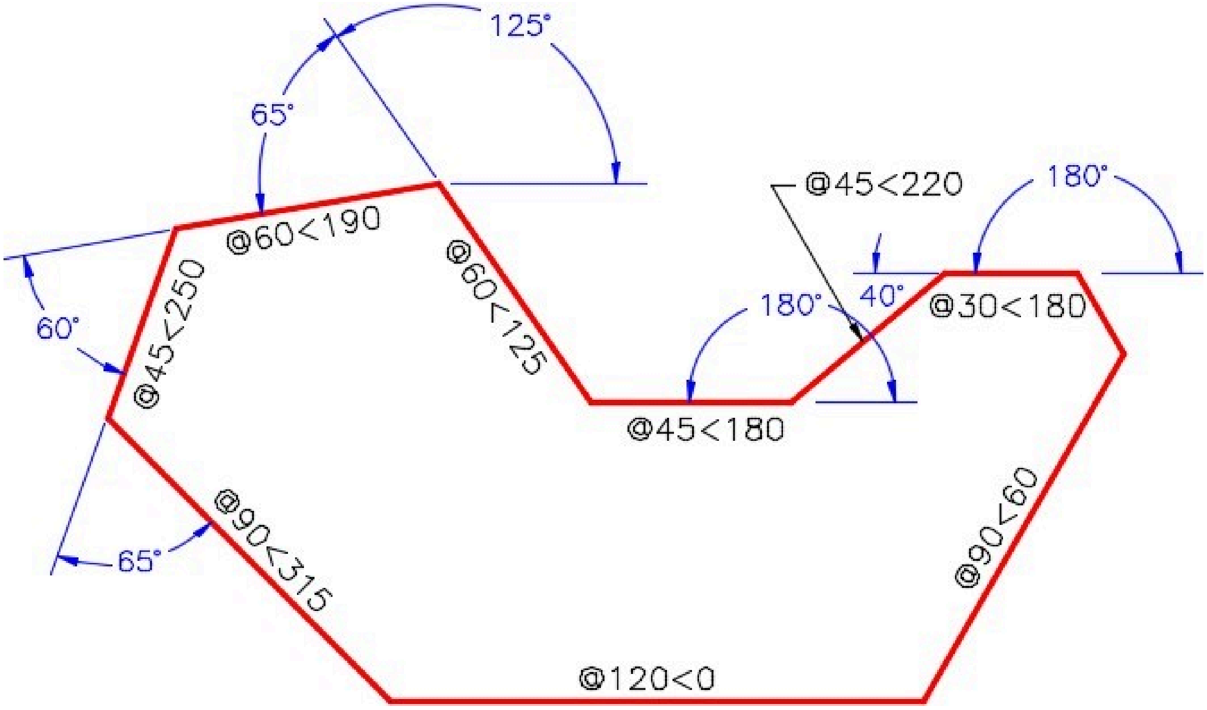

125°
65°
@60<190
@60<125
@45<220
180°
40°
@30<180
180°
60°
@45<250
@45<180
@90<315
65°
@90<60
@120<0

Figure Hint 1

Lab Exercise 10-3

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 10-3	2D Metric	Millimeters

Layering Scheme

Layer Name	Objects on Layer	Color
Angle Bracket	All drawing objects	Blue

Step 1

Draw the front view of the angle bracket shown in the dimensioned drawing using the layering scheme. (Figure Step 1A, 1B, and 1C)

Step 2

All objects should be on layer Angle Bracket and display blue.

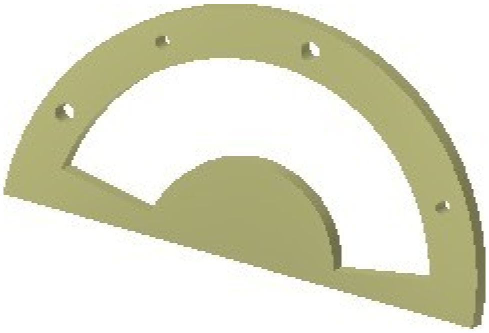


Figure Step 1A
Angle Bracket

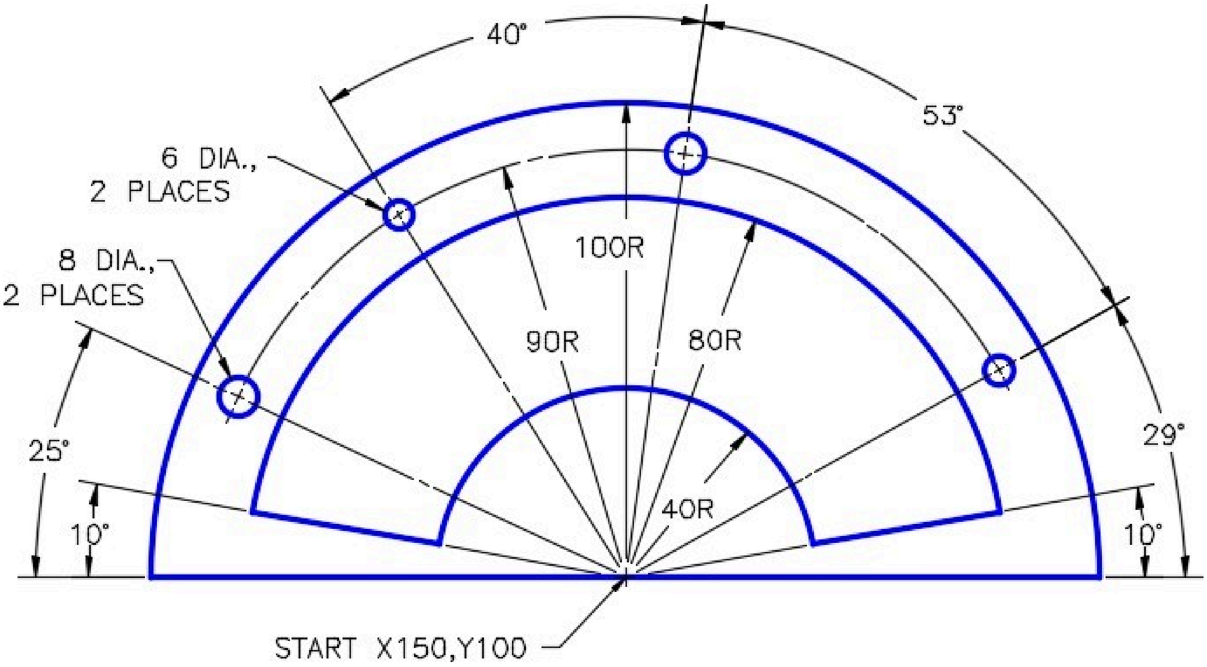

40°
53°
6 DIA.,
2 PLACES
8 DIA.,
2 PLACES
100R
90R
80R
25°
29°
40R
10°
10°
START X150,Y100

Figure Step 1B

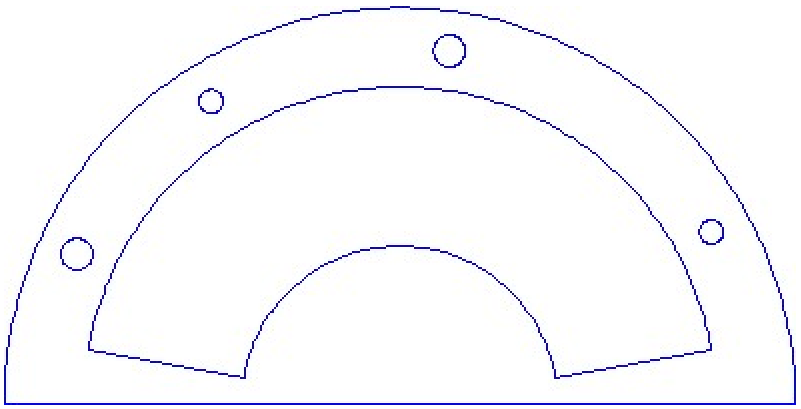


Figure Step 1C
Completed Drawing

Step 3

Set the insertion units and check your drawing with the key. Turn layer: Key off and freeze layer: Construction.

Step 4

Save and close the drawing.

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints.

Hint 1

This figure shows my construction drawing objects. Your construction drawing objects do not have to match as there are may ways to construct an object. I use circles rather an arcs, whenever possible, as they are easier to construct. (Figure Hint 1)

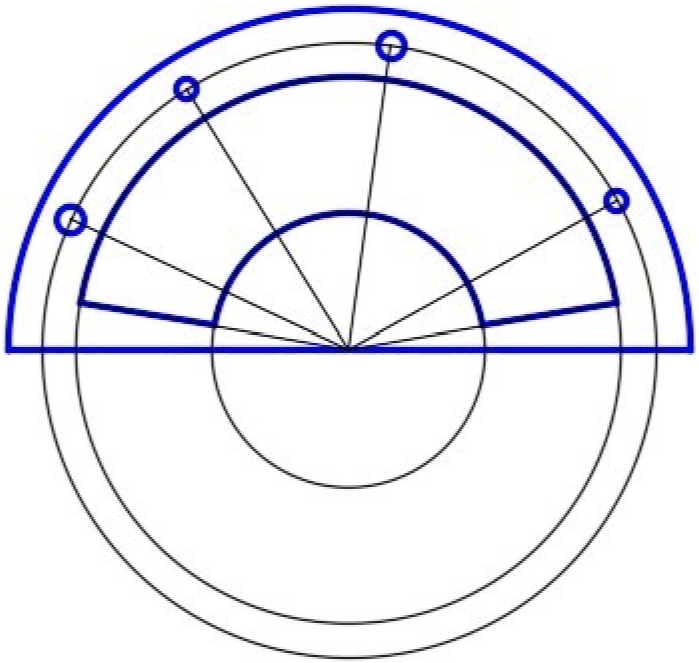


Figure Hint 1

Angle A = 40 degrees

Angle B = 125 degrees

Angle C = 150 degrees

Angle D = 30 degrees

Angle E = 215 degrees

Angle F = 35 degrees

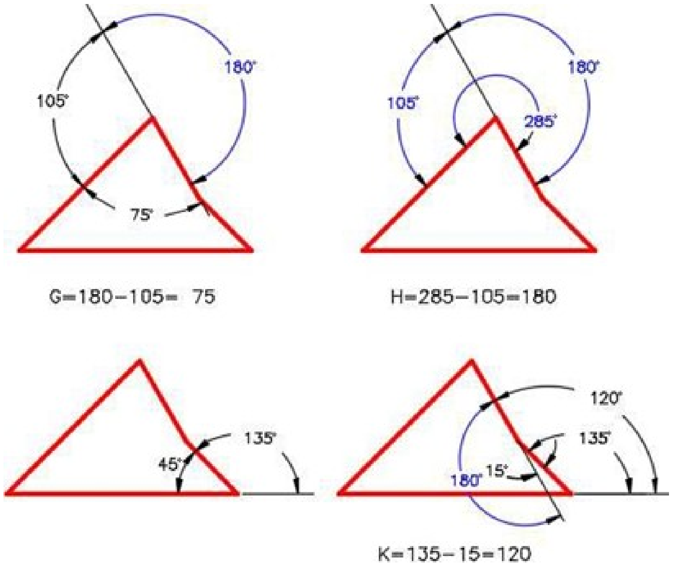
Angle G = 75 degrees

Angle H = 105 degrees

Angle J = 135 degrees

Angle K = 120 degrees

Angle L = 211 degrees


105°
180°
75°
G=180−105= 75
105°
180°
285°
H=285−105=180
135°
45°
120°
135°
180°
15°
K=135−15=120

Angle M = 324 degrees

Object A Solution:

1. Find the polar coordinate A TO B:	@2<0
2. Calculate the first angle:	180 – 150 = 30 degrees
3. Find the polar coordinate B TO C:	@2<30
4. Copy the 30 degree angle to the next corner.	
5. Calculate the second angle:	180 – 110 = 70 degrees
6. Add the two angles:	30 + 70 = 100 degrees
7. Find the polar coordinate C TO D:	@3<100
8. Calculate angle DD:	180 – 100 = 80 degrees
9. Find angle EE:	100 + 80 = 180 degrees
10. Find the polar coordinate D TO E:	@2<180

Object B Solution:

1. Find polar coordinate A TO B	@3<0
2. Calculate angle AA:	180 – 135 = 45 degrees
3. Find polar coordinate B TO C	@3<45
4. Copy angle AA to the next corner	
5. Calculate angle BB	180 – 105 = 75 degrees
6. Find angle CC	45 + 75 = 120 degrees
7. Find polar coordinate C TO D	@2<120
8. Copy angle CC to the next corner	
9. Calculate angle DD	180 – 120 = 60 degrees
10. Find angle EE120 + 60 = 180 degrees	
11. Find polar coordinate D TO E	@3<180

Module 11: Competency Test 2 Open Book

Learning Outcomes

When you have completed this module, you will be able to:

1. Within a one-hour time limit, complete a written exam and a lab exercise without the aid of a key.

The AutoCAD 2D book was written with competency based modules. What that means is that you have not completed each module until you have mastered it. The Competency Test module contains multiple choice questions and a comprehensive lab exercise to test your mastery of the set of modules that you completed. There are no answers or keys supplied in a Competency Test module since it is meant to be checked by your instructor. If there are any parts of this module that you have trouble completing, you should go back and reread the module or modules containing the information that you are having trouble with. If necessary, redo as many lab exercises required until you fully understand the material.

If you are Completing this book:

- Without the Aid of an Instructor, complete the written test and the lab exercise.
- In a Classroom with an Instructor, the instructor will give instructions on what to do after this module has been completed.

Multiple Choice Questions

Select the BEST answer.

1. What object snap mode is used to snap to the center point of a line?
    A. Halfpoint
    B. Center
    C. Quadrant
    D. Midpoint
    E. Endpoint
2. What scale should almost all objects be drawn in an AutoCAD drawing?
    A. 1:50
    B. 1:10

   C. 1:2
   D. Full Scale
   E. You don't draw in scale in AutoCAD

3. What is the name of the visual aid that helps the user locate object snap modes more efficiently?
   A. Snapmode
   B. AutoSnap
   C. SnapAuto
   D. AutoCADSnap
   E. AutoMode
4. What coordinates are the best to set the lower left corner of the limits of the drawing?
   A. 0,0
   B. 2,2
   C. 4,4
   D. 6,6
   E. 8,8
5. What is the smallest rectangle that will fit around all of the existing objects in the drawing called?
   A. The drawing border
   B. The drawing grid
   C. The drawing mode
   D. The drawing extents
   E. The drawing size
6. What command sets the number of vectors that AutoCAD uses when it regenerates circles and arcs?
   A. VIEWRES
   B. REGEN
   C. CIRCLE
   D. ARC
   E. VIEW
7. What function key toggles Autosnap on and off?
   A. F1
   B. F2
   C. F3

   D. F4
   E. F5

8. What is the name of the four object snap mode locations that are located around the circumference of a circle?
   A. Center
   B. Midpoint
   C. Endpoint
   D. Points
   E. Quadrants
9. Which one of the following best describes the polar coordinate @10<125?
   A. From the last point, go 12.5 units at an angle of negative 10 degrees
   B. From the last point, go 10 units at an angle of negative 125 degrees
   C. From the last point, go 125 units at an angle of positive 10 degrees
   D. From the last point, go 10 units at an angle of positive 125 degrees
   E. From the last point, go 125 units at an angle of negative 10 degrees
10. What polar coordinate would draw the same angle as @5<225?
   A. @5<-125
   B. @5<-225
   C. @5<-135
   D. @5<-145
   E. @5<-165

Lab Exercise 11-1 OPEN BOOK

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 11-1	2D Metric	Millimeters

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Construction	Construction objects	253
Object	Line, circle and arcs	Red

Step 1

Setup the layers using the Layering Scheme shown above.

Step 2

Draw the object shown in the figure using the layering scheme. (Figure Step 2A and 2B)

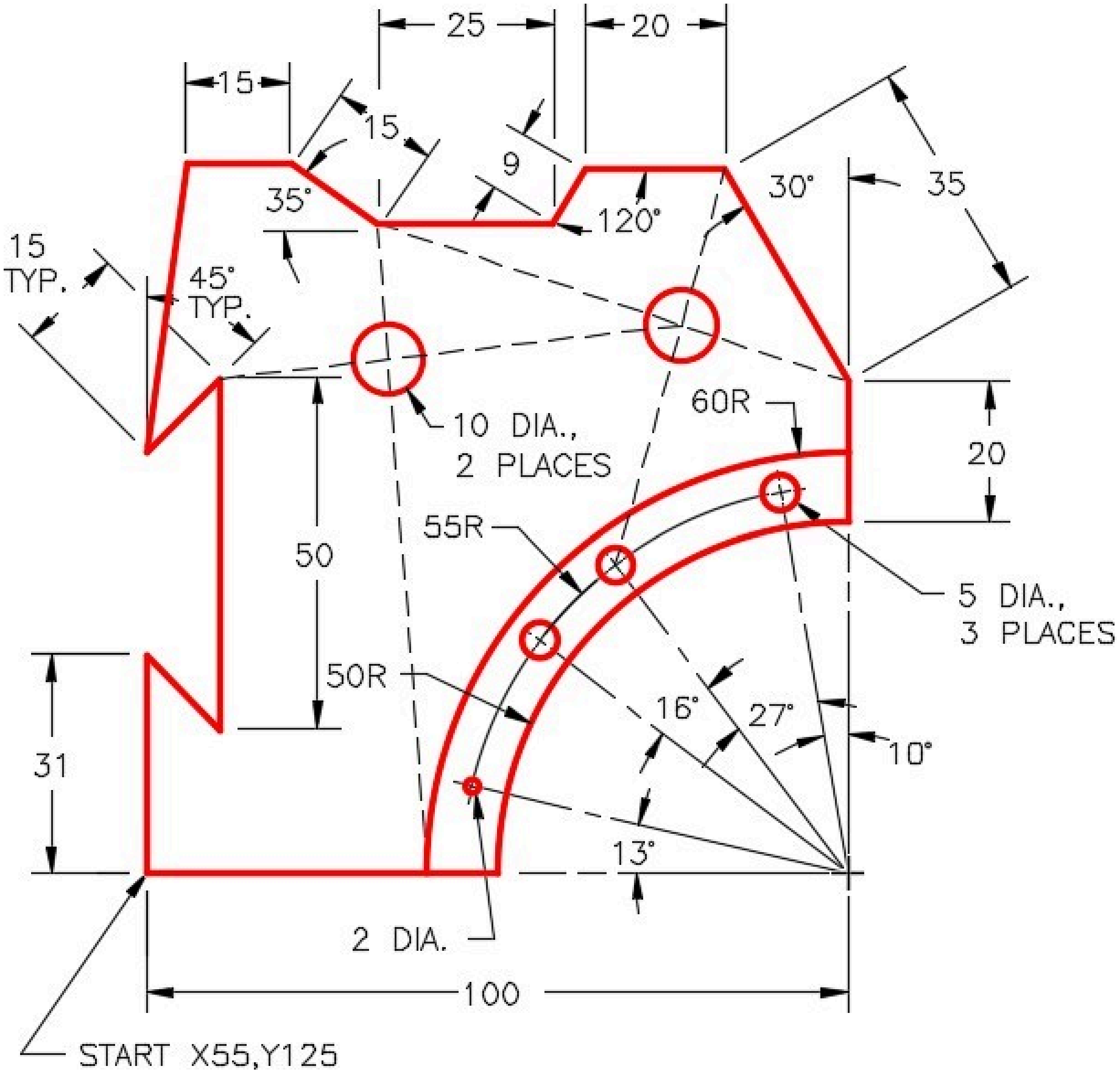

25
20
15
15
9
35°
120°
30°
35
15
TYP.
45°
TYP.
10 DIA.,
2 PLACES
60R
20
50
55R
5 DIA.,
3 PLACES
50R
16°
27°
10°
31
13°
2 DIA.
100
START X55,Y125

Figure Step 2A

AUTHOR'S COMMENTS: Do not delete the construction lines.

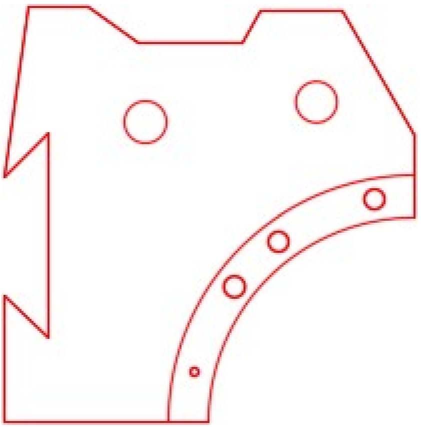


Figure Step 2B

Step 3

Draw all construction objects on layer: Construction.

AUTHOR'S COMMENTS: Draw the construction lines (shown as dashed lines in the drawing for clarity) by snapping to the endpoints of the lines, endpoint of the 60 radius arc and the center of the 5 diameter circle. Locate the 10 diameter circles with their center located at the intersections of the construction lines.

Step 4

When complete, freeze layer: Construction.

Step 5

Save and close the drawing.

AUTHOR'S COMMENTS: There is no key for this lab.

Part 3

Module 12: Fillets

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe tangency and explain how to locate the point of tengency.
2. Describe a fillet.
3. Apply the FILLET command to construct fillets between two drawing objects.
4. Describe the object snap mode Tan.
5. Apply the LINE command using the object snap mode Tan to draw lines tangent to arcs and circles.

Geometry Lesson: Tangency

A *point of tangency* is the theoretical point where a line joins an arc, two arcs join each other, or two circles join each other making a smooth transition. A line is tangent to a circle or an arc when it passes touching only one point on the circle or arc. The tangent point is located where a line drawn perpendicular from the center of the circle or arc touches the line. Study Figure 12-1.

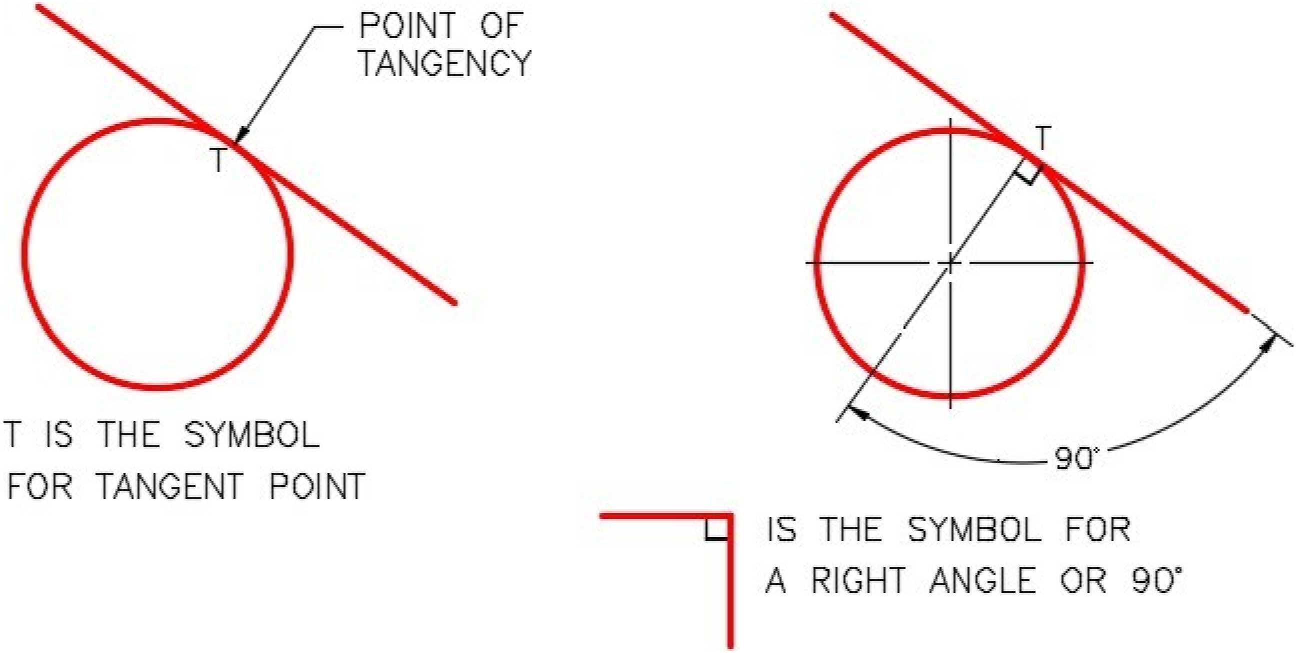

POINT OF TANGENCY
T
T IS THE SYMBOL FOR TANGENT POINT
T
90°
IS THE SYMBOL FOR A RIGHT ANGLE OR 90°

Figure 12-1
Tangency – Part 1

A *fillet* is a tangent arc. A tangent arc must be tangent at both ends where it connects to existing lines or arcs. Figure 12-2 shows five examples of fillets.

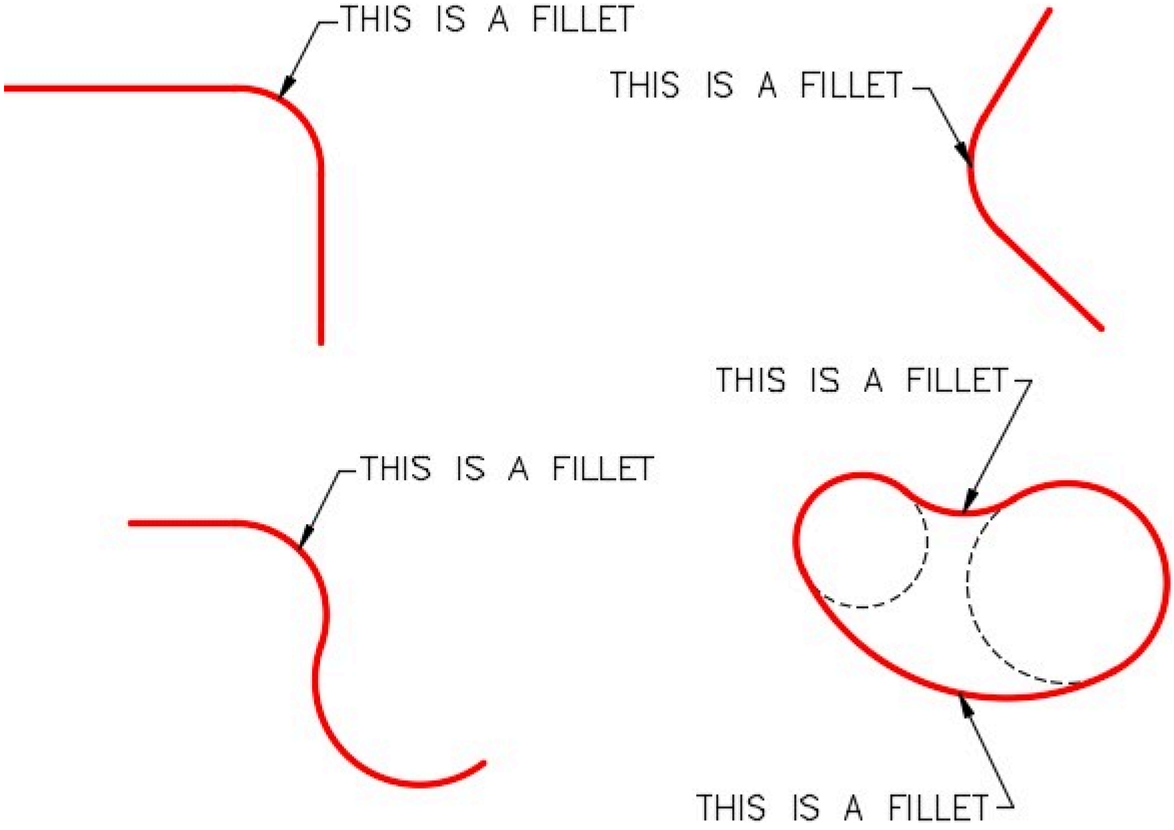
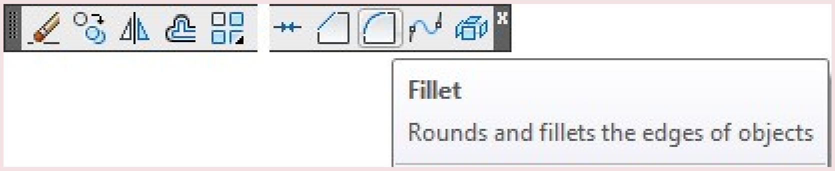

THIS IS A FILLET
THIS IS A FILLET
THIS IS A FILLET
THIS IS A FILLET
THIS IS A FILLET

Figure 12-2
Tangency – Part 2

AutoCAD Command: FILLET

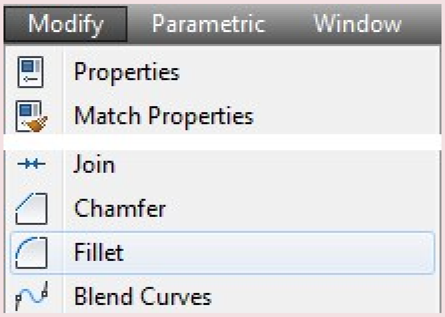
The FILLET command is used to draw a tangent arc. It inserts an arc tangent to the two existing drawing objects selected in the command.

Shortcut: **F**

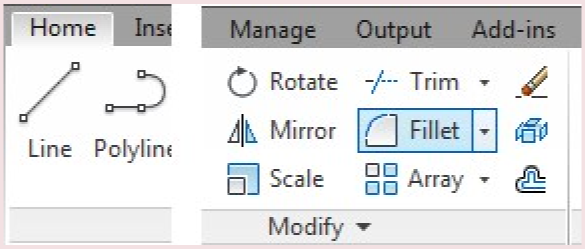


Fillet
Rounds and fillets the edges of objects

Modify Toolbar


Modify
Parametric
Window
Properties
Match Properties
Join
Chamfer
Fillet
Blend Curves

Modify Pull-down


Home
Inse
Manage
Output
Add-ins
Line
Polyline
Rotate
Trim
Mirror
Fillet
Scale
Array
Modify

Home Ribbon

WORKALONG: Using the FILLET Command

Step 1

Start a new drawing using the template: 2D English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 12-1.

Step 3

Create layers: Object and Construction. Set layer: Object as the current layer.

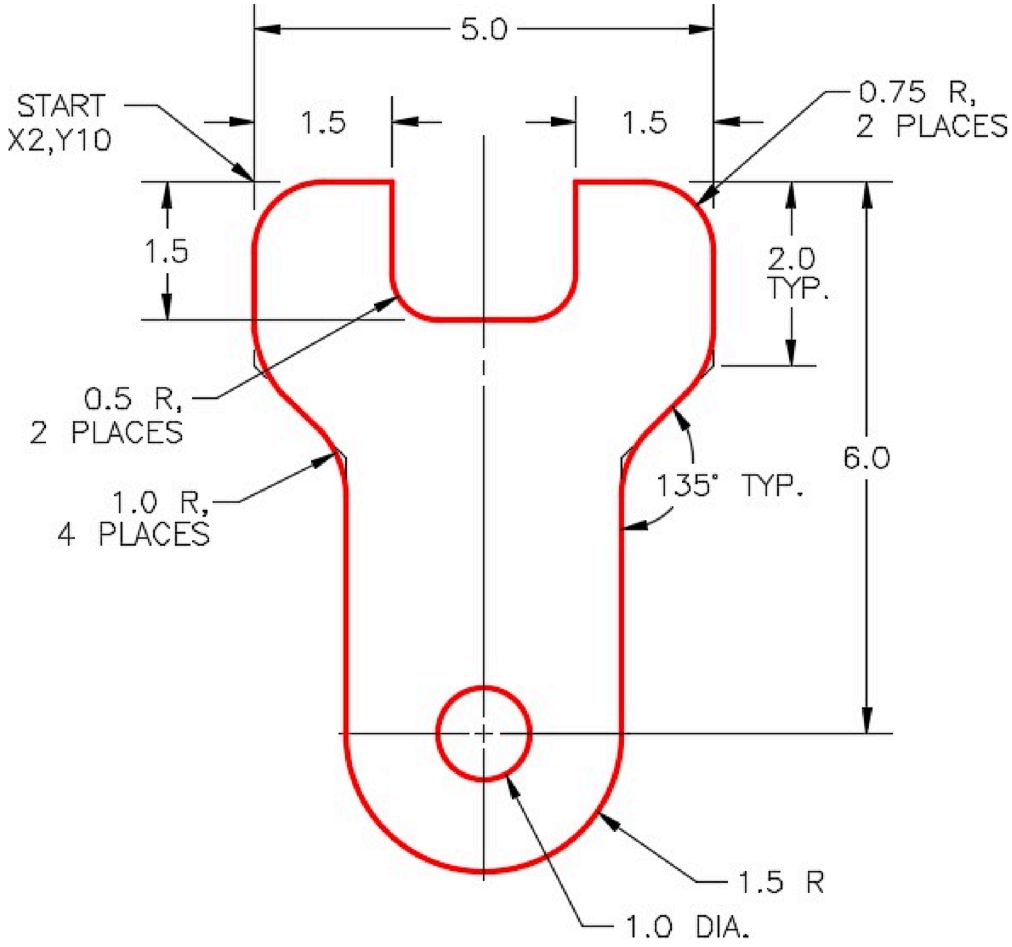

5.0
START
X2,Y10
1.5
1.5
0.75 R,
2 PLACES
1.5
2.0
TYP.
0.5 R,
2 PLACES
1.0 R,
4 PLACES
135° TYP.
6.0
1.5 R
1.0 DIA.

Figure Step 4A

Drawing Outside the Box

AUTHOR'S COMMENTS: Up to this point in the book, you have been drawing lines knowing the exact location and length of the lines. In this module, you will learn how to draw lines without knowing their exact length. You must know the exact start point of the line and the line's direction. You guess at its length either drawing it longer or shorter than its actual length. It is best to draw it longer. You then trim or extend the lines to their exact length using the FILLET command. This workalong will teach you how to do this.

Step 4

Using the LINE command and referring to the figures, draw the 6 lines to match the figure. (Figure Step 4A and 4B)

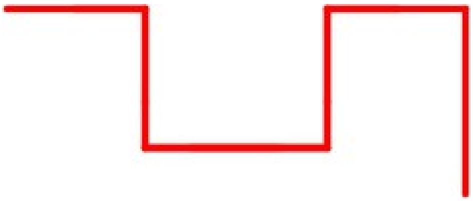


Figure Step 4B

Step 5

Using the LINE command, draw the vertical line. (Figure Step 5A and 5B)

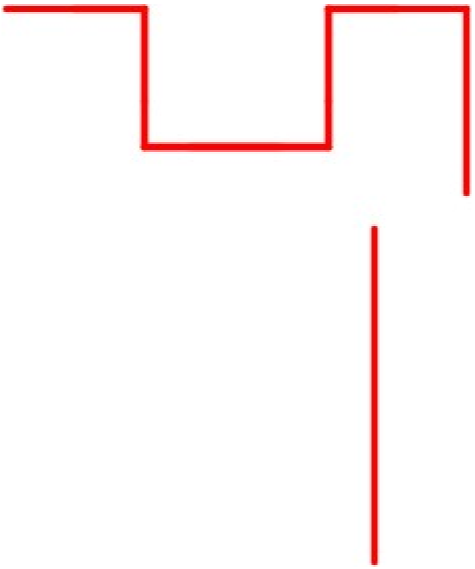


Figure Step 5A

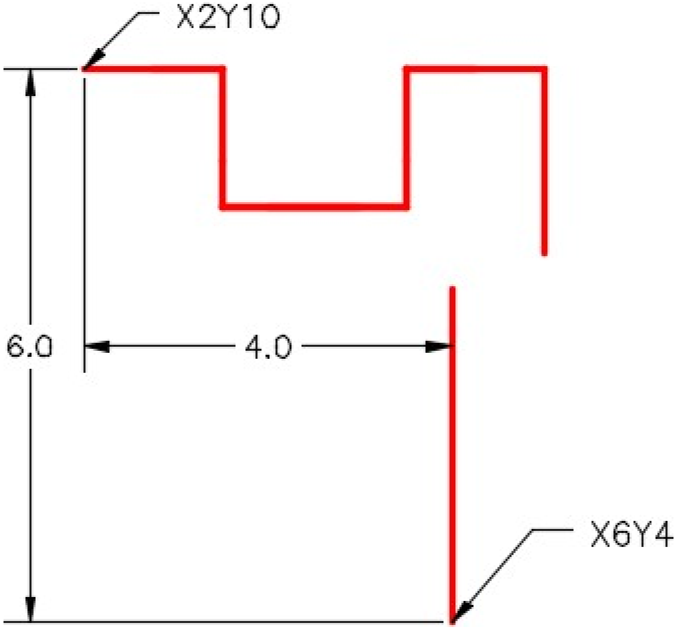


Figure Step 5B

AUTHOR'S COMMENTS: To draw this line, you must know its start point. Figure Step 5B shows how I solved the start point of the line. Since you don`t know the exact length of the line, guess at the length and try to draw it longer. I drew it 3.5 inches. Ensure you understand how I solved for the start point.

Step 6

Using the LINE command, draw the inclined line. First you have to solve the angle. Guess at the length and try to make it longer. After you know the angle and the length, snap to end point of the existing to start the line. (Figure Step 6A and 6B)

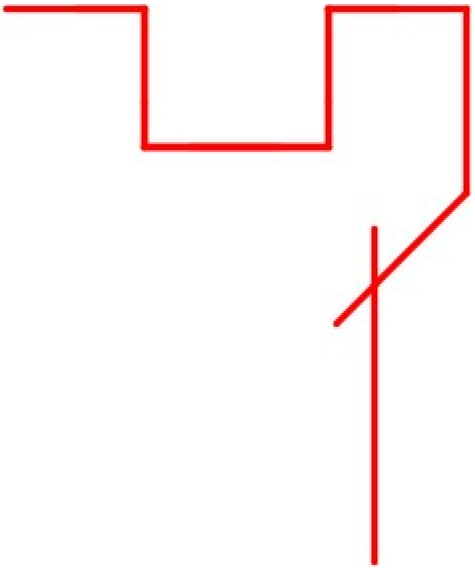


Figure Step 6A

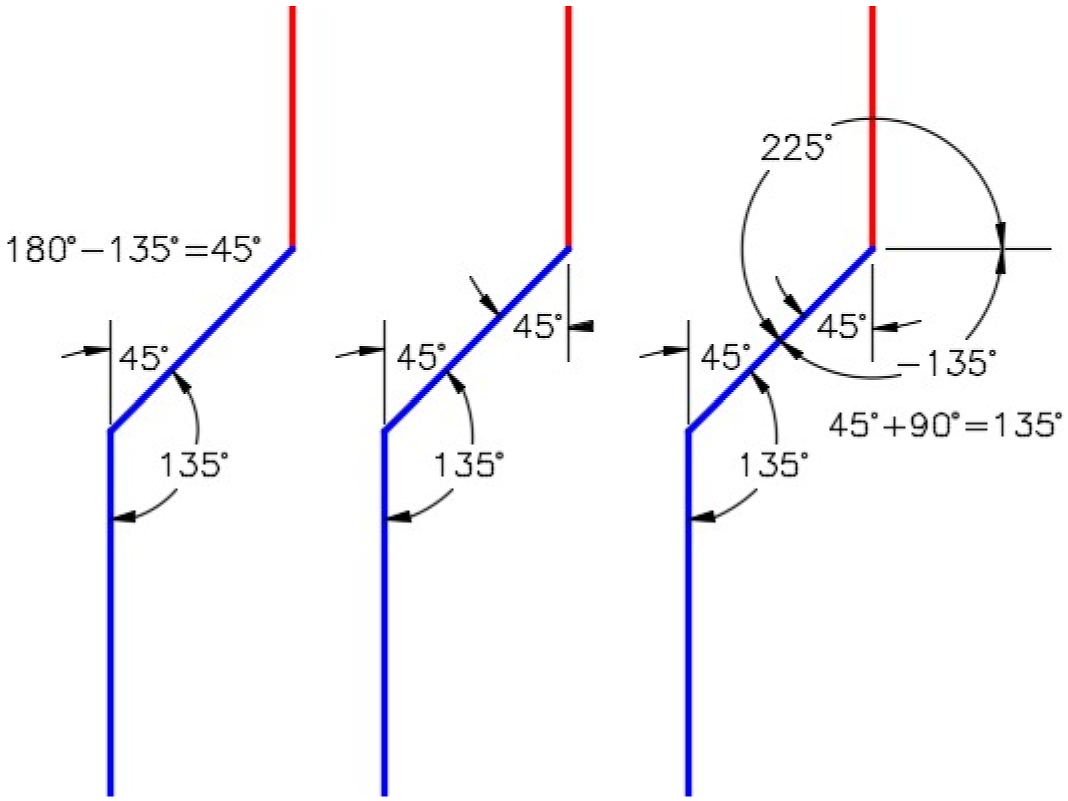

180°–135°=45°
45°
135°
45°
45°
135°
225°
45°
45°
–135°
45°+90°=135°
135°

Figure Step 6B

AUTHOR'S COMMENTS: Figure Step 6B shows the three steps I used to solve the angle of the line. Draw it longer than required since you don`t know its exact length. I drew it 2.0 inches long.

Step 7

Using the principles you just learned, use the LINE command to draw the two lines on the left side of the object. (Figure Step 7)

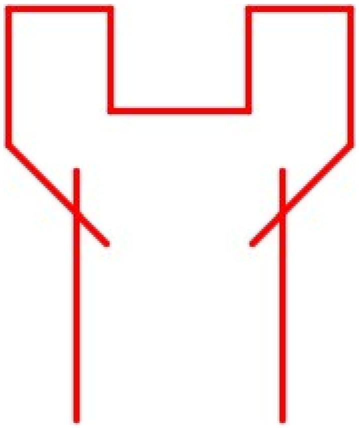


Figure Step 7

Step 8

Enter the FILLET command, as shown below, and select the two vertical lines to insert the fillet (tangent arc). (Figure Step 8A, 8B, and 8C)

Command: **FILLET**

Current settings: Mode = NOTRIM, Radius = 0.5000

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **T**

Enter Trim mode option [Trim/No trim] <No trim>: **T**

Select first object or [Undo/Polyline/Radius/Trim/Multiple]:

Command: *(This commands simply sets the FILLET command to trim objects when filleted. You only have to do this the first time you use the FILLET command or if you re-set it to NOTRIM.)*

Command: **FILLET**

Current settings: Mode = TRIM, Radius = 0.0000

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **(Pick left vertical line)**

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **(Pick right vertical line)**

Select second object or shift-select to apply corner:

Command:

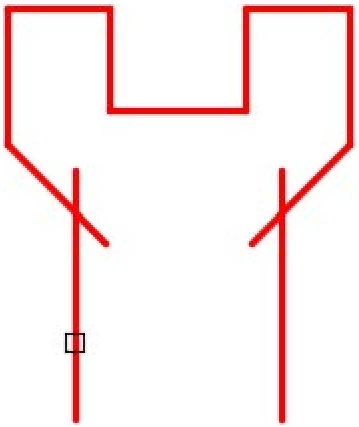


Figure Step 8A

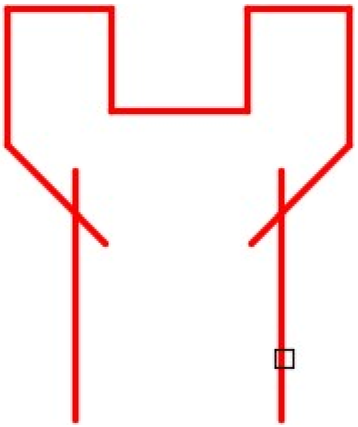


Figure Step 8B

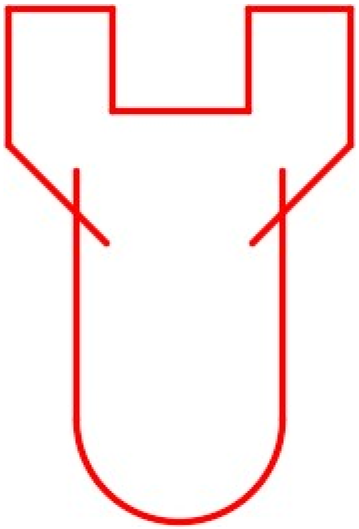


Figure Step 8C

AUTHOR’S COMMENTS: When the lines selected in the FILLET command are parallel, AutoCAD will automatically calculate the radius of the arc for you, regardless of the current radius setting. Note how the radius is set to zero in this example.

Step 9

Enter the FILLET command, as shown below, to insert a 0.5 radius fillet between the two lines. (Figure Step 9A. 9B, and 9C)

Command: **FILLET**

Current settings: Mode = TRIM, Radius = 0.0000

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **R**

Specify fillet radius <0.0000>: 0.5 *(Set the radius for the fillet.)*

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **(Pick right vertical line)**

Select second object or shift-select to apply corner: **(Pick the horizontal line)**

Command:

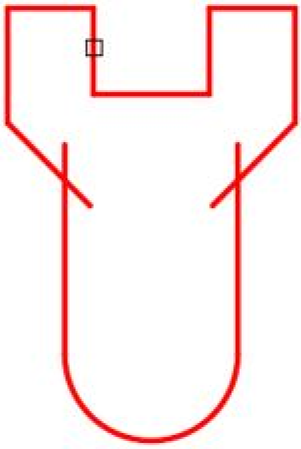


Figure Step 9A

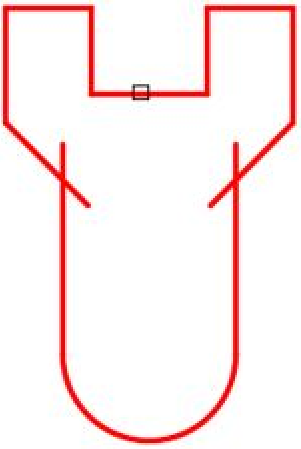


Figure Step 9B

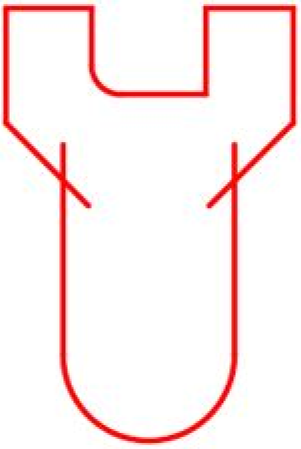


Figure Step 9C

AUTHOR'S COMMENTS: Since this fillet is inserted between lines that are not parallel, you must set the radius of the fillet. In this case, 0.5.

Step 10

Using what you learned in Step 9, insert the 0.5 radius fillet on the other side to match the figure. (Figure Step 10)

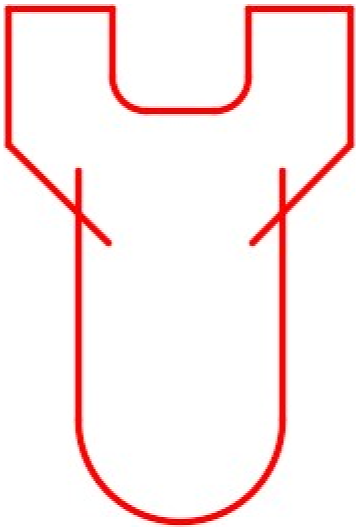


Figure Step 10

Step 11

Using what you learned in Step 9 and 10, insert the two 0.75 radius fillets on top left and right corners. (Figure Step 11)

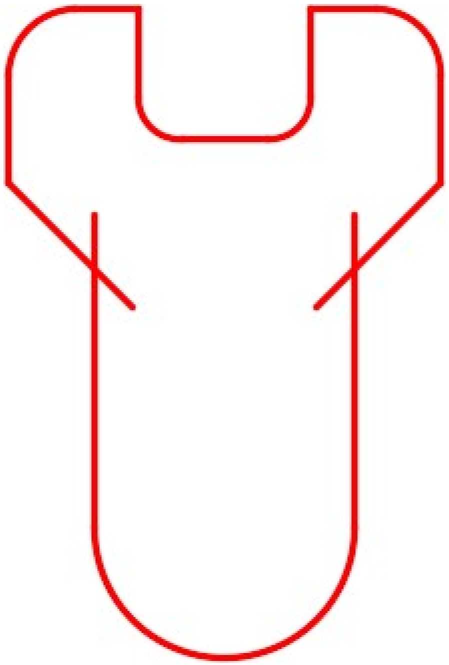


Figure Step 11

AUTHOR'S COMMENTS: Ensure that you set the radius in the FILLET command to 0.75 before you select the lines to fillet.

Step 12

Insert the four 1.0 radius fillets to match the figure. (Figure Step 12)

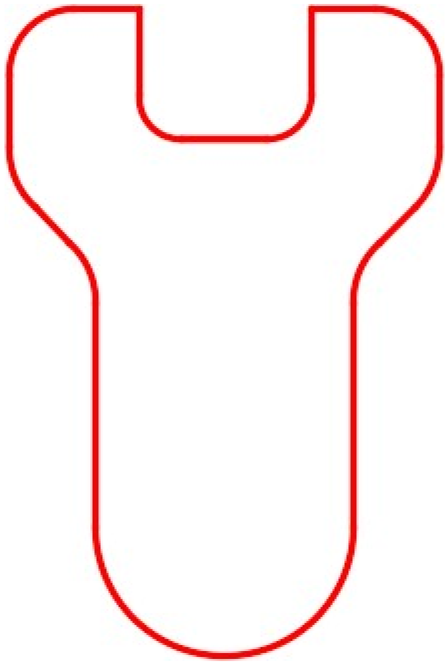


Figure Step 12

Step 13

Enter the CIRCLE command and draw 1.0 diameter circle by snapping to the center of the arc to locate the its center. The completed drawing should match the figure. (Figure Step 13A and 13B)

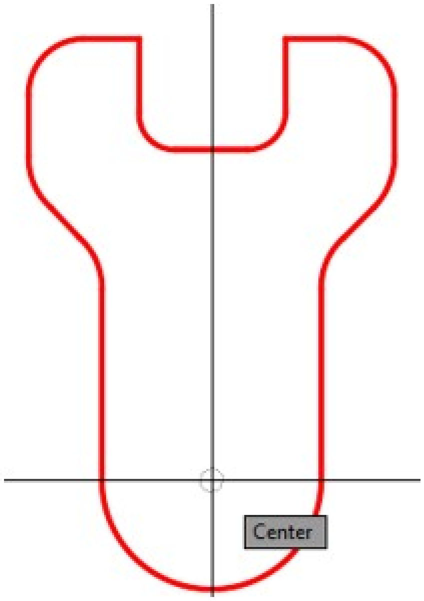


Figure Step 13A

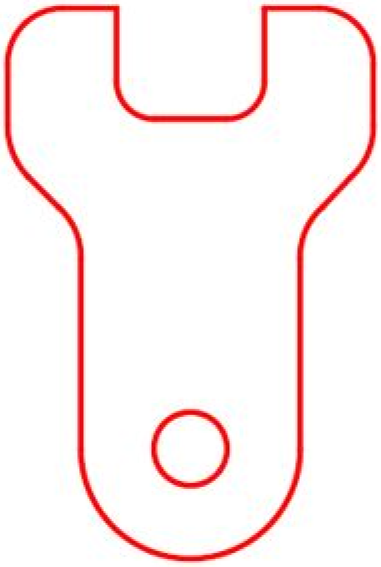
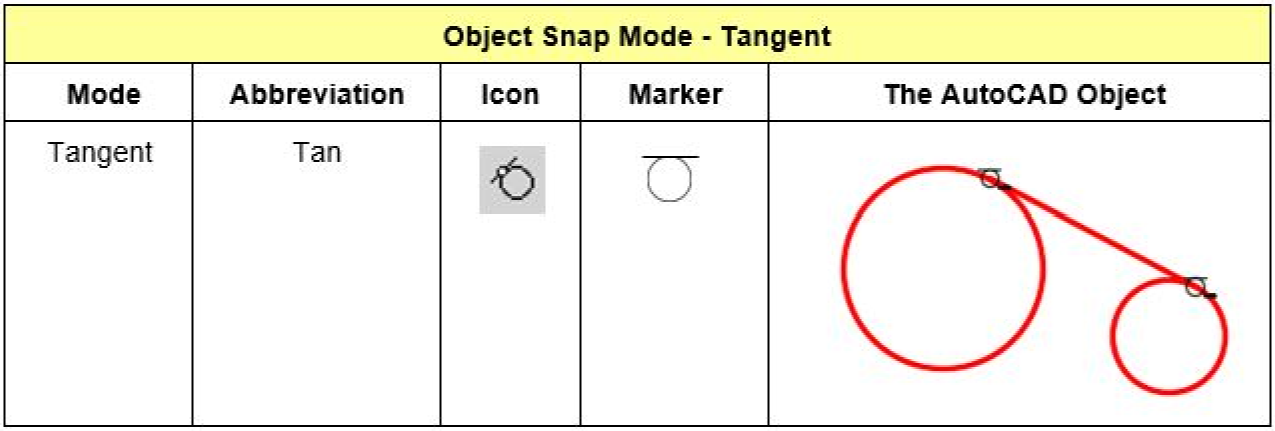


Figure Step 13B

AUTHOR'S COMMENTS: The circle and arc share the same center point. Ensure that AutoSnap is set to locate centers and Osnap is enabled. Use the CIRCLE command and snap to the center of the arc to locate the center of the circle.

Step 14

Save and close the drawing.



Object Snap Mode - Tangent				
Mode	**Abbreviation**	**Icon**	**Marker**	**The AutoCAD Object**
Tangent	Tan			

Figure 12-3
Object Snap Mode – Tangent

WORKALONG: Drawing Tangent Lines

Step 1

Start a new drawing using the template: 2D English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 12-2.

Step 3

Create the layers: Object and Construction. Set Layer: Construction as the current layer.

Step 4

Referring to the figures, draw the 2 radius circle with its center at 10,10. (Figure Step 4A and 4B)

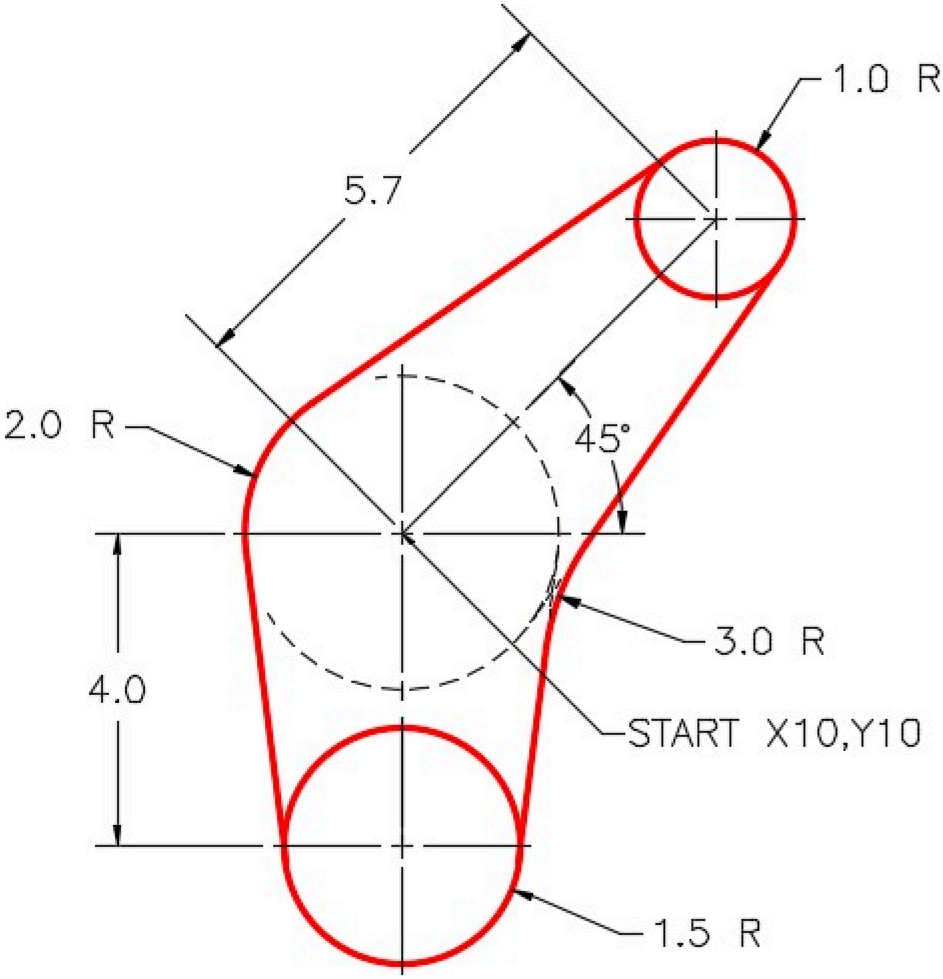

1.0 R
5.7
2.0 R
45°
3.0 R
4.0
START X10,Y10
1.5 R

Figure Step 4A

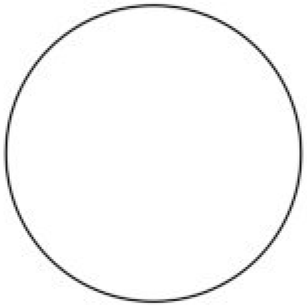


Figure Step 4B

Step 5

Using polar coordinates, enter the LINE command, two times, to draw two construction lines. Start each line starting at the center of the circle and ending at the center of the other two circles. (Figure Step 5)

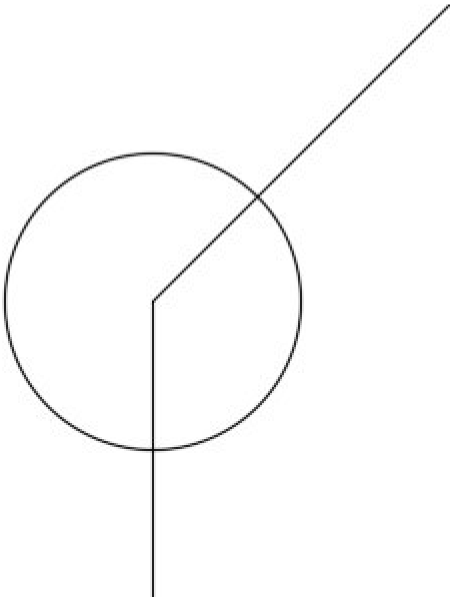


Figure Step 5

Step 6

Draw circles locating their centers at the endpoints of the lines you drew in Step 5. (Figure Step 6)

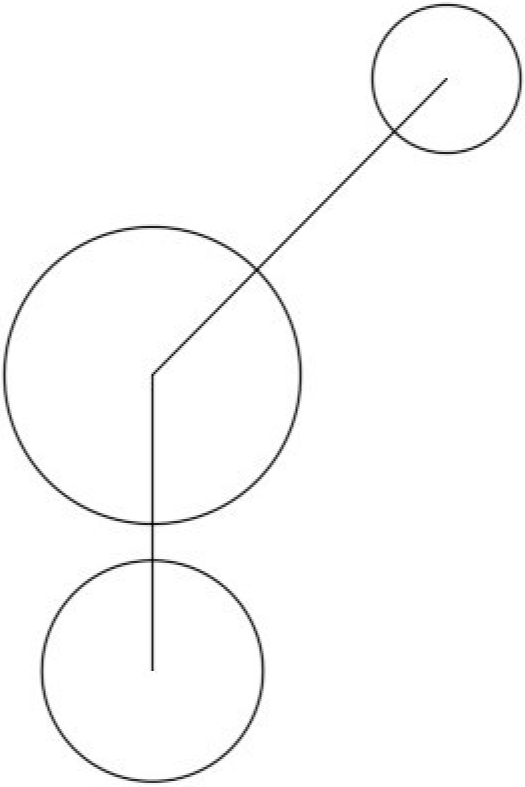


Figure Step 6

Step 7

Enter the LINE command, as shown below, to draw a line tangent to the two circles. Enter the TAN object snap mode manually by typing it in on the keyboard. (Figure Step 7)

Command: **LINE**

Specify first point: **TAN** to **P1**

Specify next point or [Undo]: **TAN** to **P2** Specify next point or [Undo]:

Command:

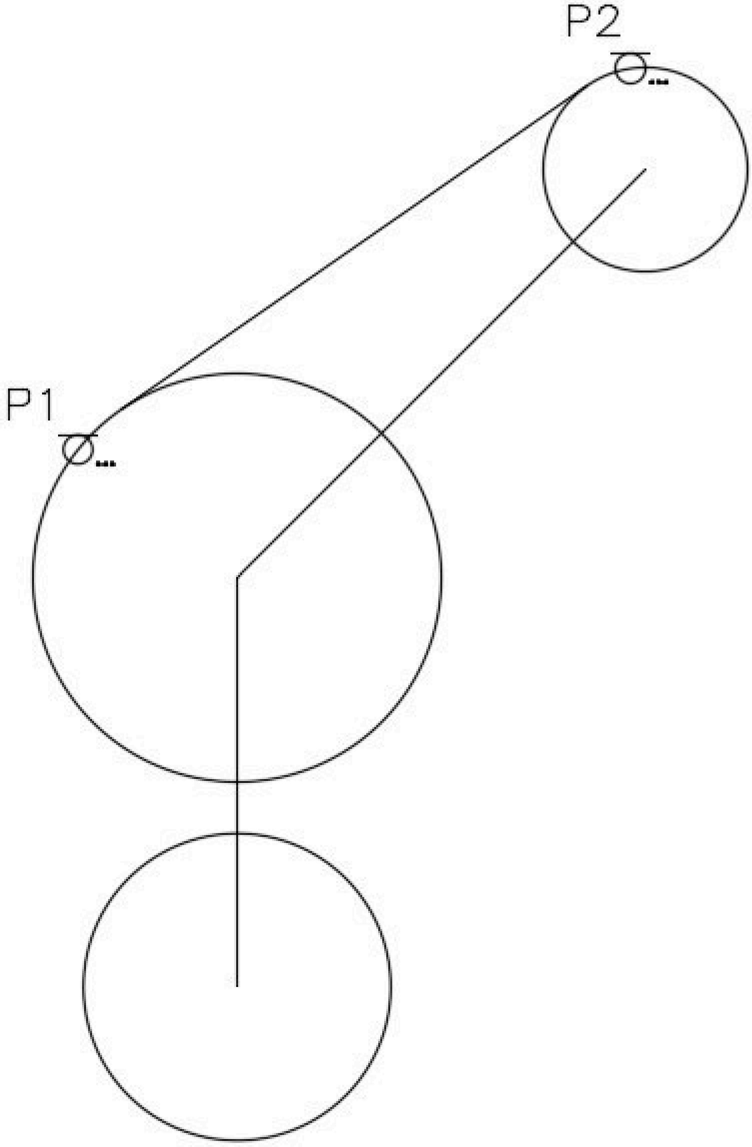


Figure Step 7

AUTHOR’S COMMENTS: Notice how you have to enter **TAN** for both ends of the line. It is a very common error to forget to enter it at the second end. Do not use AutoSnap, enter them manually by typing TAN on the keyboard and then press Enter. Note the AutoSnap marker when AutoCAD is finding the tangent location on the circle.

Step 8

Using what you learned in Step 7, enter the LINE command and draw a line tangent to the two circles as shown in the figure. (Figure Step 8)

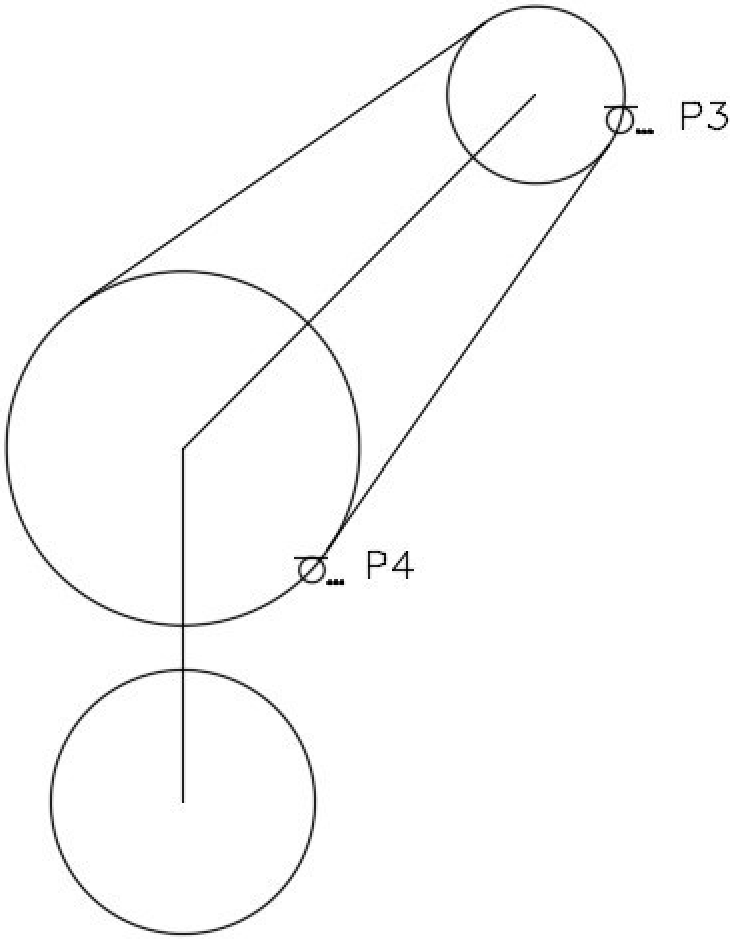


Figure Step 8

Command: **LINE**

Specify first point: **TAN** to **P3**

Specify next point or [Undo]: **TAN** to **P4**

Specify next point or [Undo]:

Command:

AUTHOR'S COMMENTS: Note the AutoSnap Marker icon that displays when you use the TAN object snap mode.

Step 9

Using what you just learned, insert the two tangent lines as shown in the figure. (Figure Step 9)

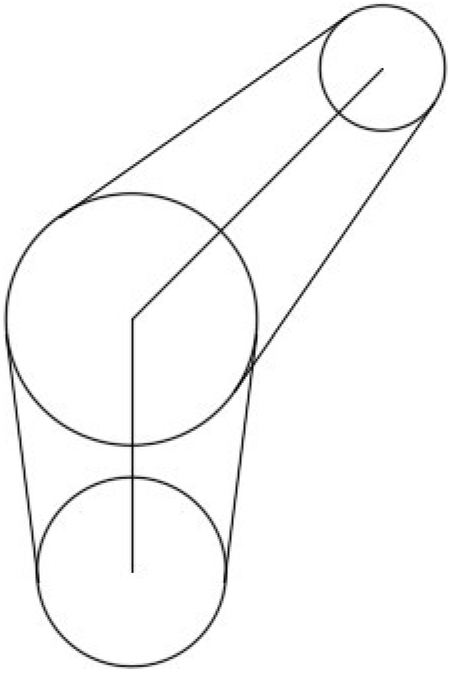


Figure Step 9

Step 10

Using the FILLET command, insert a 3.0 radius fillet as shown in the figures. (Figure Step 10A , 10B, and 10C)

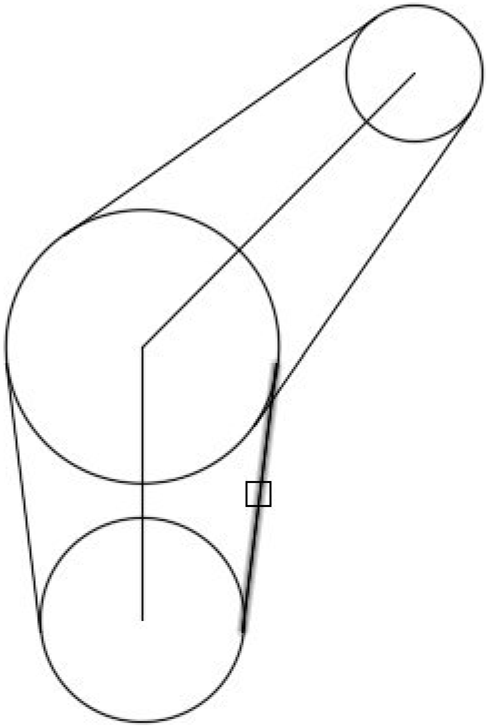


Figure Step 10A

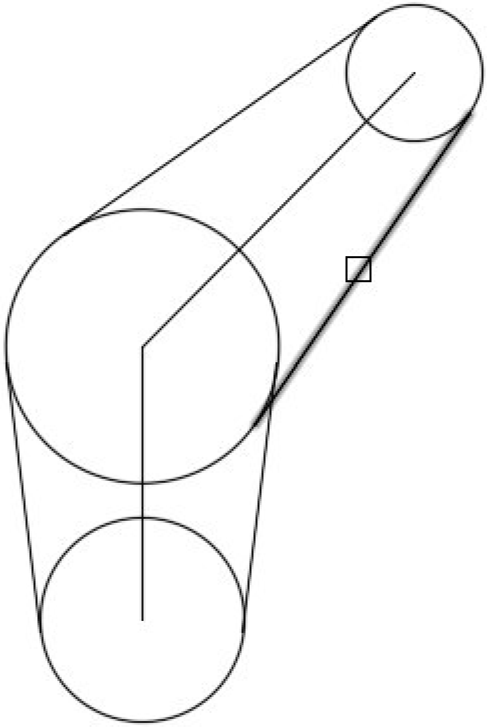


Figure Step 10B

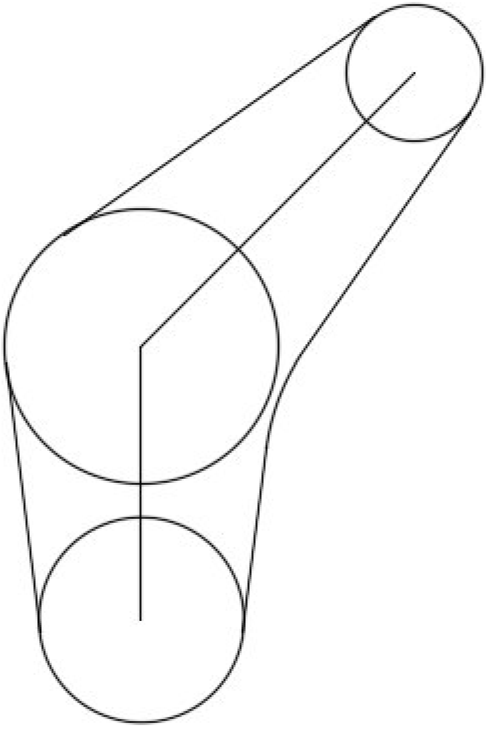


Figure Step 10C

Step 11

Change the layer of the objects from layer: Construction to layer: Object to match the figure. (Figure Step 11)

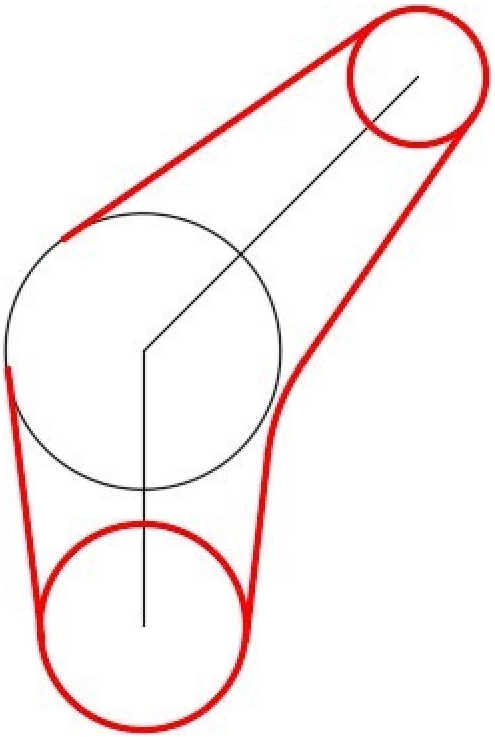


Figure Step 11

Step 12

Freeze layer: Construction. (Figure Step 12)

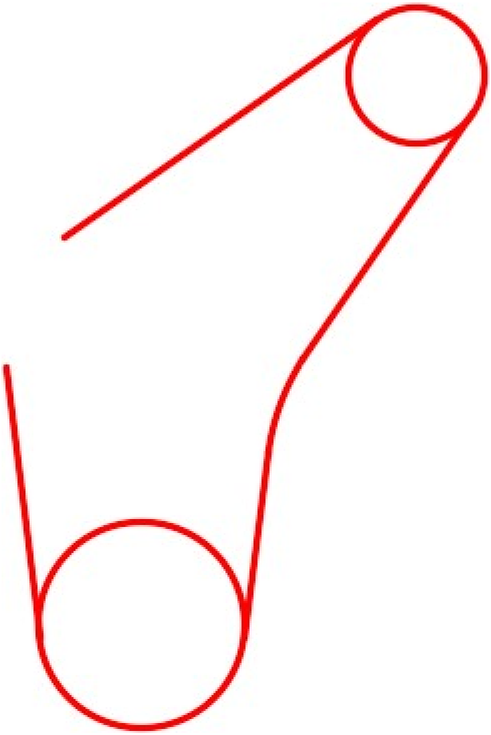


Figure Step 12

Step 13

Insert a 2.0 radius fillet to complete the drawing. (Figure Step 13)

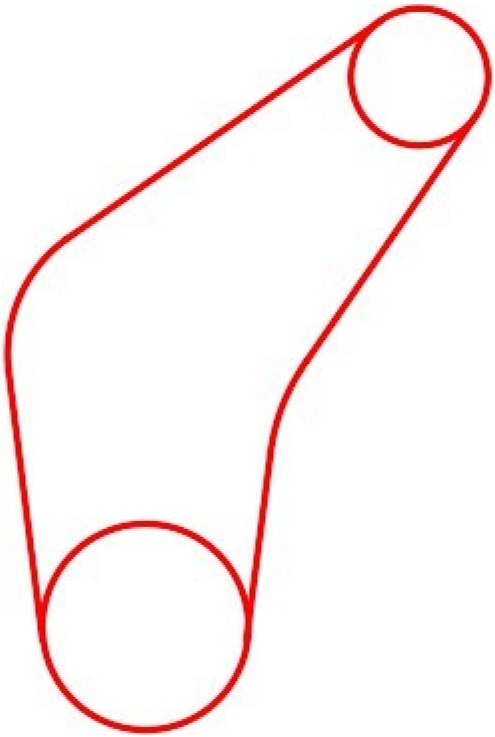


Figure Step 13

Step 14

Save and close the drawing.

WORKALONG: Drawing Construction Techniques

Step 1

Start a new drawing using the template: 2D English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 12-3.

Step 3

Create layers: Object and Construction. Set layer: Construction as the current layer.

Step 4

Referring to the figures, enter the LINE command and draw the lines shown in the figures in two steps. (Figure 4A, 4B, and 4C)

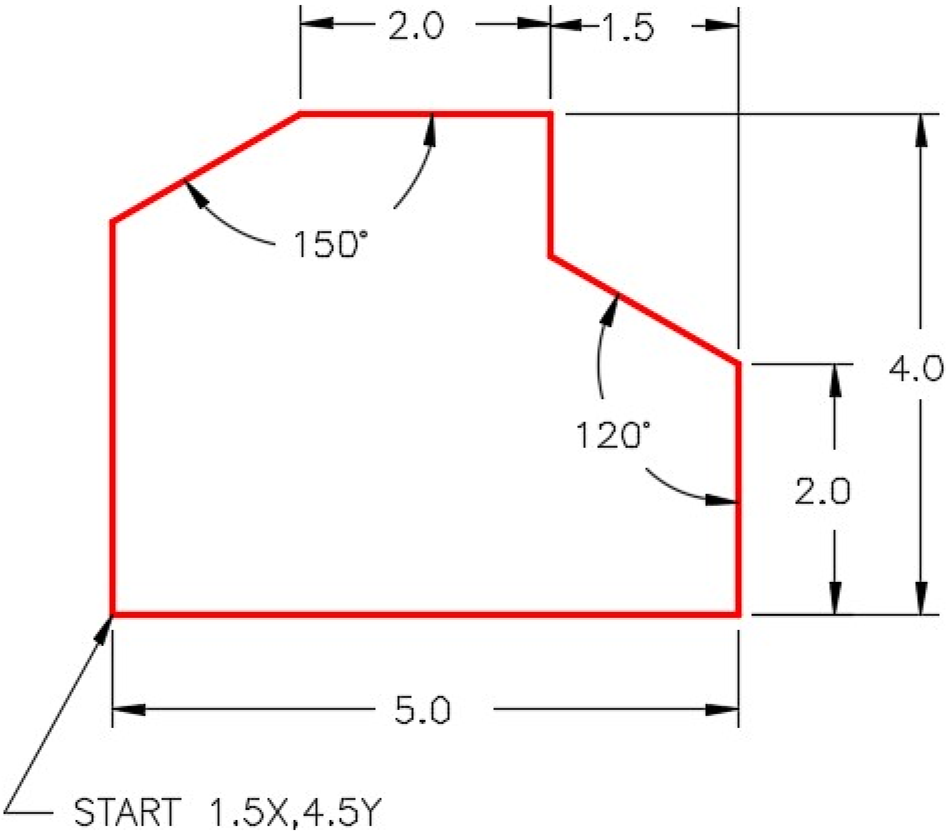


Figure Step 4A

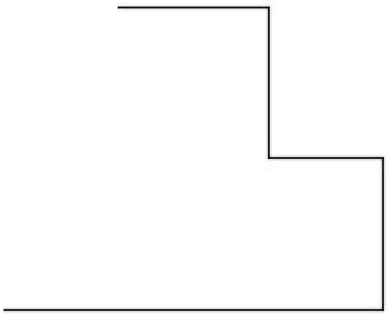


Figure Step 4B

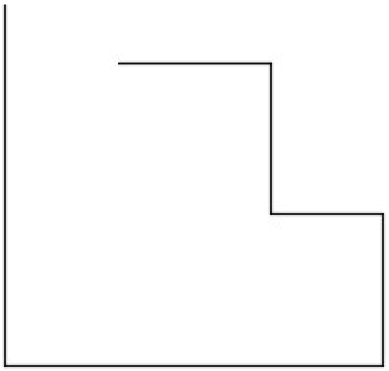


Figure Step 4C

AUTHOR'S COMMENTS: Draw the vertical line on the left side any length over 4 inches. The length is not important at this time.

Step 5

Enter the LINE command, as shown below, to draw two lines. (Figure Step 5)

Command: **L**

Specify first point:<Osnap on> *(end)* **P1**

Specify next point or [Undo]: **@2<150**

(Since you don't know the length of the line, draw a line of any length at the correct angle.)

Specify next point or [Undo]:

Command: **L**

Specify first point: *(end)* **P2**

Specify next point or [Undo]: **@3<210**

(Draw any length line at the correct angle.)

Specify next point or [Undo]:

Command:

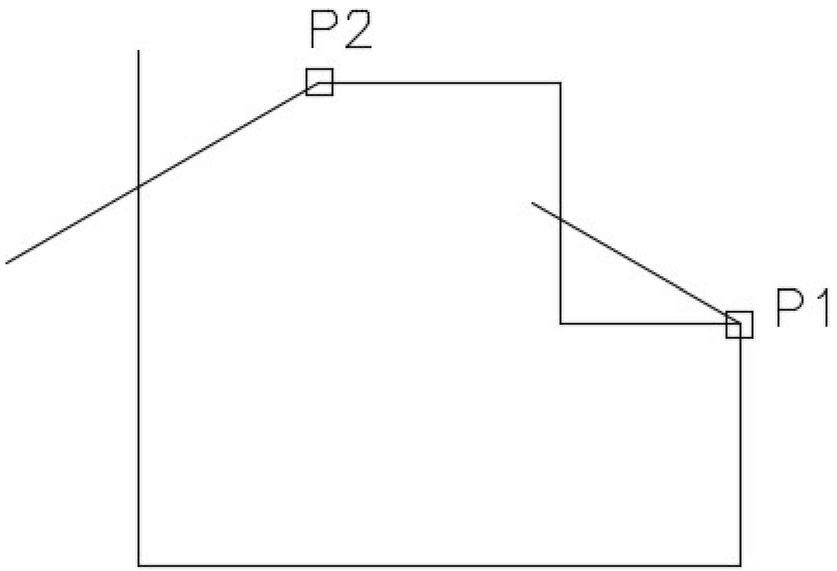


Figure Step 5

Step 6

Enter the FILLET as shown below. Set the radius to 0 to create perfect corners. (Figure Step 6A and 6B)

Command: **FILLET**

Current settings: Mode = TRIM, Radius = 0.3500

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **R**

Specify fillet radius <0.3500>: **0**

(Set the radius to zero.)

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **P3**

Select second object or shift-select to apply corner: **P4**

Command: **F**

(F is the shortcut for the command FILLET.)

Current settings: Mode = TRIM, Radius = 0.0000

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **P5**

Select second object or shift-select to apply corner: **P6**

Command:

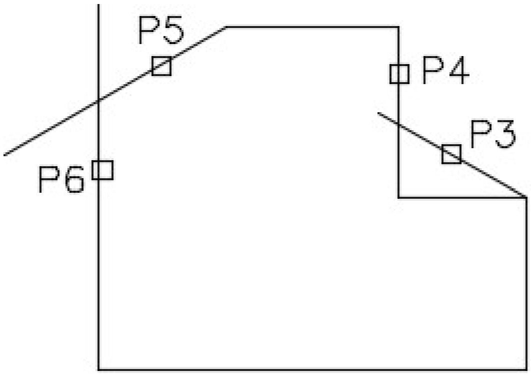


Figure Step 6A

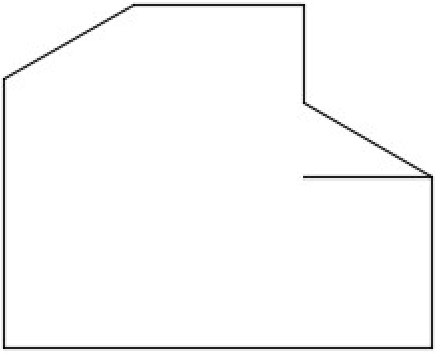


Figure Step 6B

AUTHOR'S COMMENTS: By setting the fillet radius to zero, the FILLET command will trim the lines to create perfect corners.

Step 7

Change the layer of the necessary lines to reside on layer: Object. Freeze layer: Construction. (Figure Step 7)

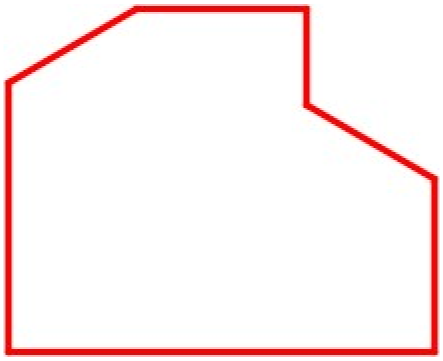


Figure Step 7

Step 8

Save and close the drawing.

USER TIP: To insert an arc at the end of two parallel lines, use the FILLET command.Regardless of what radius is currently set in the FILLET command, AutoCAD will draw a tangent arc using the one-half distance between the lines as the radius. The lines do not have to be the same length.

Command: **FILLET**

Current settings: Mode = TRIM, Radius = 0.0

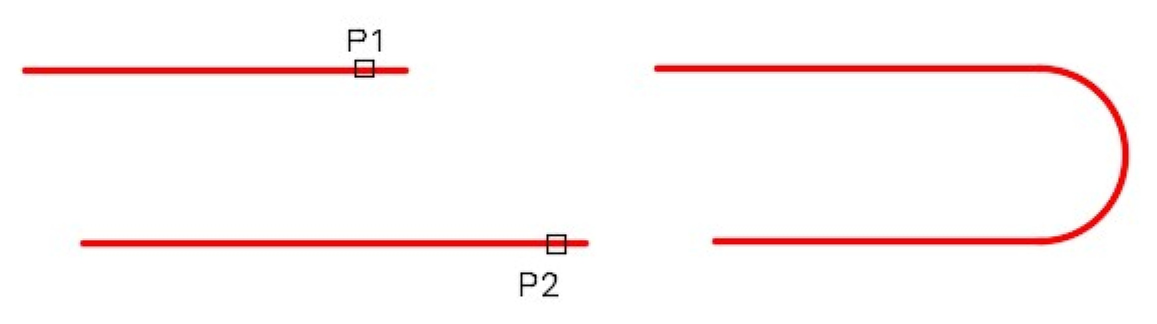
(Notice how the radius is set to zero. It can be any size.)

Select first object or [Polyline/Radius/Trim]: **P1**

(The first line you pick will retain it’s original length and the second one will be trimmed or extended.)

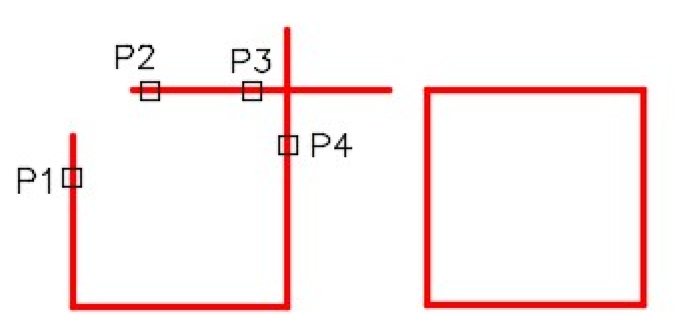
Select second object: **P2**

Command



Before

USER TIP: By setting the radius of the FILLET command to zero, the FILLET command will square the lines to make a perfect corner. The lines can be too short or too long or a combination of the two as shown in the figure on the right.



Before After

Key Principles

Key Principles in Module 12

1. Point of tangency is the theoretical point where a line joins an arc, two arcs join each other, or two circles join each other making a smooth transition.
2. A fillet is a tangent arc. To construct a fillet, use the FILLET command. The rule of thumb to follow is ‘ If the arc you are drawing is tangent to both objects it is connecting to, use the FILLET command. If the arc is tangent to one or neither of the objects, use the ARC command ‘.
3. Use the LINE command and the TAN object snap mode to draw lines tangent to two circles or arcs. Make sure that you apply it to both ends of the line.
4. By setting the radius of the FILLET command to zero, the FILLET command will square two lines to make a perfect corner.

Lab Exercise 12-1

Time allowed: 30 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 12-1	2D English	Inches

Layering Scheme

Layer Name	Objects on Layer	Color
Construction	Construction objects	253
Object	Lines, circle and arcs	Red

Step 1

Draw the object shown in the figures using the layering scheme. (Figure Step 1A and 1B)

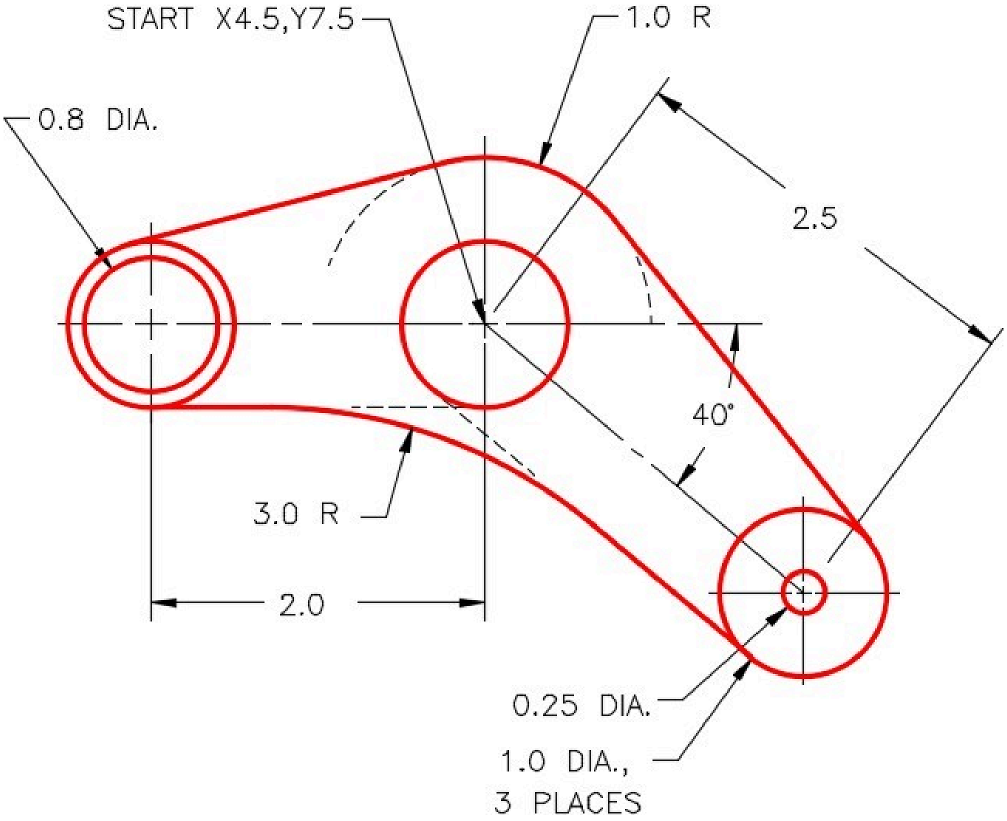

START X4.5,Y7.5
1.0 R
0.8 DIA.
2.5
40°
3.0 R
2.0
0.25 DIA.
1.0 DIA.,
3 PLACES

Figure Step 1A

AUTHOR’S COMMENTS: Do not delete the construction lines.

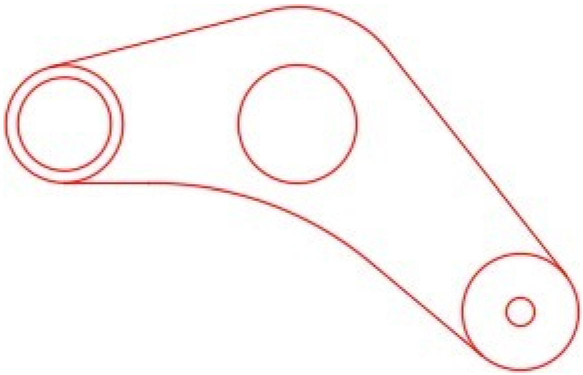


Figure Step 1B
Completed Drawing

Step 2

Set the insertion units and check your drawing with the key.

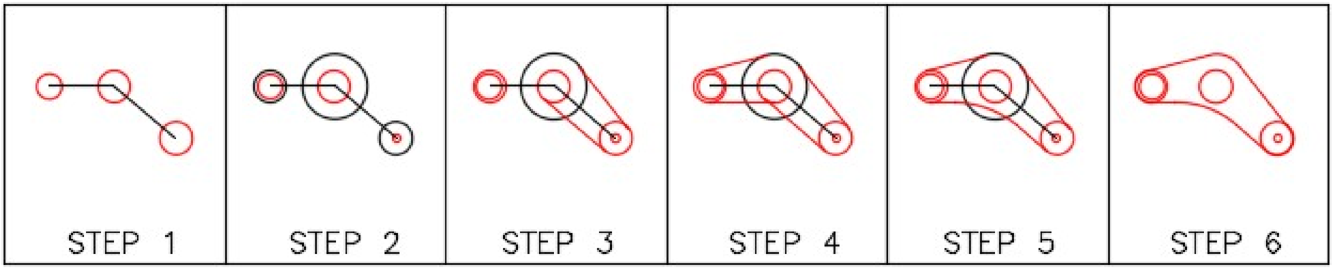
Step 3

Turn layer: Key off and freeze layer: Construction.

Step 4

Save and close the drawing.

AUTHOR'S CONSTRUCTION TECHNIQUES: The following steps are the construction technique suggested by the author to help you learn how to construct objects using AutoCAD. It is only the suggested method and if you can complete the drawing accurately using a different construction technique, that is what is important. You may want to compare your construction technique with the authors.



AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hint. If you get stuck and cannot complete it on your own, use the following hint to help you.

Hint 1

The dashed line shown in the dimensioned drawing indicates that the lines should be constructed tangent to the small circle from the circles on either end of the object. After the lines are inserted tangent to both circles, you can then insert the fillet. (Figure Hint 1)

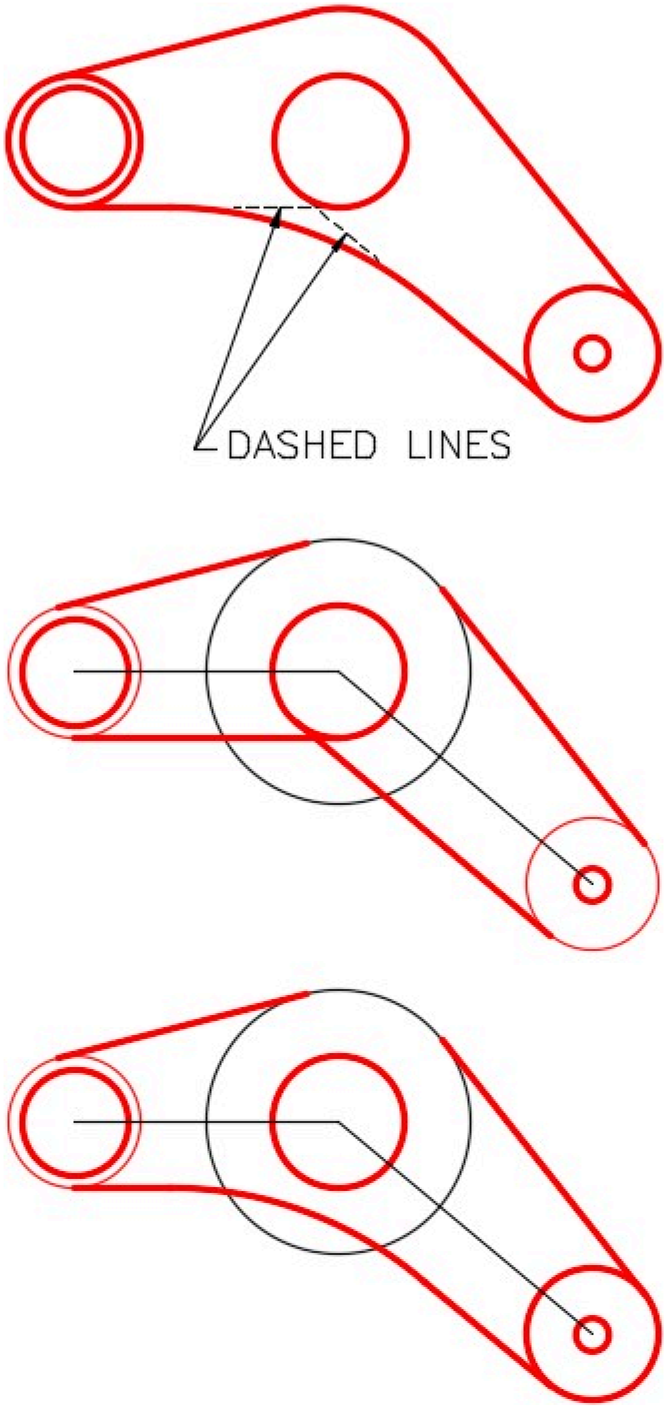


Figure Hint 1

Lab Exercise 12-2

Time allowed: 30 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 12-2	2D Metric	Millimeters

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Construction	Construction objects	253
Object	Lines, circles and arcs	Red

Step 1

Draw the object shown in the dimensioned drawing using the layering scheme. (Figure Step 1A and 1B)

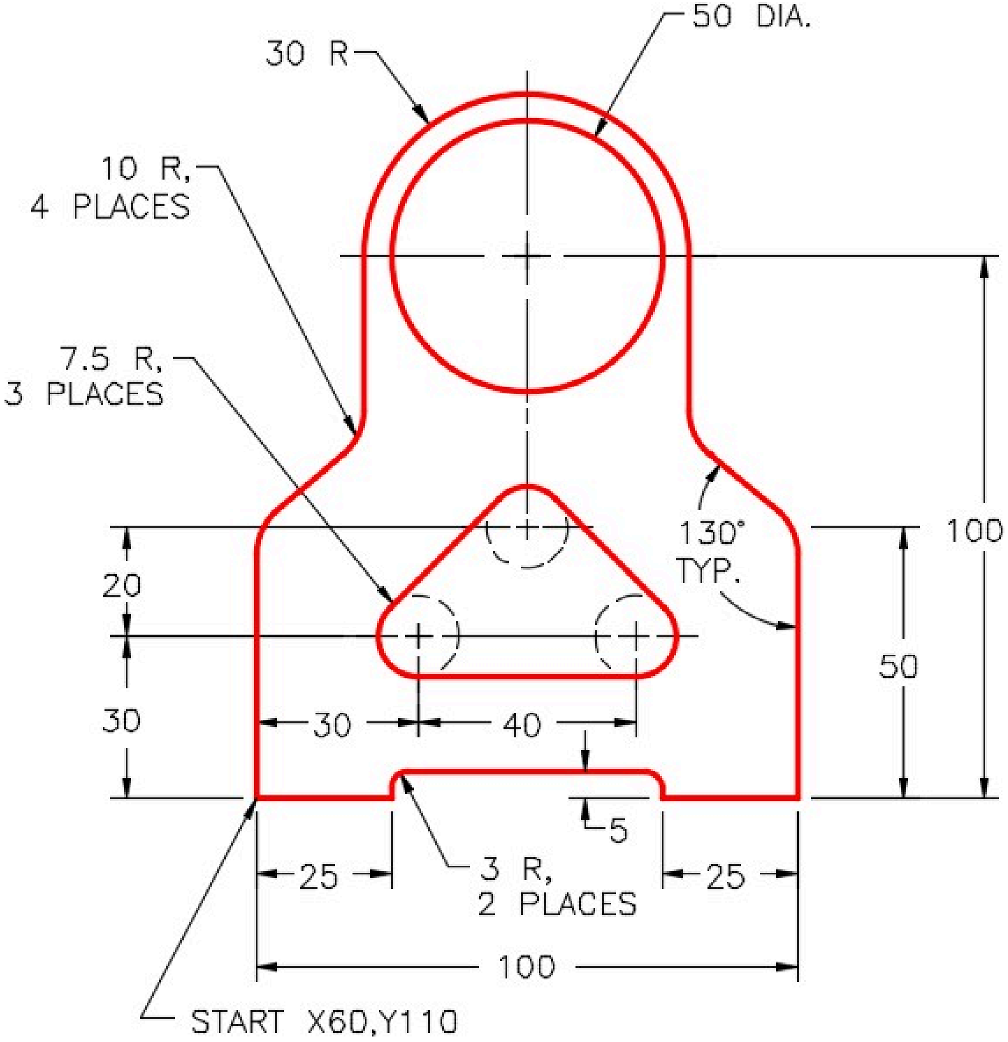

50 DIA.
30 R
10 R,
4 PLACES
7.5 R,
3 PLACES
130°
TYP.
100
20
50
30
30
40
5
25
3 R,
2 PLACES
25
100
START X60,Y110

Figure Step 1A

AUTHOR'S COMMENTS: Do not delete the construction lines

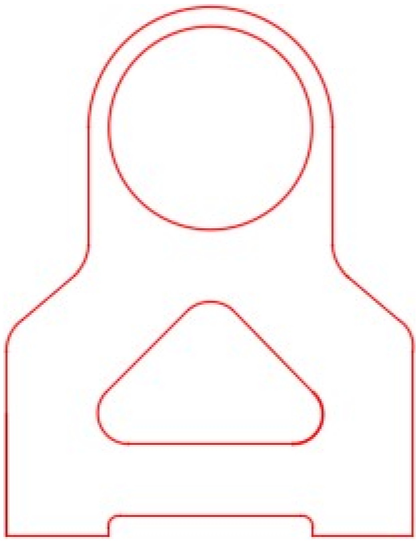


Figure Step 1B
Completed Drawing

Step 2

Set the insertion units and check your drawing with the key.

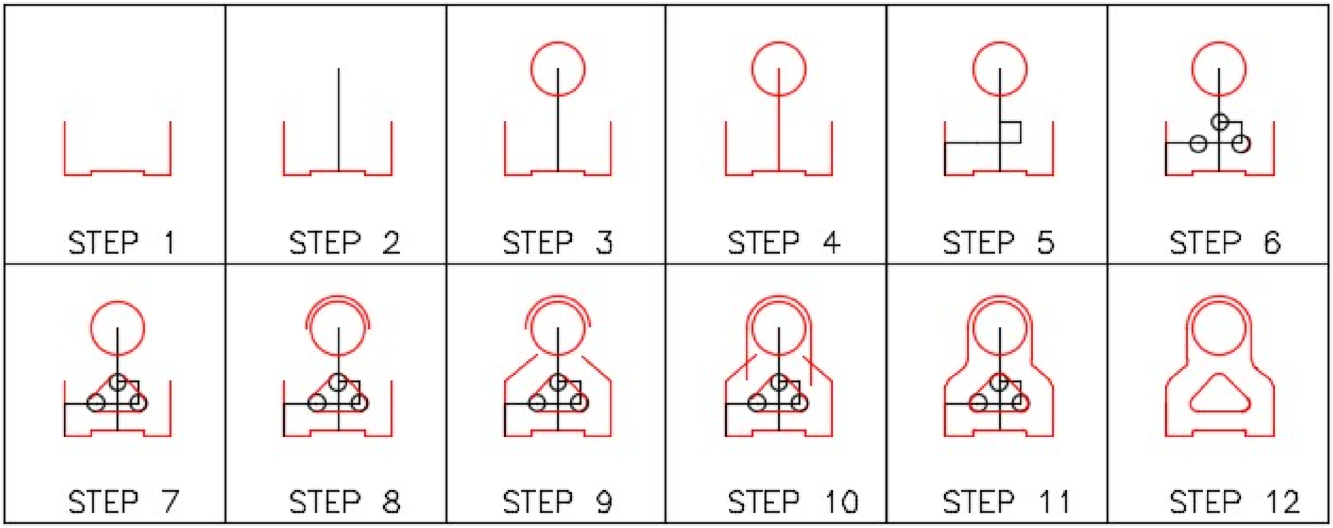
Step 3

Turn layer: Key off and freeze layer: Construction.

Step 4

Save and close the drawing.

AUTHOR'S CONSTRUCTION TECHNIQUES: The following steps are the construction technique suggested by the author to help you learn how to construct objects using AutoCAD. It is only the suggested method and if you can complete the drawing accurately using a different construction technique, that is what is important. You may want to compare your construction technique with the authors.


STEP 1
STEP 2
STEP 3
STEP 4
STEP 5
STEP 6
STEP 7
STEP 8
STEP 9
STEP 10
STEP 11
STEP 12

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

Draw a construction line from the midpoint of the line across the bottom of the slot 95 units in the positive Y direction. Draw a circle with its center at the end of the line. (Figure Hint 1)

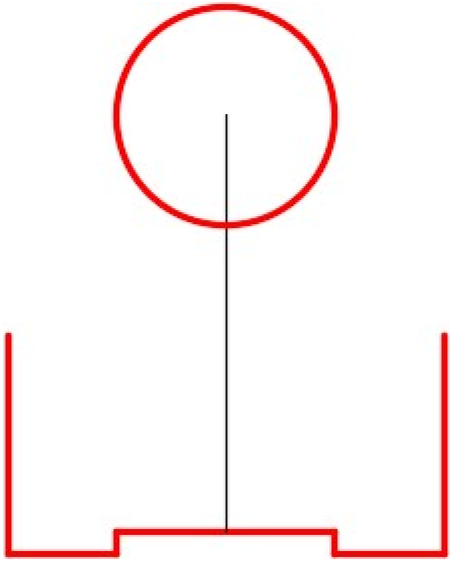


Figure Hint 1

Hint 2

Since you don't know the exact length of the lines shown in the figure, draw them any length but in the correct direction. When you insert the fillets, the line will automatically be trimmed. (Figure Hint 2)

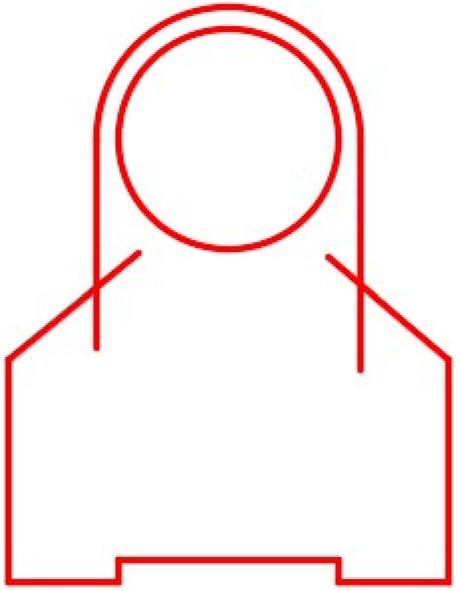


Figure Hint 2

Module 13: Object Properties

Learning Outcomes

When you have completed this module, you will be able to:

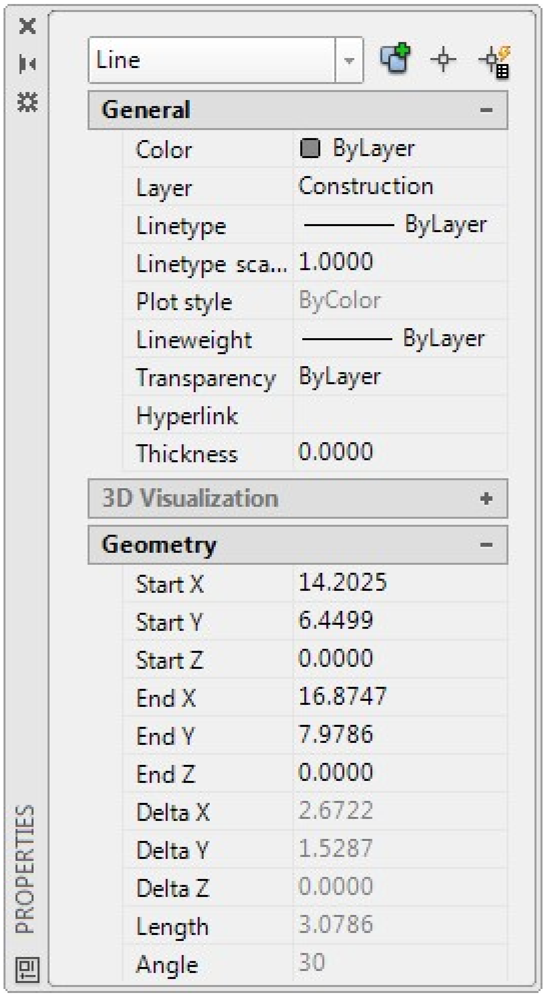
1. Describe object properties, linetype, and linetype scale.
2. Describe how to assign and edit linetypes ByLayer and set their linetype scale.
3. Apply the PROPERTIES and LTSCALE commands to display and edit properties of drawing objects and set the linetype scale of objects.
4. Describe how to select objects using a pick, a window, and a crossing window.

Object Properties

AutoCAD stores all existing objects and their properties in a database file called the *drawing file*. The name of the file is the drawing name that you assigned to the drawing and AutoCAD automatically assigns the extension .dwg. The drawing file stores all existing drawing objects along with the properties of each object in a database. The drawing objects that have been taught up to now are lines, circles, and arcs. Figure 13-1 displays the properties of a line.

When an existing drawing is opened, AutoCAD reads the drawing file and uses it to construct the objects from the database and displays the objects in the Graphic window. The REGEN command does the same thing for the current drawing.

The PROPERTIES command opens the Properties window. Properties of existing drawing objects can be modified using commands or they can be edited in the Properties window. It is faster and easier to edit the properties in the Properties window but not all of the properties can be edited using it. AutoCAD grays the boxes of the properties that cannot be changed in the window. See Figure 13-1.



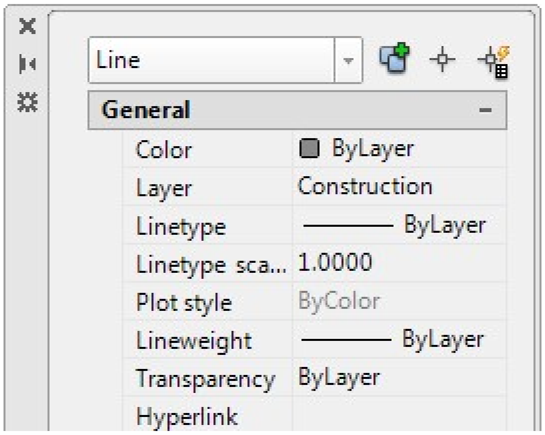
Line

General	
Color	ByLayer
Layer	Construction
Linetype	ByLayer
Linetype sca...	1.0000
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	
Thickness	0.0000
3D Visualization	
Geometry	
Start X	14.2025
Start Y	6.4499
Start Z	0.0000
End X	16.8747
End Y	7.9786
End Z	0.0000
Delta X	2.6722
Delta Y	1.5287
Delta Z	0.0000
Length	3.0786
Angle	30

Figure 13-1
Properties of a Line

Without entering a command, one or more objects in the current drawing can be selected using the pick box. The selected object(s) will highlight and the data will appear in the Properties window. If more then one object is selected, only the properties which are common to all of the selected objects will display. To unselect objects, press the Esc key.

Figure 13-2 shows the Properties window where only one line was selected by the user. This can be seen in the box located in the top left corner of the window. This box will display what object(s) is currently selected and its properties are displayed below in the window.

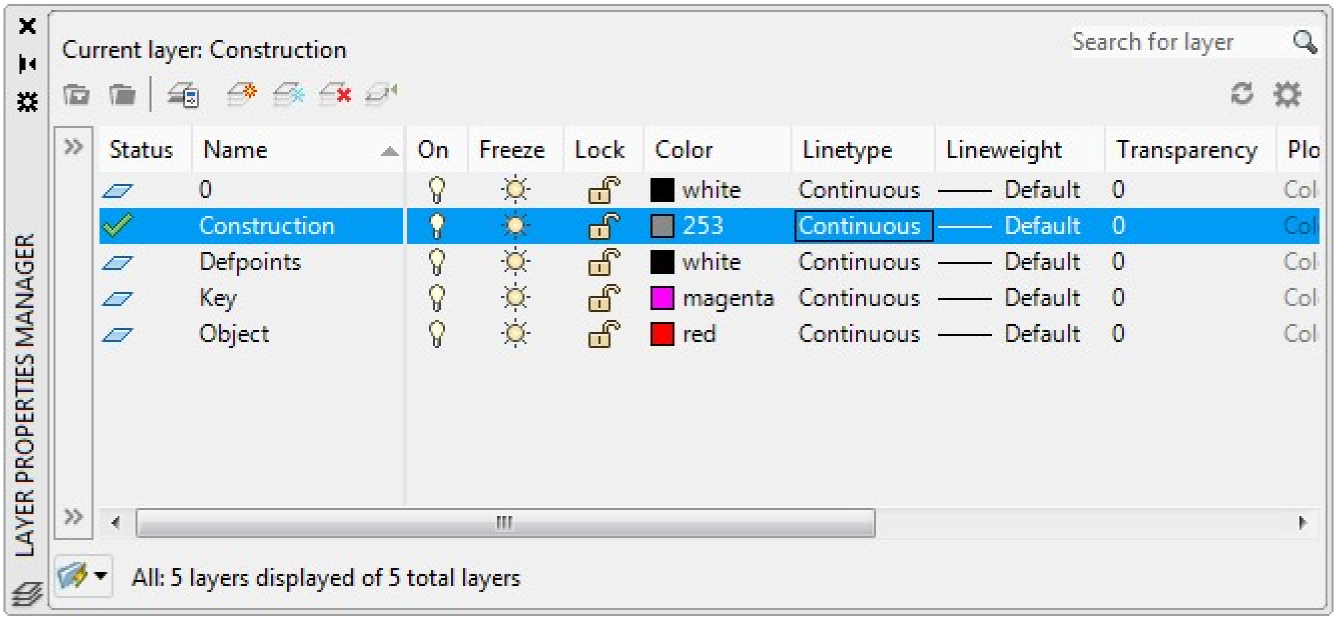


Color	ByLayer
Layer	Construction
Linetype	ByLayer
Linetype sca...	1.0000
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	

Figure 13-2
The Properties Dialogue Box

Linetypes

A *linetype* is a repeating pattern of dots, dashes, text, objects, or blank spaces displayed on a drawing object. Up to this point in the book, the default linetype *Continuous* has been assigned to all objects. See Figure 13-3. A continuous linetype displays solid. In this module, linetypes will be assigned *ByLayer*. The template files that you are using are only loaded with the Continuous linetype. Before a linetype can be assigned to a layer, it must be loaded into the current drawing's database. All loaded linetypes will remain in the drawing file for future use. If required, they can be purged out of the drawing file. Purging is taught in Module 34.



Current layer: Construction

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight	Transparency	Plo
	0				white	Continuous	Default	0	Col
✓	Construction				253	Continuous	Default	0	Col
	Defpoints				white	Continuous	Default	0	Col
	Key				magenta	Continuous	Default	0	Col
	Object				red	Continuous	Default	0	Col

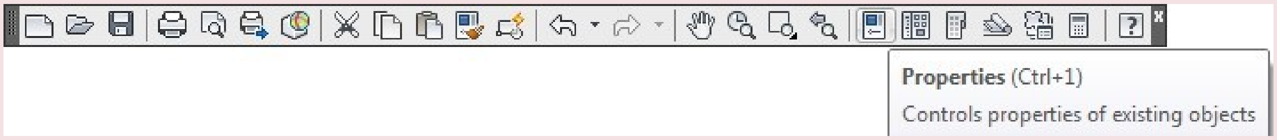
All: 5 layers displayed of 5 total layers

Figure 13-3
Continuous Linetype

AutoCAD Command: PROPERTIES

The PROPERTIES command opens the Properties window.

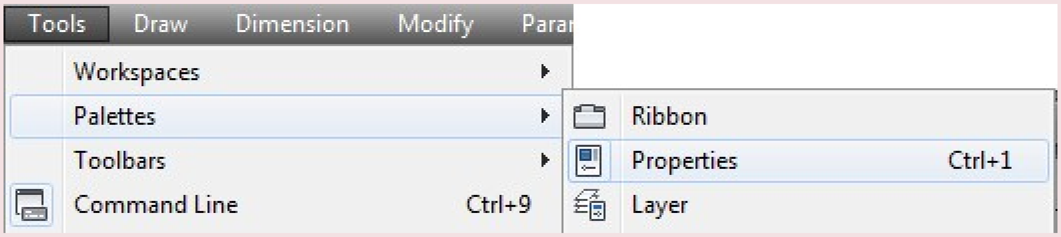
Shortcut: **CTRL+1**



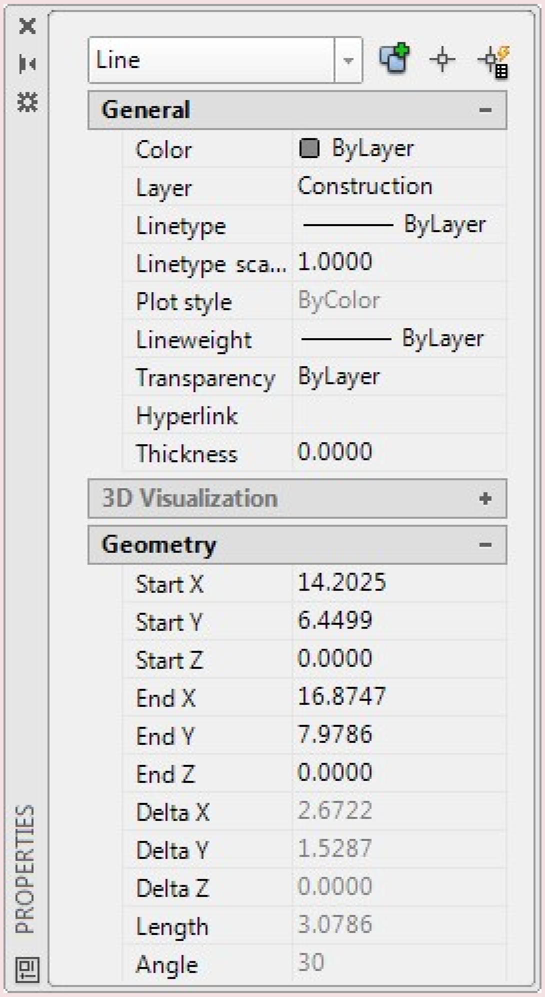
Properties (Ctrl+1)

Controls properties of existing objects

Standard Toolbar


Tools
Draw
Dimension
Modify
Parar
Workspaces
Palettes
Toolbars
Command Line
Ctrl+9
Ribbon
Properties
Ctrl+1
Layer

Tools Pull-down


PROPERTIES

Line

General

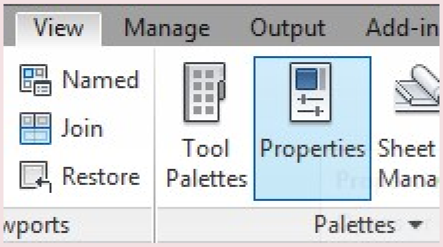
Color	ByLayer
Layer	Construction
Linetype	ByLayer
Linetype sca...	1.0000
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	
Thickness	0.0000

3D Visualization

Geometry

Start X	14.2025
Start Y	6.4499
Start Z	0.0000
End X	16.8747
End Y	7.9786
End Z	0.0000
Delta X	2.6722
Delta Y	1.5287
Delta Z	0.0000
Length	3.0786
Angle	30

Properties Window


View
Manage
Output
Add-in
Named
Join
Restore
Tool Palettes
Properties
Sheet Mana
wports
Palettes

View Ribbon

AutoCAD Command: LTSCALE

The LTSCALE system variable is used to set the global linetype scale for the drawing.

Shortcut**: LTS**

MUST KNOW: The AutoCAD drawing file contains all of the data that relates to the drawing in a database. When an existing drawing is opened, AutoCAD uses the drawing file to construct the drawing objects in the Graphic window. When the current drawing is saved, AutoCAD updates the drawing file. AutoCAD adds the extension .dwg to the user assigned drawing name. For example, if you name the drawing House Plan, AutoCAD will name the file House Plan.dwg, when the file is saved.

MUST KNOW: Before AutoCAD displays the linetype on a drawing object, it multiplies the scale factor set in LTSCALE times the linetype scale property that is assigned to the drawing object.

For example, if the LTSCALE is set to 5 and the drawing object's linetype scale property is set to 2, AutoCAD will display the object with the linetype scale of 10. In other words, it will display the linetype 10 times larger then its normal display.

It is best to set the LTSCALE to 1 and set the drawing object's linetype scale property to the setting to display the desired linetype and that makes it easier for you to set the linetype scale.

For example, if the LTSCALE is set to 1 and the objects linetype scale property is set to 10, AutoCAD will display the drawing object with the linetype scale of 10 or 10 times its normal display.

USER TIP: The Quick Properties window will open automatically when a drawing object is double clicked with the left mouse button. This saves you one step when editing or inquiring about the object's properties.

WORKALONG: Working with Linetypes and Linetype Scales

Step 1

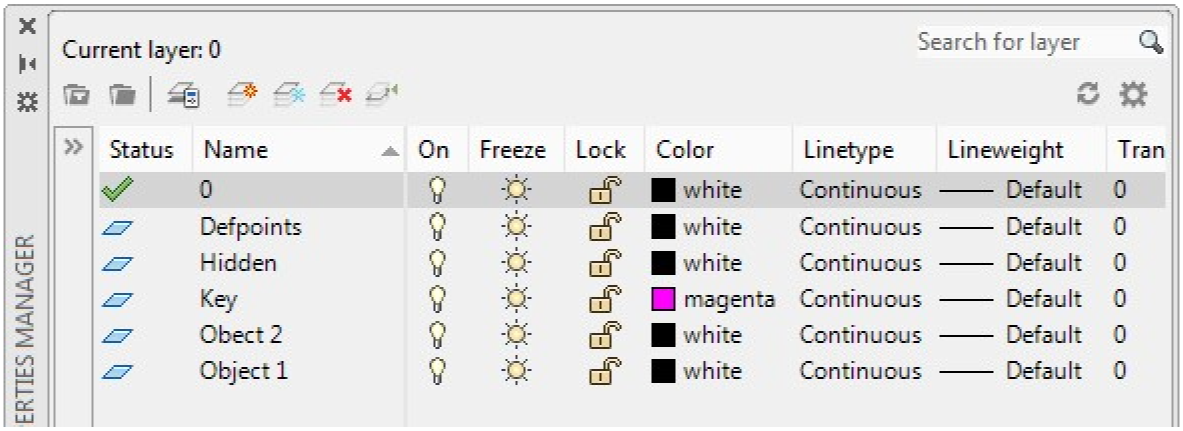
Start a new drawing using template: 2D English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 13-1.

Step 3

Open the Layer Properties Manager window and create the layers: Hidden, Object 1, and Object 2. (Figure Step 3)


ERTIES MANAGER
Search for layer

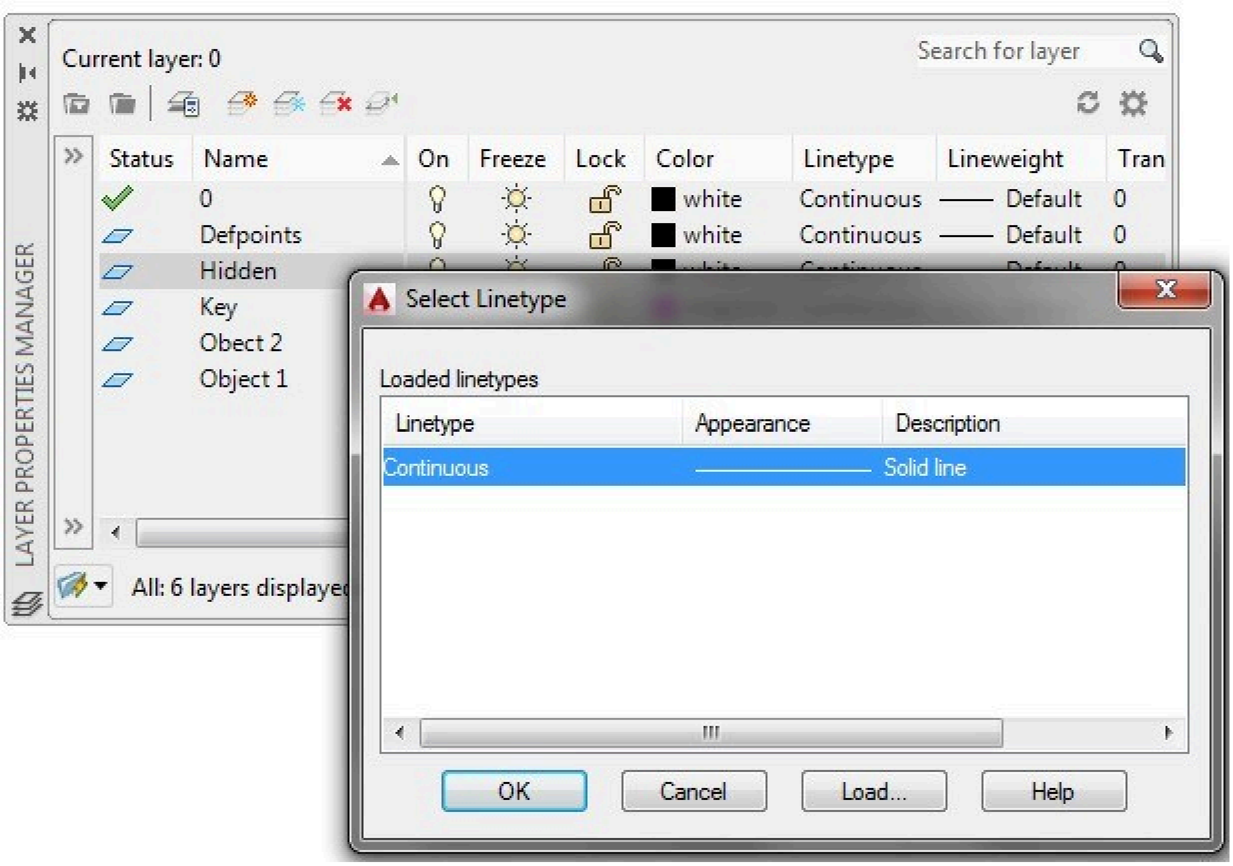
Current layer: 0

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight	Tran
✓	0				■ white	Continuous	—— Default	0
	Defpoints				■ white	Continuous	—— Default	0
	Hidden				■ white	Continuous	—— Default	0
	Key				■ magenta	Continuous	—— Default	0
	Obect 2				■ white	Continuous	—— Default	0
	Object 1				■ white	Continuous	—— Default	0

Figure Step 3

Step 4

Click layer: Hidden to select it. Select Continuous. This will open the Select Linetype dialogue box. (Figure Step 4)


LAYER PROPERTIES MANAGER
Search for layer
All: 6 layers displaye
Select Linetype
OK
Cancel
Load...
Help

Current layer: 0

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight	Tran
✓	0				■ white	Continuous	—— Default	0
	Defpoints				■ white	Continuous	—— Default	0
	Hidden							
	Key							
	Obect 2							
	Object 1							

Loaded linetypes

Linetype	Appearance	Description
Continuous	————	Solid line

Figure Step 4

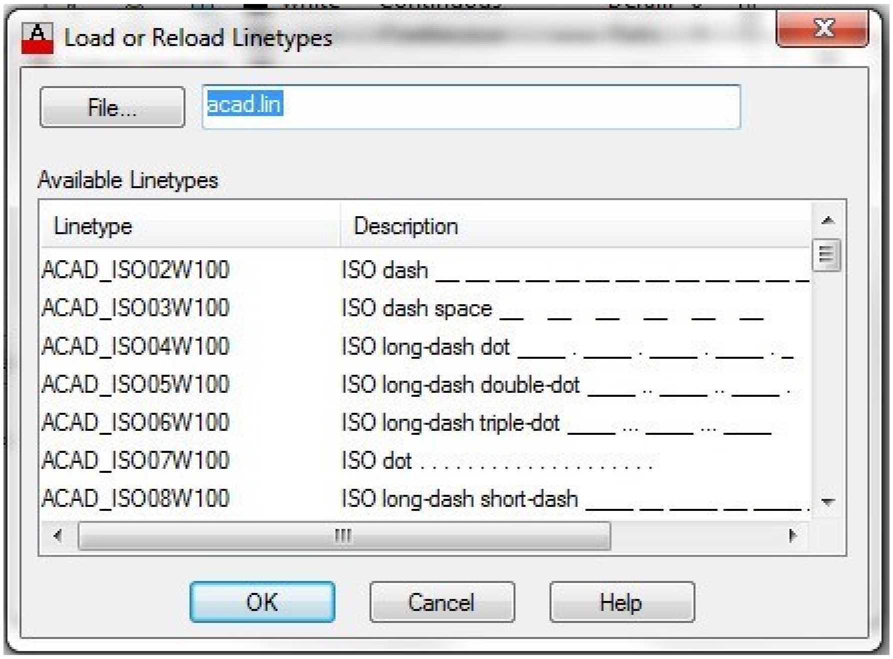
AUTHOR’S COMMENTS: As you can see, the only linetype currently loaded in the drawing is Continuous.

Step 5

Click the Load button to open the Load or Reload Linetypes dialogue box. The name in the File box will be one of the following: (Figure Step 5).

ACAD.LIN – English

ACADISO.LIN – Metric



File... acad.lin

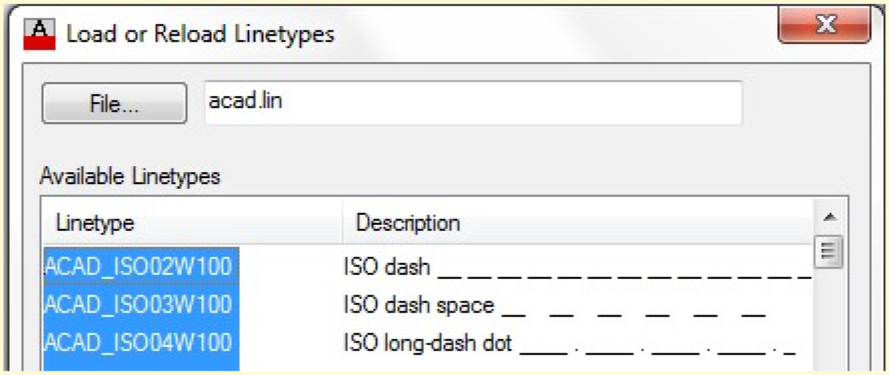
Available Linetypes

Linetype	Description
ACAD_ISO02W100	ISO dash __ __ __ __ __ __ __ __ __ __ __ __ __
ACAD_ISO03W100	ISO dash space __ __ __ __ __ __
ACAD_ISO04W100	ISO long-dash dot ____ . ____ . ____ . ____ . _
ACAD_ISO05W100	ISO long-dash double-dot ____ .. ____ .. ____ .
ACAD_ISO06W100	ISO long-dash triple-dot ____ ... ____ ... ____
ACAD_ISO07W100	ISO dot . . . .
ACAD_ISO08W100	ISO long-dash short-dash ____ __ ____ __ ____ .

Figure Step 5

AUTHOR’S COMMENTS: There is an english and a metric linetype file. The template that was used to start the drawing dictates the file that the dialogue box will display.

USER TIP: It easier to load all the linetypes into your current drawing rather then selecting them individually. To do this, click any linetype to select it and then press Ctrl+A. This will select all linetypes.

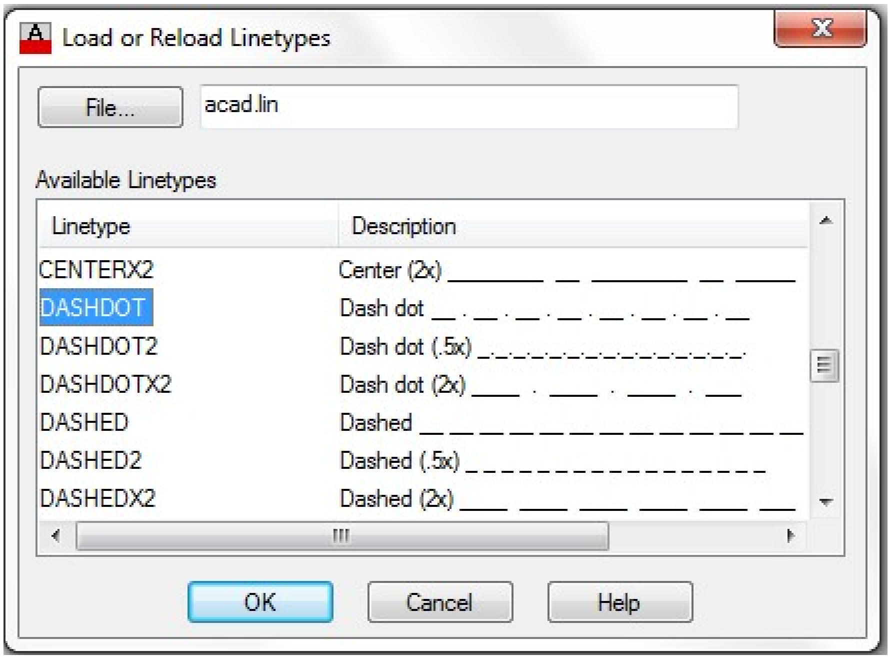

Load or Reload Linetypes
File...
Available Linetypes

acad.lin

Linetype	Description
ACAD_ISO02W100	ISO dash
ACAD_ISO03W100	ISO dash space
ACAD_ISO04W100	ISO long-dash dot

Step 6

Scroll down until you find the linetype: DASHDOT. Select it as shown in the figure. (Figure Step 6)


Load or Reload Linetypes
File...
Available Linetypes
OK
Cancel
Help

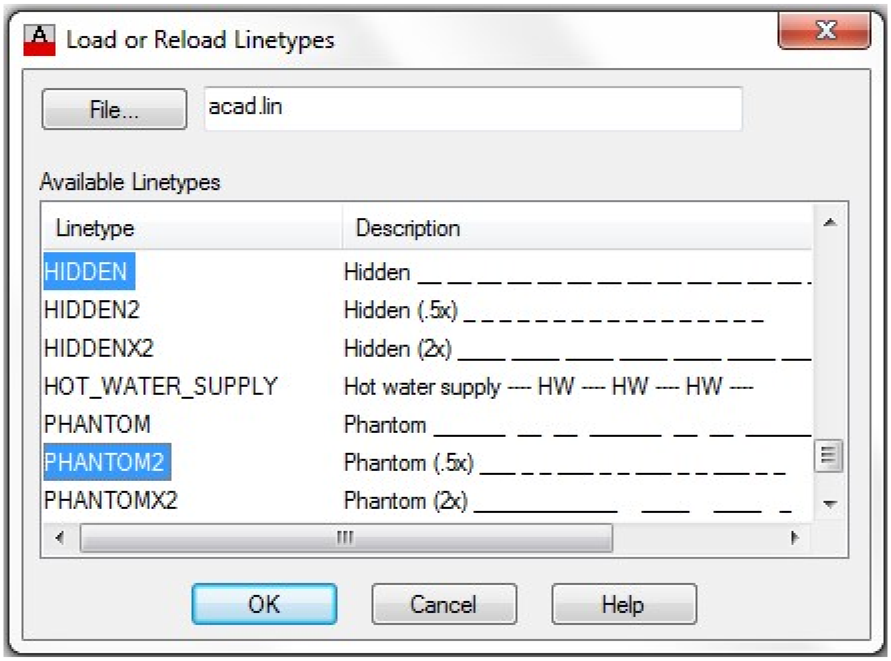
acad.lin

Linetype	Description
CENTERX2	Center (2x)
DASHDOT	Dash dot
DASHDOT2	Dash dot (.5x)
DASHDOTX2	Dash dot (2x)
DASHED	Dashed
DASHED2	Dashed (.5x)
DASHEDX2	Dashed (2x)

Figure Step 6

Step 7

While holding down the Ctrl key, scroll down until you find the linetypes: HIDDEN and PHANTOM2. Select them. They will highlight as shown in the figure. Click OK to close the dialogue box. (Figure Step 7).



acad.lin

Available Linetypes

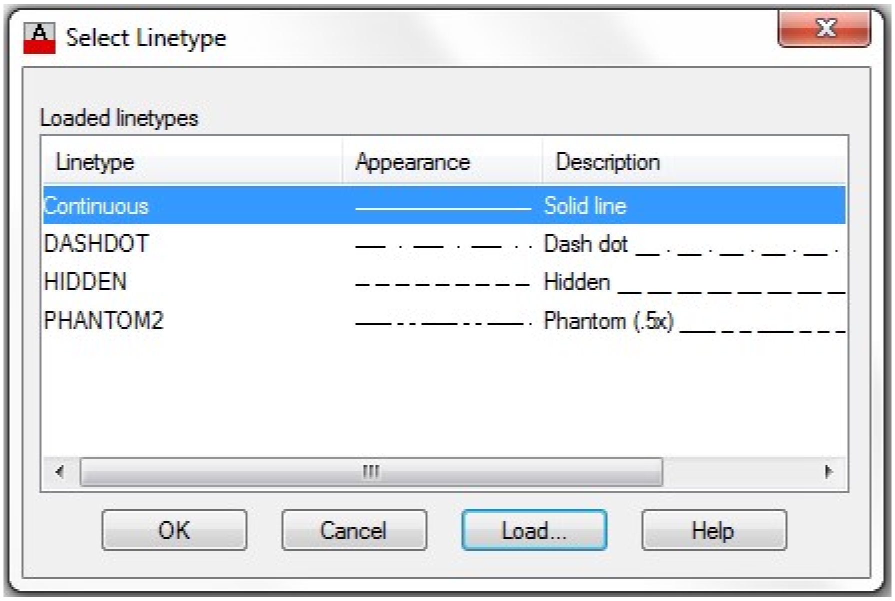
Linetype	Description
HIDDEN	Hidden __ __ __ __ __ __ __ __ __ __ __ __ __
HIDDEN2	Hidden (.5x) _ _ _ _ _ _ _ _ _ _ _ _ _ _ _ _ _
HIDDENX2	Hidden (2x) ____ ____ ____ ____ ____ ____ __
HOT_WATER_SUPPLY	Hot water supply ---- HW ---- HW ---- HW ----
PHANTOM	Phantom ______ __ __ ______ __ __ ______
PHANTOM2	Phantom (.5x) ___ _ _ ___ _ _ ___ _ _ ___ _ _
PHANTOMX2	Phantom (2x) ____________ ____ ____ _

Figure Step 7

AUTHOR'S COMMENTS: You must hold the Ctrl key down to select more then one linetype at the same time.

Step 8

The linetypes you selected in Step 7 will display in the Select Linetype dialogue box. Click OK to close the dialogue box. (Figure Step 8)



Loaded linetypes

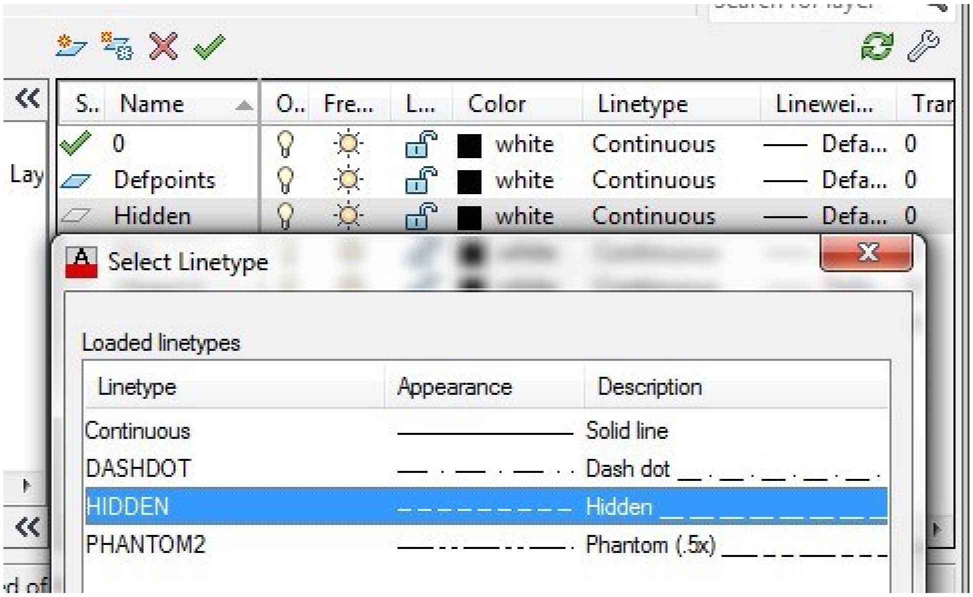
Linetype	Appearance	Description
Continuous		Solid line
DASHDOT		Dash dot __ . __ . __ . __ . __ .
HIDDEN		Hidden __ __ __ __ __ __ __ __
PHANTOM2		Phantom (.5x) ___ _ _ ___ _ _

Figure Step 8

AUTHOR'S COMMENTS: There are now 4 linetypes loaded in this drawing.

Step 9

In the Layer Properties Manager window, click layer: Hidden and click the linetype: Continuous. In the Select Linetype dialogue box, select the linetype: HIDDEN and then click OK. (Figure Step 9)



S..	Name	O..	Fre...	L...	Color	Linetype	Linewei...	Trar
✓	0				■ white	Continuous	—— Defa...	0
	Defpoints				■ white	Continuous	—— Defa...	0
	Hidden				■ white	Continuous	—— Defa...	0

Select Linetype

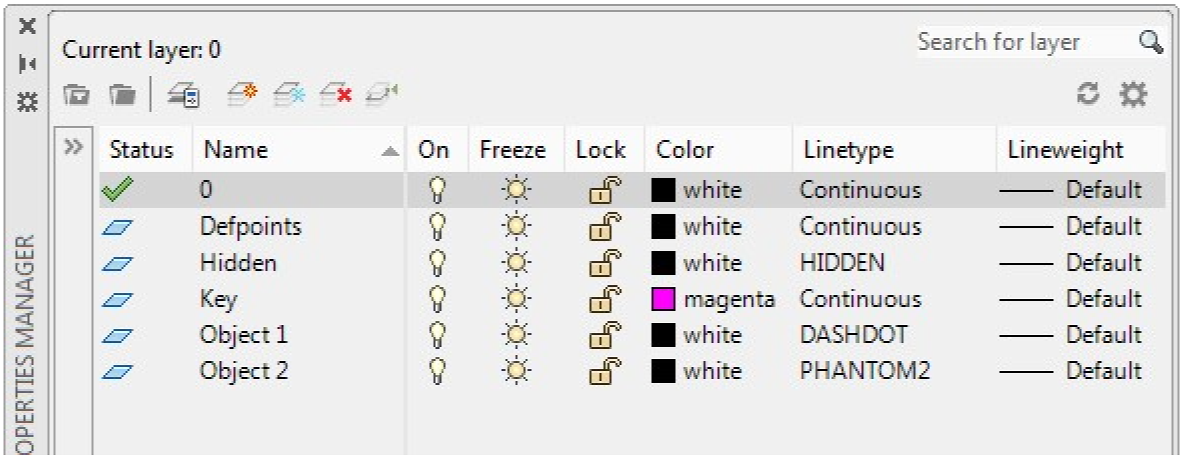
Loaded linetypes

Linetype	Appearance	Description
Continuous	————————	Solid line
DASHDOT	— · — · — · ·	Dash dot __ . __ . __ . __ . __ .
HIDDEN	– – – – – – – – –	Hidden __ __ __ __ __ __ __ __ __
PHANTOM2	—- - —- - —- ·	Phantom (.5x) ___ _ _ ___ _ _ ___

Figure Step 9

Step 10

Using what you just learned, change the linetype for layer: Object 1 to linetype: DASHDOT and layer: Object 2 to the linetype: PHANTOM2. (Figure Step 10)



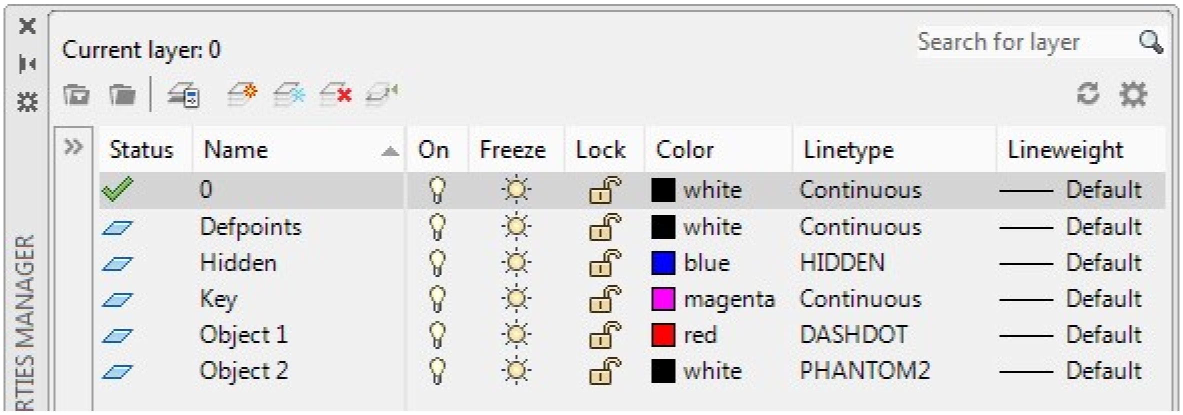
Current layer: 0

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight
✓	0				■ white	Continuous	—— Default
	Defpoints				■ white	Continuous	—— Default
	Hidden				■ white	HIDDEN	—— Default
	Key				■ magenta	Continuous	—— Default
	Object 1				■ white	DASHDOT	—— Default
	Object 2				■ white	PHANTOM2	—— Default

Figure Step 10

Step 11

Change the colors of the layer: Hidden to blue and layer: Object 1 to red to match the figure. (Figure Step 11)



Current layer: 0

Search for layer

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight
✓	0	💡	☼	🔓	⬛ white	Continuous	—— Default
▱	Defpoints	💡	☼	🔓	⬛ white	Continuous	—— Default
▱	Hidden	💡	☼	🔓	🟦 blue	HIDDEN	—— Default
▱	Key	💡	☼	🔓	🟪 magenta	Continuous	—— Default
▱	Object 1	💡	☼	🔓	🟥 red	DASHDOT	—— Default
▱	Object 2	💡	☼	🔓	⬛ white	PHANTOM2	—— Default

Figure Step 11

Step 12

Draw three 2-inch diameter circles on layer 0 locating their centers at the coordinates shown in the figure. (Figure Step 12)

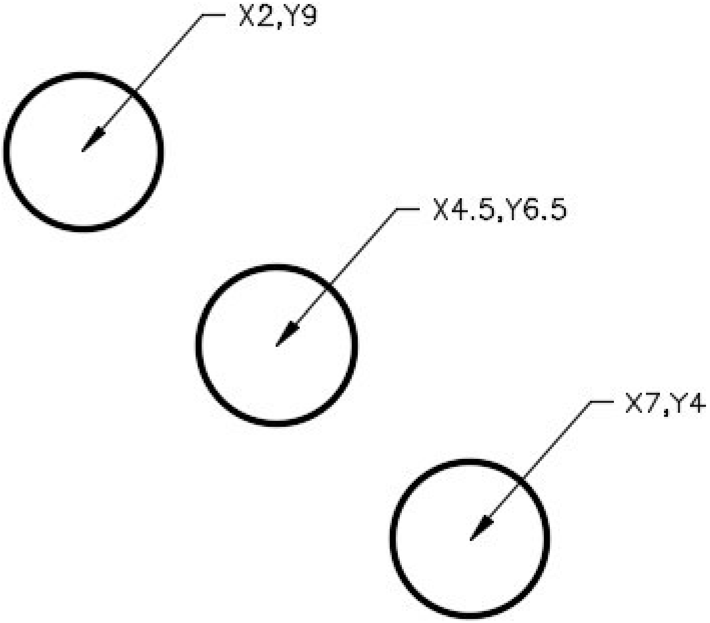
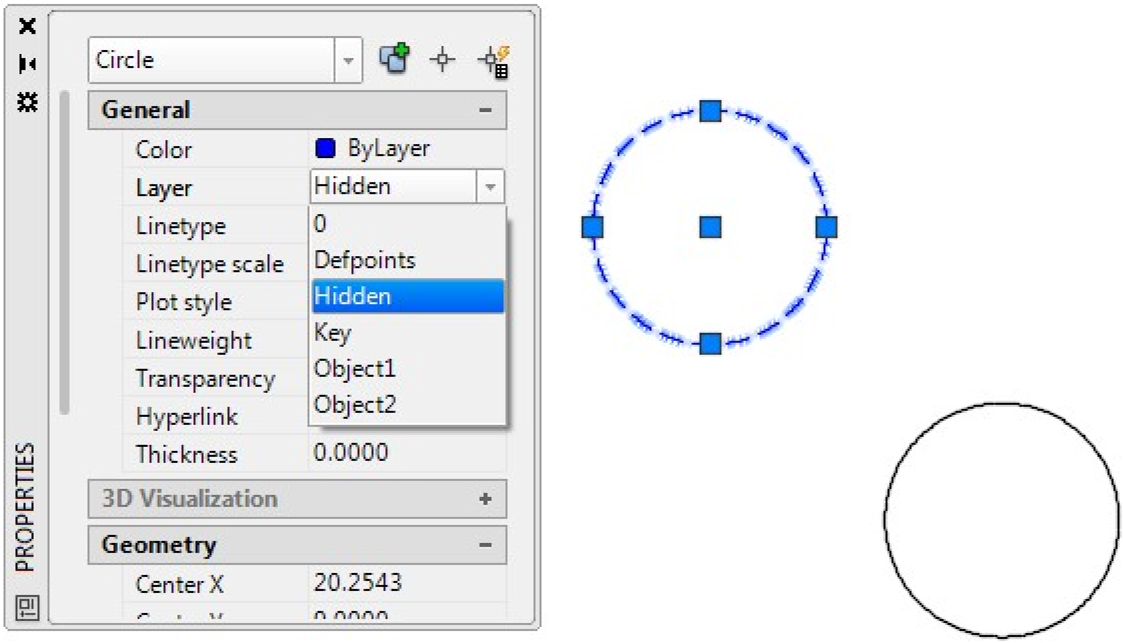


Figure Step 12

Step 13

Open the Properties window and select the top left circle. While it is selected, change it to layer: Hidden as shown in the figure. (Figure Step 13)


Circle

General

Color ■ ByLayer

Layer Hidden

Thickness 0.0000

Geometry

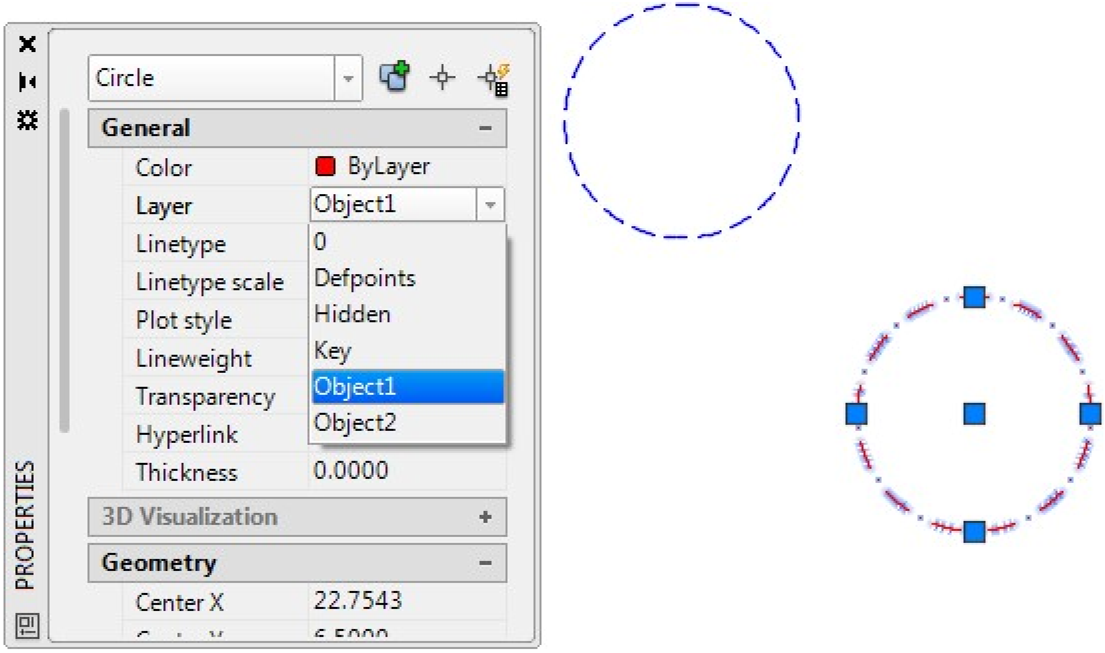
Center X 20.2543

PROPERTIES
Linetype
Linetype scale
Plot style
Lineweight
Transparency
Hyperlink
0
Defpoints
Hidden
Key
Object1
Object2
3D Visualization

Figure Step 13

Step 14

Select the center circle. In the Properties window, change it to layer Object 1. (Figure Step 14)


Circle

General

Color ■ ByLayer

Layer Object1

Thickness 0.0000

Geometry

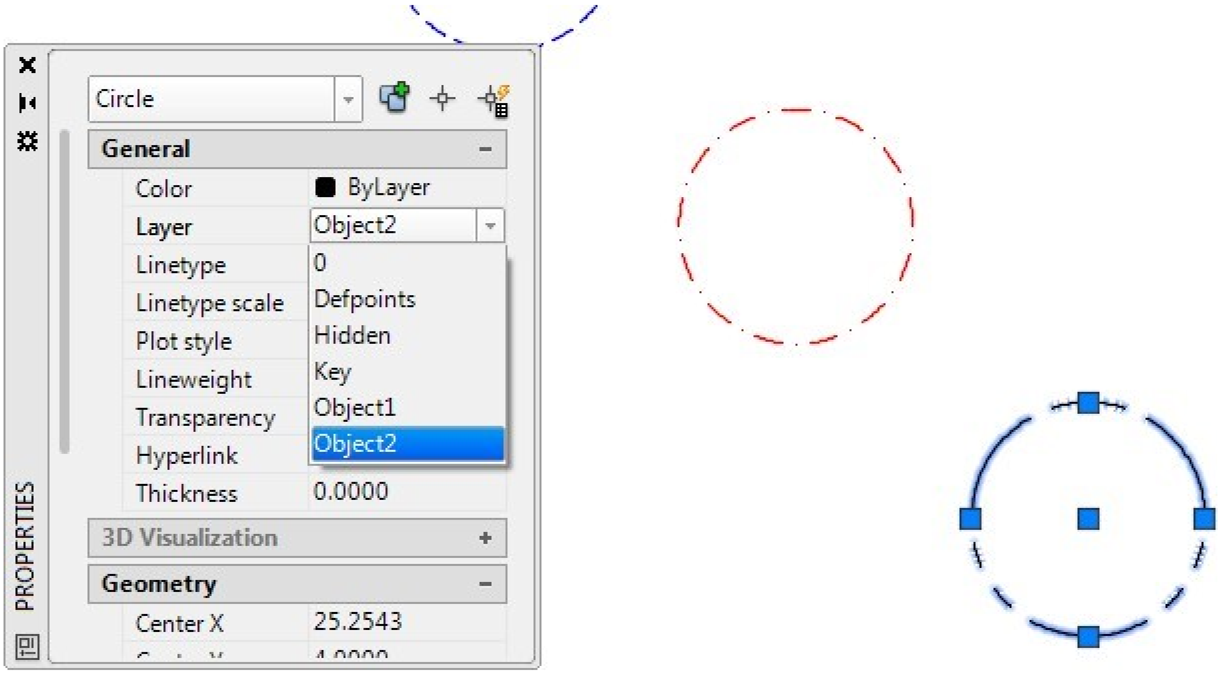
Center X 22.7543

PROPERTIES
Linetype
Linetype scale
Plot style
Lineweight
Transparency
Hyperlink
0
Defpoints
Hidden
Key
Object1
Object2
3D Visualization

Figure Step 14

Step 15

Select the bottom right circle. In the Properties window, change it to layer: Object 2. Your drawing should now appear as shown in figure. (Figure Step 15A and 15B)



PROPERTIES
Circle

General

Color	ByLayer
Layer	Object2
Linetype	
Linetype scale	
Plot style	
Lineweight	
Transparency	
Hyperlink	
Thickness	0.0000

0
Defpoints
Hidden
Key
Object1
Object2

3D Visualization

Geometry

Center X	25.2543

Figure Step 15A

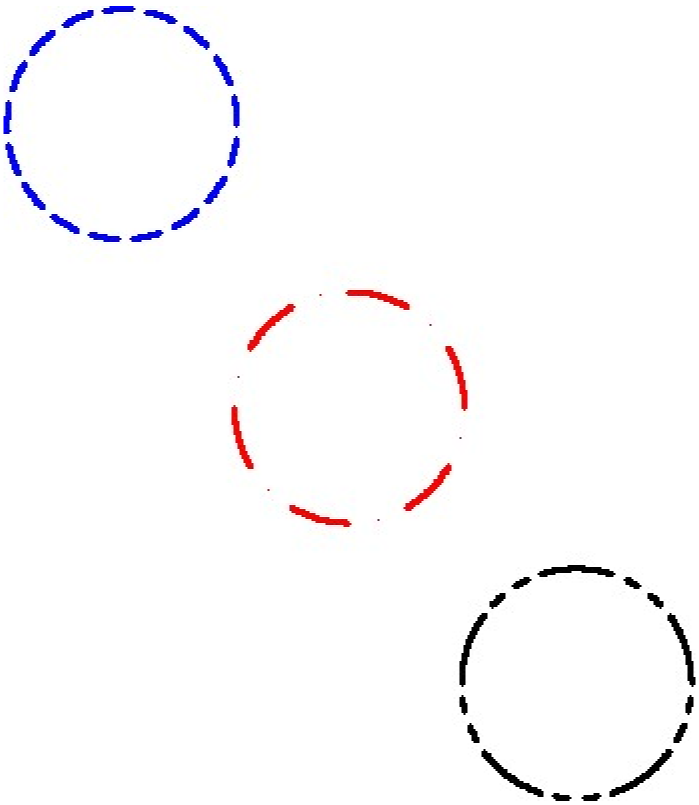
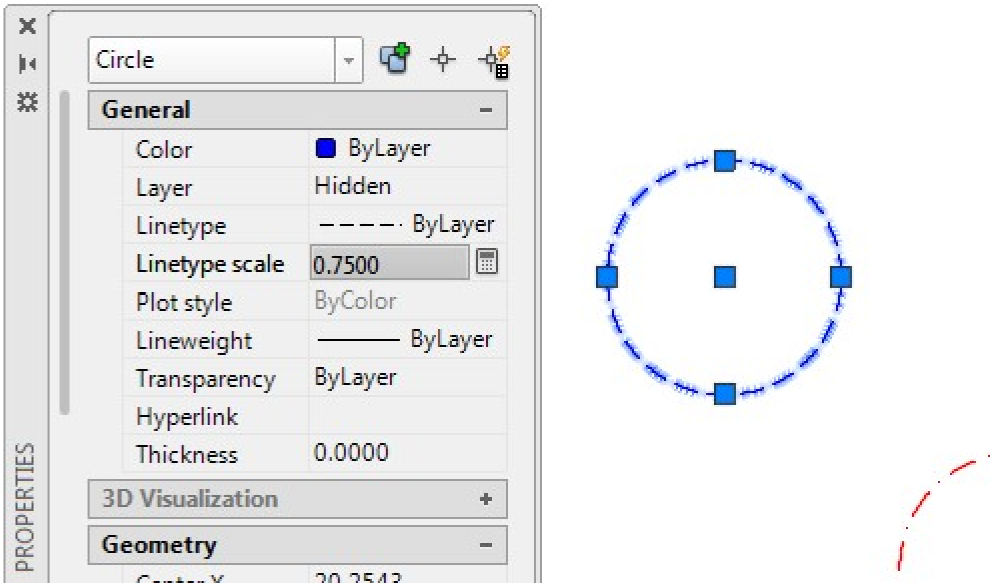


Figure Step 15B

Step 16

Select the top left circle. In the Properties window, change the Linetype scale property to 0.750. (Figure Step 16)



Circle

General

Color	ByLayer
Layer	Hidden
Linetype	ByLayer
Linetype scale	0.7500
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	
Thickness	0.0000

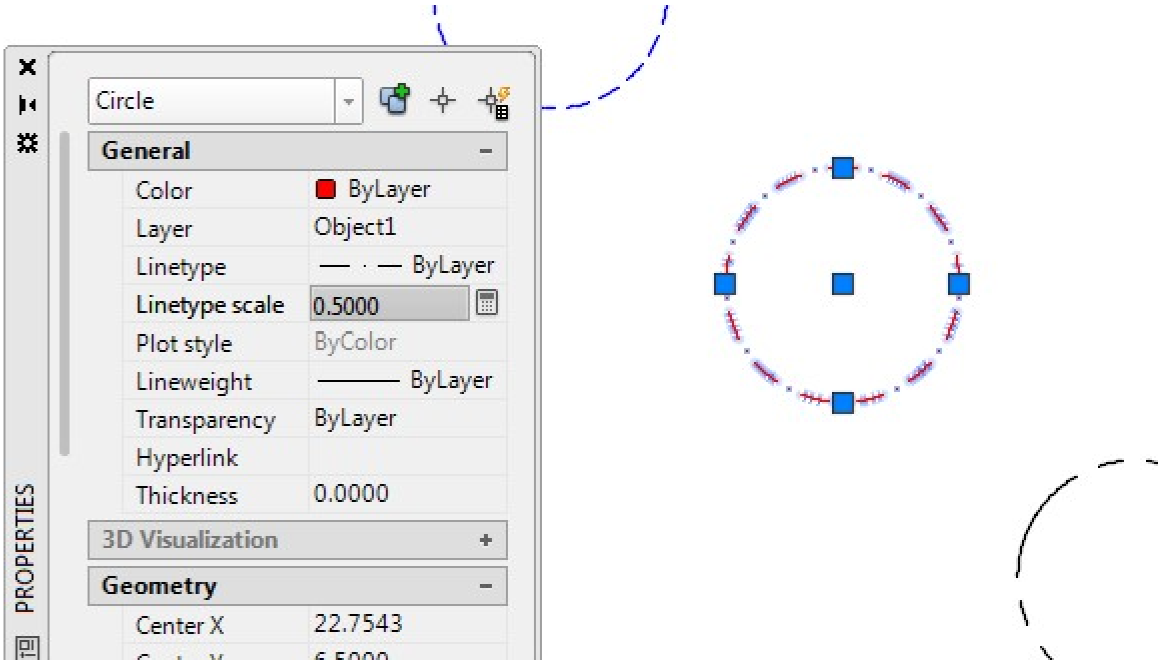
3D Visualization

Geometry

Figure Step 16

Step 17

Select the middle circle. In the Properties window, change the Linetype scale property to 0.500. (Figure Step 17)



Circle

General

Color	ByLayer
Layer	Object1
Linetype	ByLayer
Linetype scale	0.5000
Plot style	ByColor
Lineweight	ByLayer
Transparency	ByLayer
Hyperlink	
Thickness	0.0000

3D Visualization

Geometry

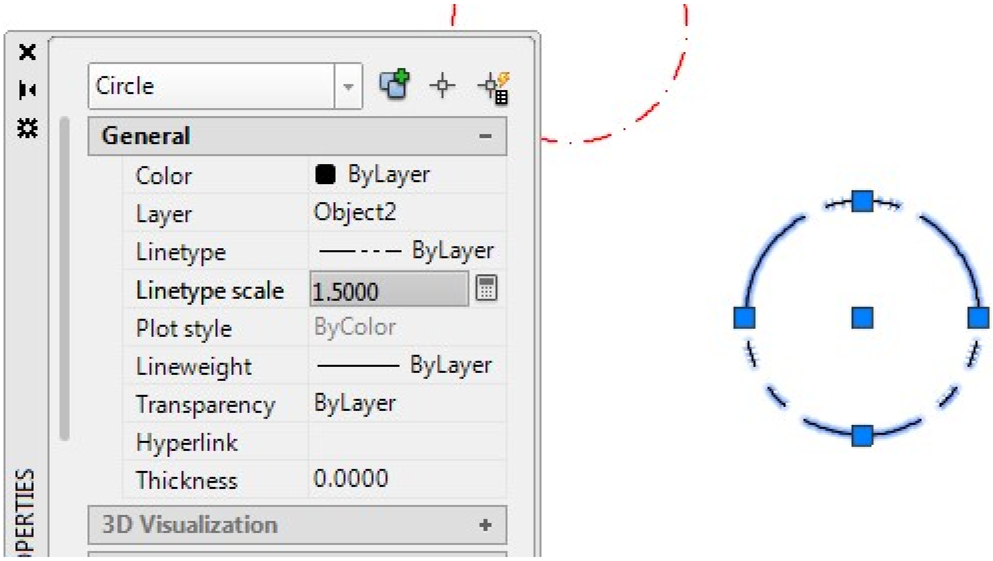
Center X	22.7543

Figure Step 17

USER TIP: Understanding and using object properties to edit and inquire about drawing objects is very important to your drawing speed and production. Whenever possible, use the Properties window to edit objects rather than using AutoCAD commands to do the same thing.

Step 18

Select the bottom circle. In the Properties window, change the Linetype scale property to 1.500. (Figure Step 18)



Color	ByLayer
Layer	Object2
Linetype	——- - - — ByLayer
Linetype scale	1.5000
Plot style	ByColor
Lineweight	———— ByLayer
Transparency	ByLayer
Hyperlink	
Thickness	0.0000

Figure Step 18

AUTHOR'S COMMENTS: You can see that by changing the Linetype scale property of the object, It changes it's appearance.

Step 19

Enter the LTSCALE system variable as shown below and ensure that it is set to 1.

Command: **LTSCALE**

Enter new linetype scale factor <1.0000>:

Command:

AUTHOR'S COMMENTS: If it is not set to 1, enter 1 to the prompt.

AUTHOR'S COMMENTS: If possible, always leave the LTSCALE system variable set to 1.

Step 20

Your completed drawing should appear as shown in the figure. (Figure Step 20)

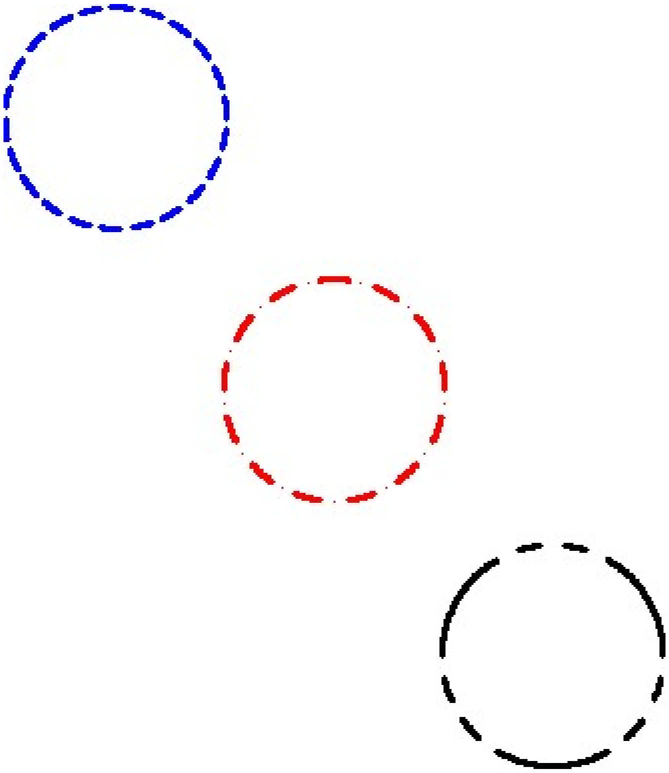


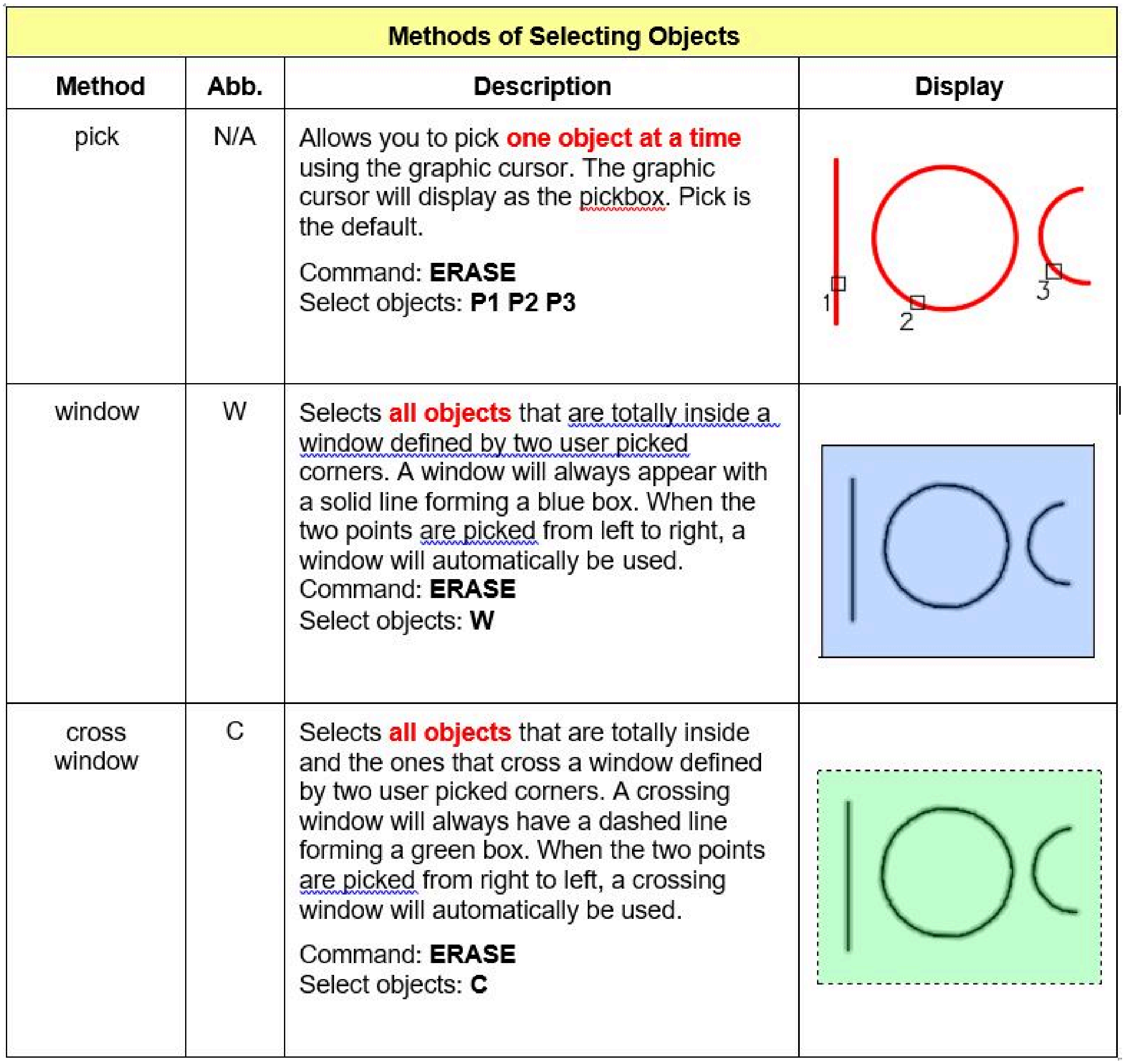
Figure Step 20

Step 21

Save and close the drawing.

Selecting Objects

After entering a command that requires you to select one or more objects, AutoCAD will display the Select object: prompt. To draw faster, it is important for you to be able to select objects in the most time efficient method since many commands require object selection. There are many methods of selecting objects and they will be taught in future module. In this module, the pick, window, and crossing window methods are taught. See Figure 13-4.



Methods of Selecting Objects			
Method	**Abb.**	**Description**	**Display**
pick	N/A	Allows you to pick **one object at a time** using the graphic cursor. The graphic cursor will display as the pickbox. Pick is the default. Command: **ERASE** Select objects: **P1 P2 P3**	1 2 3
window	W	Selects **all objects** that are totally inside a window defined by two user picked corners. A window will always appear with a solid line forming a blue box. When the two points are picked from left to right, a window will automatically be used. Command: **ERASE** Select objects: **W**	
cross window	C	Selects **all objects** that are totally inside and the ones that cross a window defined by two user picked corners. A crossing window will always have a dashed line forming a green box. When the two points are picked from right to left, a crossing window will automatically be used. Command: **ERASE** Select objects: **C**	

Figure 13-4 Selecting Objects Using Pick, Window, and Cross Window

WORKALONG: Selecting Objects

Step 1

Open the drawing: AutoCAD 2D Lab 12-2. (Figure Step 1)

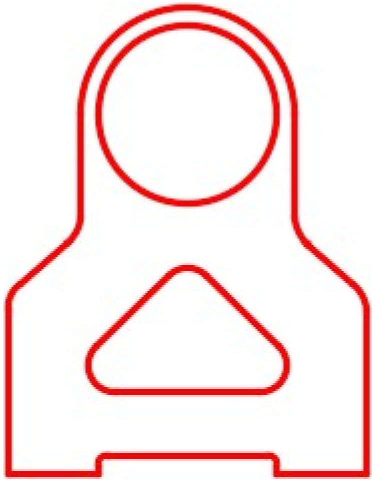


Figure Step 1

Step 2

Using the SAVEAS command, save and name the drawing:

AutoCAD 2D Workalong 13-2.

Step 3

Enter the ERASE command as shown below. When you are prompted to Select objects, select the circle with the pickbox and press the Enter key. (Figure Step 3A and 3B)

Command: **ERASE**

Select objects:

Command:

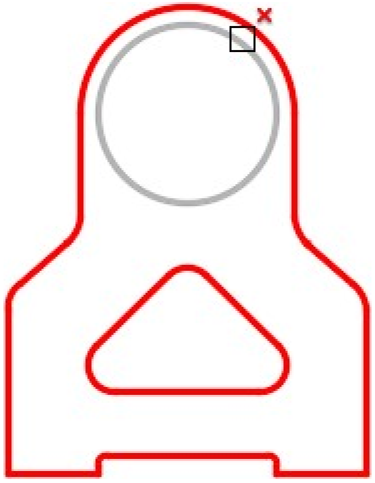


Figure Step 3A

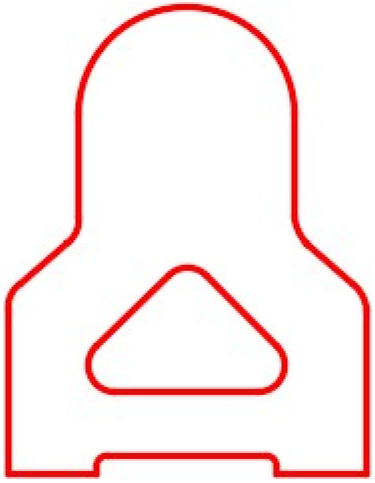


Figure Step 3B

AUTHOR'S COMMENTS: It is important to understand that when you select objects by selecting them with the pickbox, only one object is selected per pick. If there are more then one circle with the same diameter at the same location, there would still be a circle remaining after you execute the ERASE command.

Step 4

Enter the ERASE command. When prompted to Select objects, select two locations to form a window. You must select the locations on opposite corners from left to right. A window is colored blue and has a solid line around it. Press Enter to execute the command. (Figure Step 4A and 4B)

AUTHOR'S COMMENTS: A window must be selected from left to right and will select only the objects that are located totally inside the window. In this case, the arc. The lines that are not totally inside the window will not be selected.

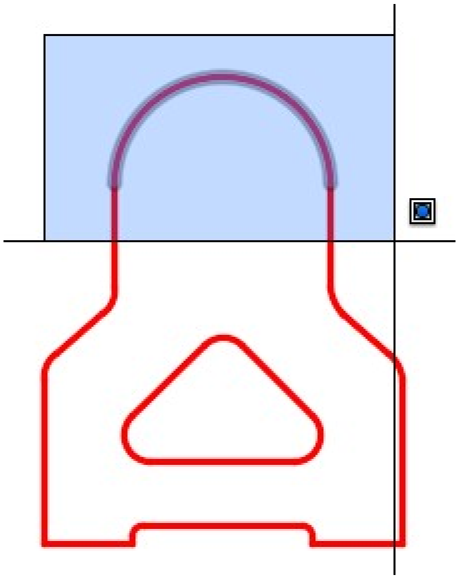


Figure Step 4A

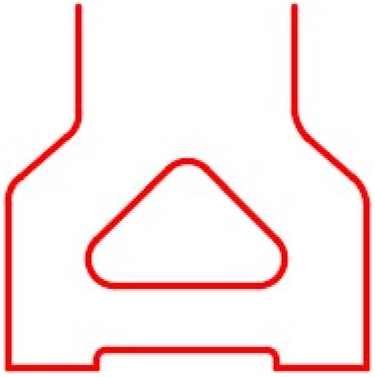


Figure Step 4B

AUTHOR'S COMMENTS: It is important to understand that when you select objects using a window, all of the objects that are totally inside the window are selected. If there is more then one arc with the same radius at the same location, all of them would also be erased.

Step 5

Enter the U command to undo that last command. The arc you just erased will redisplay on the drawing. (Figure Step 5)

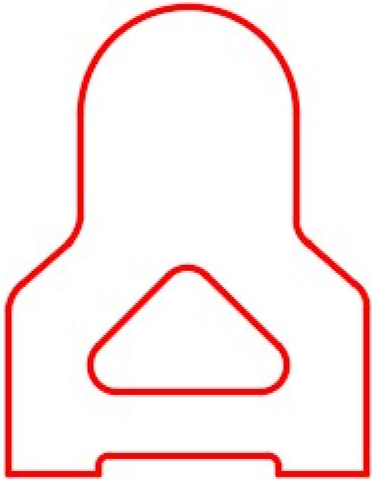


Figure Step 5

MUST KNOW: It is very important to understand how objects are selected. If you use the pick method to select the objects, AutoCAD will select only **one** object, even though two or more objects are located exactly at the same location. If you select the objects using either the window or cross window method, AutoCAD will select **all** of the objects, even though two or more objects are located exactly at the same location.

Step 6

Enter the ERASE command. When you are prompted with the Select objects prompt, select two locations moving from right to left. The two locations must be the opposite corners of the crossing window. Press Enter to execute the command. (Figure Step 6A and 6B)

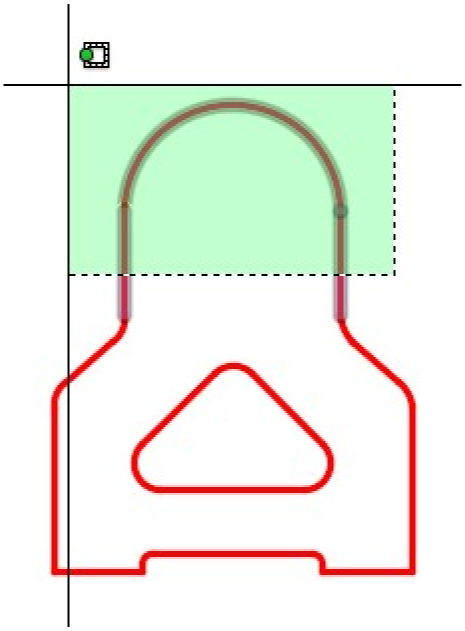


Figure Step 6A

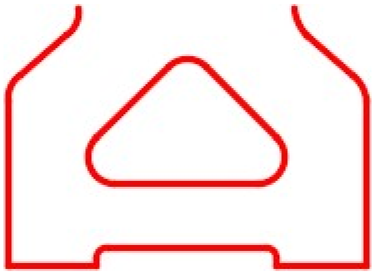


Figure Step 6B

AUTHOR'S COMMENTS: A crossing window must be selected from right to left and will select all of the objects that the window crosses and all of the objects that are located totally inside the crossing window. In this case, the arc and the two lines.

Step 7

Enter the U command as many times as required to display the drawing as shown in the figure. (Figure Step 7)

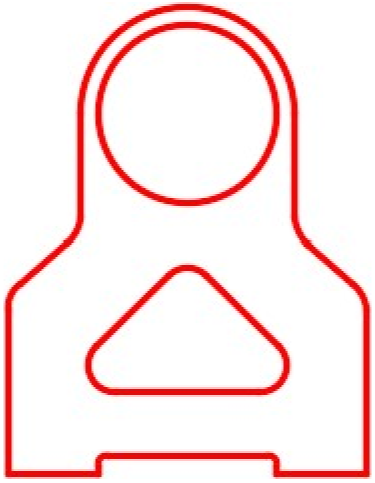


Figure Step 7

MUST KNOW: It is very important to understand how objects are selected. If you use the pick method to select the objects, AutoCAD will select only **one** object, even though two or more objects are located exactly at the same location. If you select the objects using either the window or cross window method, AutoCAD will select **all** of the objects, even though two or more objects are located exactly at the same location.

Step 8

Without entering a command and using what you learned earlier, place a window as shown in the figure. All of the objects that are totally inside the window will highlight and display grips. Press the Delete key to delete the objects. (Figure Step 8A, 8B, and 8C)

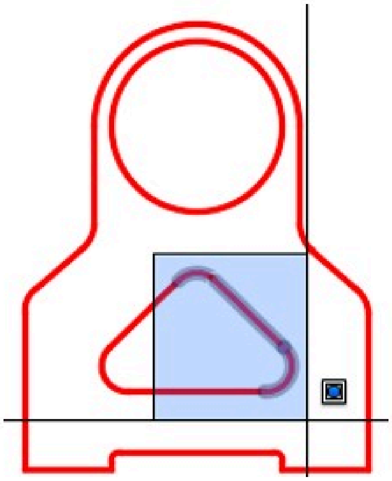


Figure Step 8A

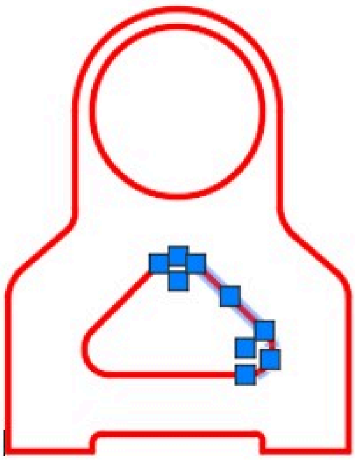


Figure Step 8B

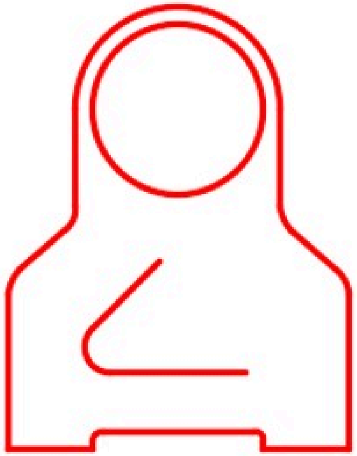


Figure Step 8C

Step 9

Enter the U command to undo that last command. The arcs and line you just erased will redisplay on the drawing.

Step 10

Without entering a command and using what you learned earlier, place a crossing window as shown in the figure. All of the objects that cross the window and all of the objects that are totally inside the window will be selected and display grips. Press the Delete key to delete the objects. (Figure Step 10A, 10B, and 10C)

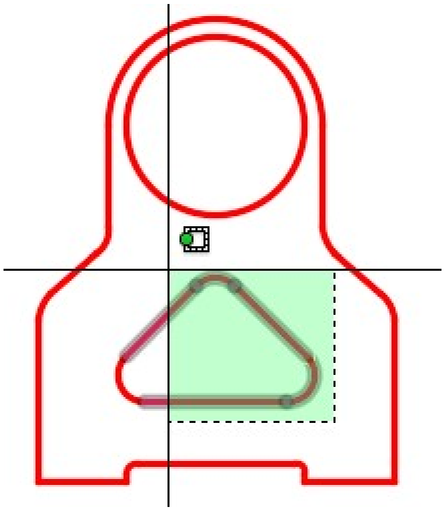


Figure Step 10A

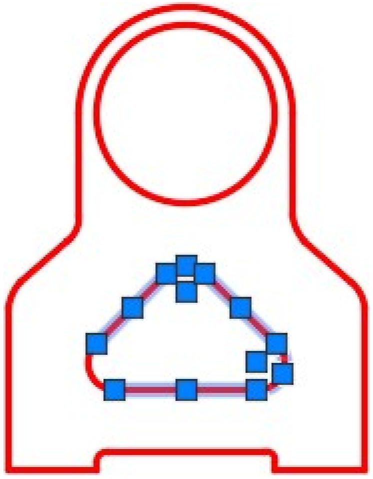


Figure Step 10B

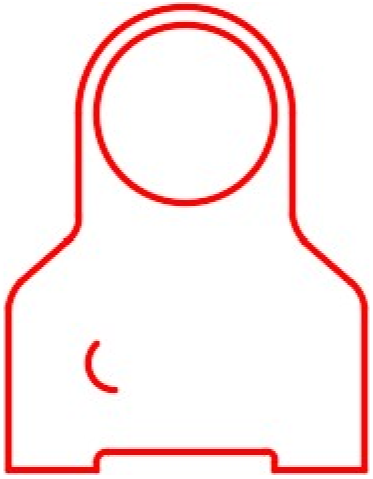


Figure Step 10C

Step 11

Enter the U command to undo that last command.

Step 12

Save and close the drawing.

Geometry Lesson: A Compound Fillet

A *compound fillet* is two or more tangent arcs used to create a continuous curve. Study Figure 13-5. Note the small dashed line. It is a drafting standard to indicate that the 3 R (Radius) arc is tangent to the line and small circle as shown in Step 1. The 1 R tangent arc is then inserted tangent to 3 R arc and the large circle as shown in Step 2. Since they are tangent arcs, you would use the FILLET command, two times, to draw them.

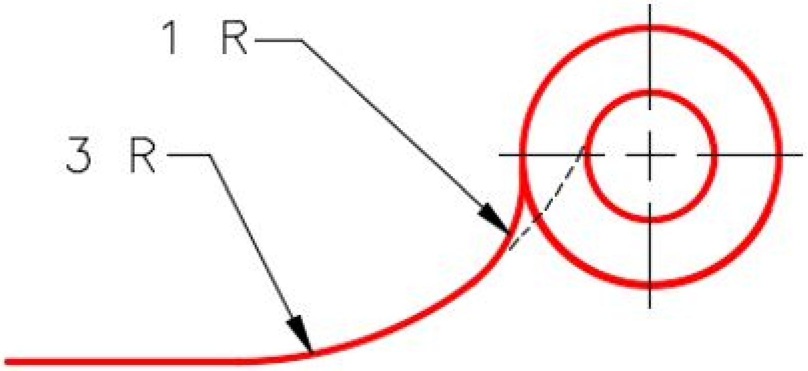
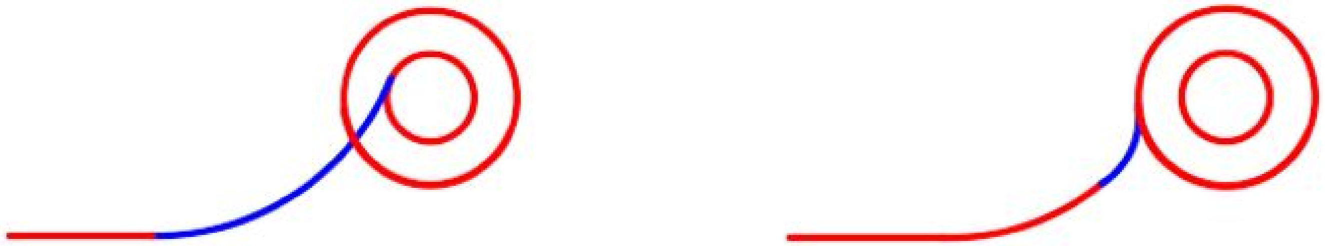


Figure 13-5
A Compound Fillet



Step 1 *Step 2*

USER TIP: Always leave the setting for LTSCALE to 1 and set the drawing object's linetype scale property to the setting to display the desired linetype.

Key Principles

Key Principles in Module 13

1. AutoCAD stores all existing drawing objects and their properties in a database file called the drawing file. The name of the file is the drawing name with the extension .dwg.
2. A linetype is a repeating pattern of dots, dashes, text, objects, or blank spaces displayed on the object. It is an appearance only and does not change the object in any way.
3. The individual linetype scale property multiplies the scale factor it is set to times the existing scale factor of the global linetype scale (LTSCALE). It is best to set LTSCALE to 1 at all times.
4. If you use the pick method to select the objects, AutoCAD will select only one object, even though two objects are exactly located at the same location. If the same objects are selected using either the window or cross window method, AutoCAD will select all objects even if they are at the exact same location.

Lab Exercise 13-1

Time allowed: 30 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 13-1	2D English	Inches

Layering Scheme

Layer Name	Objects on Layer	Color
Construction	Construction objects	253
Object	All objects	Red

Step 1

Draw the object shown in the figure using the layering scheme. (Figure Step 1A and 1B)

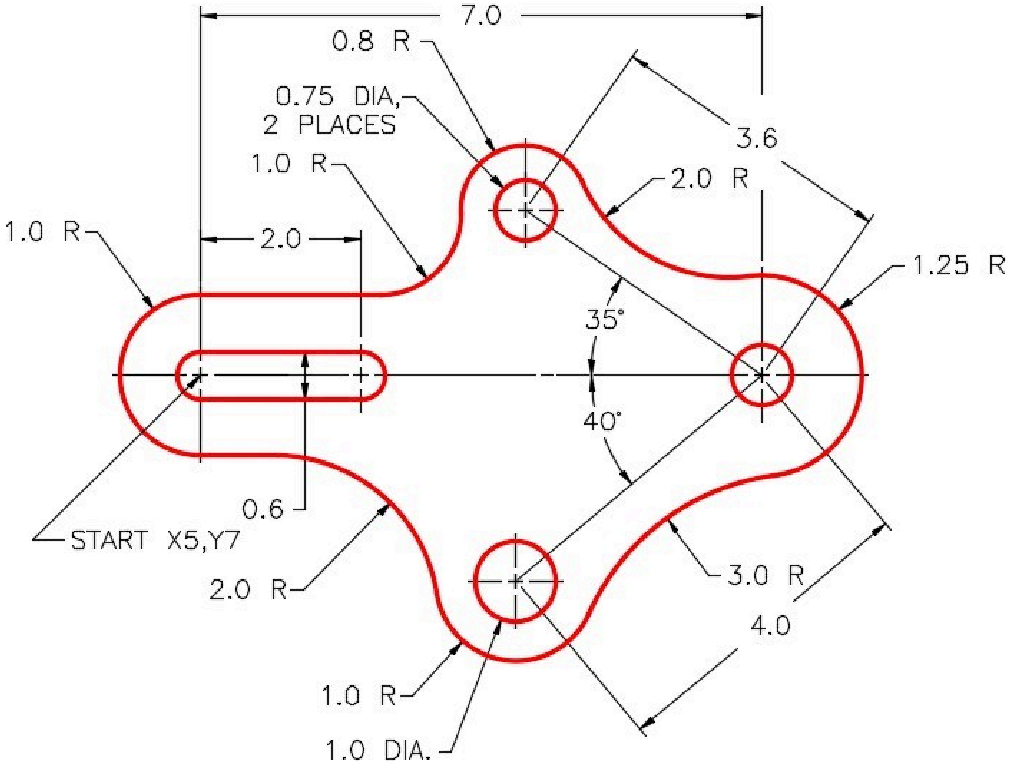

7.0
0.8 R
0.75 DIA,
2 PLACES
1.0 R
3.6
2.0 R
1.0 R
2.0
1.25 R
35°
40°
START X5,Y7
0.6
2.0 R
3.0 R
4.0
1.0 R
1.0 DIA.

Figure Step 1A

AUTHOR'S COMMENTS: Do not delete the construction lines.

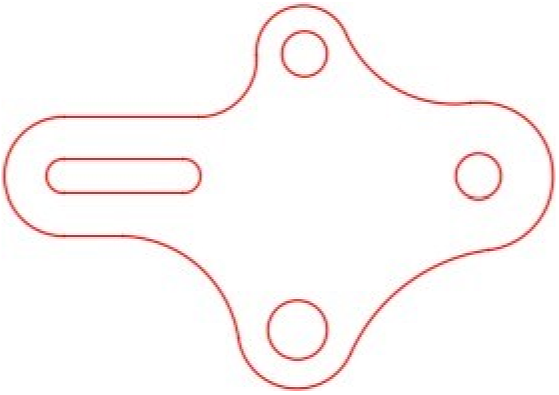


Figure Step 1B
Completed Drawing

Step 2

Set the insertion units and check your drawing with the key.

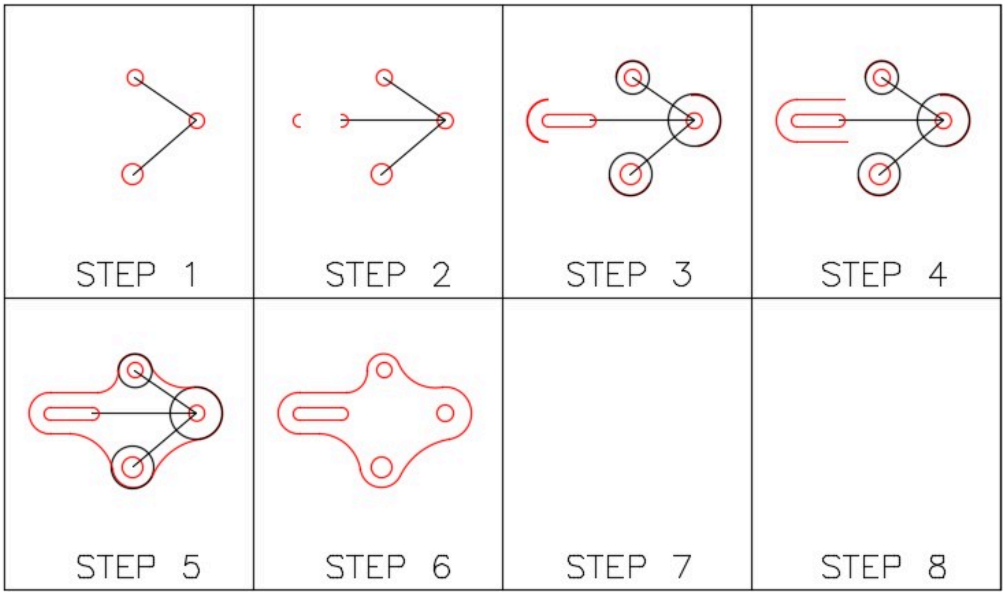
Step 3

Turn layer: Key off and freeze layer: Construction.

Step 4

Save and close the drawing.

AUTHOR'S CONSTRUCTION TECHNIQUES: The following steps are the construction technique suggested by the author to help you learn how to construct objects using AutoCAD. It is only the suggested method and if you can complete the drawing accurately using a different construction technique, that is what is important. You may want to compare your construction technique with the authors.



AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

See the following step:

Step 1

Draw the lines, circles and arcs as shown in the figure. (Figure Hint Step 1)

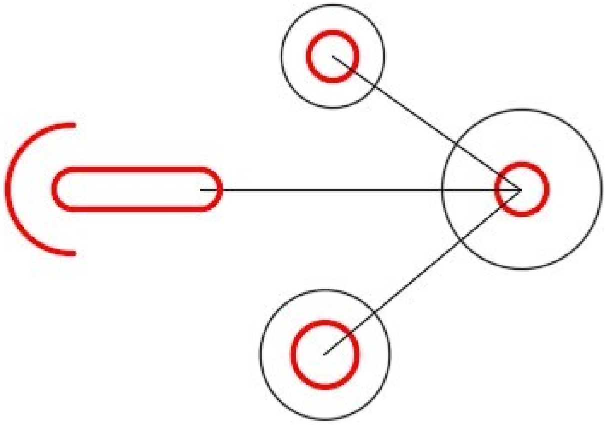


Figure Hint Step 1

Step 2

Draw two horizontal lines, of any length, starting at the ends of the arc on the left. (Figure Hint Step 2)

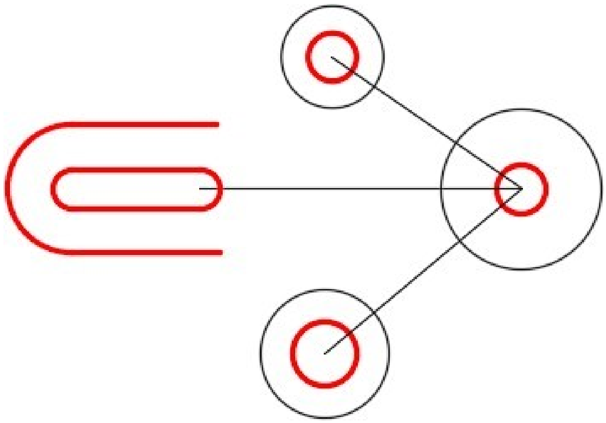


Figure Hint Step 2

Step 3

Insert a 1 radius fillet between the horizontal line and the circle. (Figure Hint Step 3)

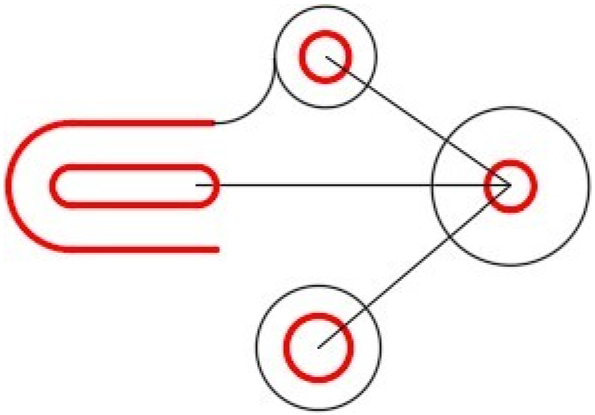


Figure Hint Step 3

Step 4

Insert a 2 radius fillet between the two circles on the top right and another one between the horizontal line and circle on the bottom left. (Figure Hint Step 4)

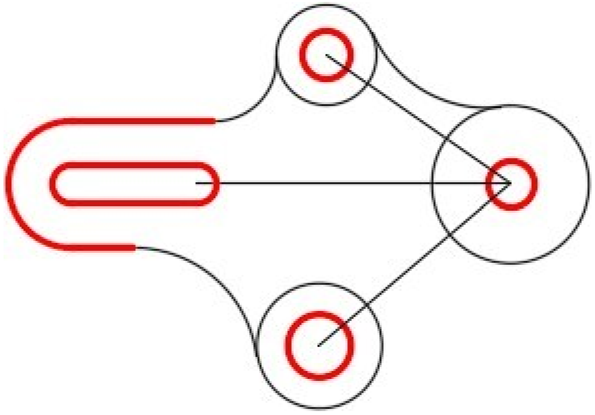


Figure Hint Step 4

Step 5

Insert a 3 radius fillet between the two circles on the bottom right. (Figure Hint Step 5)

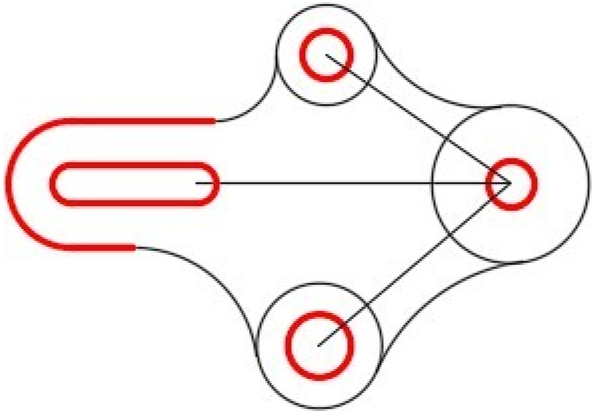


Figure Hint Step 5

Step 6

Using the Properties window, change the layer of the arcs to layer: Object. (Figure Hint Step 6)

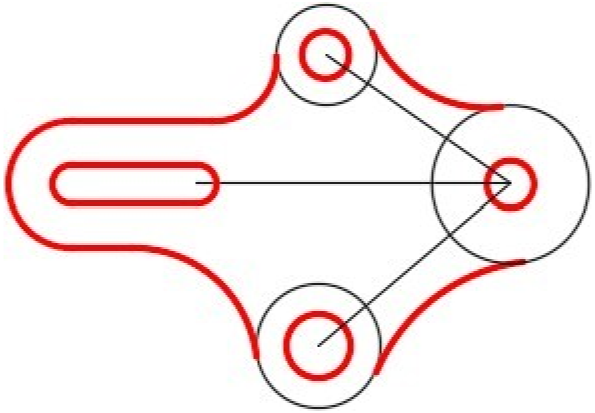


Figure Hint Step 6

Step 7

Freeze layer: Construction. (Figure Hint Step 7)

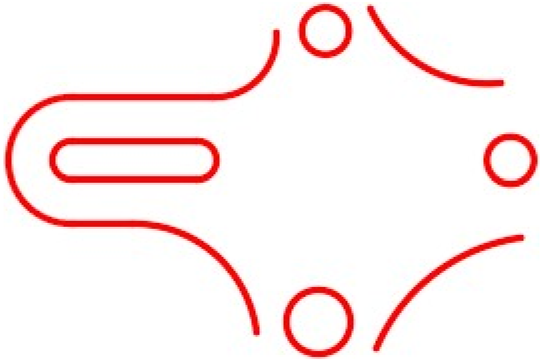


Figure Hint Step 7

Step 8

Using the FILLET command, insert three fillets, using the applicable radius, between the arcs. (Figure Hint Step 8)

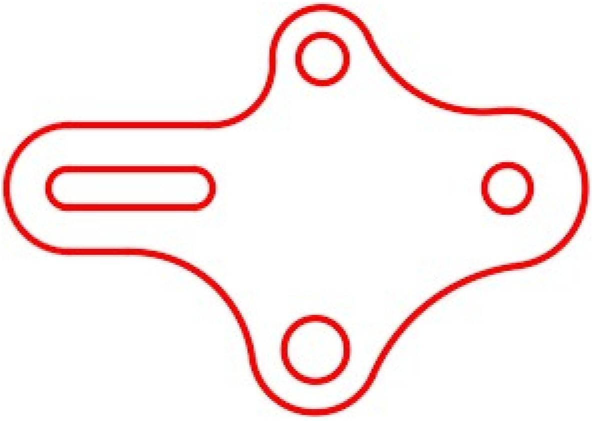


Figure Hint Step 8

Lab Exercise 13-2

Time allowed: 20 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 13-2	N/A	N/A

Layering Scheme

Layer Name	**Objects on Layer**	**Color**	**Line Type**
Outline	Outline of object	Red	Phantom 2
Circle	Full circles	Green	Border 2
Slot	The slot	Blue	Dashed 2

Step 1

Open the drawing: AutoCAD 2D Lab 13-1.

Step 2

Using the SAVEAS command, save the drawing with the name: AutoCAD 2D Lab 13-2.

Step 3

Open the Layer Property Manager window and change the linetypes of the layers to match the layering scheme shown above.

Step 4

Using the Properties window, change the layers of the objects to match the layering scheme shown above.

Step 5

Change the linetype scale of each object to match the figure. (Figure Step 5)

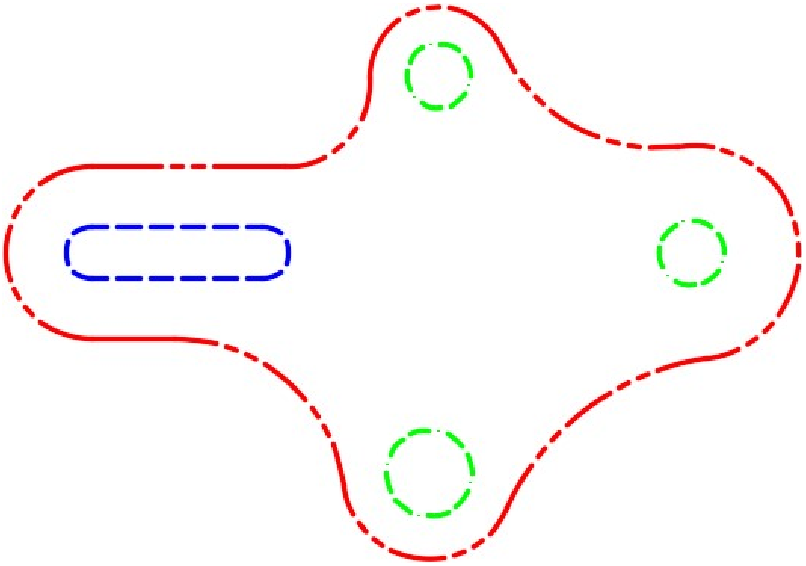


Figure Step 5

AUTHOR'S COMMENTS: You will have to guess at the linetype scale and then adjust it. Do your best to match the figure, by eye, as closely as you can.

Lab Exercise 13-3

Time allowed: 30 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 13-3	2D Metric	Millimeters

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Construction	Construction objects	253
Object	All objects	Red

Step 1

Draw the object shown in the figure using the layering scheme. (Figure Step 1A and 1B)

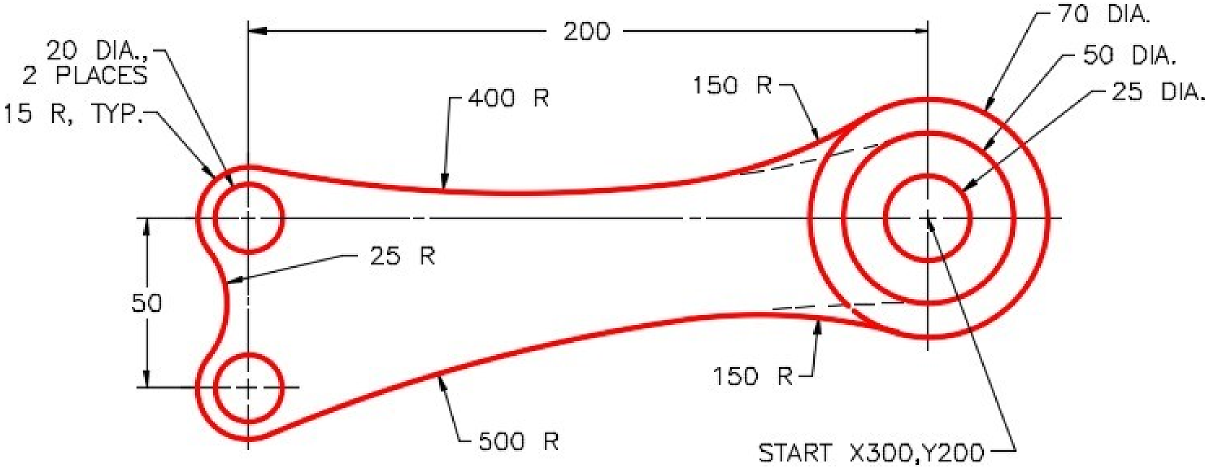


Figure Step 1A

AUTHOR'S COMMENTS: Do not delete the construction lines.

Step 2

Set the insertion units and check your drawing with the key.

Step 3

Turn layer: Key off and freeze layer: Construction.

Step 4

Save and close the drawing.

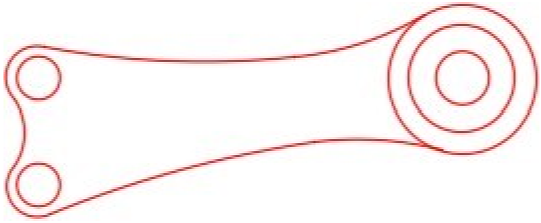
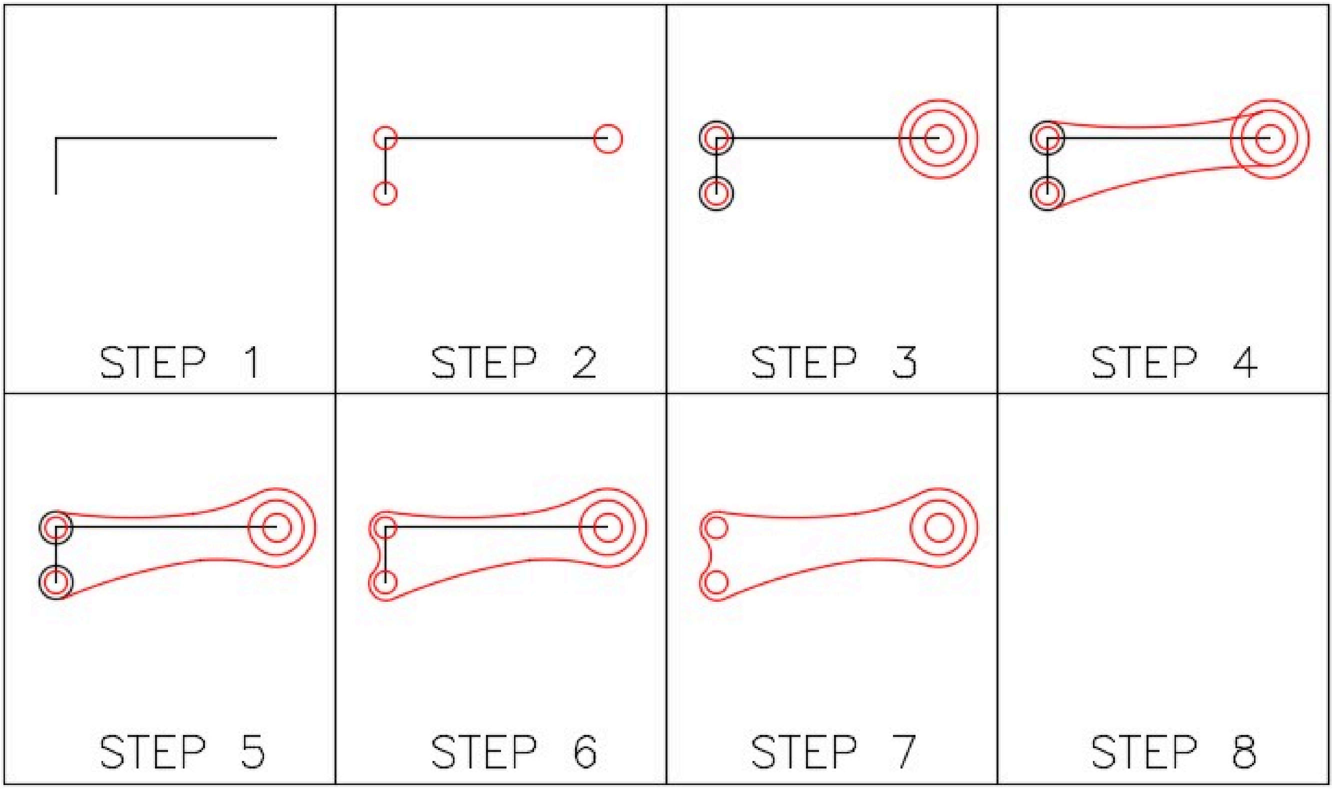


Figure Step 1B
Completed Drawing

AUTHOR'S CONSTRUCTION TECHNIQUES: The following steps are the construction technique suggested by the author to help you learn how to construct objects using AutoCAD. It is only the suggested method and if you can complete the drawing accurately using a different construction technique, that is what is important. You may want to compare your construction technique with the authors.


STEP 1
STEP 2
STEP 3
STEP 4
STEP 5
STEP 6
STEP 7
STEP 8

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

See the following steps:

Step 1

Draw the lines and circles as shown in the figure. (Figure Hint Step 1)

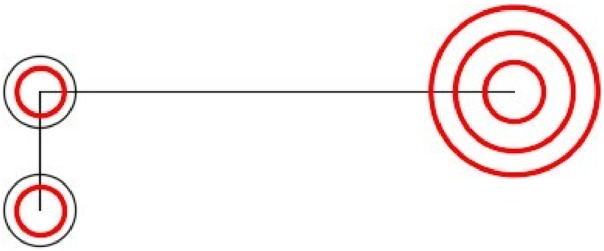


Figure Hint Step 1

Step 2

Enter the FILLET command and set the radius to 400. For the first object, select the large circle on the left. For the second object, select the middle circle on the right. (Figure Hint Step 2A, 2B, and 2C)

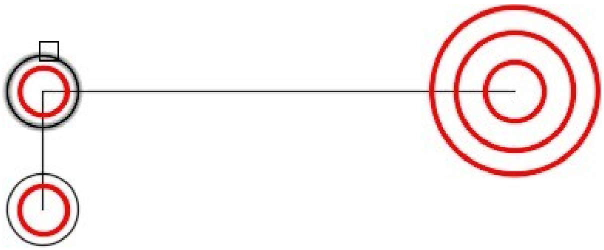


Figure Hint Step 2A

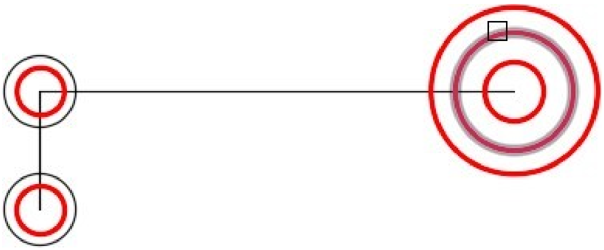


Figure Hint Step 2B

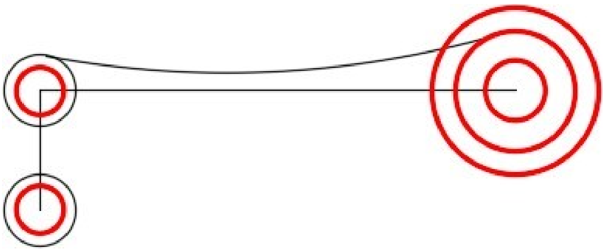


Figure Hint Step 2C

Step 3

Using what you learned in Step 2, insert the compound bottom fillet. You will have to change the radius to 500. (Figure Hint Step 3)

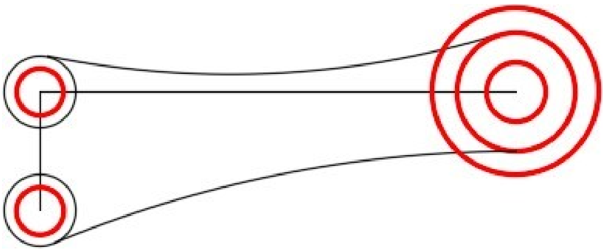


Figure Hint Step 3

Step 4

Enter the FILLET command and set the radius to 150. For the first object, select the fillet inserted in Step 2. For the second object, select the large circle on the right. Figure Hint Step 4A, 4B, and 4C)

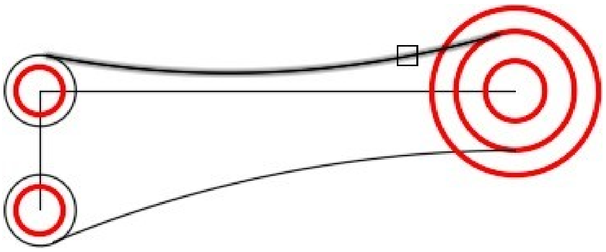


Figure Hint Step 4A

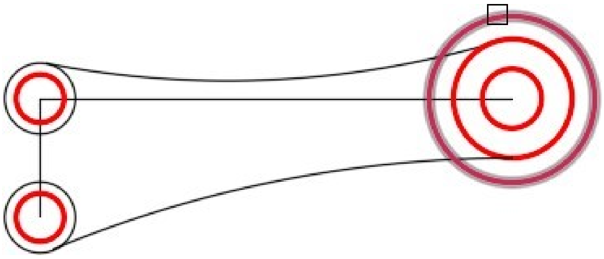


Figure Hint Step 4B

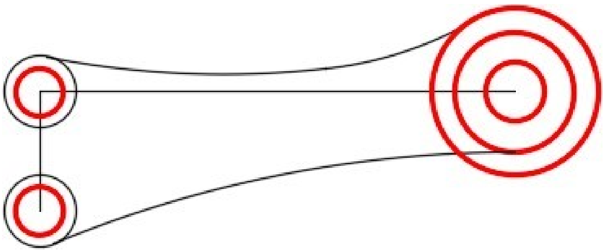


Figure Hint Step 4C

Step 5

Using what you learned in Step 4, insert the fillet on the bottom. (Figure Hint Step 5)

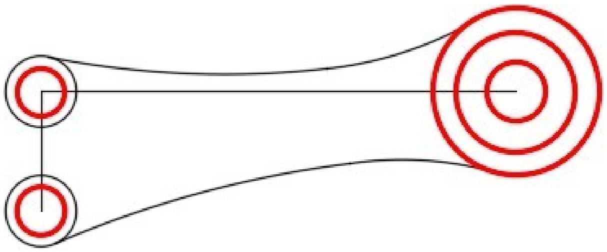


Figure Hint Step 5

Step 6

Insert a 25 radius fillet between the circles on the left. (Figure Hint Step 6)

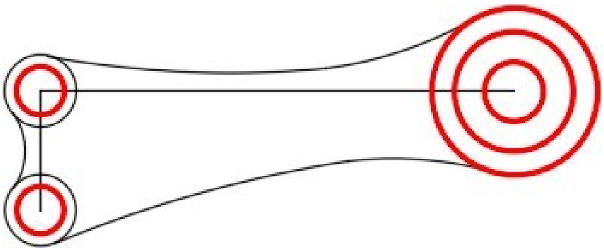


Figure Hint Step 6

Step 7

Change the layer of the fillets to layer: Object. (Figure Hint Step 7)

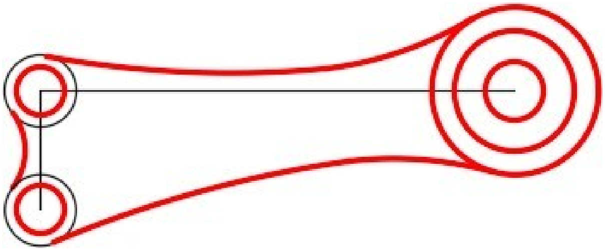


Figure Hint Step 7

Step 8

Freeze layer: Construction. (Figure Hint Step 8)

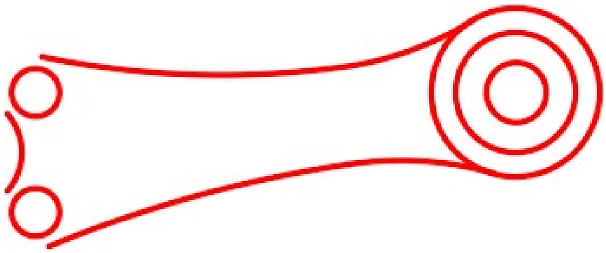


Figure Hint Step 8

Step 9

Insert two 15 radius fillets between the arcs on the left. (Figure Hint Step 9)

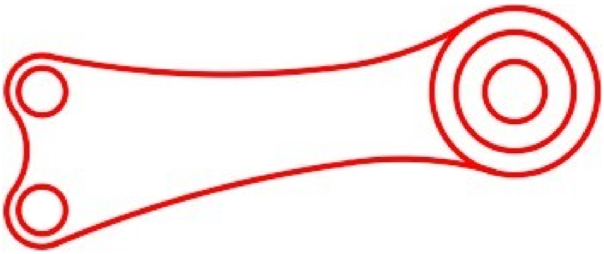


Figure Hint Step 9

Lab Exercise 13-4

Time allowed: 20 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 13-4	N/A	N/A

Layering Scheme

Layer Name	**Objects on Layer**	**Color**	**Line Type**
Circles	All circles	Blue	Dashed
Arcs	All arcs	Magenta	Phantom

Step 1

Open the drawing: AutoCAD 2D Lab 13-3.

Step 2

Using the SAVEAS command, save the drawing with the name: AutoCAD 2D Lab 13-4.

Step 3

Open the Layer Property Manager window and change the linetypes of the layers to match the layering scheme shown above.

Step 4

Using the Properties window, change the layers of the objects to match the layering scheme shown above.

Step 5

Change the linetype scale of each object to match the figure. (Figure Step 5)

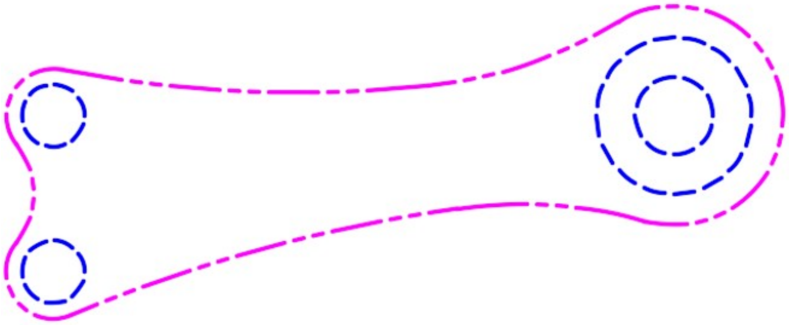


Figure Step 5

AUTHOR'S COMMENTS: You will have to guess at the linetype scale and then adjust it. Do your best to match the figure, by eye, as closely as you can.

Module 14: Trimming and Extending

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe the geometry of regular polygons.
2. Apply the TRIM and EXTEND commands to shorten and lengthen drawing objects at selected cutting edges.

Geometry Lesson: Regular Polygons

A *polygon* is defined as any plane figure bounded by straight lines. A *regular polygon* is a polygon that has equal angles, equal sides, and can be inscribed or circumscribed around a circle. AutoCAD has a command to automatically construct polygons and it will be taught in a future module. For now, it is important for you to be able to construct polygons using geometry.

The first nine regular polygons are shown in Figure 14-1.

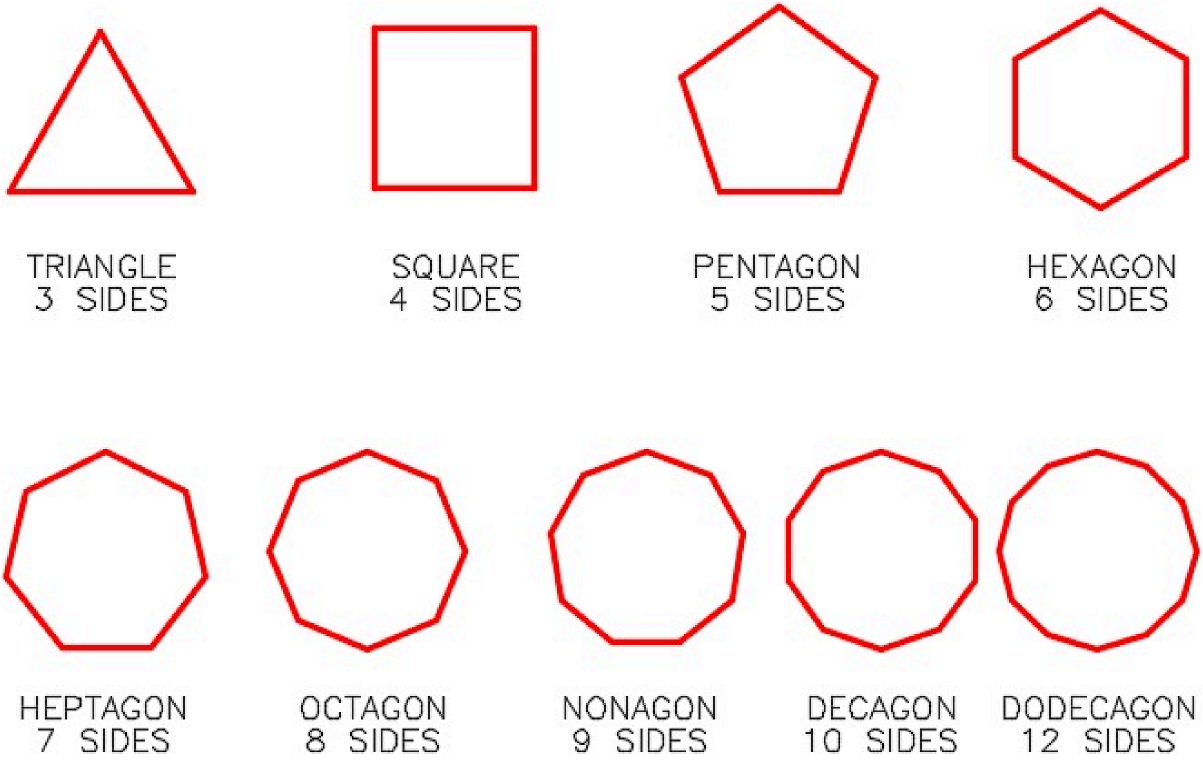

TRIANGLE
3 SIDES
SQUARE
4 SIDES
PENTAGON
5 SIDES
HEXAGON
6 SIDES
HEPTAGON
7 SIDES
OCTAGON
8 SIDES
NONAGON
9 SIDES
DECAGON
10 SIDES
DODECAGON
12 SIDES

Figure 14-1
Regular Polygons

Any regular polygon can be inscribed in or circumscribed around a circle as shown, using a hexagon, in Figure 14-2.

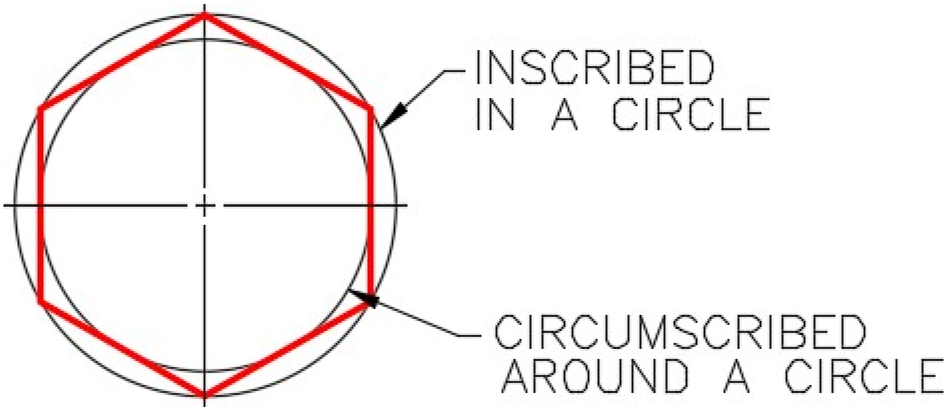

INSCRIBED
IN A CIRCLE
CIRCUMSCRIBED
AROUND A CIRCLE

Figure 14-2
Inscribed and Circumscribed Regular Polygon

It is important to understand and know the geometry involved to construct a triangle, square, hexagon, and octagon as shown in Figure 14-3. Study each one and try to understand how they are constructed and the angles used to construct them.

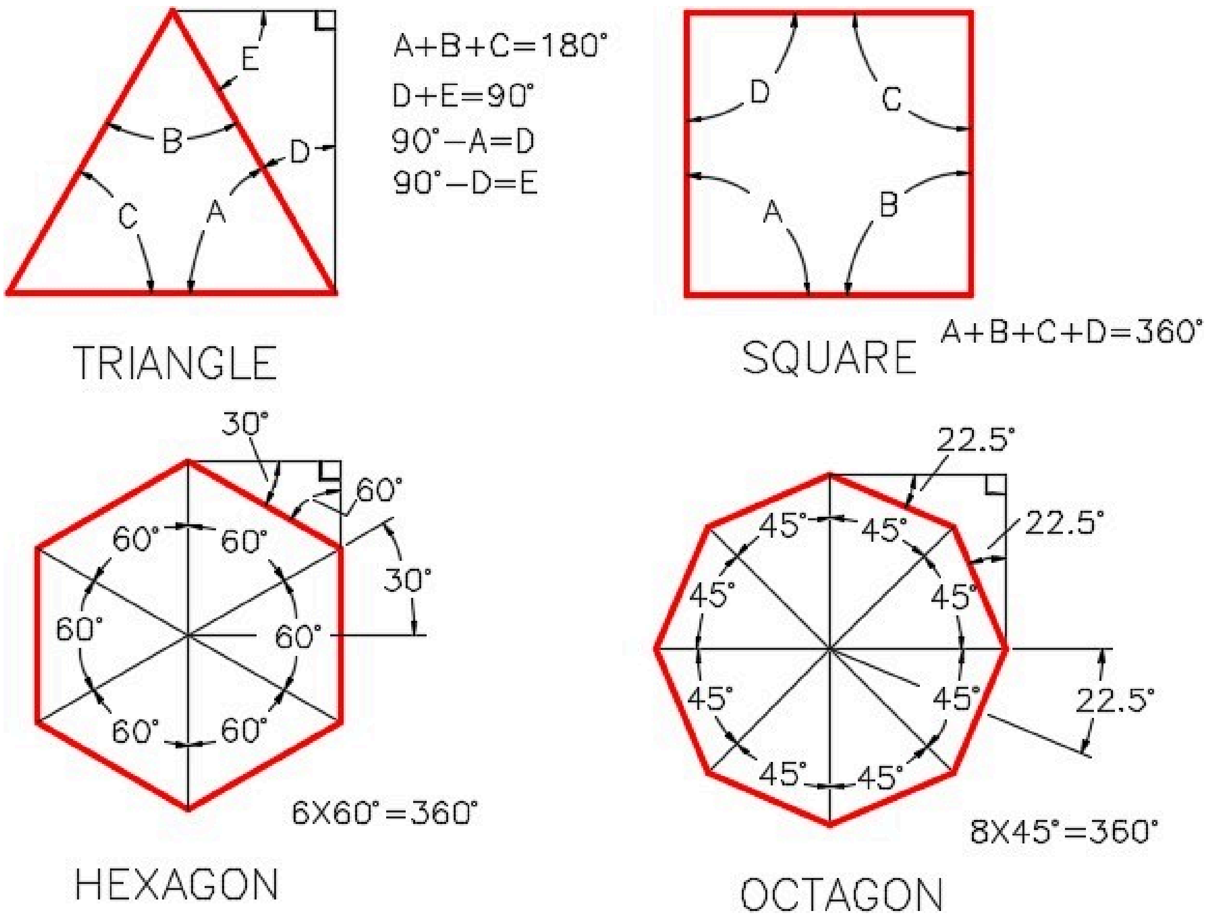

E
A+B+C=180°
D+E=90°
90°−A=D
90°−D=E
B
D
C
A
TRIANGLE
D
C
A
B
SQUARE
A+B+C+D=360°
30°
60°
60°
60°
30°
60°
60°
60°
60°
60°
6X60°=360°
HEXAGON
22.5°
45°
45°
22.5°
45°
45°
45°
45°
22.5°
45°
45°
8X45°=360°
OCTAGON

Figure 14-3
Geometry of Four Regular Polygons

The following are the steps used to draw a hexagon in an AutoCAD drawing. See Figure 14-4.

Step 1

Draw 6 lines on layer Construction from the center to each corner, 60 degrees apart.

Step 2

Draw six lines snapping from the end of one line to the end of the next line.

Step 3

Turn layer Construction off.

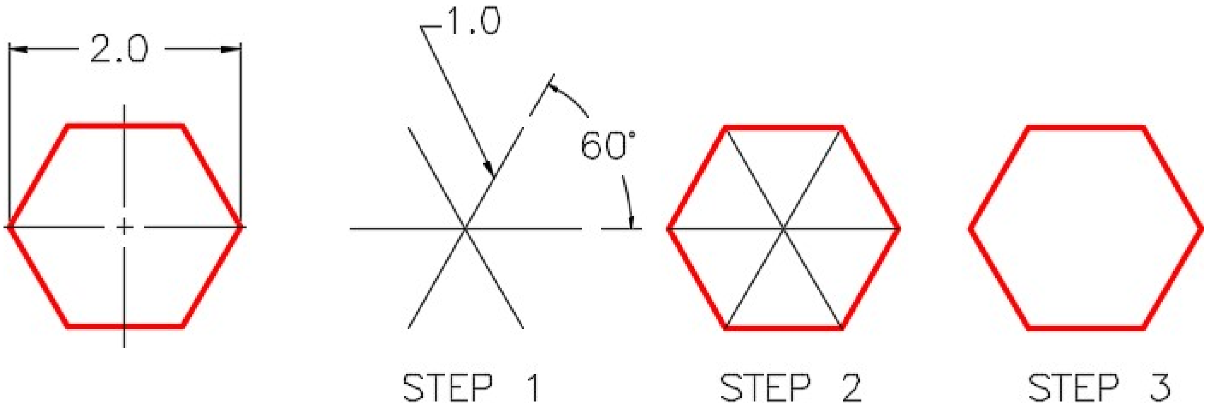


Figure 14-4
Drawing a Hexagon

Trimming and Extending

Not every object can be drawn using exact coordinates or locations as has been taught to this point in the book. There are times where it is essential that objects be drawn to approximate lengths and sizes and then trimmed or extended to their exact size and location. This does not mean that you can guess at the size and location of the objects. Objects must still be drawn with 100 percent accuracy. The length can be approximated and then adjusted using the TRIM or EXTEND commands. These two commands are a big part of you becoming more productive.

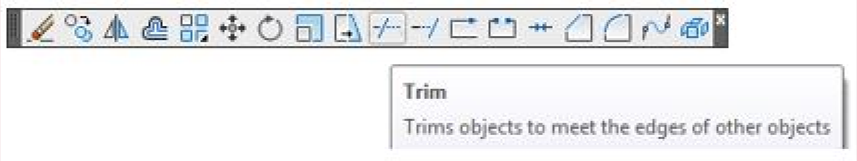
In AutoCAD 2021 there have been changes to the Trim and extend commands and the default settings will not produce the expected results in the following tutorials. The default MODE setting, QUICK, will give access to the **[cutting edges/Crossing/mode/Project/erase]** options, which should be set as explained later in this tutorial. Then, as a last step, the mode should be set to standard. This will produce the expected behavior of TRIM and EXTEND. If you do not see the above options, set the MODE to Quick, make the setting changes and then restore the MODE setting to Standard.

USER TIPS: With both the TRIM and EXTEND commands, a very common beginner's error is picking the wrong objects first. Always select the cutting edge object(s) first, press the Enter or Space next, and then select the objects to be trimmed or extended.

AutoCAD Command: TRIM

The TRIM command is used to shorten objects to meet the edges of other objects. It is not required for the object being trimmed to intersect with the object forming the cutting edge.

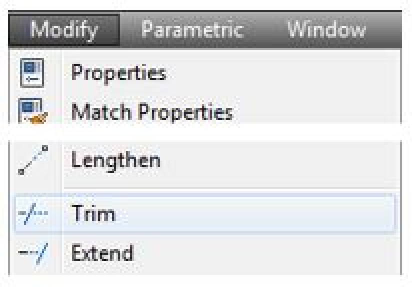
Shortcut: **TR**



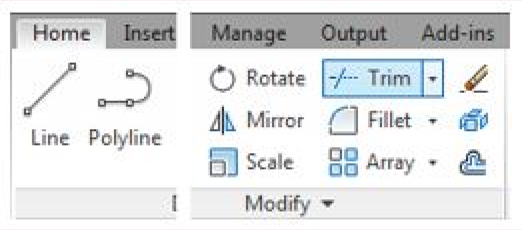
Trim

Trims objects to meet the edges of other objects

Modify Toolbar


Modify
Parametric
Window
Properties
Match Properties
Lengthen
Trim
Extend

Modify Pull-down

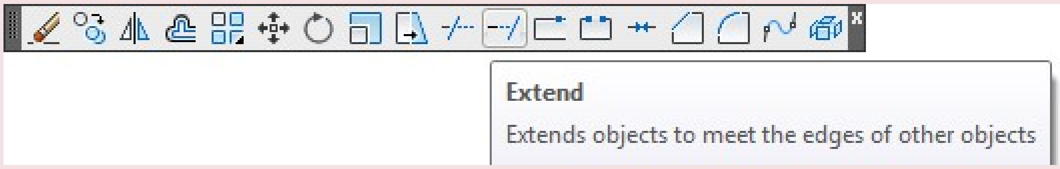

Home
Insert
Manage
Output
Add-ins
Line
Polyline
Rotate
Trim
Mirror
Fillet
Scale
Array
Modify

Home Ribbon

AutoCAD Command: EXTEND

The EXTEND command is used to lengthen objects to meet the edges of other objects. It is not required for the object being extended to intersect with the object forming the cutting edge.

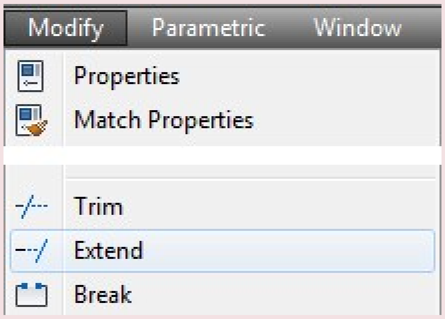
Shortcut: **EX**



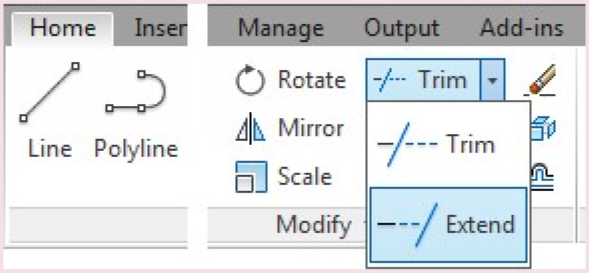
Extend

Extends objects to meet the edges of other objects

Modify Toolbar


Modify
Parametric
Window
Properties
Match Properties
Trim
Extend
Break

Modify Pull-down


Home
Inser
Manage
Output
Add-ins
Line
Polyline
Rotate
Mirror
Scale
Trim
Trim
Extend
Modify

Home Ribbon

WORKALONG: Using the TRIM and EXTEND Commands

Step 1

Start a new drawing using the template: 2D English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 14-1.

Step 3

Create the layers: Object and Construction. Set layer: Construction as the current layer.

Step 4

Using the figures as a reference, enter the LINE command, as shown on the below, to draw the construction lines shown in Figure Step 4B. (Figure Step 4A and 4B)

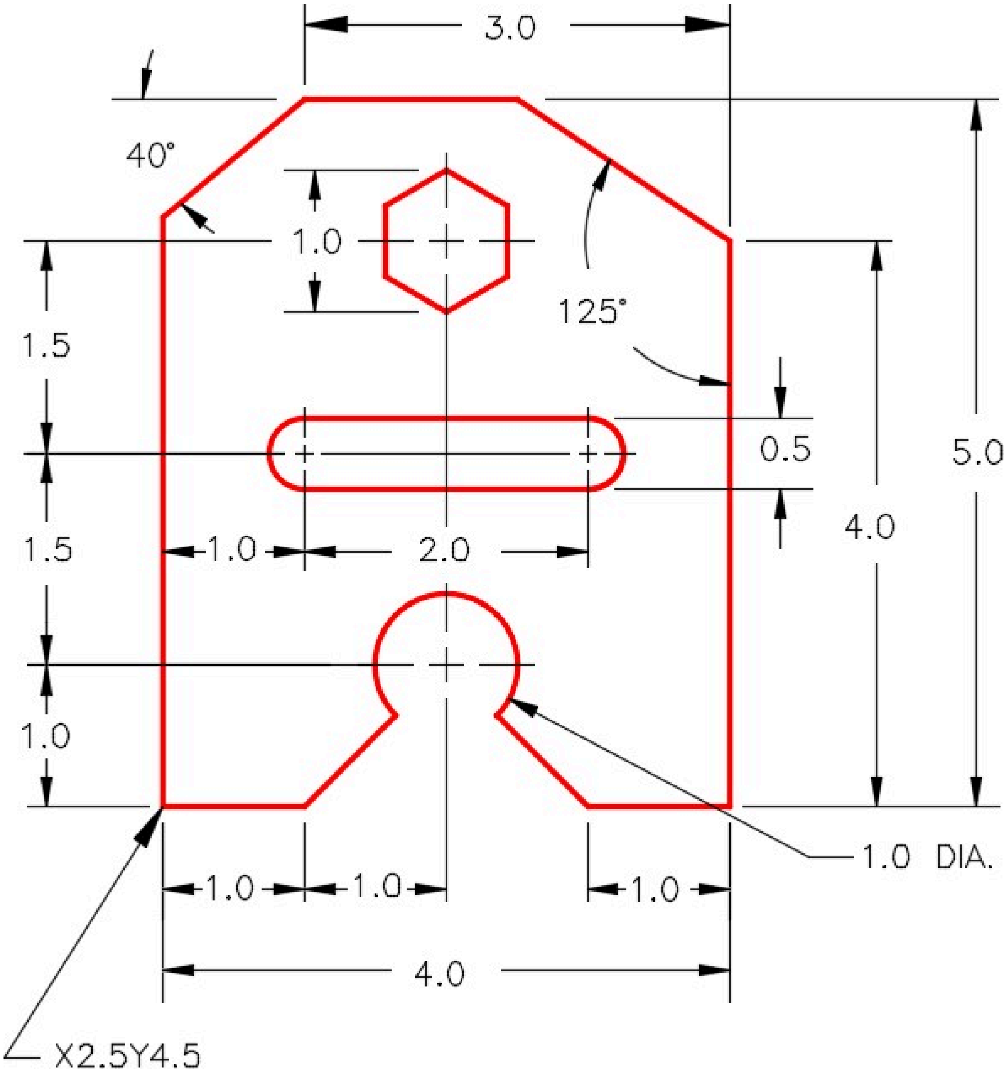

3.0
40°
1.0
125°
1.5
0.5
5.0
1.5
1.0
2.0
4.0
1.0
1.0 DIA.
1.0
1.0
1.0
4.0
X2.5Y4.5

Figure Step 4A

Command: **L**

Specify first point: **2.5,4.5**

Specify next point or [Undo]: **@1,0**

Specify next point or [Undo]: **@2,0**

Specify next point or [Close/Undo]: **@1,0**

Specify next point or [Close/Undo]: **@0,4**

Specify next point or [Close/Undo]: **@0,1**

Specify next point or [Close/Undo]: **@-3,0**

Specify next point or [Close/Undo]: **@2<220**

Specify next point or [Close/Undo]:

Command: **L**

Specify first point: (end) **P1**

Specify next point or [Undo]: **@0,5**

Command: **L**

Specify first point: (end) **P2**

Specify next point or [Undo]: **@1<145**

(The length of this line can be any length but the direction must be accurate. I purposely drew it shorter to demonstrate the EXTEND command.)

Command:

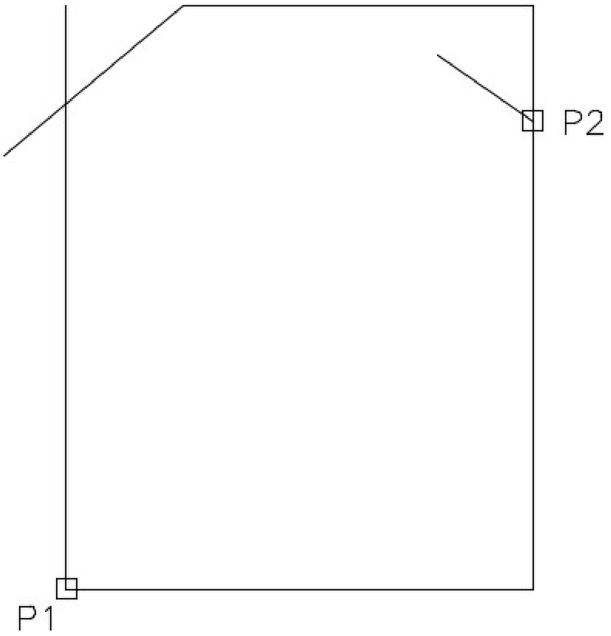


Figure Step 4B

AUTHOR'S COMMENTS: Notice how the object is drawn with a series of construction lines that go from point to point on the object rather than from corner to corner. This is done so that the endpoints of the lines can be used as snap point later in the construction.

Step 5

Enter the TRIM command as shown below. (Figure Step 5).

Command: **TRIM**

Current settings: Projection=UCS, Edge=None

Select cutting edges …

Select objects or <select all>:

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: **E**

Enter an implied edge extension mode [Extend/No extend] <No extend>: **E**

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]:

Command:

> **AUTHOR’S COMMENTS:** The Edge option in the TRIM command should be set to Extend most of the time. Once it is set on your computer, it will stay that way until you change it.

Command: **TRIM**

Current settings: Projection=UCS, Edge=Extend

Select cutting edges …

Select objects or <select all>: **P3** 1 found

Select objects: **P4** 1 found, 2 total

(These lines are the cutting edges.)

Select objects:

(Press enter or space.)

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: **P5**

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: **P6**

(These are the lines to trim. Select the line close to the end to be trimmed.)

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]:

Command:

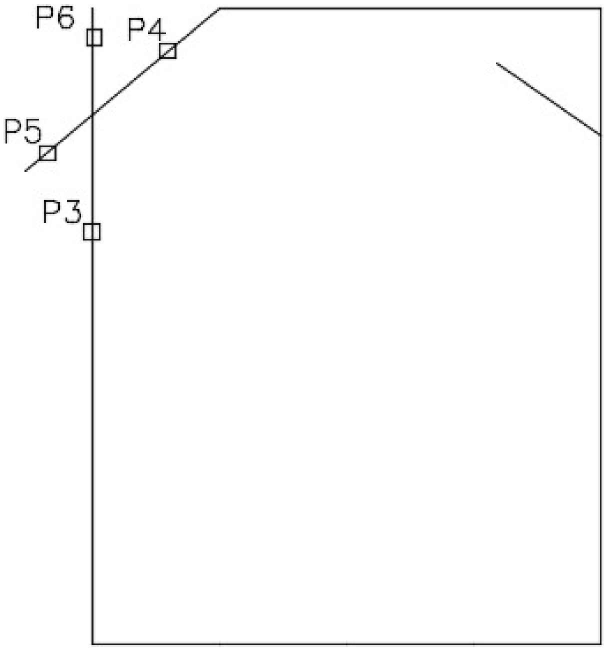


Figure Step 5

Step 6

Enter the EXTEND command, as shown below, to extend the line to top line in object. (Figure Step 6A and 6B)

Command: **EXTEND**

Current settings: Projection=UCS, Edge=Extend

Select boundary edges …

Select objects or <select all>: **P7** 1 found

(The cutting edge.)

Select objects:

(Press enter or space.)

Select object to extend or shift-select to trim or [Fence/Crossing/Project/Edge/Undo]: **P8**

(The line to extend. Select the line close to the end that is to be extended.)

Select object to extend or shift-select to trim or [Fence/Crossing/Project/Edge/Undo]:

Command:

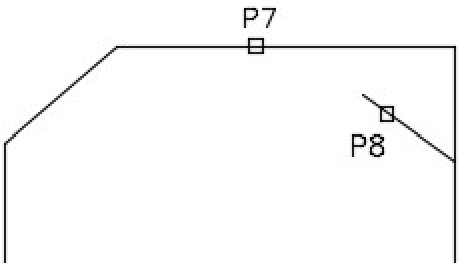


Figure Step 6A

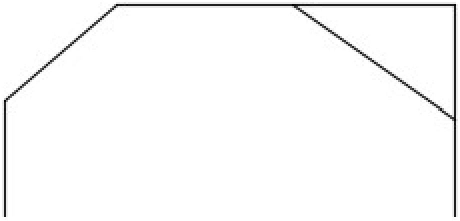


Figure Step 6B

Step 7

Draw three construction lines starting at the midpoint of the bottom line. (Figures Step 7)

Command: **L**

Specify first point: *(mid)* **P9**

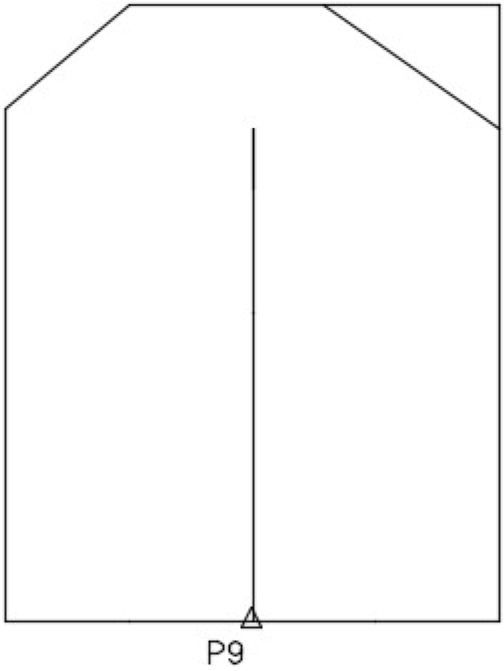
Specify next point or [Undo]: **@0,1**

Specify next point or [Undo]: **@0,1.5**

Specify next point or [Close/Undo]: **@0,1.5**

Specify next point or [Close/Undo]:

Command:



Figures Step 7

AUTHOR’S COMMENTS: Notice how the three lines are drawn end to end even though they look like one line. This is done to establish the endpoints so that they can be snapped onto later to construct the object.

Step 8

Enter the LINE commands, as shown below, to insert the construction lines for the hexagon. Each line starts at the end of the construction line drawn in Step 7. (Figure Step 8)

Command: **L**

Specify first point: *(end)* **P10**

Specify next point or [Undo]: **@.5<30**

Specify next point or [Close/Undo]:

Command: **L**

Specify first point: *(end)* **P10**

Specify next point or [Undo]: **@.5<90**

Specify next point or [Close/Undo]:

Command: **L**

Specify first point: *(end)* **P10**

```
Specify next point or [Undo]: @.5<150

Specify next point or [Close/Undo]:

Command: L

Specify first point: (end) P10

Specify next point or [Close/Undo]: @.5<210

Specify next point or [Close/Undo]:

Command: L

Specify first point: (end) P10

Specify next point or [Undo]: @.5<270

Specify next point or [Close/Undo]:

Command: LINE Specify first point:

Command: L

Specify first point: (end) P10

Specify next point or [Undo]: @.5<330

Specify next point or [Close/Undo]:

Command:
```

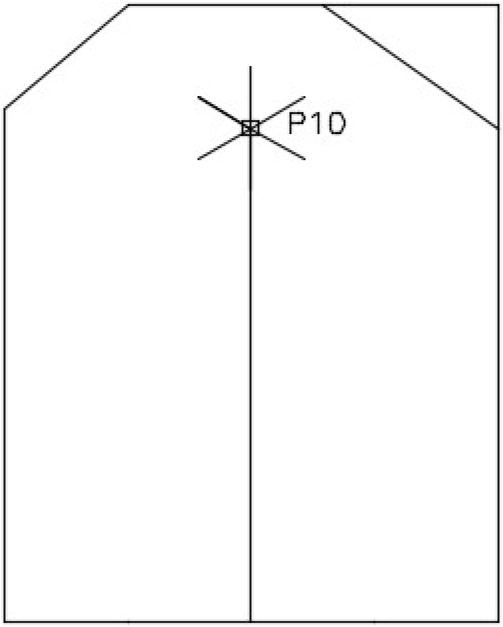


Figure Step 8

Step 9

Enter the LINE command, as shown below, to construct the lines for the hexagon. Ensure that you have Autosnap enabled and you snap to the endpoints of the lines. (Figure Step 9)

Command: **L**

Specify first point: *(end)* **P11**

Specify next point or [Undo]: *(end)* **P12**

Specify next point or [Undo]: *(end)* **P13**

Specify next point or [Close/Undo]: *(end)* **P14**

Specify next point or [Close/Undo]: *(end)* **P15**

Specify next point or [Close/Undo]: *(end)* **P16**

Specify next point or [Close/Undo]: *(end)* **P11**

Specify next point or [Close/Undo]:

Command:

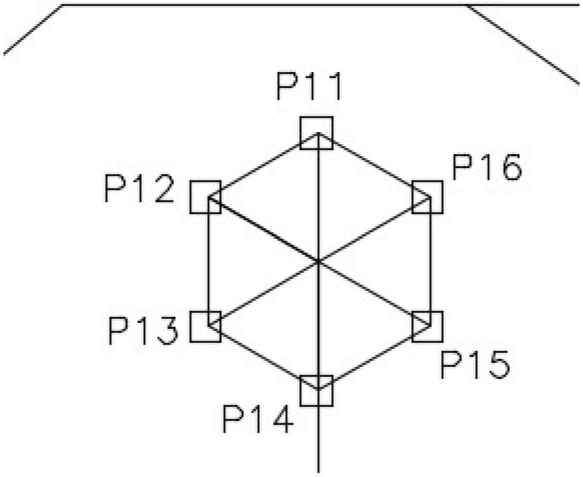


Figure Step 9

Step 10

Enter the LINE and CIRCLE commands, as shown below, to draw the slot. (Figure Step 10A and 10B)

Command: **L**

Specify first point: *(end)* **P17**

Specify next point or [Undo]: **@1,0**

Specify next point or [Undo]:

Command: **L**

Specify first point: *(end)* **P17**

Specify next point or [Undo]: **@-1,0**

Specify next point or [Undo]:

Command: **C**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: *(end)* **P18**

Specify radius of circle or [Diameter]: **D**

Specify diameter of circle: **.5**

Command: **C**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: (end) **P19**

Specify radius of circle or [Diameter] <0.2500>:

Command: **L**

Specify first point: *(quad)* **P20**

Specify next point or [Undo]: *(quad)* **P21**

Specify next point or [Undo]: Command: **L**

Specify first point: *(quad)* **P22**

Specify next point or [Undo]: *(quad)* **P23**

Specify next point or [Undo]:

Command:

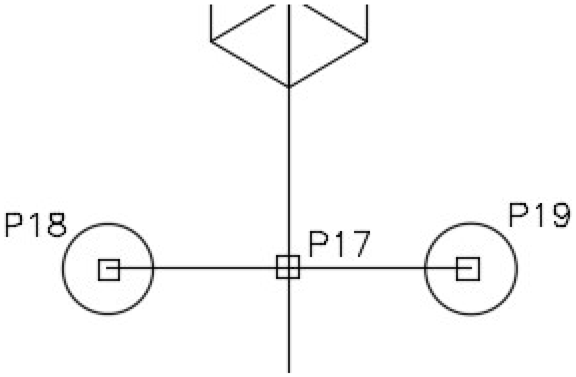


Figure Step 10A

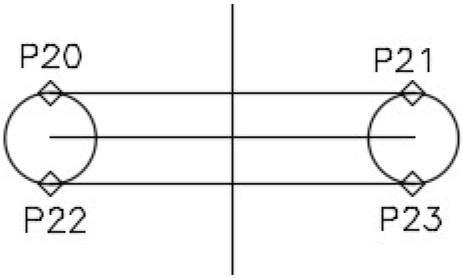


Figure Step 10B

AUTHOR'S COMMENTS: Ensure that you snap to the object snap mode shown before each pick in the command.

Step 11

Enter the commands, as shown below, to draw the construction objects for the bottom arc. (Figure Step 11A and 11B)

Command: **C**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: *(end)* **P24**

Specify radius of circle or [Diameter]: **D**

Specify diameter of circle: **1**

Command: **L**

Specify first point: *(cen)* **P25**

Specify next point or [Undo]: *(end)* **P26**

Specify next point or [Undo]:

Command: **L**

Specify first point: *(cen)* **P25**

Specify next point or [Undo]: *(end)* **P27**

Specify next point or [Undo]:

Command:

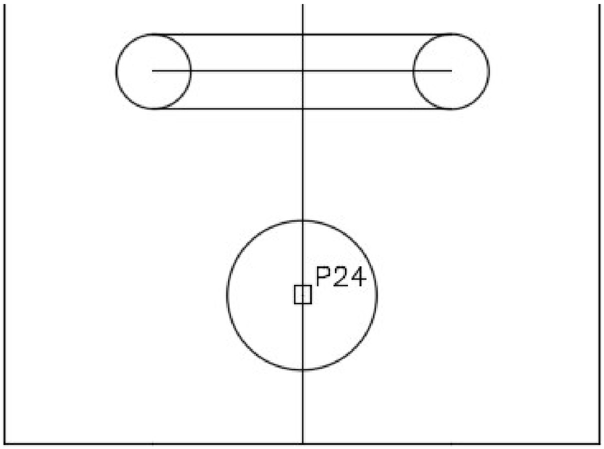


Figure Step 11A

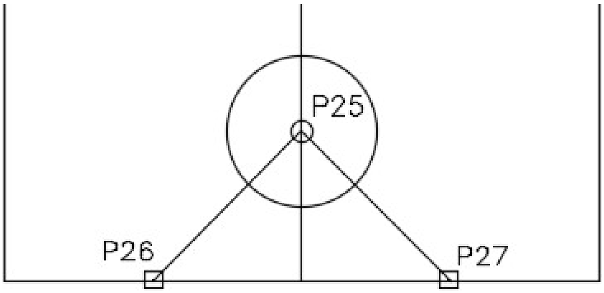


Figure Step 11B

AUTHOR'S COMMENTS: I find it easier to draw circles rather than arcs. Then I trim them to construct the required arc.

Step 12

Enter the TRIM command, as shown below, to create the slot. (Figure Step 12A and 12B)

Command: **TRIM**

Current settings: Projection=UCS, Edge=Extend

Select cutting edges …

Select objects or <select all>: **P28** 1 found

Select objects: **P29** 1 found, 2 total

Select objects:

(Press enter or space.)

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: **P30**

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: **P31**

```
Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]:

Command:
```

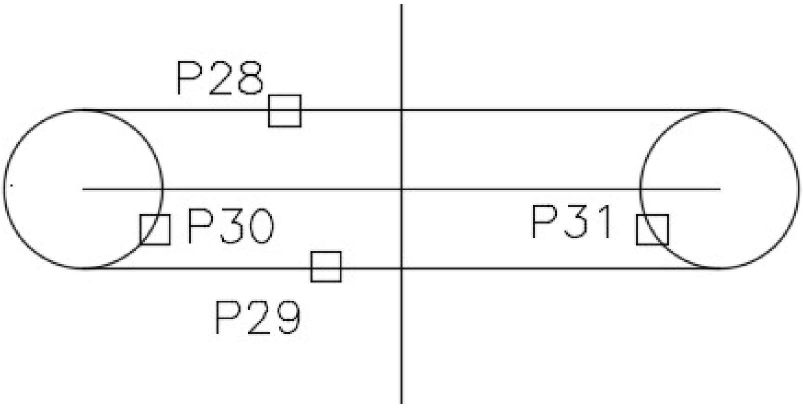


Figure Step 12A

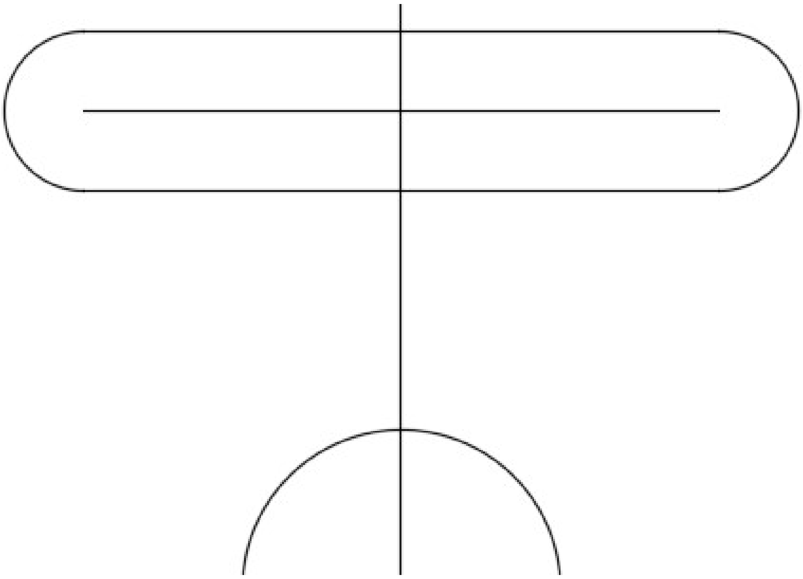


Figure Step 12B

Step 13

Enter the TRIM command, as shown below, to create the bottom arc. (Figure Step 13A and 13B)

Command: **TRIM**

Current settings: Projection=UCS, Edge=Extend

Select cutting edges …

Select objects or <select all>: **P32** 1 found

Select objects: **P33** 1 found, 2 total

Select objects: **P34** 1 found, 3 total

Select objects:

(Press enter or space)

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: **P35**

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]:**P36**

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: **P37**

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]:

Command:

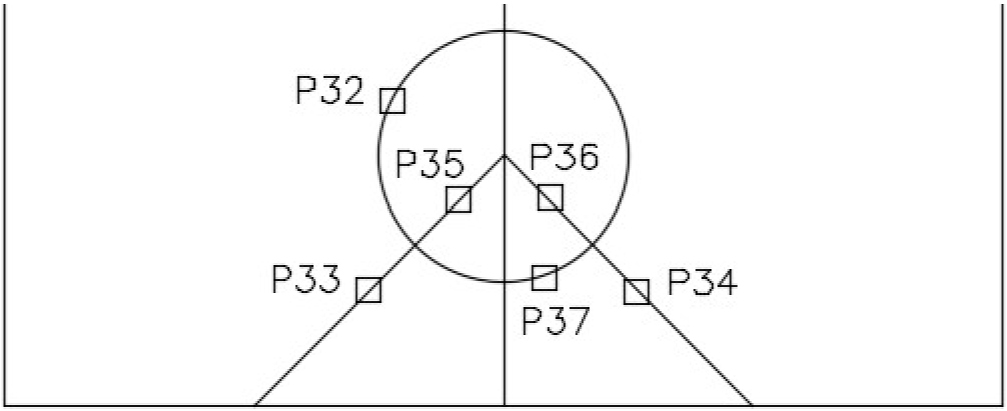


Figure Step 13A

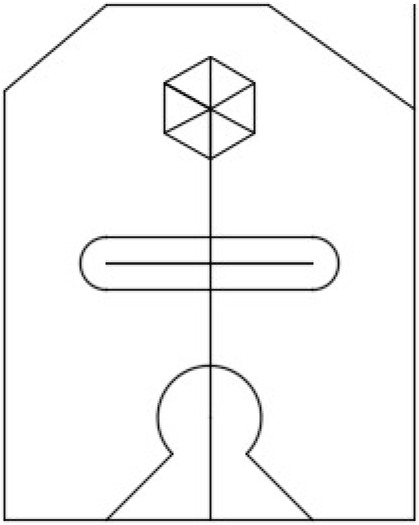


Figure Step 13B

Step 14

Enter the TRIM command, as shown below, to trim the line on the top right. (Figure Step 14A and 14B)

Command: **TRIM**

Current settings: Projection=UCS, Edge=Extend

Select cutting edges …

Select objects or <select all>: **P38** 1 found

Select objects:

(Press enter or space)

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: **P39**

Select object to trim or shift-select to extend or

[Fence/Crossing/Project/Edge/eRase/Undo]:

Command:

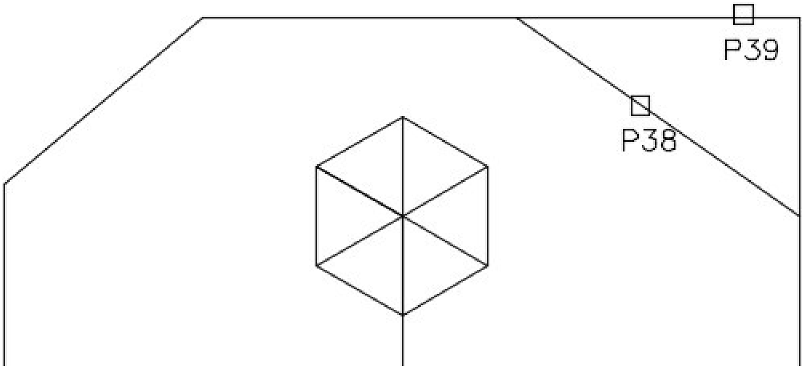


Figure Step 14A

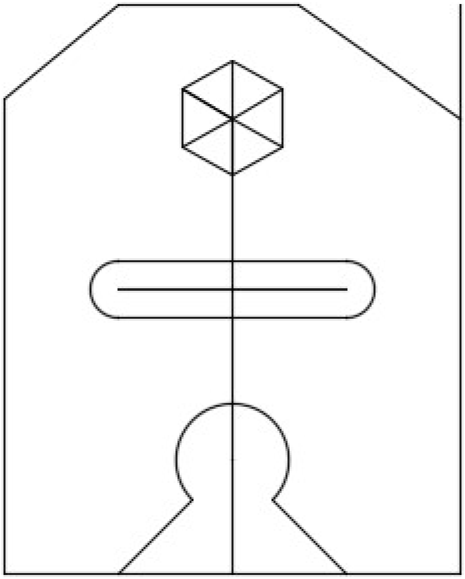


Figure Step 14B

Step 15

Change the layer of the objects that you want to reside on layer: Object. (Figure Step 15)

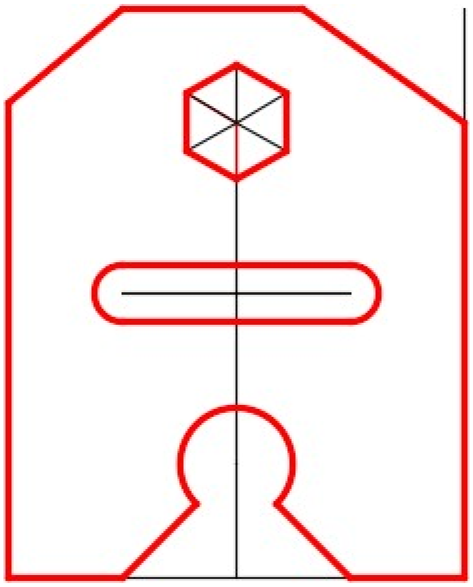


Figure Step 15

Step 16

Freeze layer: Construction. Your completed drawing should appear as shown in the figure. (Figure Step 16)

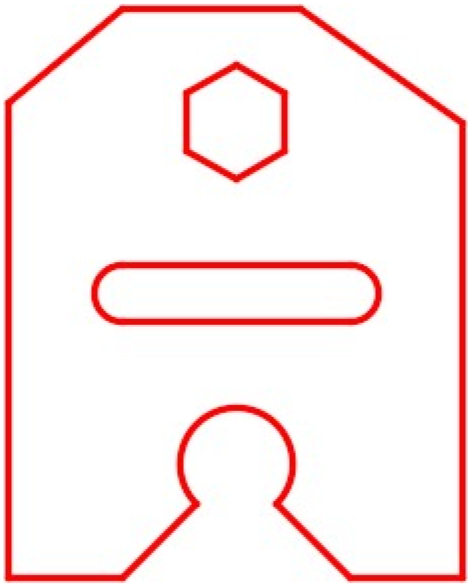


Figure Step 16

Step 17

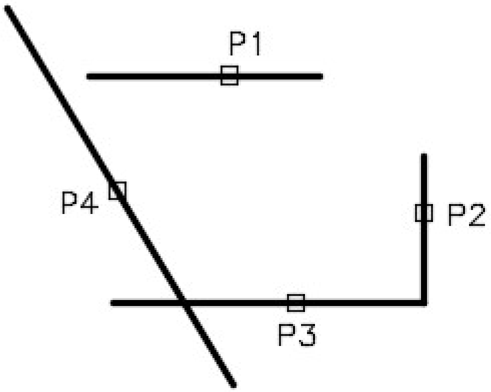
Save and close the drawing.

MUST KNOW: The Edge option in the TRIM and EXTEND commands should always be set to Extend as shown below. Step 5 will show you how to set it. See also the 2nd paragraph in the notes on Trim an Extend.

Command: **TRIM**

Current settings: Projection=UCS, Edge=Extend

USER TIP: The TRIM and EXTEND commands areinterchangeable. If you enter either command, you can change over to the other command by holding down the Shift key. i.e.
Command: TRIM *(See Figure Step 1.)*



Current settings: Projection=UCS, Edge=None
Select cutting edges …Select objects or <select all>: **P1** 1 found
Select objects: **P2** 1 found, 2 total
Select objects: **P3** 1 found, 3 total
Select objects: **P4** 1 found, 4 total

Select objects:

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: **E**

(Enter the Edge option.)

Enter an implied edge extension mode [Extend/No extend] <No extend>: **E**

(Set it to Extend. This allows you to trim or extend when the objects do not physically touch.)

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: **P5**

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: **P6**

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: **P7**

(Press and hold down the Shift key to change to the EXTEND command.)

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: **P8**

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: **P9**

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: **P10**

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]:

Command:

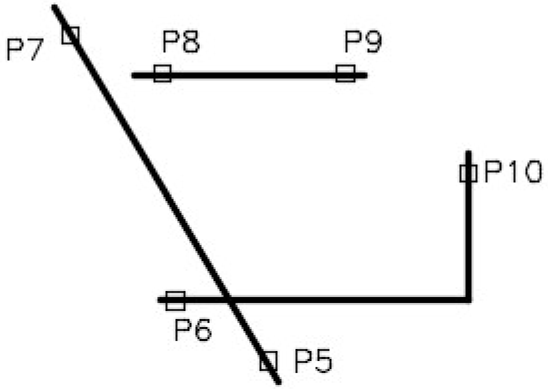
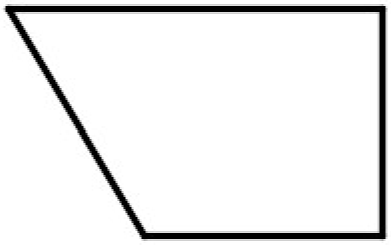


Figure Step 2



Completed Drawing

Key Principles

Key Principles in Module 14

1. To improve your drawing productivity, draw the location and direction of objects with 100 percent accuracy but at an arbitrary length. Adjust the length with the TRIM or EXTEND commands.
2. With both the TRIM and EXTEND commands, select the cutting edge objects first, press the Enter or Space key, and then select the object(s) to trim or extend.
3. The TRIM and EXTEND commands are interchangeable by holding down the Shift key to change from one command to the other.

Lab Exercise 14-1

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 14-1	2D English	Inches

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Construction	Construction objects	253
Object 1	All objects	Red

Step 1

Draw the object shown in the figure using the layering scheme. (Figure Step 1A and 1B)

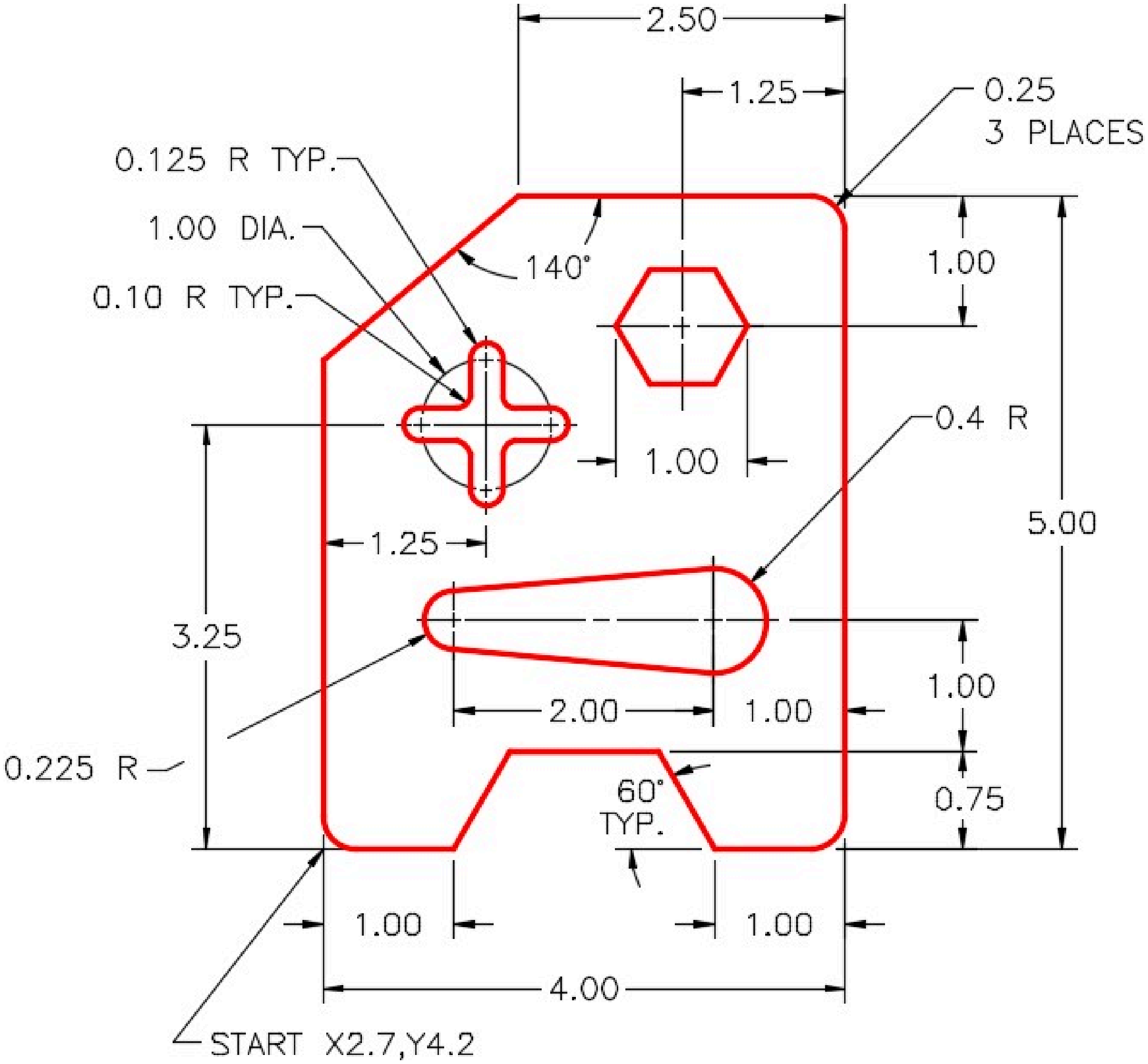

2.50
1.25
0.25
3 PLACES
0.125 R TYP.
1.00 DIA.
0.10 R TYP.
140°
1.00
1.00
0.4 R
5.00
1.25
3.25
2.00
1.00
1.00
0.225 R
60°
TYP.
0.75
1.00
1.00
4.00
START X2.7,Y4.2

Figure Step 1A

AUTHOR'S COMMENTS: Do not delete the construction lines.

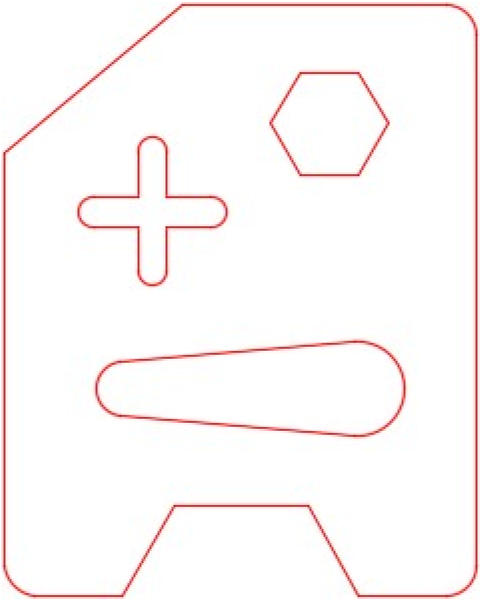


Figure Step 1B

Step 2

Set the insertion units and check your drawing with the key.

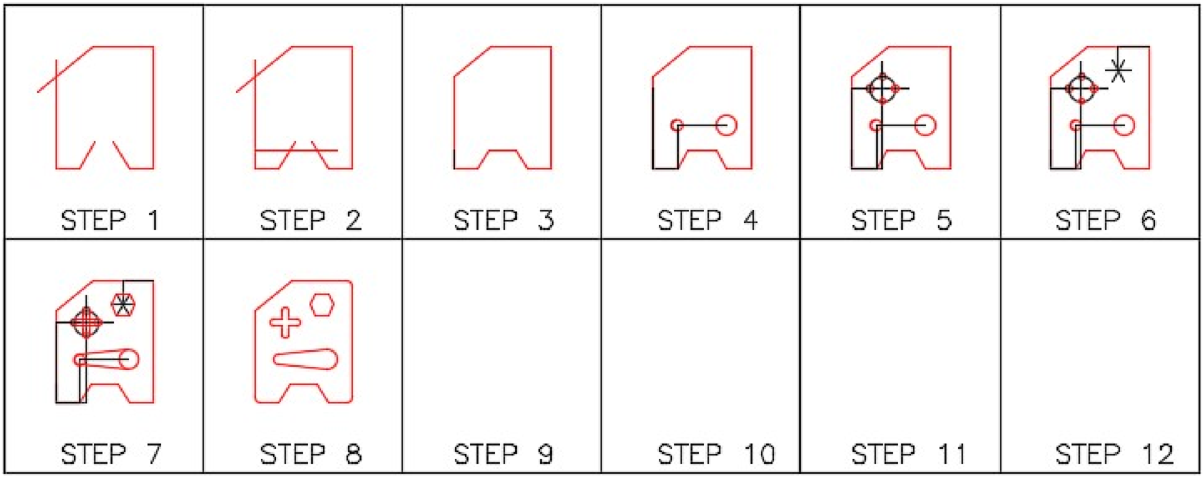
Step 3

Turn layer: Key off and freeze layer: Construction.

Step 4

Save and close the drawing.

AUTHOR'S CONSTRUCTION TECHNIQUES: The following steps are the construction technique suggested by the author to help you learn how to construct objects using AutoCAD. It is only the suggested method and if you can complete the drawing accurately using a different construction technique, that is what is important. You may want to compare your construction technique with the authors.


STEP 1
STEP 2
STEP 3
STEP 4
STEP 5
STEP 6
STEP 7
STEP 8
STEP 9
STEP 10
STEP 11
STEP 12

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

Draw the lines in the correct location and direction but to any length and then trim or extend them. (Figure Hint 1)

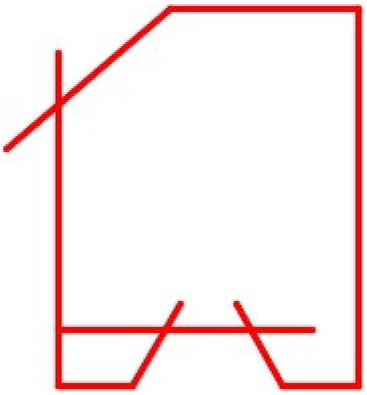


Figure Hint 1

Hint 2

The lines in Figure Hint 2A must be drawn tangent to tangent (see Module 12) while the lines in Figure Hint 2B can be draw either tangent to tangent or quad to quad. (Figure Hint 2A and 2B)

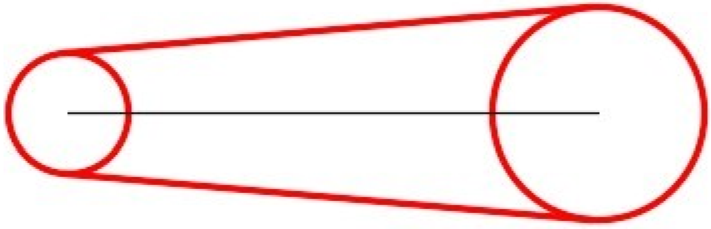


Figure Hint 2A

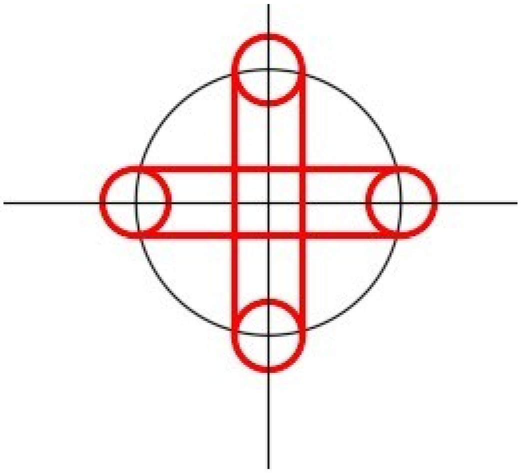


Figure Hint 2

Hint 3

The figure shows the construction lines. (Figure Hint 3)

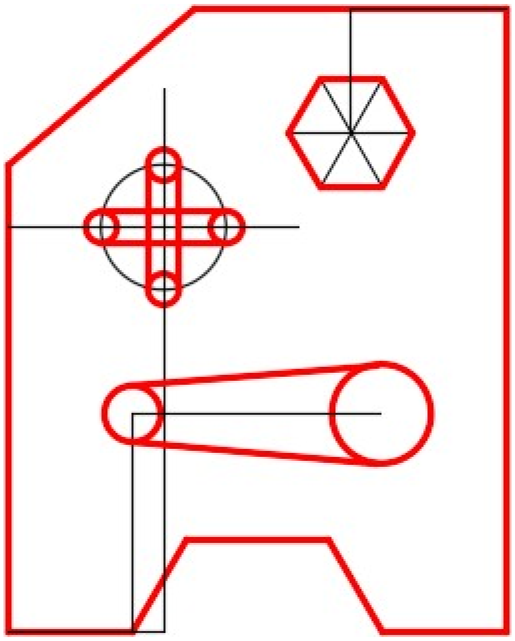


Figure Hint 3

Lab Exercise 14-2

Time allowed: 30 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 14-2	2D Metric	Millimeters

Layering Scheme

Layer Name	Objects on Layer	Linetype	Color
Construction	Construction objects	Continuous	253
Circle 1	Outside circle	Center2	Red
Circle 2	Inside circles	Continuous	Red
Slots	Slots	Dashdot2	Blue

Step 1

Draw the object shown in the figure using the layering scheme. (Figure Step 1A and 1B)

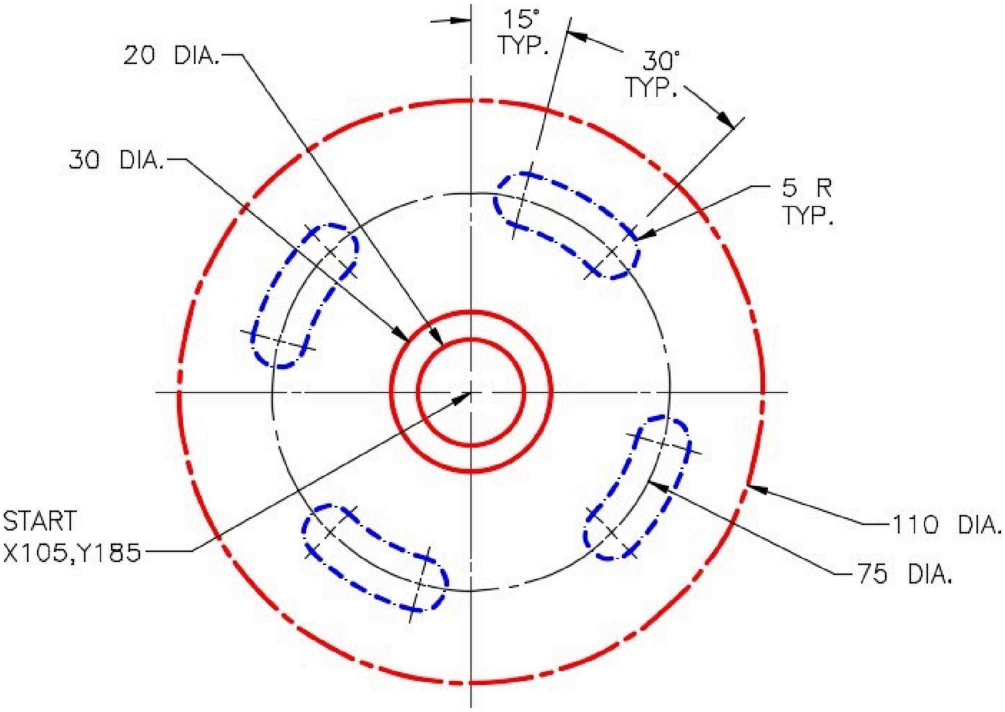

15°
TYP.
30°
TYP.
20 DIA.
30 DIA.
5 R
TYP.
START
X105,Y185
110 DIA.
75 DIA.

Figure Step 1A

AUTHOR'S COMMENTS: Do not delete the construction lines.

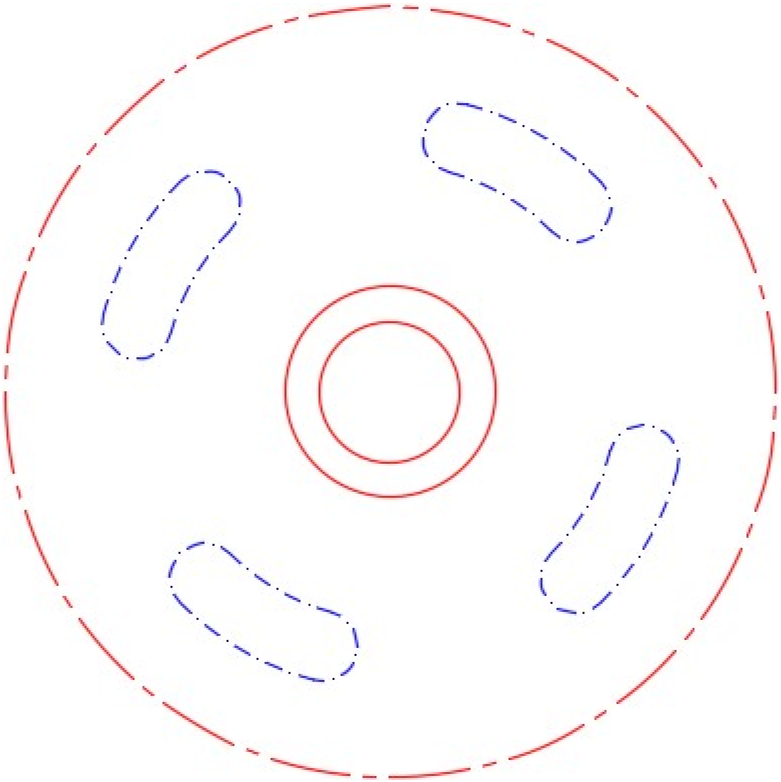


Figure Step 1B
Completed Drawing

Step 2

Set the insertion units and check your drawing with the key.

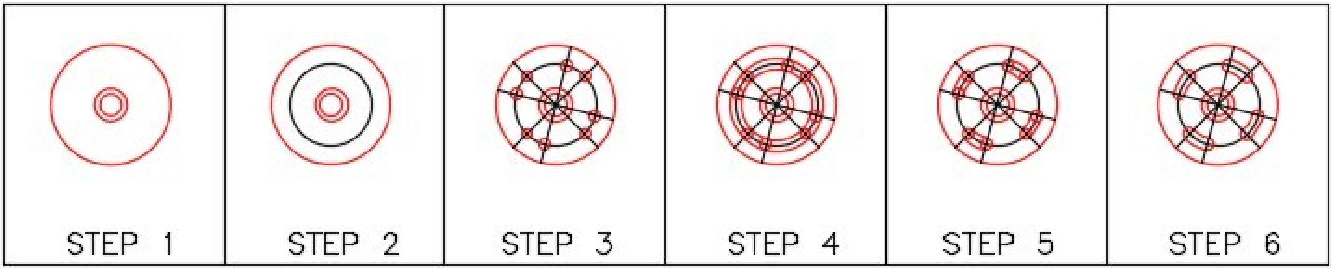
Step 3

Turn layer: Key off and freeze layer: Construction.

Step 4

Save and close the drawing.

AUTHOR'S CONSTRUCTION TECHNIQUES: The following steps are the construction technique suggested by the author to help you learn how to construct objects using AutoCAD. It is only the suggested method and if you can complete the drawing accurately using a different construction technique, that is what is important. You may want to compare your construction technique with the authors.



AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

Step 1

Draw only four construction lines at the correct angle as shown in the figure. Extend them to other side of the large circle. That will save you from drawing all 8 lines. (Figure Step 1A and 1B)

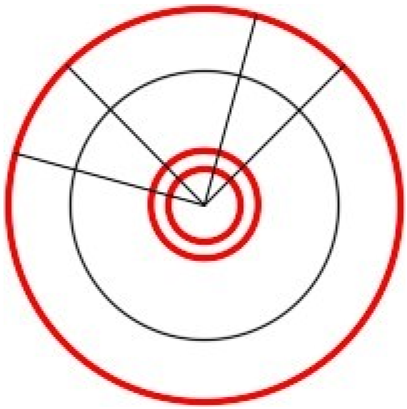


Figure Step 1A

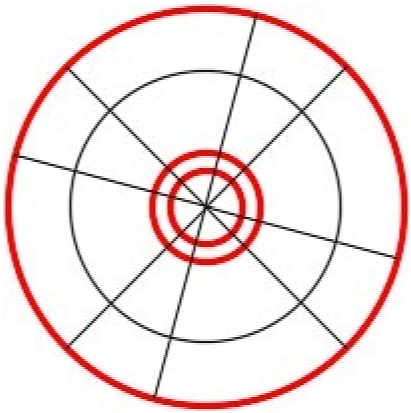


Figure Step 1B

Step 2

Draw 8 circles locating their centers at the intersection of the lines and circle. (Figure Step 2)

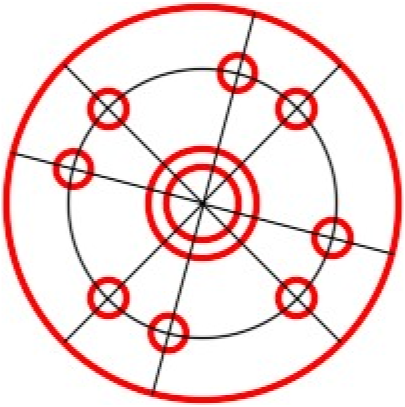


Figure Step 2

Step 3

Draw 2 large circles by showing AutoCAD the radius by snapping to the intersection of the small circle and line. (Figure Step 3)

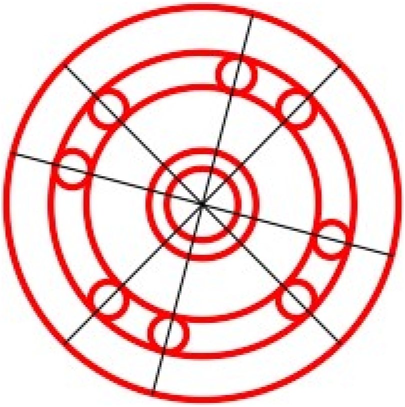


Figure Step 3

Step 4

Trim the large circles. (Figure Step 4)

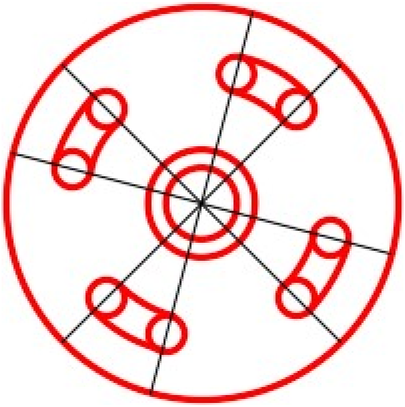


Figure Step 4

Step 5

Trim the slots. (Figure Step 5)

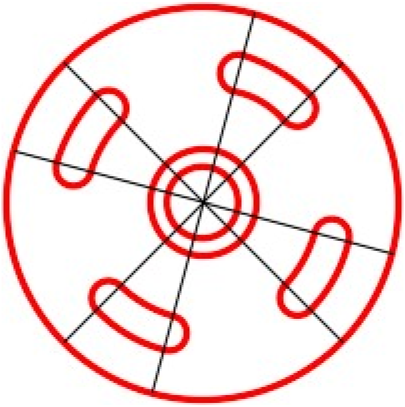


Figure Step 5

Lab Exercise 14-3

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 14-3	2D Metric	Millimeters

Layering Scheme

Layer Name	Objects on Layer	Color
Concrete Block	All objects	Blue

Step 1

Draw the top view of the concrete block shown in the figure using the layering scheme. (Figure Step 1A, 1B, and 1C)

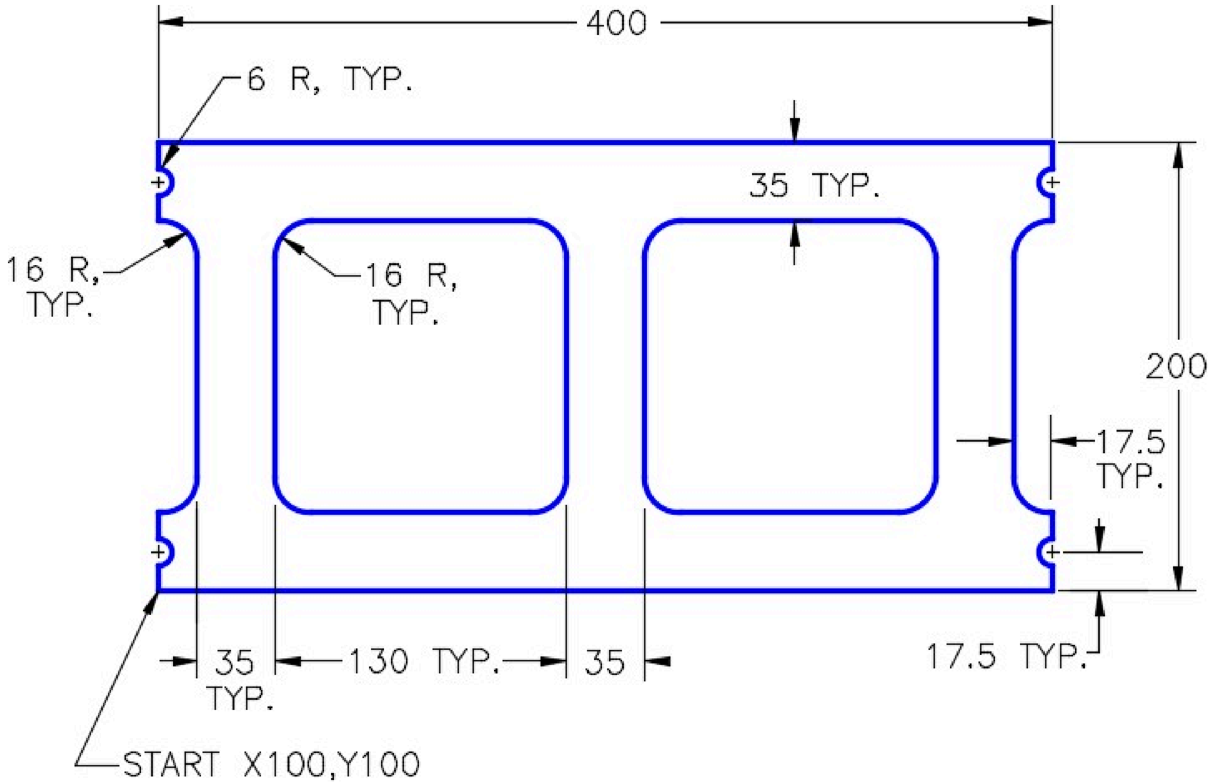

400
6 R, TYP.
35 TYP.
16 R, TYP.
16 R, TYP.
200
17.5 TYP.
35 TYP.
130 TYP.
35
17.5 TYP.
START X100,Y100

Figure Step 1A

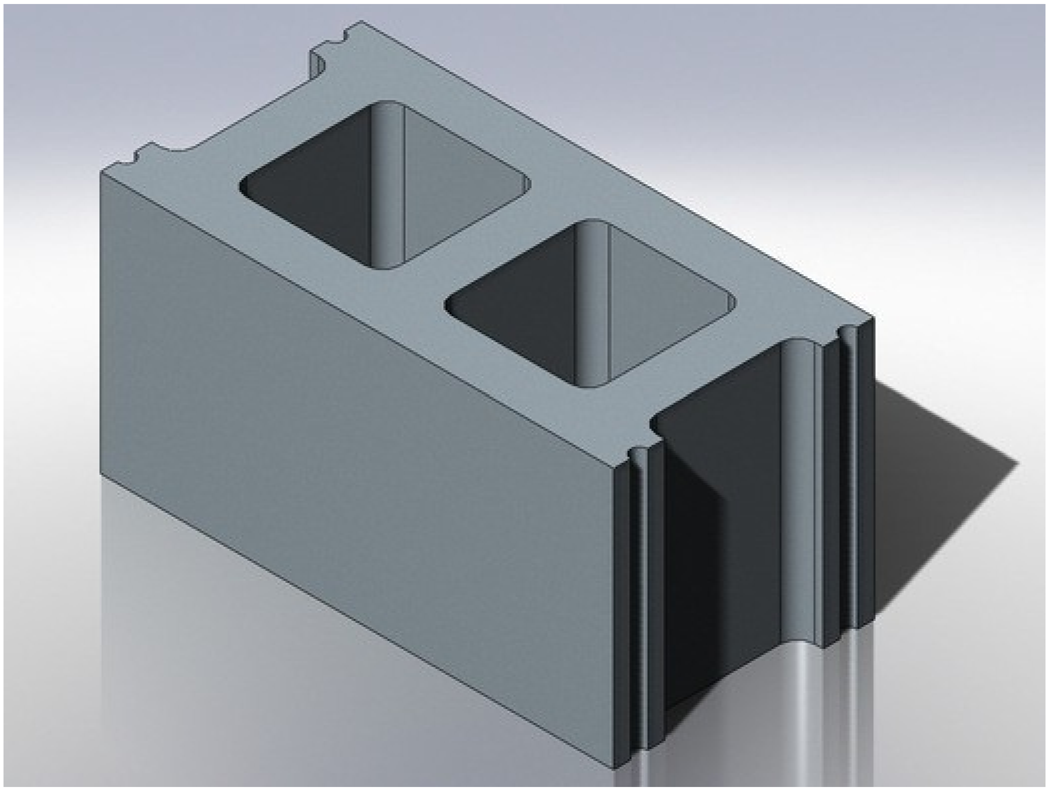


Figure Step 1B Concrete Block

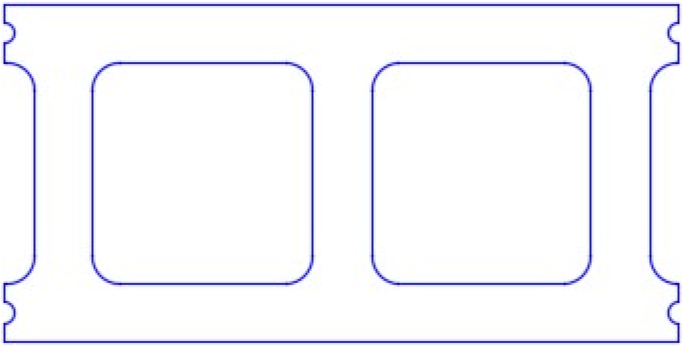


Figure Step 1C Completed Drawing

Step 2

Set the insertion units and check your drawing with the key. Turn layer: Key off and freeze layer: Construction.

Step 3

Save and close the drawing.

Module 15: Offsetting Objects

Learning Outcomes

When you have completed this module, you will be able to:

1. Apply the OFFSET command to insert objects parallel to existing objects.
2. Apply the ID command to establish temporary reference locations.
3. Describe and apply the object snap mode Perpendicular.

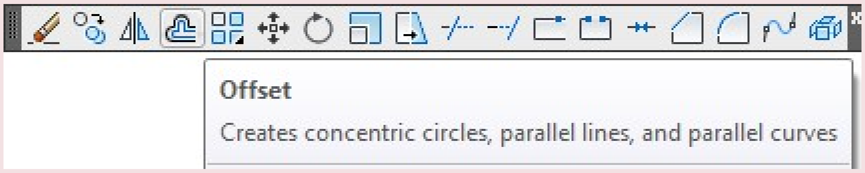
Offsetting

The OFFSET command allows you to create a drawing object parallel to an existing drawing object at a specified distance from the original object. OFFSET is a powerful command and if used when applicable, will change the way you construct objects and improve your drawing productivity. The drawing objects lines, circles, and arcs can be offset plus many others that you will be learning later in the book.

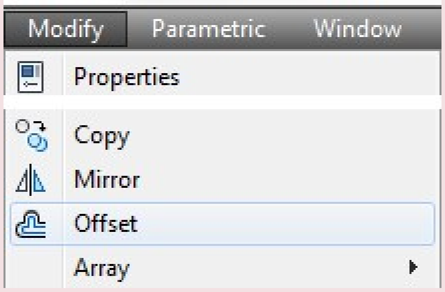
AutoCAD Command: OFFSET

The OFFSET command is used to create a drawing object parallel to an existing drawing object at a specified distance from the original object.

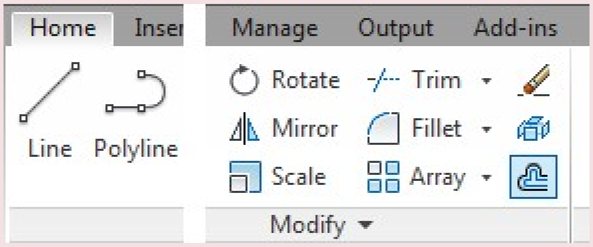
Shortcut: **O**


Offset
Creates concentric circles, parallel lines, and parallel curves

Modify Toolbar


Modify
Parametric
Window
Properties
Copy
Mirror
Offset
Array

Modify Pull-down

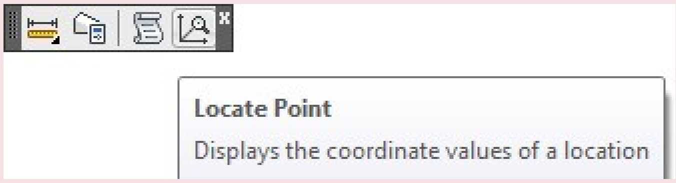

Home
Inser
Manage
Output
Add-ins
Line
Polyline
Rotate
Trim
Mirror
Fillet
Scale
Array
Modify

Home Ribbon

AutoCAD Command: ID

The ID command is used to return the coordinate location for a point in Model or Paper space.

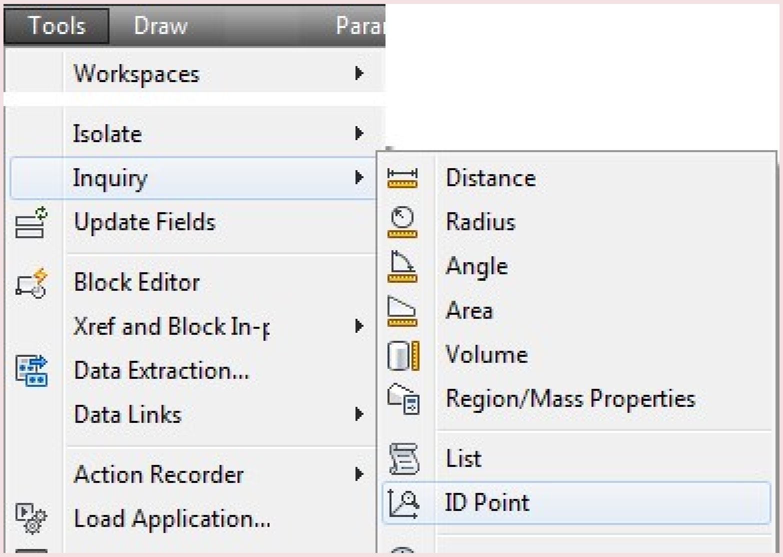
Shortcut: none



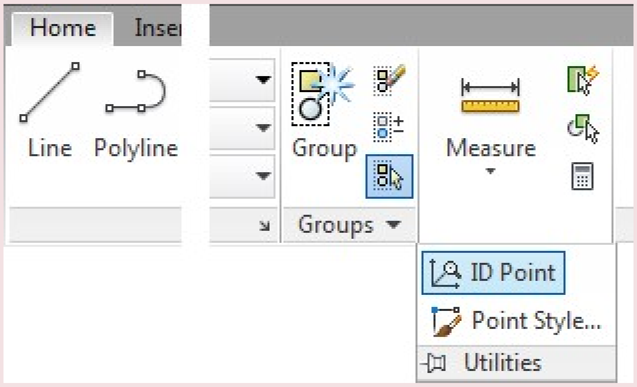
Locate Point

Displays the coordinate values of a location

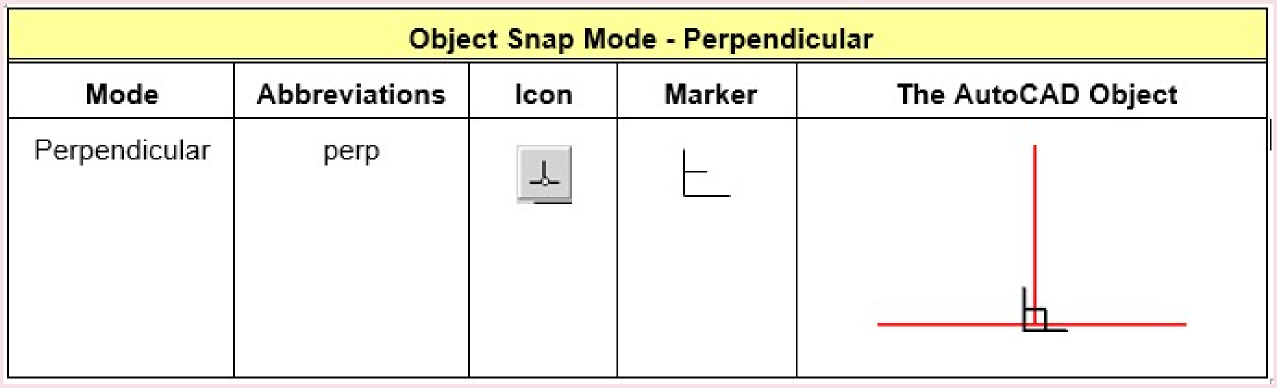
Inquiry Toolbar


Tools
Draw
Para
Workspaces
Isolate
Inquiry
Update Fields
Block Editor
Xref and Block In-p
Data Extraction...
Data Links
Action Recorder
Load Application...
Distance
Radius
Angle
Area
Volume
Region/Mass Properties
List
ID Point

Tools Pull-down


Home
Inser
Line
Polyline
Group
Groups
Measure
ID Point
Point Style...
Utilities

Home Ribbon



Object Snap Mode - Perpendicular				
Mode	**Abbreviations**	**Icon**	**Marker**	**The AutoCAD Object**
Perpendicular	perp	⊥	∟	

Figure 15-1
Object Snap Mode – Perpendicular

MUST KNOW: Using the ID command to set a reference point or the lastpoint is a useful drawing tool. Once you pick an ID point, ignore the actual coordinates. On the next command, use an @ to locate a coordinate location from the ID point.

WORKALONG: Offsetting Objects

Step 1

Start a new drawing using the template: 2D English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 15-1. (Figure Step 2)

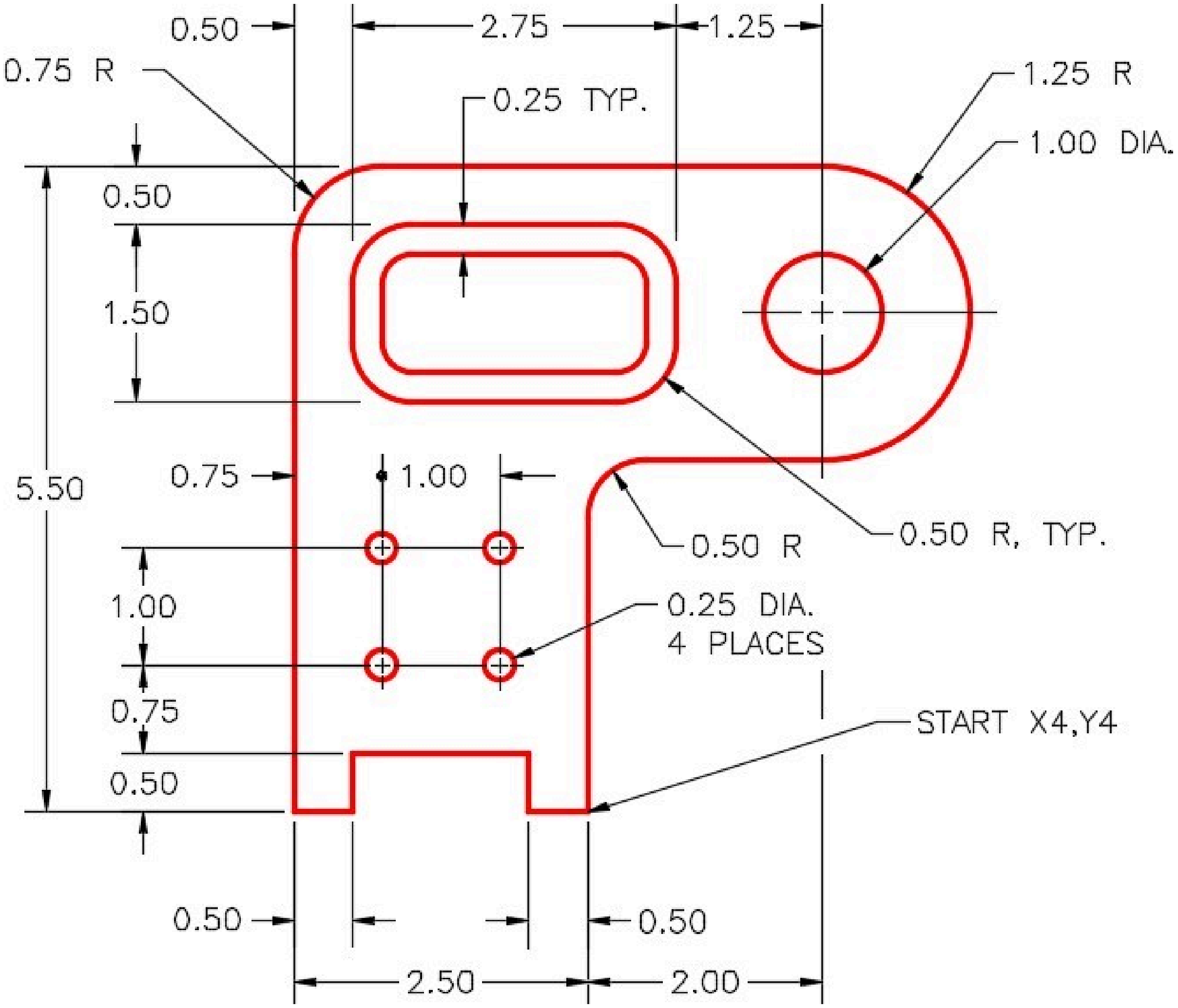

0.50
2.75
1.25
0.75 R
1.25 R
0.25 TYP.
1.00 DIA.
0.50
1.50
5.50
0.75
1.00
0.50 R
0.50 R, TYP.
1.00
0.25 DIA.
4 PLACES
0.75
START X4,Y4
0.50
0.50
0.50
2.50
2.00

Figure Step 2

Step 3

Create the layers: Object and Construction. Set layer: Construction as the current layer.

MUST KNOW: When a drawing object is offset, the newly constructed offset drawing object will reside on the same layer as the original object, regardless of the current layer. If you want the newly created object to be on a different layer than the original object, you will have to change it's properties after it is offset.

Step 4

Enter the LINE command, as shown below, to create the outline of the object. (Figure Step 4)

```
Command: L

Specify first point: 4,4

Specify next point or [Undo]: @-2.5,0

Specify next point or [Undo]: @0,5.5

Specify next point or [Close/Undo]: @4.5,0

Specify next point or [Close/Undo]: @0,-2.5

Specify next point or [Close/Undo]: @-2,0

Specify next point or [Close/Undo]: C

Command:
```

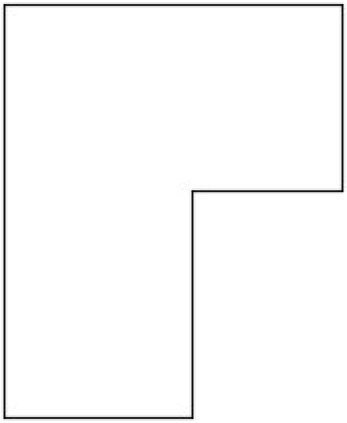


Figure Step 4

Step 5

Enter the commands, as shown below, to draw the circles and fillets. (Figure Step 5A and 5B)

Command: **C**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: *(mid)* **P1**

Specify radius of circle or [Diameter] <0.1000>: *(end)* **P2**

(When possible, it is always better to show AutoCAD a distance rather then entering it on the keyboard.)

Command: **C**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: *(mid)* **P1**

Specify radius of circle or [Diameter] <0.7500>: **D**

Specify diameter of circle <1.5000>: **1**

Command: **F**

Current settings: Mode = TRIM, Radius = 0.0000

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **R**

Specify fillet radius <0.0000>: **0.75**

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **P3**

Select second object or shift-select to apply corner: **P4**

Command: **F**

Current settings: Mode = TRIM, Radius = 0.7500

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **R**

Specify fillet radius <0.7500>: **0.5**

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **P5**

Select second object or shift-select to apply corner: **P6**

Command:

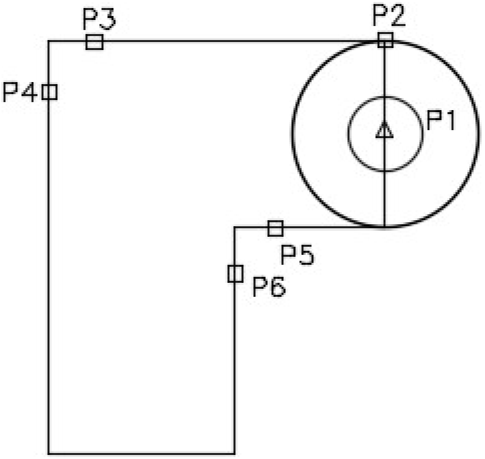

P3
P2
P4
P1
P5
P6

Figure Step 5A

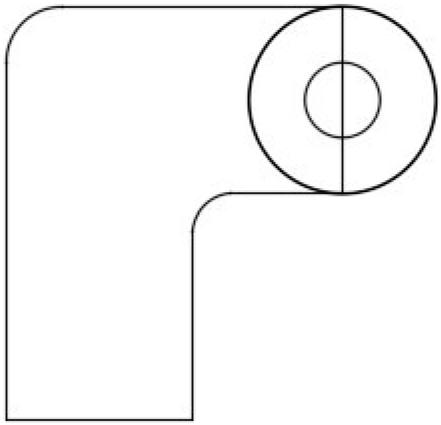


Figure Step 5B

Step 6

Use the TRIM command to trim the circle. (Figure Step 6)

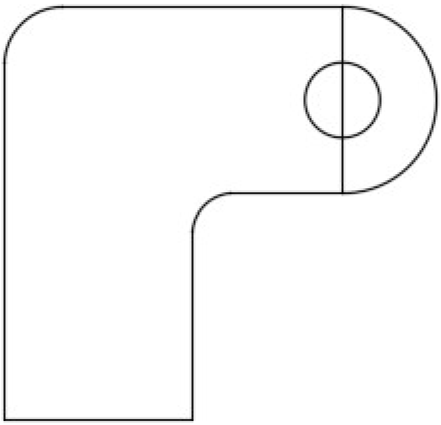


Figure Step 6

Step 7

Enter the OFFSET commands, as shown below. (Figure Step 7)

Command: **OFFSET**

Current settings: Erase source=No Layer=Source OFFSETGAPTYPE=0

Specify offset distance or [Through/Erase/Layer] <Through>: **0.5**

(Set the offset distance.)

Select object to offset or [Exit/Undo] <Exit>: **P10**

(Select the object to offset.)

Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: **P11**

(Select which side of the object to offset it on.)

Select object to offset or [Exit/Undo] <Exit>: **P12**

Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: **P13**

Select object to offset or [Exit/Undo] <Exit>:**P14**

Command: **OFFSET**

Current settings: Erase source=No Layer=Source OFFSETGAPTYPE=0

Specify offset distance or [Through/Erase/Layer] <0.5000>: **1.5**

(Set the offset distance.)

Select object to offset or [Exit/Undo] <Exit>: **P14**

(Select the object to offset.)

Specify point on side to offset or [Exit/Multiple/Undo] <Exit>:**P15**

(Select which side of the object to offset it on.)

Select object to offset or [Exit/Undo] <Exit>:

Command: **OFFSET**

Current settings: Erase source=No Layer=Source OFFSETGAPTYPE=0

Specify offset distance or [Through/Erase/Layer] <1.5000>: **2.75**

Select object to offset or [Exit/Undo] <Exit>: **P16**

Specify point on side to offset or [Exit/Multiple/Undo]

<Exit>:**P17**

Select object to offset or [Exit/Undo] <Exit>:

Command:

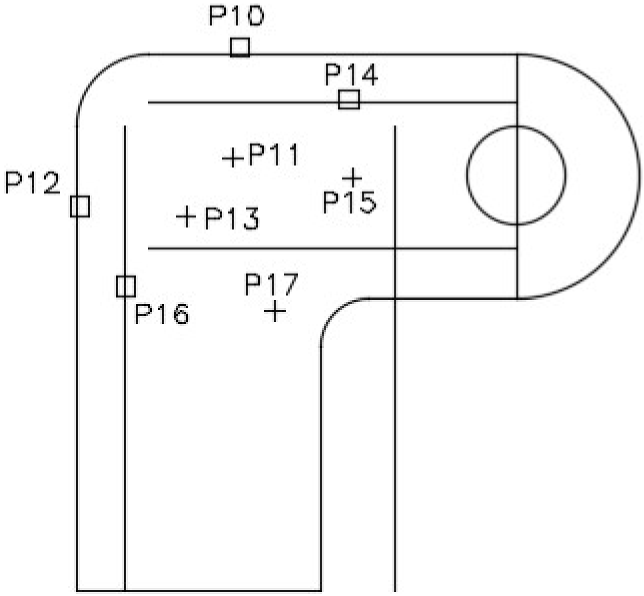


Figure Step 7

Step 8

Using the FILLET command, insert a 0.5 radius fillet on all four corners. (Figure Step 8)

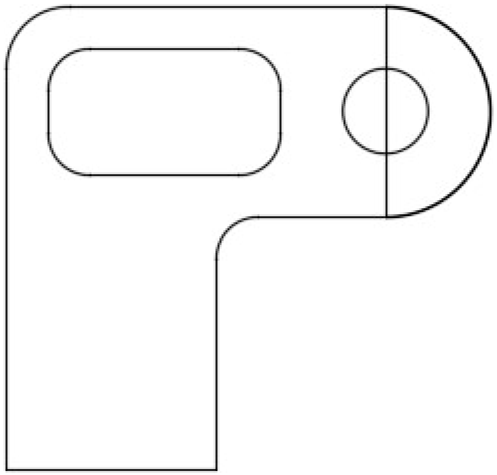


Figure Step 8

Step 9

Enter the OFFSET command, as shown below. (Figure Step 9)

```
Command: O
Current settings: Erase source=No Layer=Source
OFFSETGAPTYPE=0
Specify offset distance or [Through/Erase/Layer] <Through>: 0.5
Select object to offset or [Exit/Undo] <Exit>: P26
Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: P27
Select object to offset or [Exit/Undo] <Exit>: P28
Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: P29
Select object to offset or [Exit/Undo] <Exit>: P30
Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: P31
Select object to offset or [Exit/Undo] <Exit>:
Command:
```

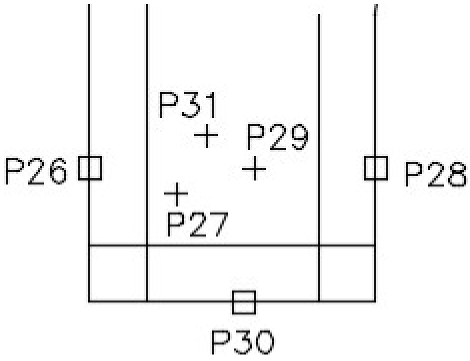


Figure Step 9

Step 10

Using what you already learned, trim the lines to finish the bottom cutout. (Figure Step 10A and 10B)

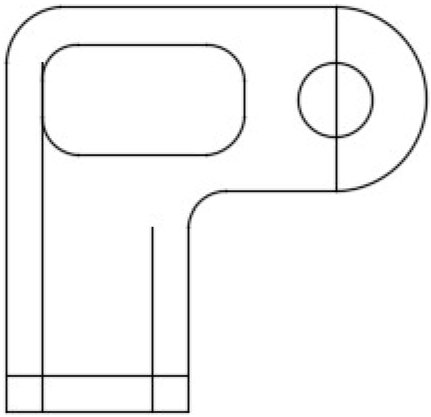


Figure Step 10A

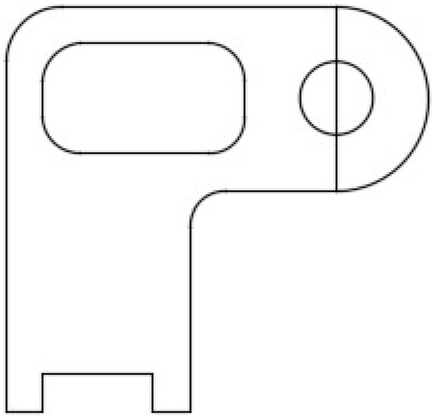


Figure Step 10B

Step 11

Enter the OFFSET command, as shown below, to offset the inside feature. (Figure Step 11A and 11B)

```
Command: O

Current settings: Erase source=No Layer=Source OFFSETGAPTYPE=0

Specify offset distance or [Through/Erase/Layer] <Through>: 0.25

Select object to offset or [Exit/Undo] <Exit>: P32

Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: P33

Select object to offset or [Exit/Undo] <Exit>: P34

Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: P33

Select object to offset or [Exit/Undo] <Exit>: P35

Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: P33

Select object to offset or [Exit/Undo] <Exit>: P36
```

```
Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: P33

Select object to offset or [Exit/Undo] <Exit>: P37

Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: P33

Select object to offset or [Exit/Undo] <Exit>: P38

Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: P33

Select object to offset or [Exit/Undo] <Exit>: P39

Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: P33

Select object to offset or [Exit/Undo] <Exit>: P40

Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: P33

Select object to offset or [Exit/Undo] <Exit>:

Command:
```

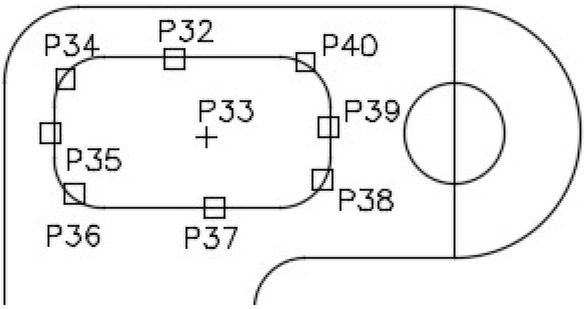


Figure Step 11A

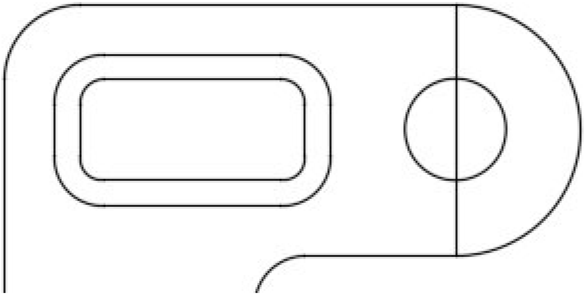


Figure Step 11B

USER TIP: It is faster to draw circles than it is to draw arcs. To improve your drawing speed, when an arc is required draw a circle instead. Then trim the circle to complete the required arc.

Step 12

Use the ID command to establish a lastpoint. Enter the CIRCLE command immediately following to insert the four circles using the lastpoint as a reference. (Figure Step 12)

Command: **ID**

Specify point: *(end)* **P41**

X = 1.5000

Y = 4.000

Z = 0.0000

(The ID command establishes a known reference point or an @ or lastpoint.)

Command: **C**

CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **@0.75,1.25**

(Now the @ can be used to measure the center of the circle from the end of the line selected in the ID command.)

Specify radius of circle or [Diameter]: **D**

Specify diameter of circle: **0.25**

Command: **C**

CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **@1,0**

Specify radius of circle or [Diameter] <0.1250>: *(Accept the default.)*

Command: **C**

CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **@0,1**

Specify radius of circle or [Diameter] <0.1250>: *(Accept the default.)*

Command: **C**

CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **@-1,0**

Specify radius of circle or [Diameter] <0.1250>: *(Accept the default.)*

Command:

AUTHOR'S COMMENTS: Using the ID command to set a reference point is a very useful tool for you. Once you pick an ID point, ignore the actual coordinates. On the next command, use an @ to locate a coordinate location from the ID point. Make sure you understand the concept.

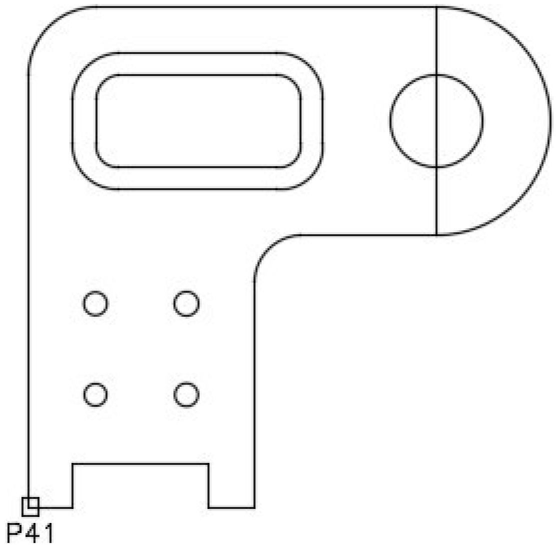


Figure Step 12

Step 13

Change the layer of the objects that you want to reside on layer: Object.

Step 14

Freeze layer: Construction and your drawing should appear as shown in the figure. (Figure Step 14)

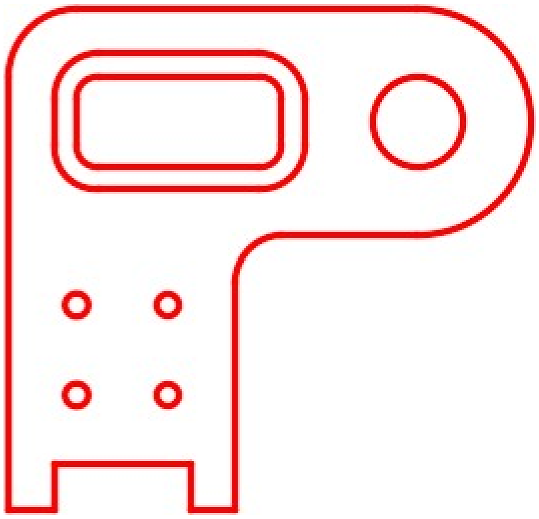


Figure Step 14

Step 15

Save and close the drawing.

WORKALONG: Drawing Construction Techniques

Step 1

Start a new drawing using the template: 2D English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 15-2. (Figure Step 2)

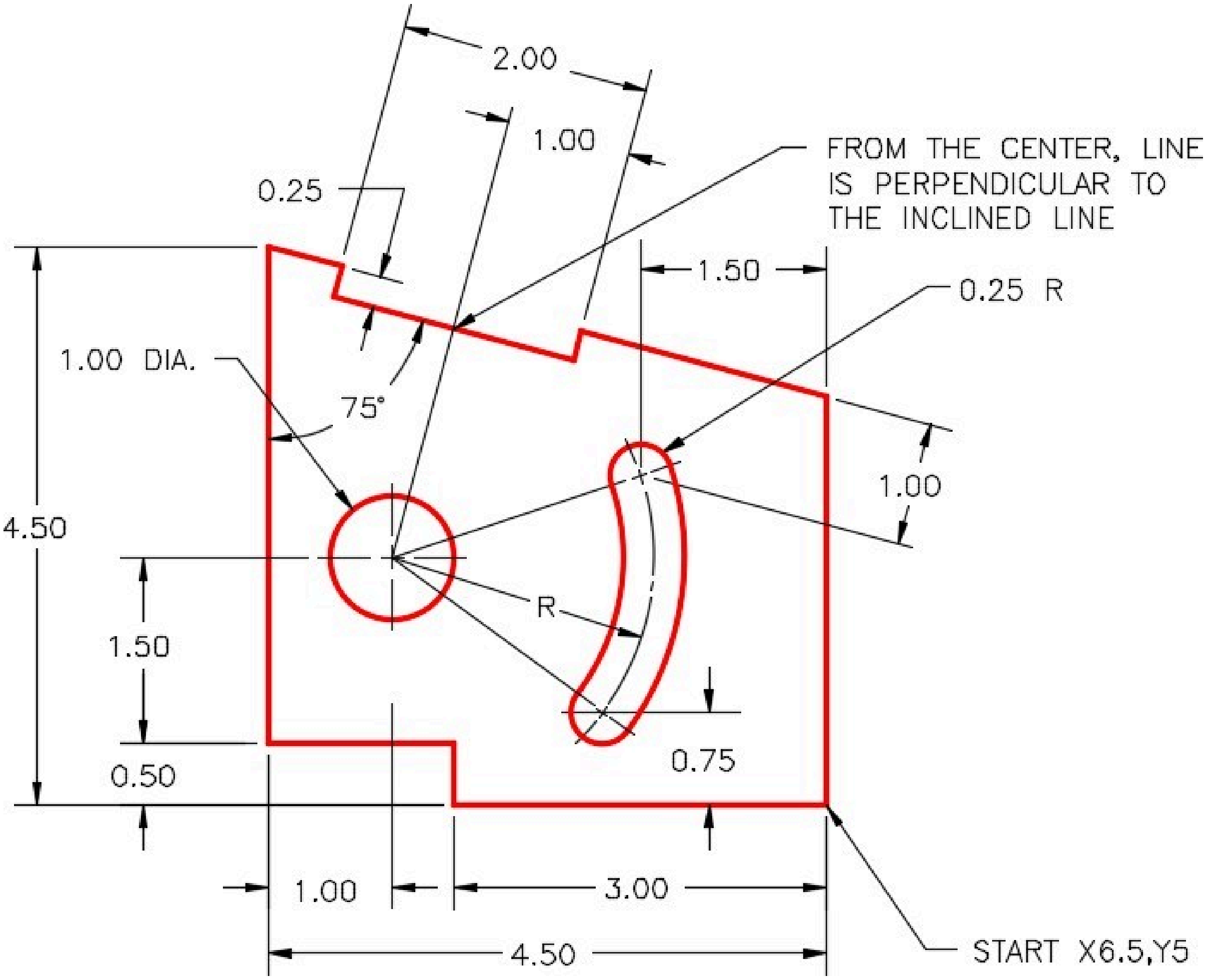

2.00
1.00
0.25
FROM THE CENTER, LINE IS PERPENDICULAR TO THE INCLINED LINE
1.50
0.25 R
1.00 DIA.
75°
4.50
1.00
R
1.50
0.75
0.50
1.00
3.00
4.50
START X6.5,Y5

Figure Step 2

Step 3

Create the layers: Object and Construction. Set layer: Construction as the current layer.

Step 4

Using Figure Step 2 as an reference, draw lines to construct the perimeter of the object. (Figure Step 4)

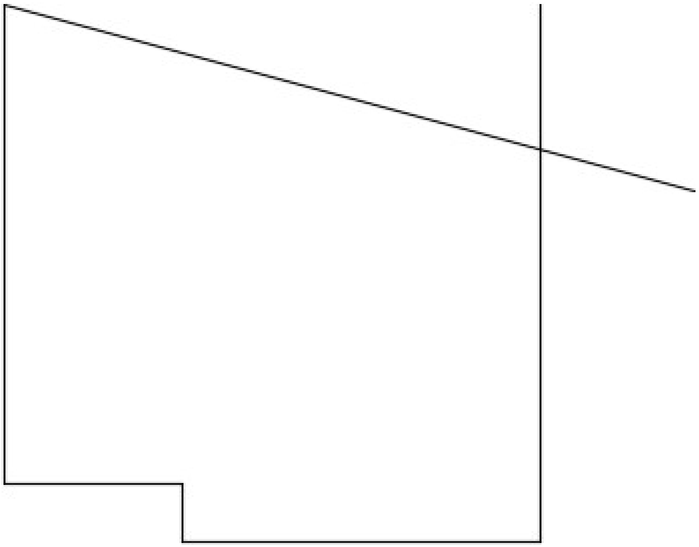


Figure Step 4

Step 5

Using what you already learned, trim the lines to complete the outline. (Figure Step 5)

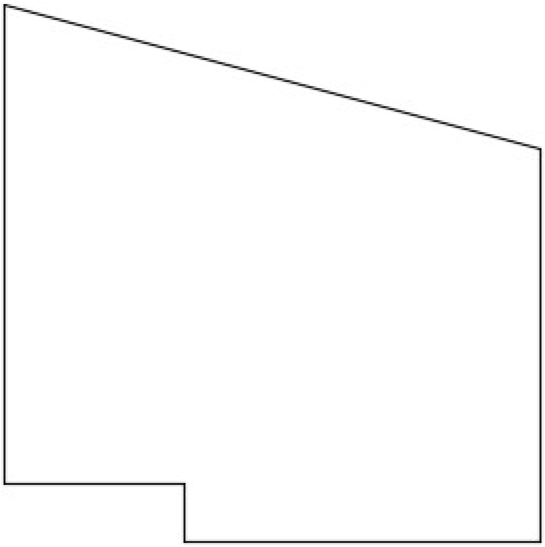


Figure Step 5

Step 6

Enter the ID command, as shown below, to establish a lastpoint. Draw the circle using the reference point you established with the ID command. (Figure Step 6)

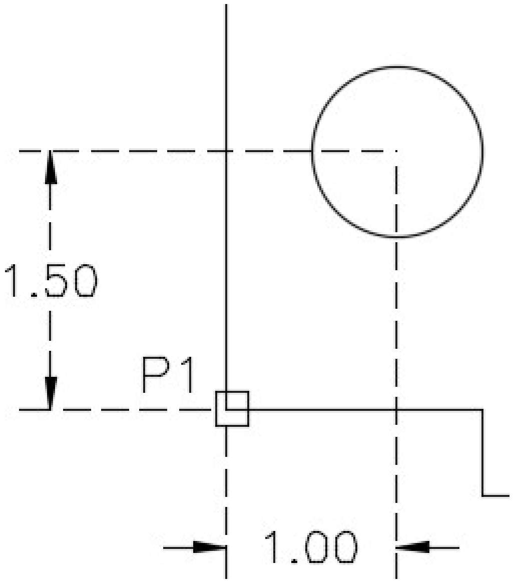


Figure Step 6

Command: **ID**

pecify point: *(end)* **P1**

X = 2.0000

Y = 5.5000

Z = 0.0000

(The lastpoint or the @ being established.)

Command: **C**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: @**1,1.5**

(The center of the circle is now located using the reference point established in the ID command above. These commands must be entered back to back.)

Specify radius of circle or [Diameter] <2.6770>: **D**

Specify diameter of circle <5.3539>: **1**

Command:

Step 7

Enter the OFFSET command, as shown below, to insert the inside offsets. (Figure Step 7)

Command: **O**

(Shortcut for the OFFSET command)

Current settings: Erase source=No Layer=Source OFFSETGAPTYPE=0

Specify offset distance or [Through/Erase/Layer] <0.00>: **0.75**

(Setting the offset distance.)

Select object to offset or [Exit/Undo] <Exit>: **P2**

Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: **P3**

(First the object is selected (P2) and then the side of the object (P3) to place the offset. The location for P3 can be anywhere as long as it is above the line to be offset.)

Select object to offset or [Exit/Undo] <Exit>:

(With what you already learned, finish the other two offsets as shown in Figure Step 7.)

Command:

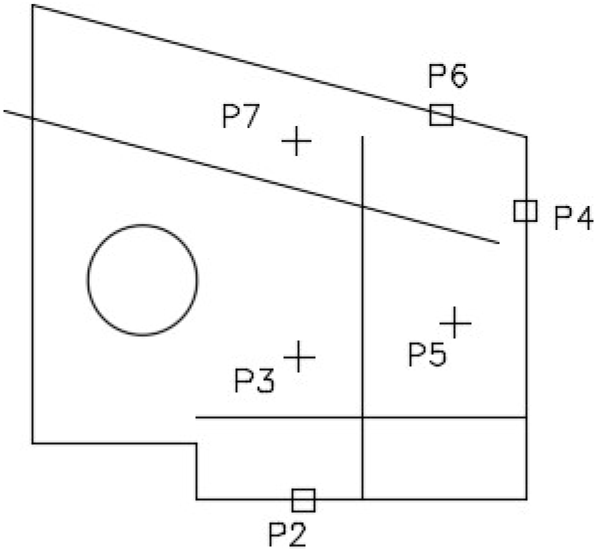


Figure Step 7

Step 8

Enter the CIRCLE command, as shown below, to insert a circle by snapping its center to the intersection of the offset lines. (Figure Step 8)

Command: **C**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: *(int)* **P8**

(Snap to locate the center of the circle at the intersection of the two offset lines.)

Specify radius of circle or [Diameter] <0.5000>: **.25**

Command:

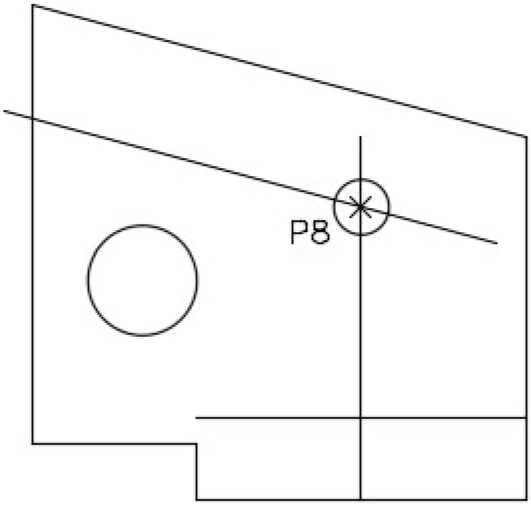


Figure Step 8

Step 9

Draw a line between the centers of the circles. (Figure Step 9)

Command: **L**

Specify first point: *(cen)* **P9**

Specify next point or [Undo]: *(cen)* **P10**

(Draw a construction line by snapping to center point of the circles.)

Specify next point or [Undo]:

Command:

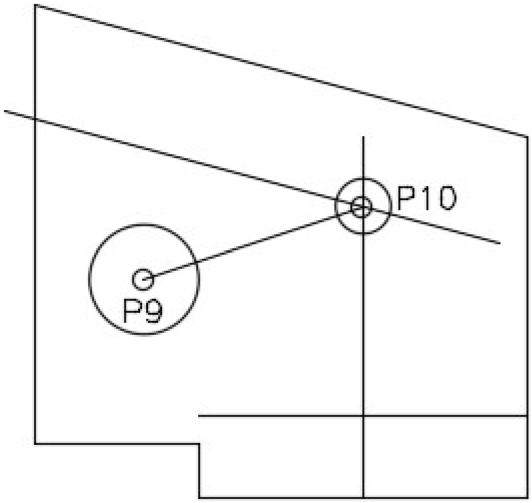


Figure Step 9

Step 10

Extend the construction line to the circle`s circumference. (Figure Step 10A and 10B)

Command: **EX**

Current settings: Projection=UCS, Edge=Extend

Select boundary edges …

Select objects or <select all>: **P11** 1 found

Select objects:

(Press enter or space.)

Select object to extend or shift-select to trim or

[Fence/Crossing/Project/Edge/Undo]: **P12**

Select object to extend or shift-select to trim or [Fence/Crossing/Project/Edge/Undo]:

Command:

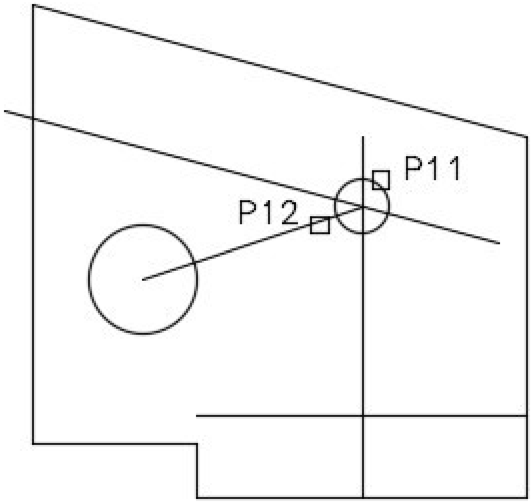


Figure Step 10A

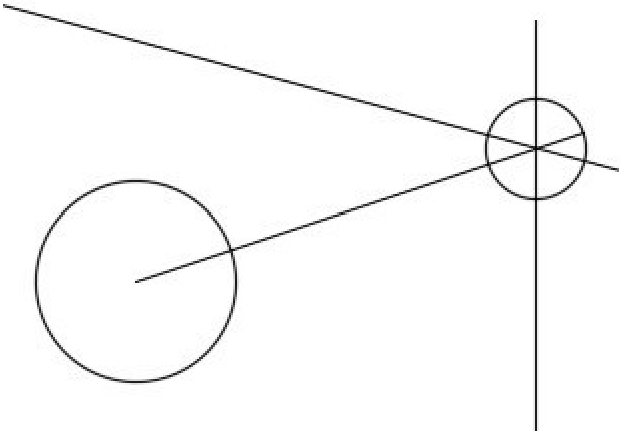


Figure Step 10B

AUTHOR'S COMMENTS: The construction line was drawn from center to center and then extended to the circumference of the circle to locate the tangent point for the arc to be inserted. A line between centers locates the tangent point of the arcs.

Step 11

Draw a circle snapping to the center of the large circle for its center location and snapping to the intersection of the construction line and the lower circumference of the small circle for the radius. (Figure Step 11)

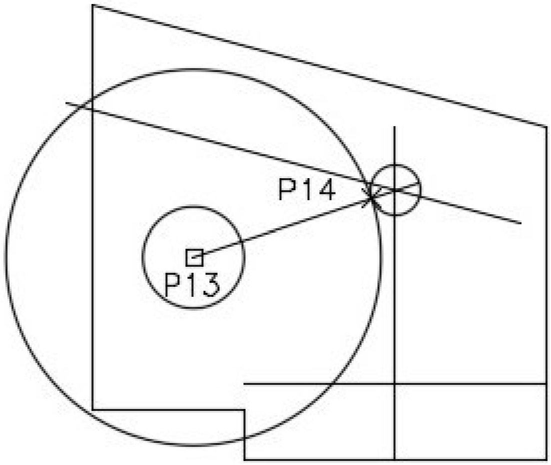


Figure Step 11

AUTHOR'S COMMENTS: If possible, always show AutoCAD a measurement instead of entering the number on the keyboard. Always snap to exiting geometry when you do this.

Step 12

Using the same principle that you learned in Step 11, insert the other two circles. Ensure that you snap one circle to the center of the small circle and the other to the intersection of the tangent line and circle circumference. (Figure Step 12A and 12B)

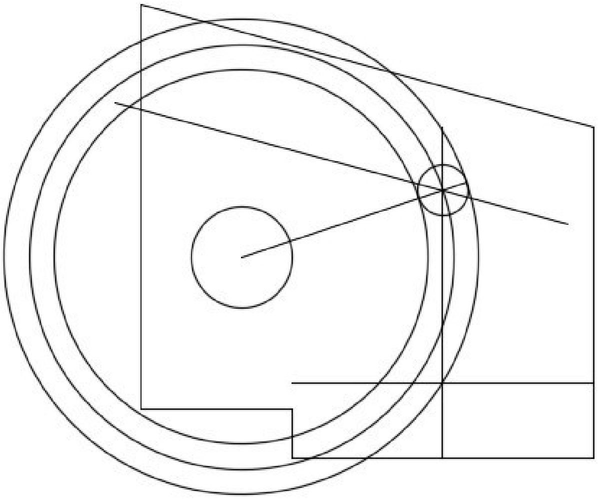


Figure Step 12A

AUTHOR'S COMMENTS: Circles were used rather then arcs because it is easier to draw circles than arcs. Once the circle is drawn, it can easily be trimmed to complete the arc. This is a much faster drawing technique.

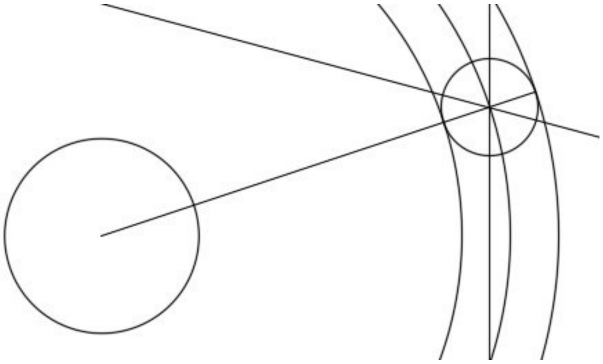


Figure Step 12B

AUTHOR'S COMMENTS: In Step 11 and 12, you are showing AutoCAD the radius of the circles by snapping to the intersection of the construction line and circle. Whenever you can, always show AutoCAD and distance rather then typing it on the keyboard.

Step 13

Draw a circle locating its center at the intersection of the large circle's centerline and the horizontal construction line. (Figure Step 13)

Command: **C**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: (int) **P15**

Specify radius of circle or [Diameter]: **.25**

Command:

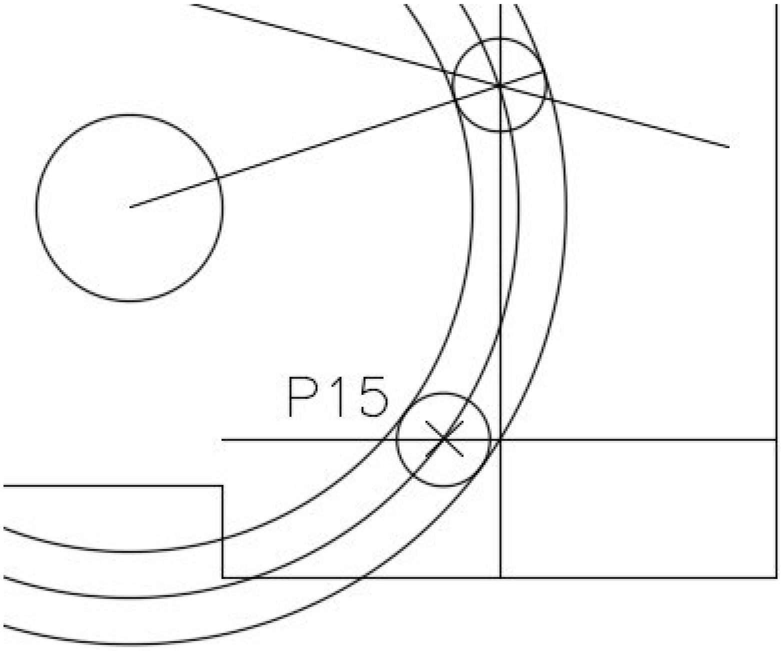


Figure Step 13

Step 14

Using what you already learned, trim the circles to complete the slot. (Figure Step 14)

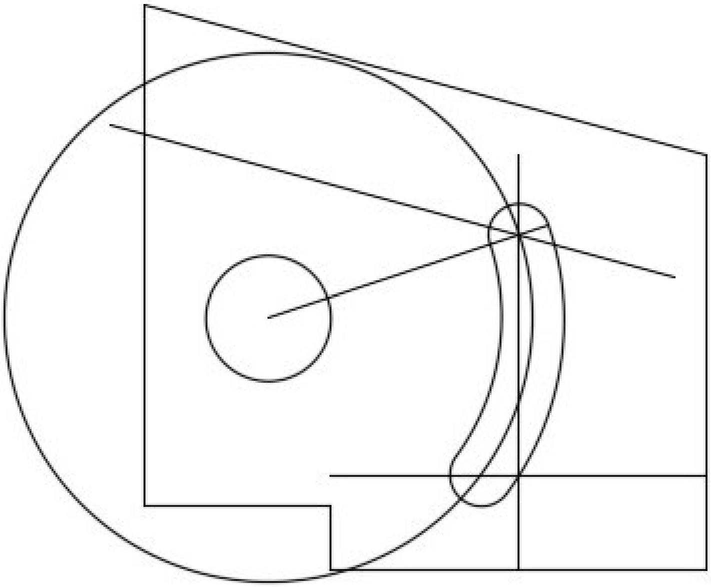


Figure Step 14

Step 15

Enter the LINE command, as shown below, to draw a line from the center of the larger circle and snapping perpendicular to the inclined line. (Figure Step 15)

Command: **L**

Specify first point: (cen) **P16**

Specify next point or [Undo]: **perp** to **P17**

(It is much faster to type in the osnap mode perp then setting it in Autosnap.)

Specify next point or [Undo]:

Command:

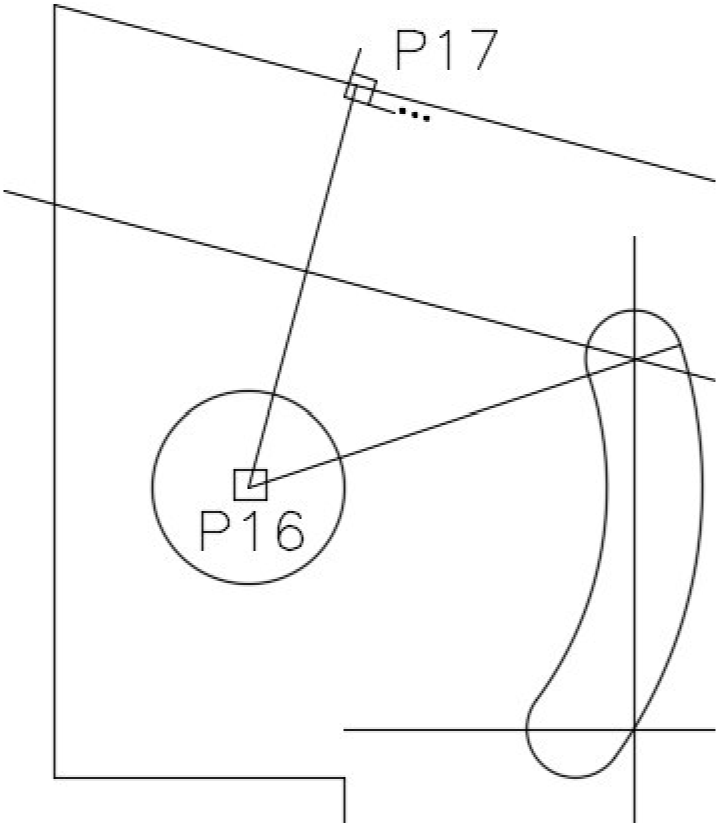


Figure Step 15

Step 16

Using what you already learned, offset the lines to create top feature. (Figure Step 16)

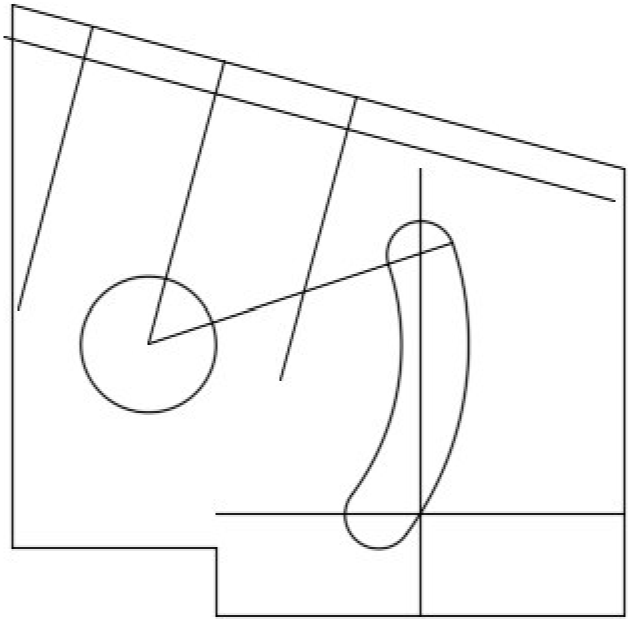


Figure Step 16

Step 17

Trim the lines to complete the top cutout. (Figure Step 17)

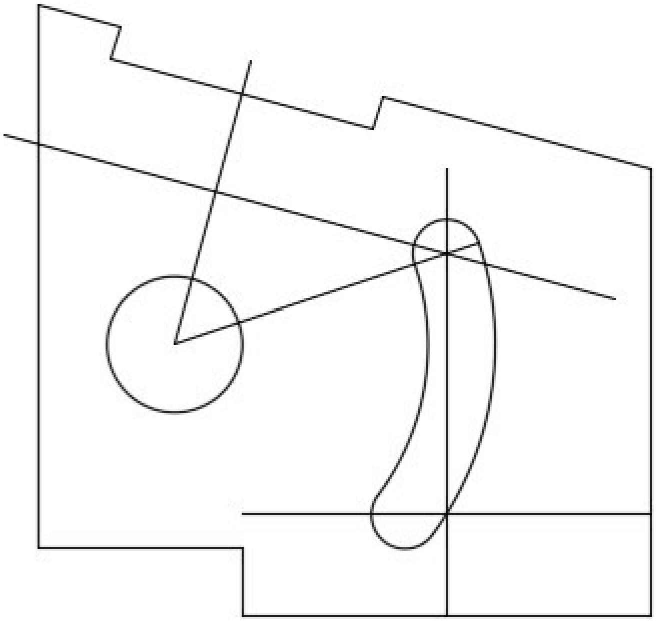


Figure Step 17

Step 18

Change the layer of the objects to layer: Object. Freeze layer: Construction to complete the drawing. (Figure Step 18)

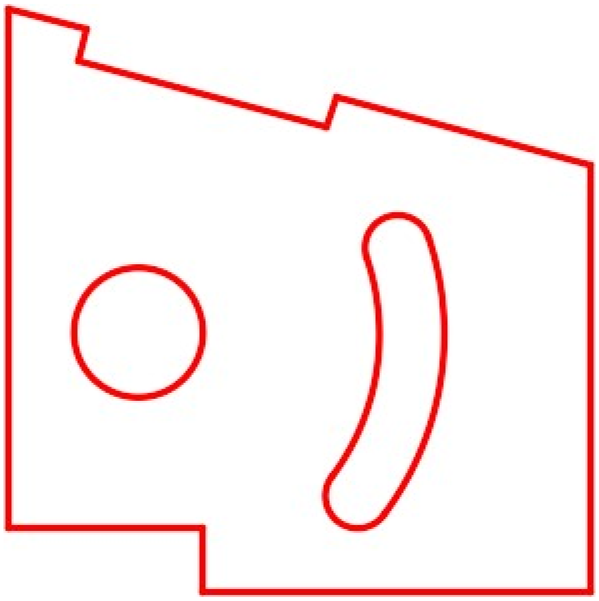


Figure Step 18

Step 19

Save and close the drawing.

Key Principles

Key Principles in Module 15

1. The OFFSET command is a very useful command. When used at the right times, it will improve your drawing speed and change the way you construct objects in AutoCAD.
2. When a drawing object is offset, the newly constructed offset drawing object will reside on the same layer as the original object, regardless of the current layer.
3. The ID command can be used to established a known reference point or an @. The reference point can then be used, in the next command, to measure from.

Lab Exercise 15-1

Time allowed: 40 minutes.

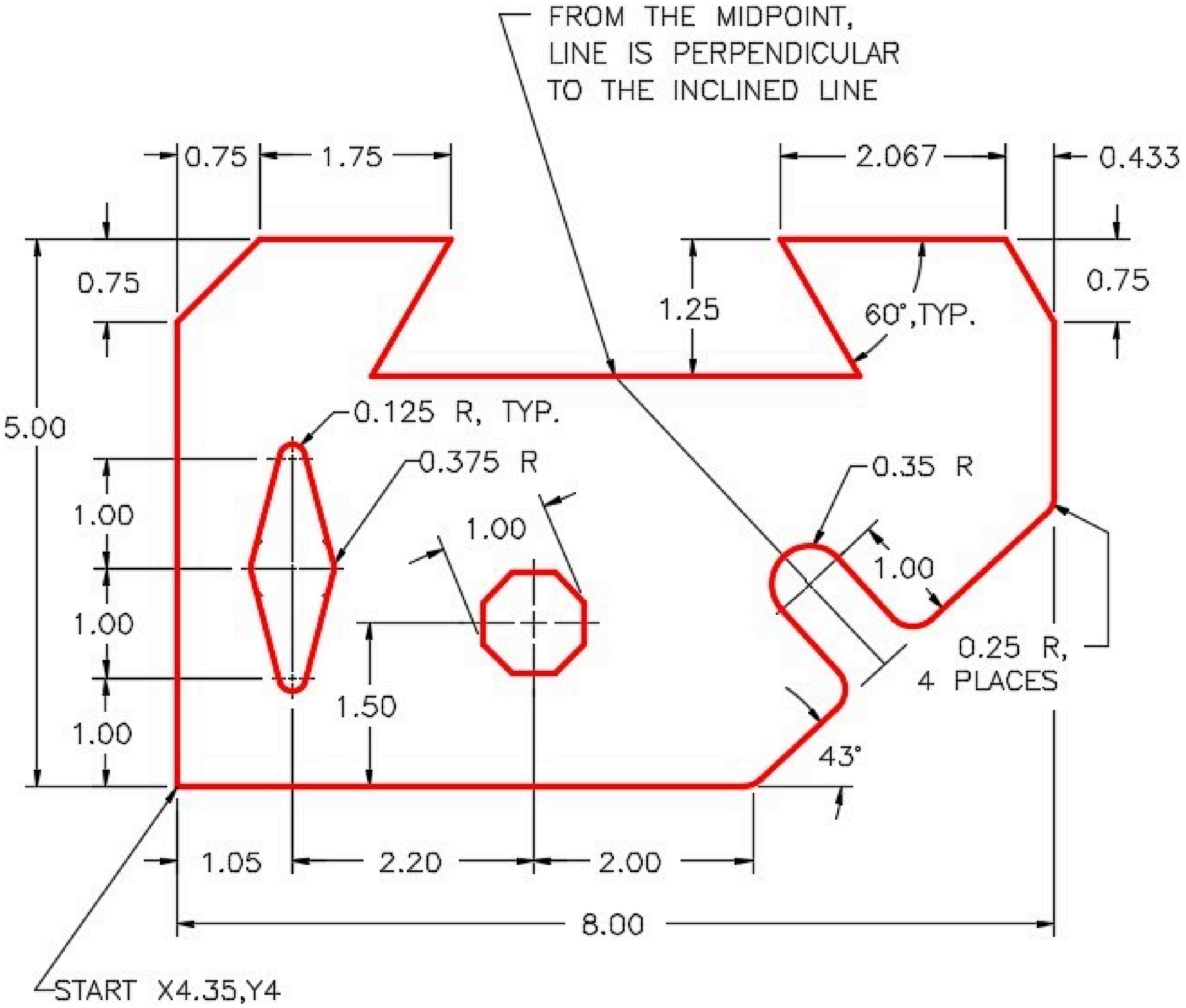
Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 15-1	2D English	Inches

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Construction	Construction objects	253
Object	All objects	Red

Step 1

Draw the object shown in the figure using the layering scheme. (Figure Step 1A and 1B)


FROM THE MIDPOINT, LINE IS PERPENDICULAR TO THE INCLINED LINE
0.75
1.75
2.067
0.433
0.75
0.75
1.25
60°,TYP.
5.00
0.125 R, TYP.
0.375 R
0.35 R
1.00
1.00
1.00
1.00
1.00
1.00
0.25 R, 4 PLACES
1.50
43°
1.05
2.20
2.00
8.00
START X4.35,Y4

AUTHOR'S COMMENTS: Do not delete the construction lines.

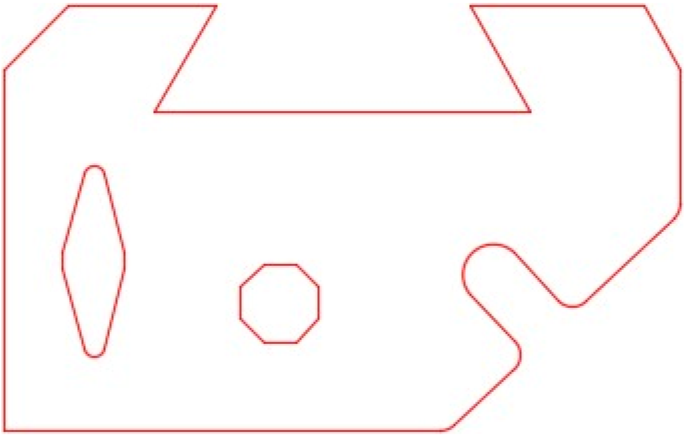


Figure Step 1B Completed Drawing

Step 2

Set the insertion units and check the drawing's accuracy.

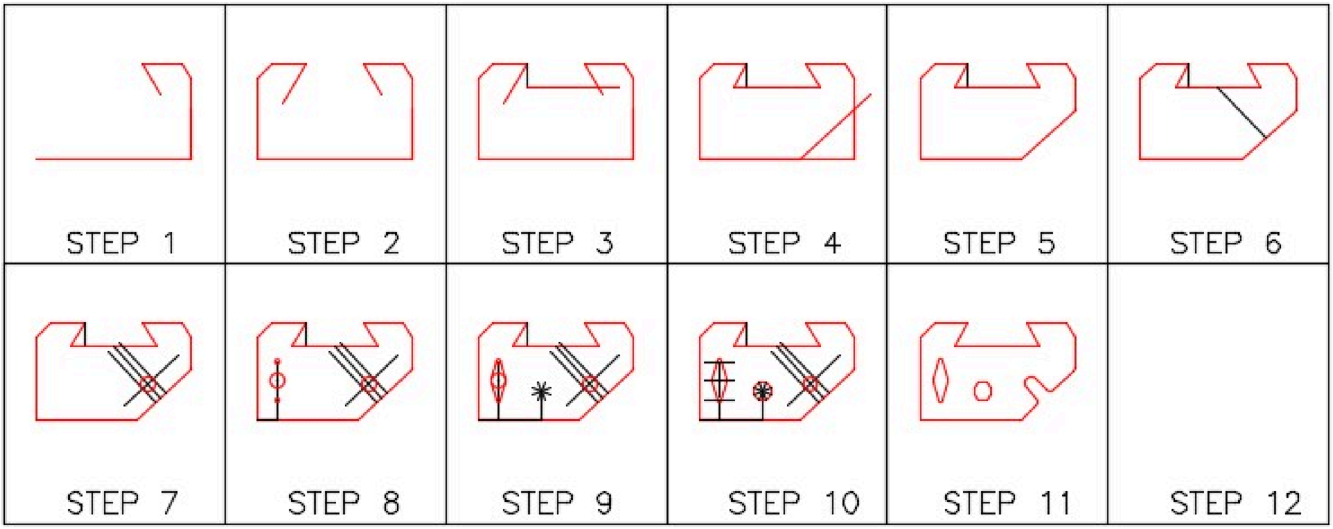
Step 3

Turn layer: Key off and freeze layer: Construction.

Step 4

Save and close the drawing.

AUTHOR'S CONSTRUCTION TECHNIQUES: The following steps are the construction technique suggested by the author to help you learn how to construct objects using AutoCAD. It is only the suggested method and if you can complete the drawing accurately using a different construction technique, that is what is important. You may want to compare your construction technique with the authors.


STEP 1
STEP 2
STEP 3
STEP 4
STEP 5
STEP 6
STEP 7
STEP 8
STEP 9
STEP 10
STEP 11
STEP 12

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

Draw the line perpendicular to inclined line and then offset the inclined line. Draw a circle with its center located at the intersection of the lines. Offset the perpendicular line using the radius of the circle as the distance and trim. (Figure Hint 1)

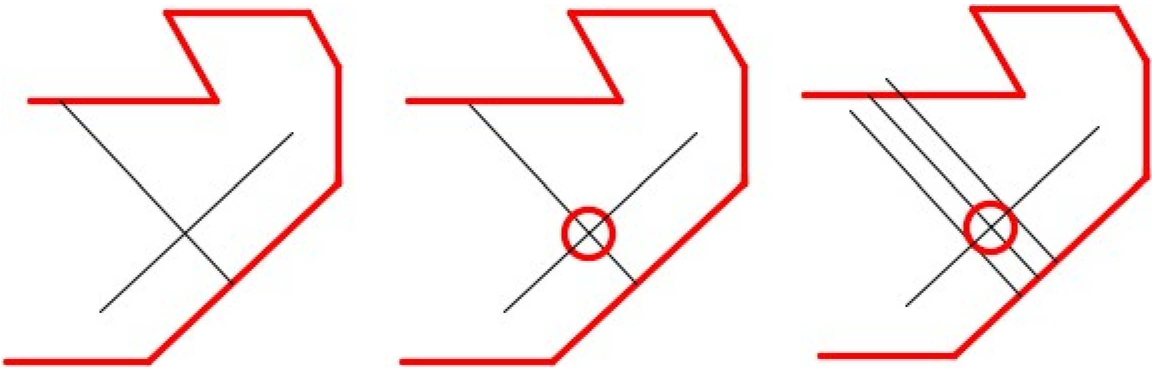


Figure Hint 1

Hint 2

Insert offset lines as construction lines to locate the centers of the three circles. Then draw the lines tangent to the circles. (Figure Hint 2)

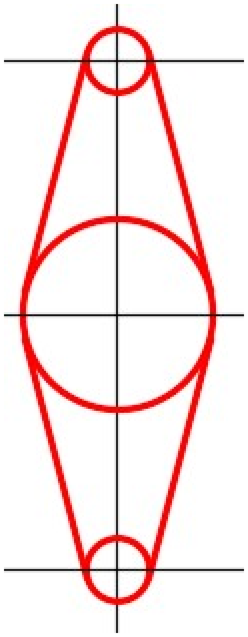


Figure Hint 2

Hint 3

The figure shows the construction lines. (Figure Hint 3)

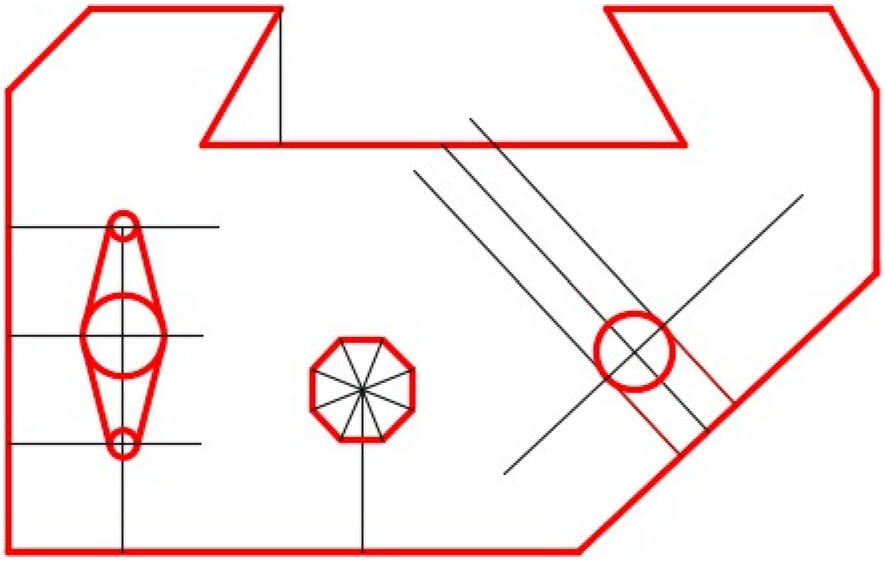


Figure Hint 3

Lab Exercise 15-2

Time allowed: 60 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 15-2	2D Metric	Millimeters

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Construction	Construction objects	253
Object	All objects	Red

Step 1

Draw the object shown in the figure using the layering scheme. (Figure Step 1A and 1B)

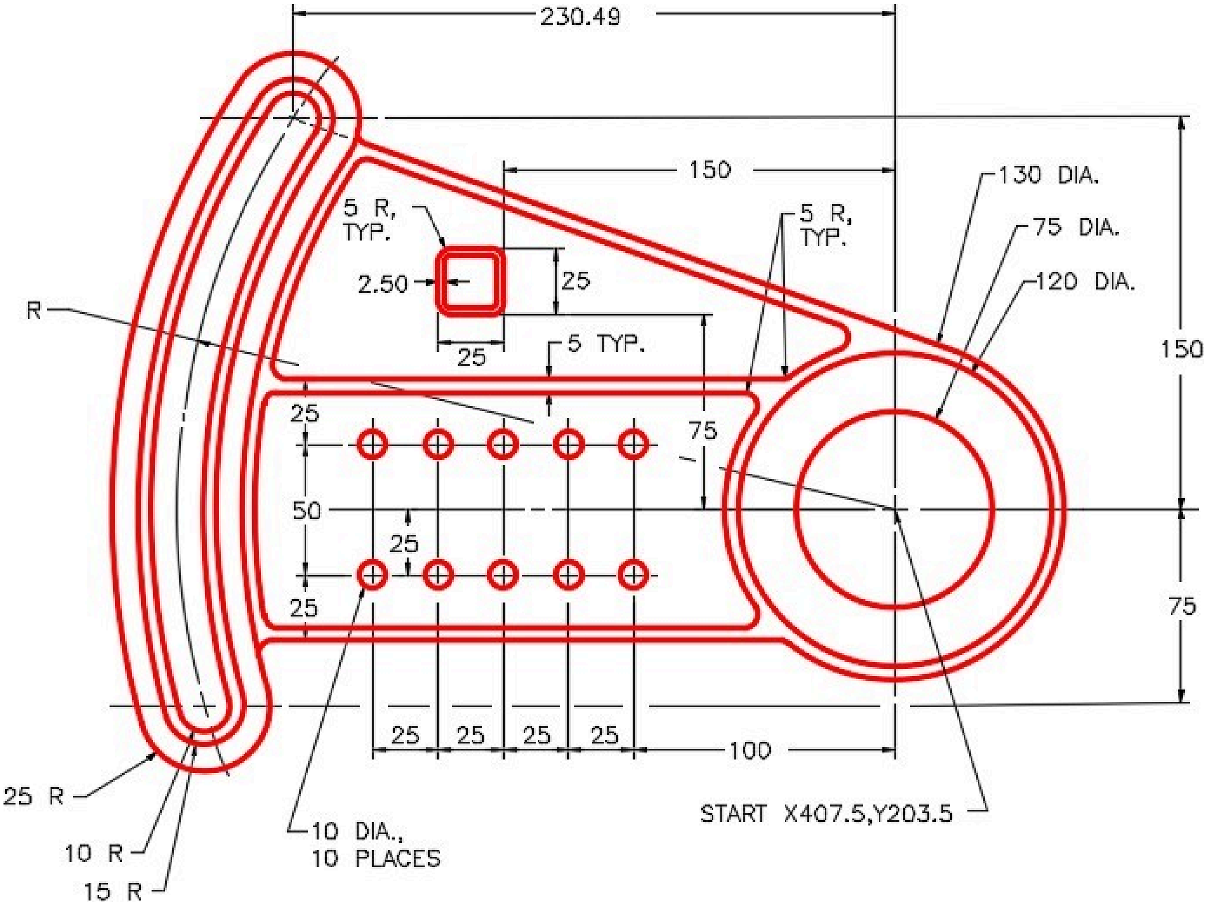

230.49
150
5 R,
TYP.
5 R,
TYP.
130 DIA.
75 DIA.
120 DIA.
2.50
25
R
25
5 TYP.
150
75
25
50
25
25
75
25
25
25
25
100
25 R
10 R
15 R
10 DIA.,
10 PLACES
START X407.5,Y203.5

Figure Step 1A

AUTHOR'S COMMENTS: Do not delete the construction lines.

Step 2

Set the insertion units and check the drawing's accuracy.

Step 3

Turn layer Key off and freeze layer Construction.

Step 4

Save and close the drawing.

AUTHOR'S CONSTRUCTION TECHNIQUES: The following steps are the construction technique suggested by the author to help you learn how to construct objects using AutoCAD. It is only the suggested method and if you can complete the drawing accurately using a different construction technique, that is what is important. You may want to compare your construction technique with the authors.

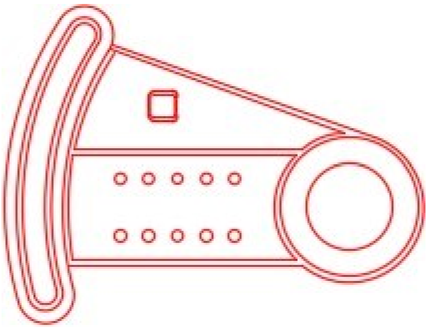
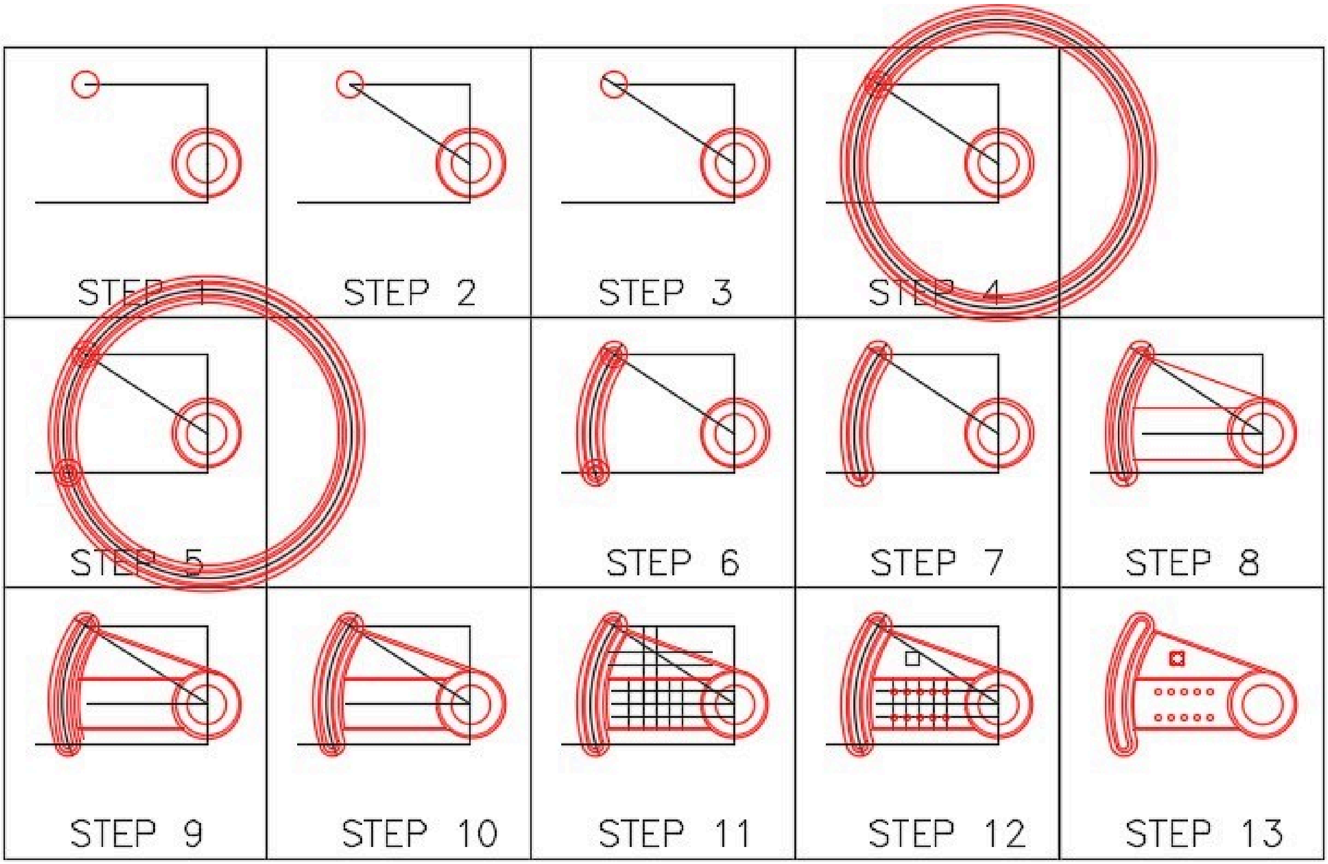


Figure Step 1B
Completed Drawing


STEP 1
STEP 2
STEP 3
STEP 4
STEP 5
STEP 6
STEP 7
STEP 8
STEP 9
STEP 10
STEP 11
STEP 12
STEP 13

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hint. If you get stuck and cannot complete it on your own, use the following hint to help you.

Hint 1

See the figure and the following seven steps:

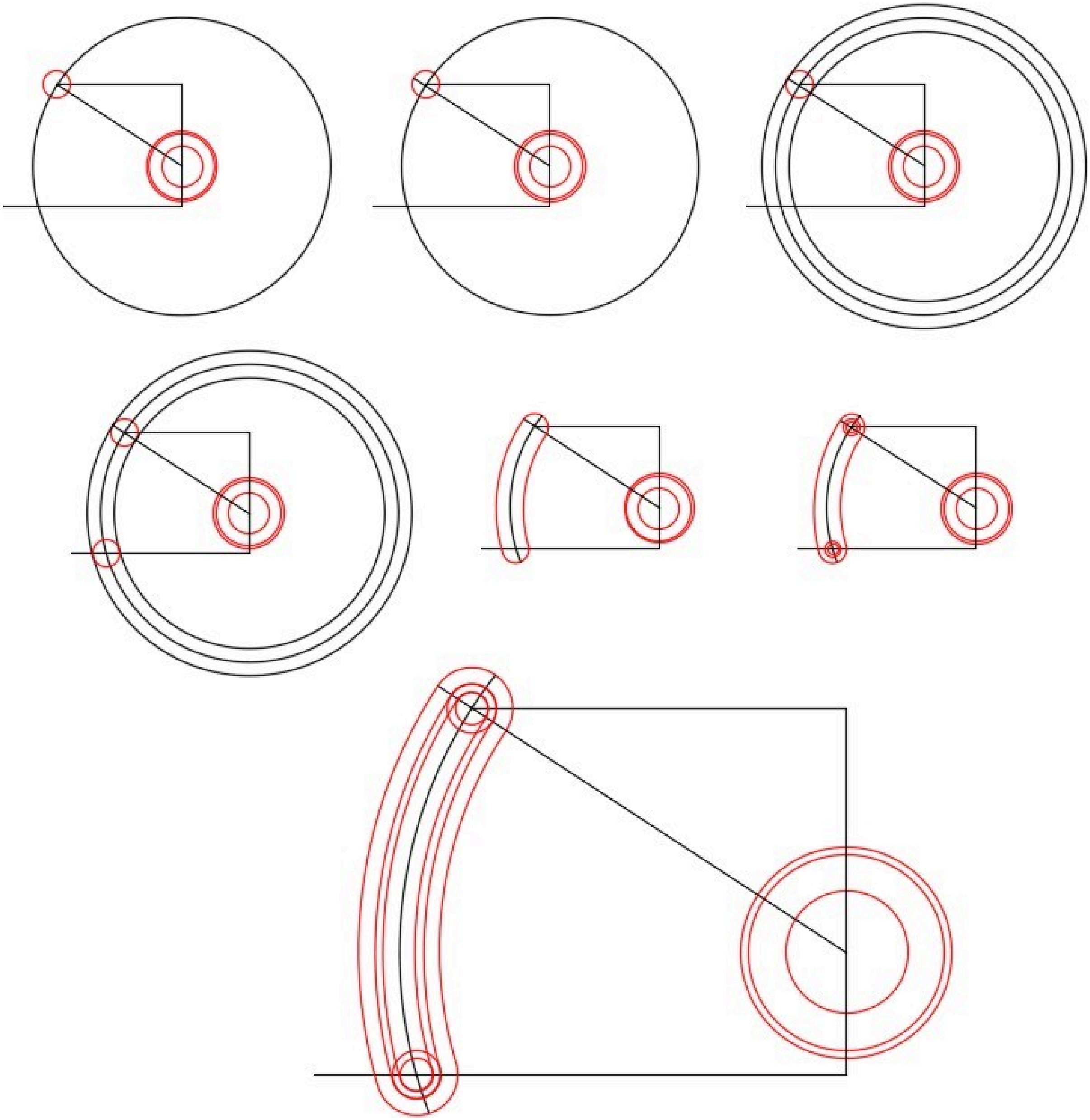


Figure Step 1

Step 1

Draw a construction line from the center of the bottom circle to the center of the upper circle.

Step 2

Extend the line to the upper circumference of the circle.

Step 3

Draw circles using the intersection of the tangent line and circle to show AutoCAD the radius.

Step 4

Draw the bottom circle at the intersection of the construction circle and line.

Step 5

Trim the circles.

Step 6

Draw the inner circles.

Step 7

Offset the arcs and trim to complete. (Figure Hint 1)

Lab Exercise 15-3

Time allowed: 40 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 15-3	2D English	Inches

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Construction	Construction objects	253
Door	All lines	Blue

Step 1

Set the LIMITS as follows:

The lower left corner to: 0.0,0.0

The upper right corner to: 100,150

Step 2

Enter the ZOOM Extents command.

Step 3

Draw the object shown in the figures using the layering scheme. (Figure Step 3A, 3B, and 3C)

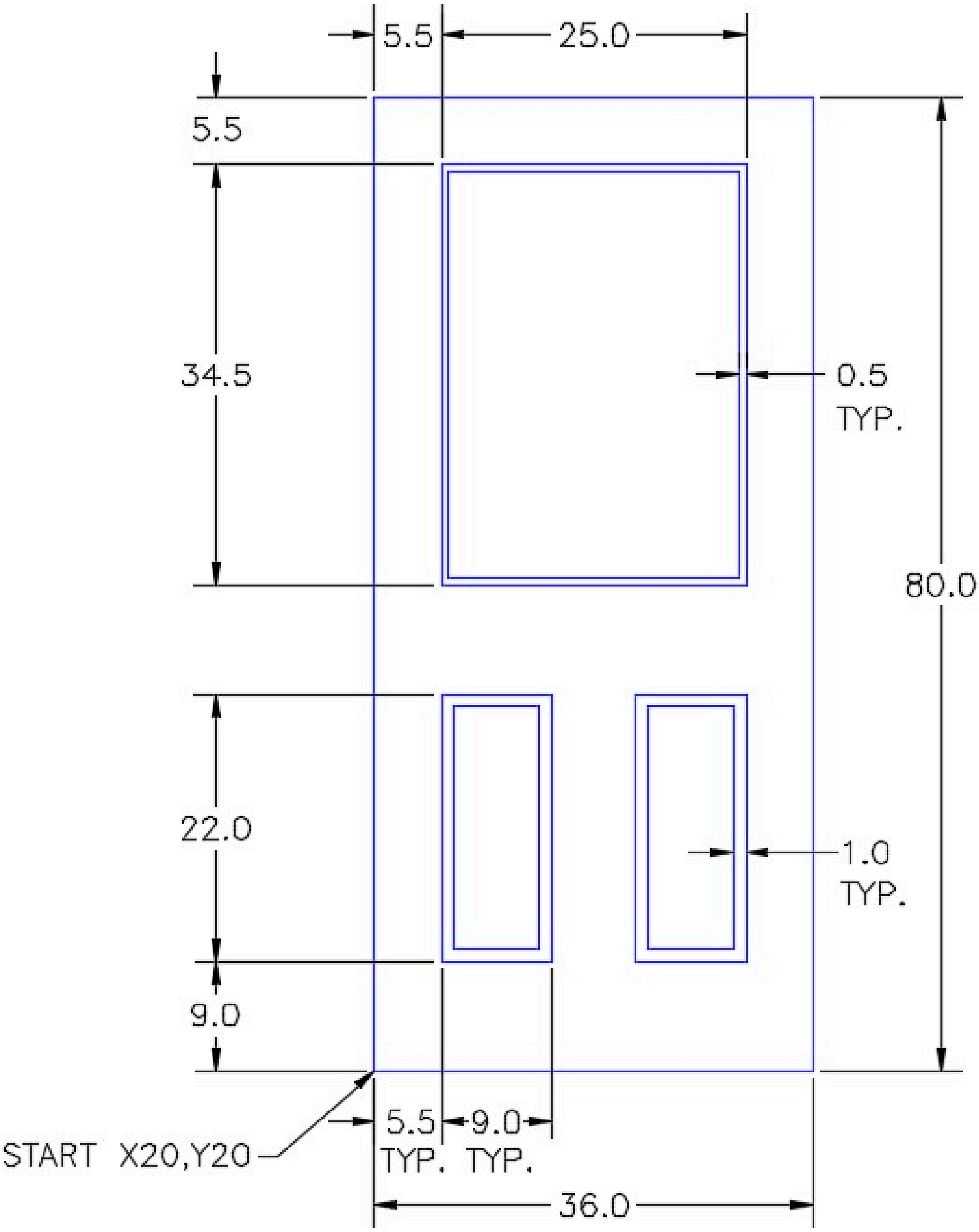

5.5
25.0
5.5
34.5
0.5
TYP.
80.0
22.0
1.0
TYP.
9.0
START X20,Y20
5.5
TYP.
9.0
TYP.
36.0

Figure Step 3A

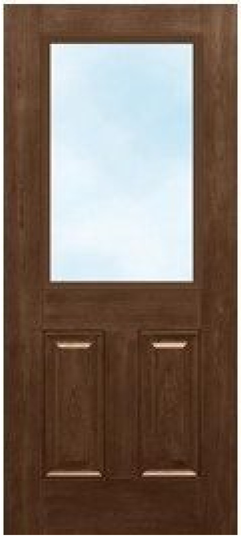


Figure Step 3B
Exterior Door

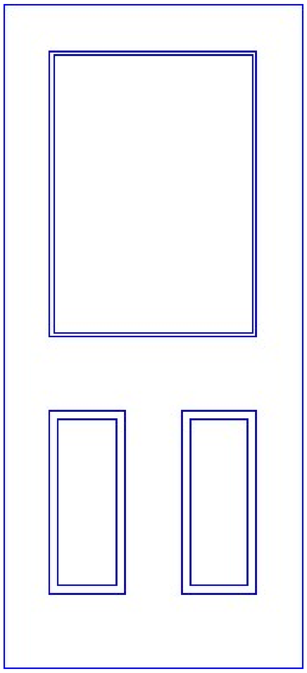


Figure Step 3C
Completed Drawing

Step 4

Set the insertion units and check your drawing with the key. Turn layer: Key off and freeze layer: Construction.

Step 5

Save and close the drawing.

Module 16: Competency Test 3 Open Book

Learning Outcomes

When you have completed this module, you will be able to:

1. Within a one-hour time limit, complete a written exam and a lab exercise without the aid of a key.

The AutoCAD 2D book was written with competency based modules. What that means is that you have not completed each module until you have mastered it. The Competency Test module contains multiple choice questions and a comprehensive lab exercise to test your mastery of the set of modules that you completed. There are no answers or keys supplied in a Competency Test module since it is meant to be checked by your instructor. If there are any parts of this module that you have trouble completing, you should go back and reread the module or modules containing the information that you are having trouble with. If necessary, redo as many lab exercises required until you fully understand the material.

If you are Completing this book:

- Without the Aid of an Instructor, complete the written test and the lab exercise.
- In a Classroom with an Instructor, the instructor will give instructions on what to do after this module has been completed.

Multiple Choice Questions

Select the BEST answer.

1. Objects can be trimmed and extended in the same command. After it is trimmed or extended, depending on the initial command entered, which key can be pressed to apply the opposite command?
    A. Shift
    B. Alt
    C. Esc
    D. Ctrl
    E. Space
2. What extension does AutoCAD automatically add to the users assigned drawing name when a drawing is saved?

- A. .drw
- B. .dgw
- C. .drg
- D. .dwg
- E. .dgn

3. Which one of the following statements best describes a good use for the ID command?
    - A. To extend an object to where it intersects another
    - B. To trim an object to where it intersects another
    - C. To insert a tangent
    - D. To change the properties of an
    - E. To set a reference
4. What command is used to change the global linetype scale for the current drawing?
    - A. LINE
    - B. PLINE
    - C. LTSCALE
    - D. LINETYPE
    - E. POLYGON
5. Which one of the following best describes a crossing window when selecting objects?
    - A. Selects all objects totally inside the window but not the ones that it crosses.
    - B. Selects all objects that it crosses and the ones outside the window.
    - C. Selects all objects totally inside the window and the ones it crosses.
    - D. Selects only the objects that totally cross the window.
    - E. Selects all the objects that are outside the window.
6. What type of AutoCAD drawing object does the FILLET command insert?
    - A. circle
    - B. pline
    - C. arc
    - D. line
    - E. point
7. What command is used to insert a tangent arc between two lines?
    - A. POLYGON
    - B. FILLET
    - C. LINE

   D. TRIM
   E. EXTEND

8. What layer will the newly constructed drawing object reside on when it is created using the OFFSET command? Select the BEST answer.
   A. It won't be on any layer.
   B. The same layer as the object being offset.
   C. On the layer you specify in the command.
   D. The current Layer
   E. Layer zero
9. Which one of the following statements best describes the OFFSET command?
   A. Creates an object perpendicular to an existing object at a specified distance.
   B. Creates an object tangent to an existing object at a specified distance.
   C. Creates an object parallel to an existing object at a specified distance.
   D. Creates an object where it intersects an existing object at a specified distance.
   E. Creates an object at the same location as the existing object.
10. What command is used to insert a line tangent to two circles?
    A. FILLET
    B. INSERT
    C. LINE
    D. EXTEND
    E. POLYGON

Lab Exercise 16-1 OPEN BOOK

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 16-1	2D English	Inches

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Construction	Construction objects	253
Object	All objects	Red

Step 1

Draw the object shown in the figure. (Figure Step 1A and 1B)

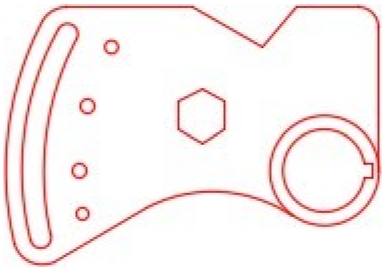


Figure Step 1A
Completed Drawing

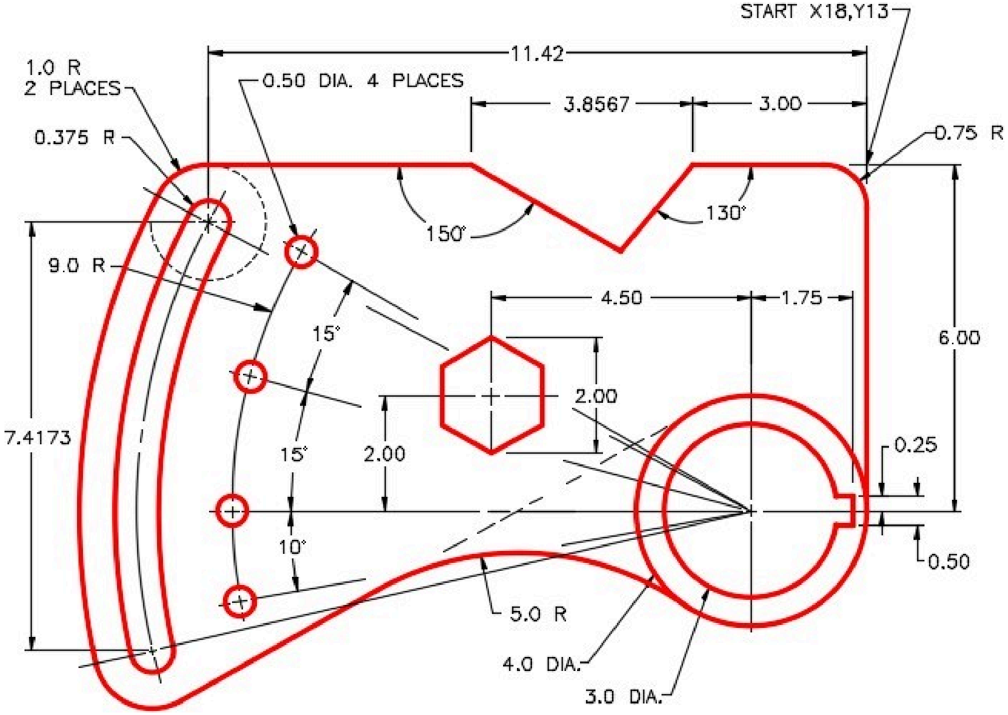

START X18,Y13
11.42
1.0 R
2 PLACES
0.50 DIA. 4 PLACES
3.8567
3.00
0.75 R
0.375 R
150°
130°
9.0 R
4.50
1.75
6.00
15°
2.00
2.00
7.4173
15°
0.25
10°
0.50
5.0 R
4.0 DIA.
3.0 DIA.

Figure Step 1B

Step 2

Draw all construction objects on layer: Construction. When complete, freeze layer: Construction.

Step 3

Save and close the drawing.

AUTHOR'S COMMENTS: There is no key for this lab. Do not delete the construction objects.

Part 4

Module 17: Inquiry and Measurement

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe the geometry of a right angle triangle.
2. Describe a point, point styles, and object snap mode node.
3. Apply the system variable SNAPANG to rotate the graphic cursor.
4. Apply the ORTHO, POINT, DIVIDE, and MEASURE commands.
5. Apply the UNITS, DIST, and PROPERTIES commands to set the display units, make inquiries about drawing objects, measure distances, and angles.

Geometry Lesson: A Right Angle Triangle

Figure 17-1 and 17-2 show the geometry of a right angle triangle. Understanding this theory will help you to complete your AutoCAD drawings.

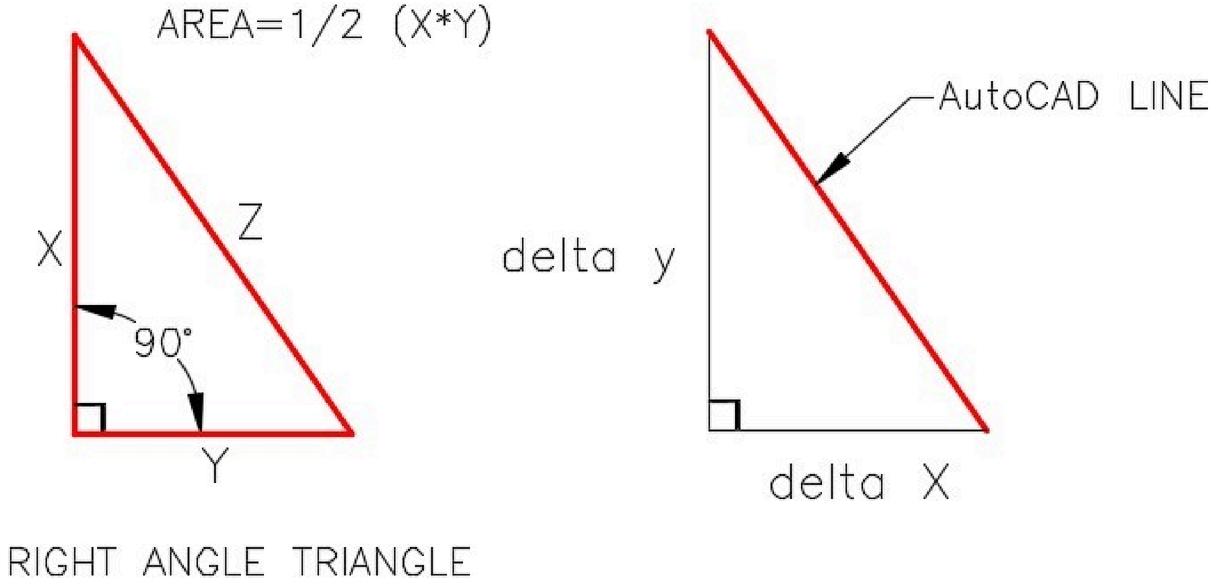


Figure 17-1
Geometry of a Right Angle Triangle

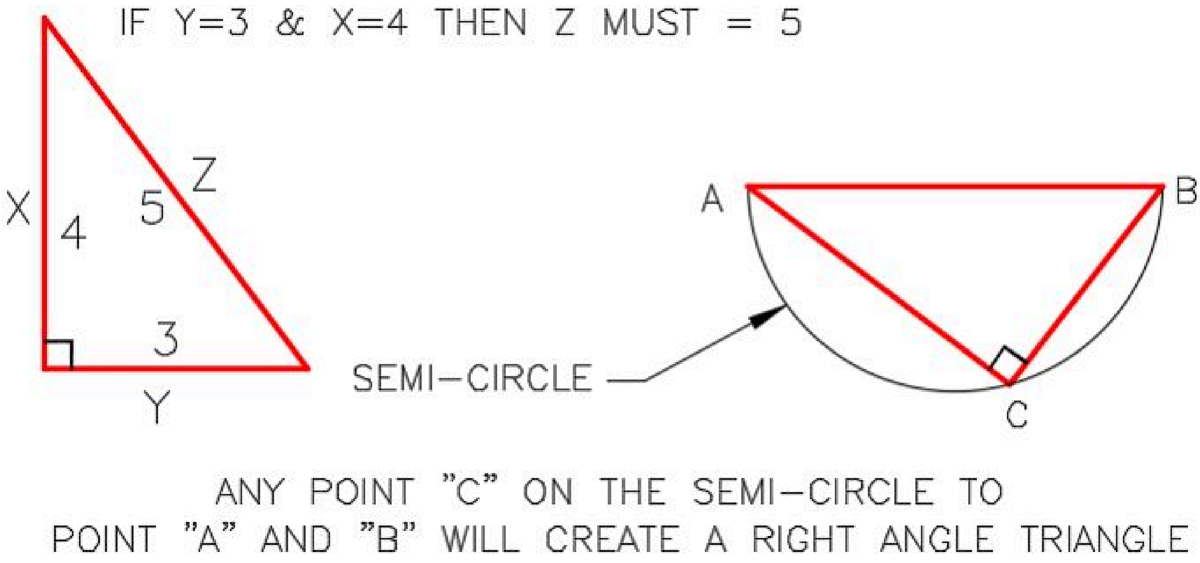
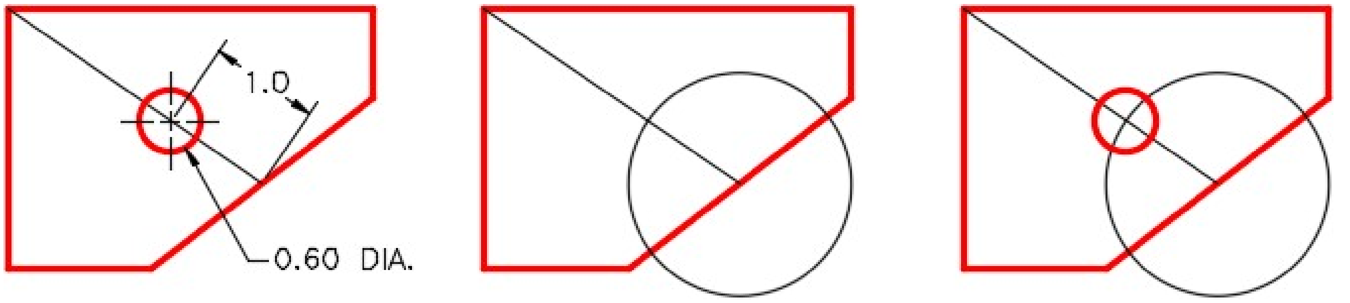

IF Y=3 & X=4 THEN Z MUST = 5
X
4
5
Z
3
Y
A
B
C
SEMI–CIRCLE
ANY POINT "C" ON THE SEMI–CIRCLE TO
POINT "A" AND "B" WILL CREATE A RIGHT ANGLE TRIANGLE

Figure 17-2
Geometry of a Right Angle Triangle

Geometry Lesson: Using Circles to Measure

When a measurement is required along an inclined line as shown in Step 1, Figure 17-3, the OFFSET command cannot easily be used to make that measurement. The best method of making the measurement is by drawing a construction circle with a 1 unit radius. Use the end of the inclined line as the center location for the circle. All points on the circumference of the circle will be the radius distance (1 unit) from the center of the circle as shown in the Step 2. Draw the 0.60 diameter circle locating its center at the intersection of the construction circle and the inclined line as shown in the Step 3.



Step 1 *Step 2* *Step 3*

Figure 17-3
Using Circles to Measure

Points

A point is an AutoCAD drawing object that is one pixel in size. Its only geometry property is one XY coordinate.

Point Style

Since a point is nothing more then one pixel in size, it will not display very well in the drawing. If it is located under an existing drawing object, it will not be visible at all. Points can be set to display with an assigned style to make them easier to see and use. The point style for the current drawing is set with the Point Style dialogue box. To open the Point Style dialogue box, select Point Style in the Format Pull-down menu, See Figure 17-4 and 17-5. The size of the points can be set relative with the screen size or an absolute size. To display the points style at the correct size, when the point size is set relative to the screen, the drawing must be regenerated using the REGEN command.

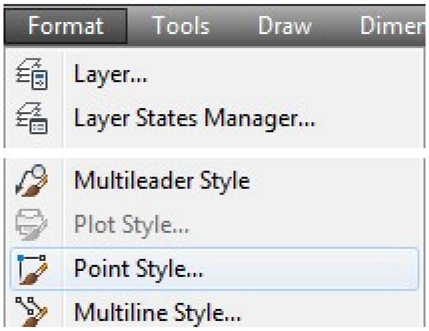
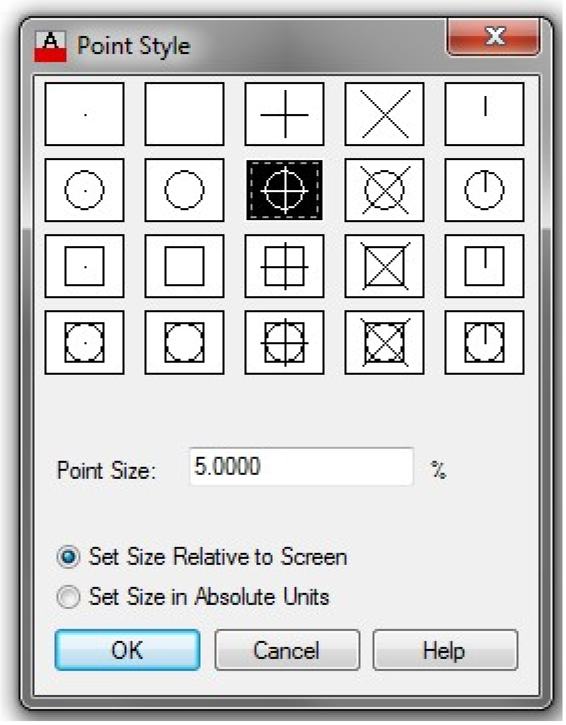

Format
Tools
Draw
Dimen
Layer...
Layer States Manager...
Multileader Style
Plot Style...
Point Style...
Multiline Style...

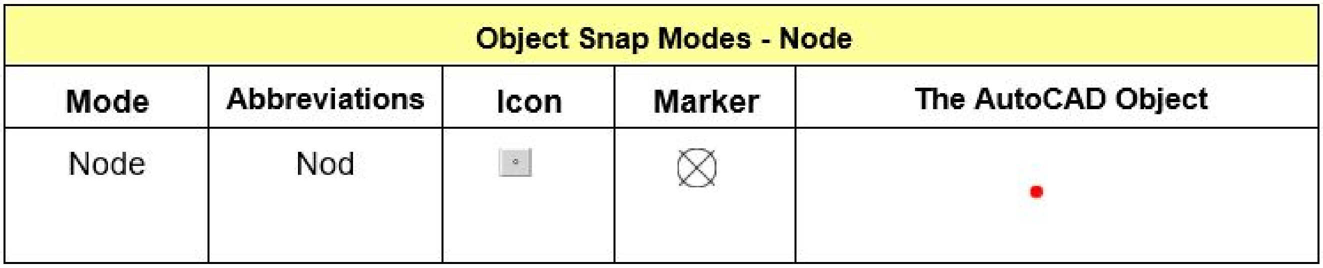
Figure 17-4
Point Style


Point Style
OK
Cancel
Help

Point Size: 5.0000 %

● Set Size Relative to Screen
○ Set Size in Absolute Units

Figure 17-5
Point Style Dialogue Box



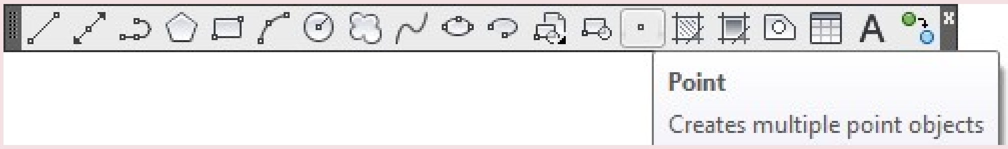
Object Snap Modes - Node				
Mode	**Abbreviations**	**Icon**	**Marker**	**The AutoCAD Object**
Node	Nod	⊡	⊗	●

Figure 17-6
Object Snap Mode – Node

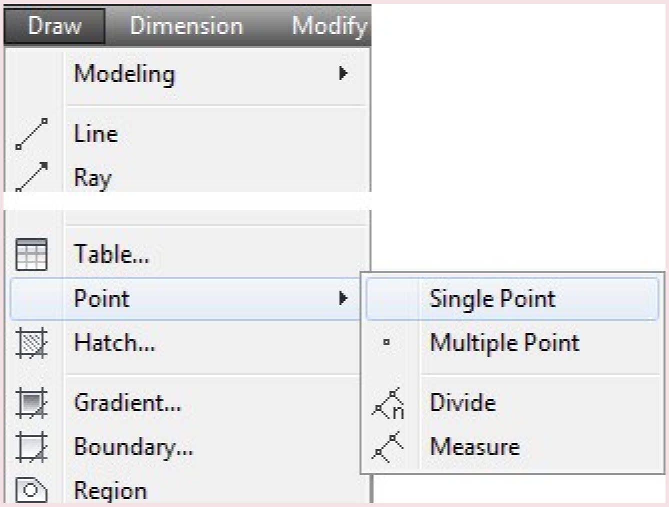
AutoCAD Command: POINT

The POINT command is used to insert a point.

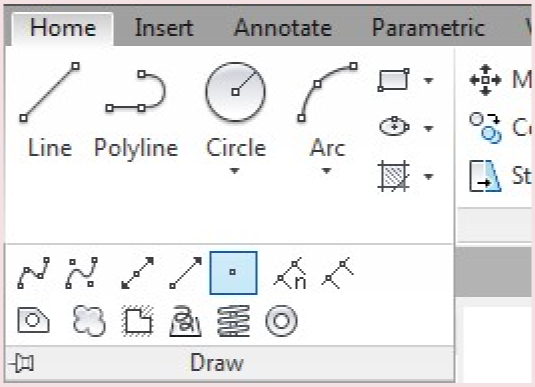
Shortcut: **PO**


Point
Creates multiple point objects

Draw Toolbar


Draw
Dimension
Modify
Modeling
Line
Ray
Table...
Point
Hatch...
Gradient...
Boundary...
Region
Single Point
Multiple Point
Divide
Measure

Draw Pull-down


Home
Insert
Annotate
Parametric
Line
Polyline
Circle
Arc
Draw

Home Ribbon

Rotating the Crosshairs

The crosshairs can be rotated to any angle. Its default rotation angle is 0 degrees, as shown in Figure 17-7. When the rotation angle is 0 degrees, the X axis is horizontal and the Y axis is vertical. There are times when it can be very helpful if the crosshairs are rotated to an angle other than 0 degrees, as shown in Figure 17-8. The SNAPANG system variable is used to set the rotation angle of the crosshairs.

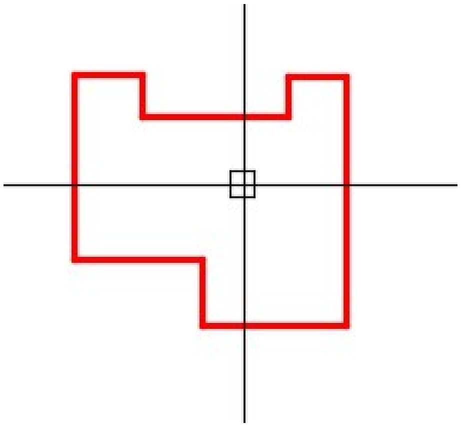


Figure 17-7
Crosshairs in The Normal Position
Rotated 0 degrees

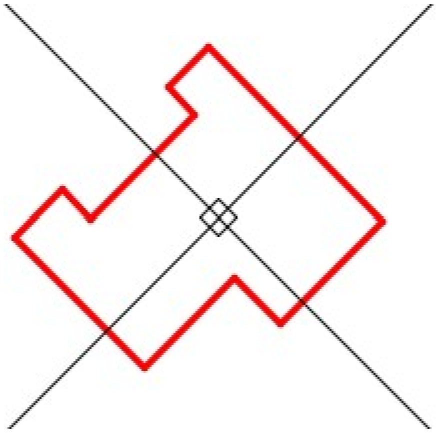


Figure 17-8 Crosshairs Rotated 45 degrees

MUST KNOW: It can be very confusing if SNAPANG is set to 90, 180, or 270 degrees since the crosshairs looks just like the default or 0 degrees. You can check its setting by entering the SNAPANG system variable on the command line and ensure it is set to 0.

The DIVIDE Command

The DIVIDE command is used to divide a drawing object by inserting points on the object at calculated XY locations. It does not change the object in any way, it simply inserts points on it. It is important that the current layer is set to layer Construction when the DIVIDE command is used. That way, the points can be easily isolated from the object it is dividing. On lines and arcs, the DIVIDE command starts calculating from the endpoint of the selected object.

The DIVIDE command works different when dividing circles. Since a circle does not have an endpoint, the DIVIDE command will place the first point where the positive X axis of the crosshairs intersects the circle.

The MEASURE Command

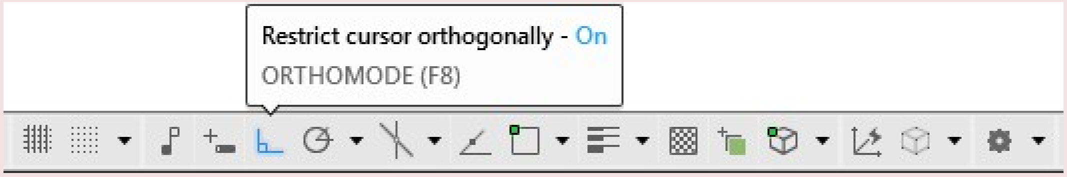
The MEASURE command is used to divide a drawing object by inserting points on the object at specified distances along the object. It does not change the object in any way, it simply inserts points on it. It is important that the current layer is set to layer Construction when the MEASURE command is used. That way, the points can easily be isolated from the object being measured. The MEASURE command always starts the first measurement from the closest endpoint to the location where the object is selected.

The MEASURE command works different when measuring circles. Since a circle does not have an endpoint, the MEASURE command will start measuring at the point where the positive X axis of the crosshairs intersects the circle.

AutoCAD Command: ORTHO

The ORTHO command is used to lock the crosshairs to allow it to move only horizontal or vertical. When enabled, it applies to most commands. The current Ortho mode is displayed on the Status bar. The function F8 key toggles ortho mode off and on.

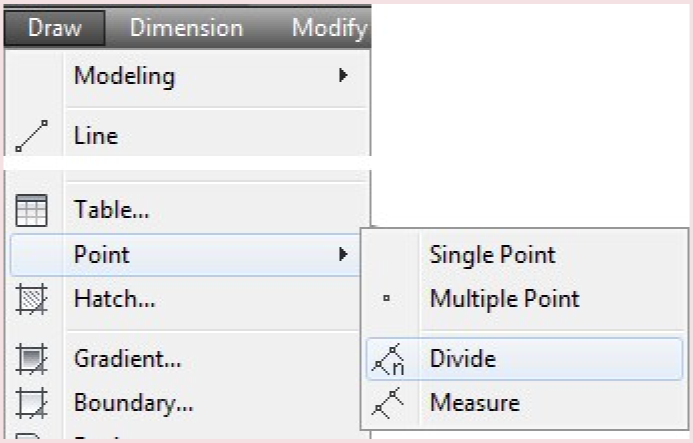
Shortcut: **F8**


Restrict cursor orthogonally - On
ORTHOMODE (F8)

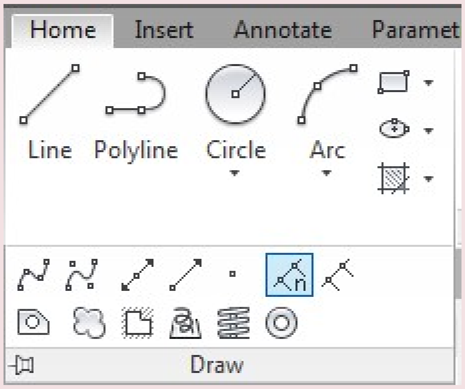
AutoCAD Command: DIVIDE

The DIVIDE command is used to insert points an equal distance apart by dividing the length of the drawing object selected by the number of divisions entered. The object being divided is not changed in any way by the command.

Shortcut: **DIV**


Draw
Dimension
Modify
Modeling
Line
Table...
Point
Hatch...
Gradient...
Boundary...
Single Point
Multiple Point
Divide
Measure

Draw Pull-down

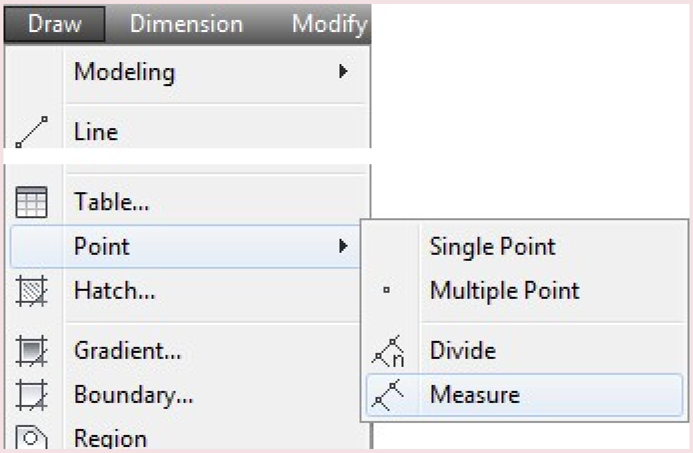

Home
Insert
Annotate
Paramet
Line
Polyline
Circle
Arc
Draw

Home Ribbon

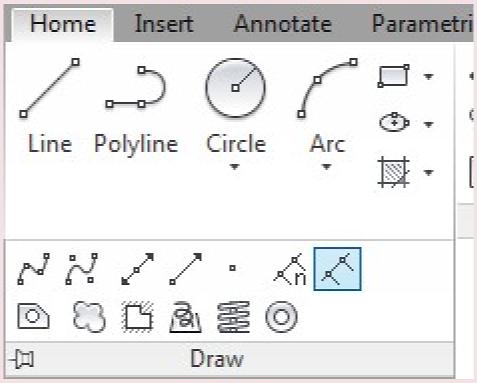
AutoCAD Command: MEASURE

The MEASURE command is used to insert points on a drawing object at a specified distance. The object being measured is not be changed in any way by the command.

Shortcut: **ME**


Draw
Dimension
Modify
Modeling
Line
Table...
Point
Hatch...
Gradient...
Boundary...
Region
Single Point
Multiple Point
Divide
Measure

Draw Pull-down


Home
Insert
Annotate
Parametri
Line
Polyline
Circle
Arc
Draw

WORKALONG: Using the POINT, DIVIDE and MEASURE Commands

Step 1

Start a new drawing using the template: 2D English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 17-1. (Figure Step 2)

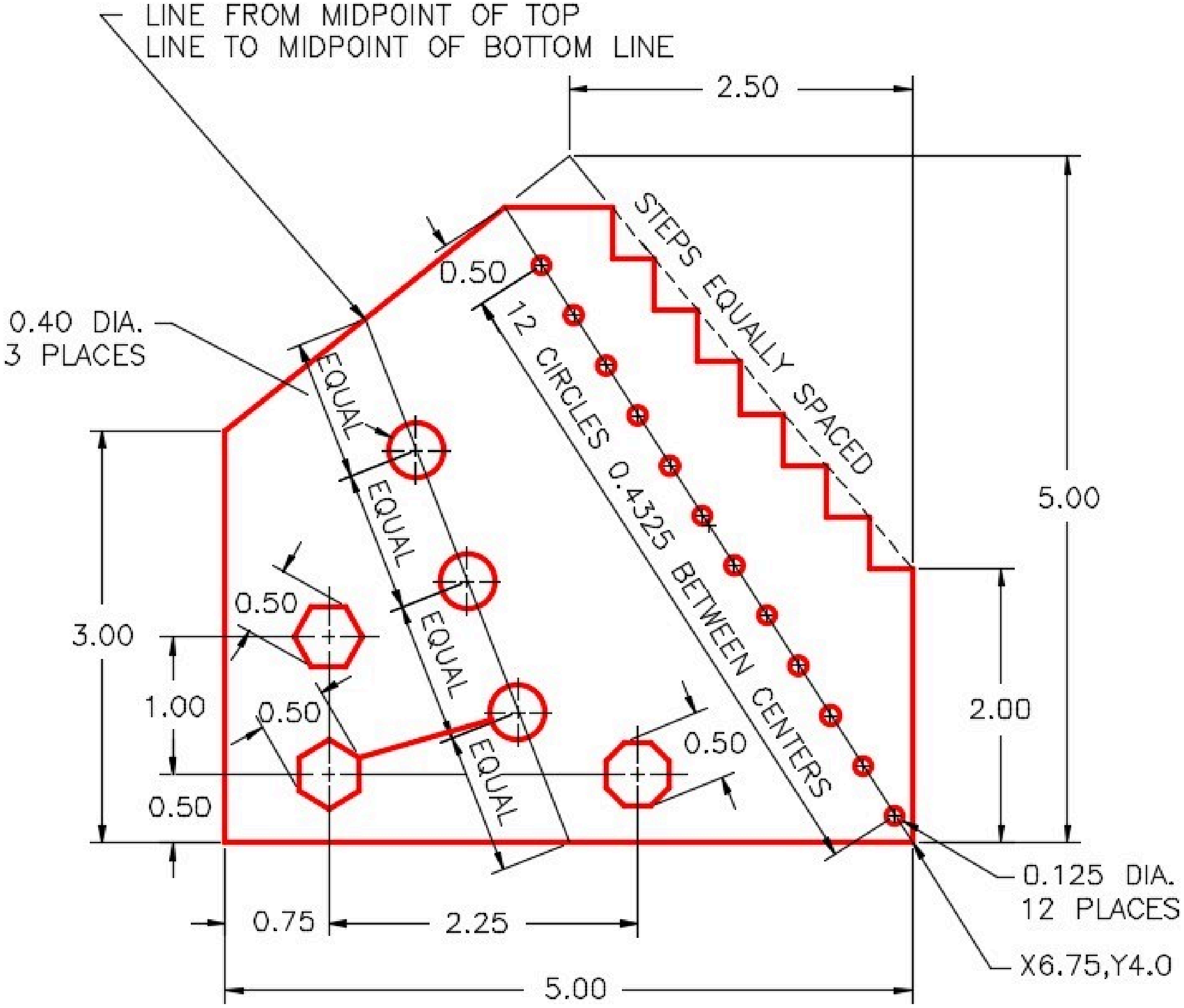

LINE FROM MIDPOINT OF TOP
LINE TO MIDPOINT OF BOTTOM LINE
2.50
0.50
STEPS EQUALLY SPACED
0.40 DIA.
3 PLACES
EQUAL
12 CIRCLES 0.4325 BETWEEN CENTERS
EQUAL
5.00
EQUAL
3.00
0.50
2.00
1.00
0.50
0.50
EQUAL
0.50
0.125 DIA.
12 PLACES
0.75
2.25
X6.75,Y4.0
5.00

Figure Step 2

Step 3

Create the layers: Object and Construction. Set layer: Construction as the current layer.

Step 4

Using what you learned earlier in the book, refer to Figure Step 2 and draw lines around the perimeter of the object. (Figure Step 4)

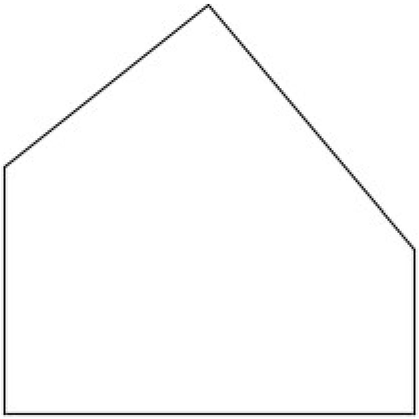
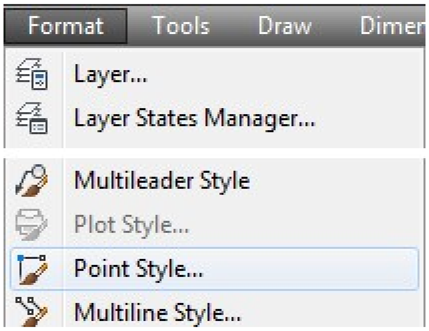


Figure Step 4

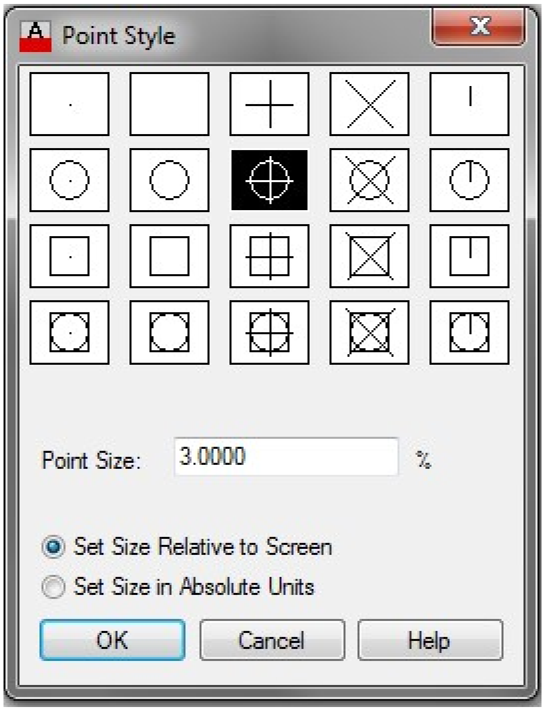
Step 5

From the Format Pull-down menu, select Point Style . This will open the Point Style dialogue box. Select the point style (center with black background) as shown in the figure. Set the Point Size to 3% and enable Set Size Relative to Screen. (Figure Step 5A and 5B)



Format
Tools
Draw
Dimen
Layer...
Layer States Manager...
Multileader Style
Plot Style...
Point Style...
Multiline Style...

Figure Step 5A



Point Size: 3.0000 %

● Set Size Relative to Screen

○ Set Size in Absolute Units

Figure Step 5B

AUTHOR’S COMMENTS: Setting the point relative to the screen means that regardless of the zoom factor of the screen, the points will always display the same size. You must execute the REGEN command to force AutoCAD to display the points size relative to the current zoom factor.

Step 6

Enter the DIVIDE command, as shown below, and select the inclined line on the right. (Figure Step 6A and 6B)

Command: **DIVIDE**

Select object to divide: **P1**

Enter the number of segments or [Block]: **8**

Command:

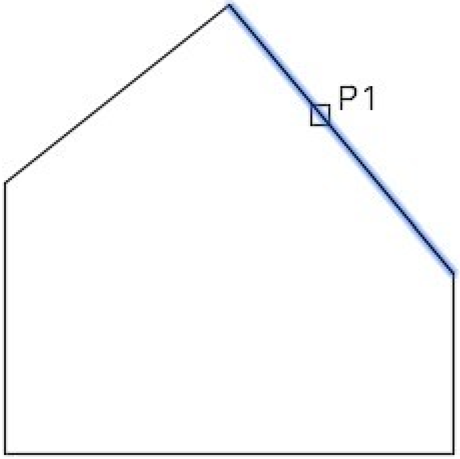


Figure Step 6A

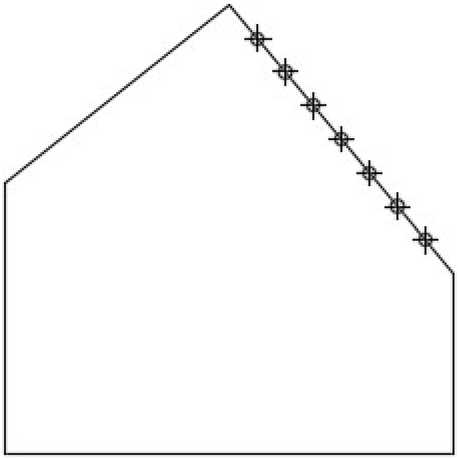
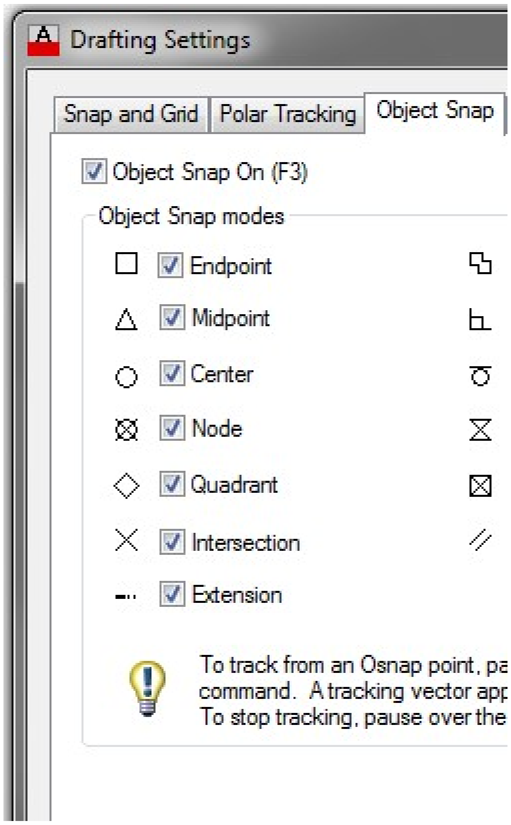


Figure Step 6B

AUTHOR'S COMMENTS: Since the line is divided into 8 segments, seven points are inserted along the line. Points are not inserted at the ends of the lines.

Step 7

As taught earlier in the book, use the OSNAP command to open the Drafting Settings dialogue box. Enable the Node object snap mode. (Figure Step 7)



Drafting Settings

Snap and Grid | Polar Tracking | Object Snap

☑ Object Snap On (F3)

Object Snap modes

- ☑ Endpoint
- ☑ Midpoint
- ☑ Center
- ☑ Node
- ☑ Quadrant
- ☑ Intersection
- ☑ Extension

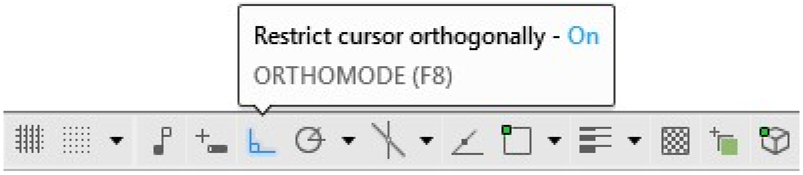
To track from an Osnap point, pa
command. A tracking vector app
To stop tracking, pause over the

Figure Step 7

AUTHOR'S COMMENTS: A node is the object snap mode for a point.

Step 8

Press F8 to enable Ortho mode. Check to ensure that it is enabled by checking the Status bar. (Figure Step 8)



Restrict cursor orthogonally - On
ORTHOMODE (F8)

Figure Step 8

AUTHOR'S COMMENTS: When Ortho mode is enabled, the LINE command will only draw horizontal or vertical lines assuming that the crosshairs is set to the default angle.

Step 9

Enable object snap and draw 8 horizontal lines by snapping to the points (nodes) as the first point of each line. The length of the lines is not important. (Figure Step 9)

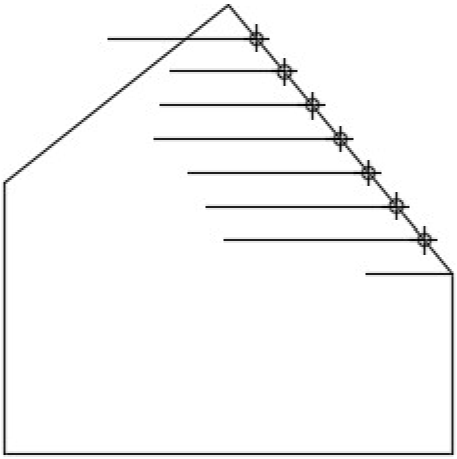


Figure Step 9

AUTHOR'S COMMENTS: You must enter the LINE command 8 times to draw the 8 lines. To work faster, press the Enter or Space key to renter the last command.

Step 10

Draw 7 vertical lines by snapping to the points (nodes) as the first point of each line. The length of the lines is not important. Disable Ortho mode. (Figure Step 10)

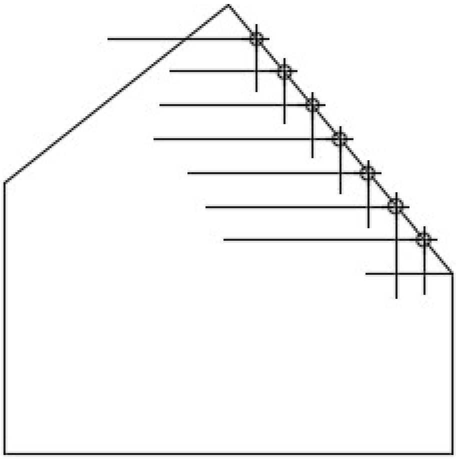


Figure Step 10

Step 11

Trim the lines to form the steps as shown in the figure. (Figure Step 11)

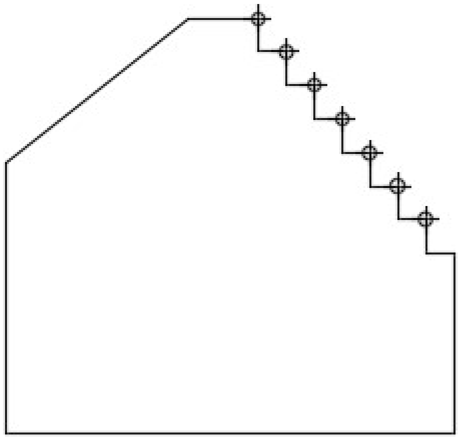


Figure Step 11

Step 12

Draw a line by snapping to midpoint on the top left inclined line and the midpoint of the bottom line.

AUTHOR'S COMMENTS: Ensure that you snap to midpoint of both lines.

Step 13

Using the DIVIDE command, divide the line into four segments. (Figure Step 13)

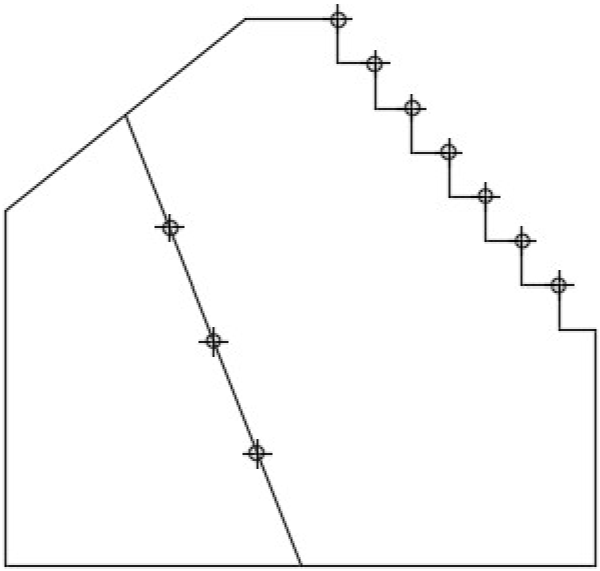


Figure Step 13

Step 14

Set layer: Object as the current layer. Enter the CIRCLE command and draw the three 0.40 diameter circles by snapping to the points (nodes) to locate the center of each circle. (Figure Step 14)

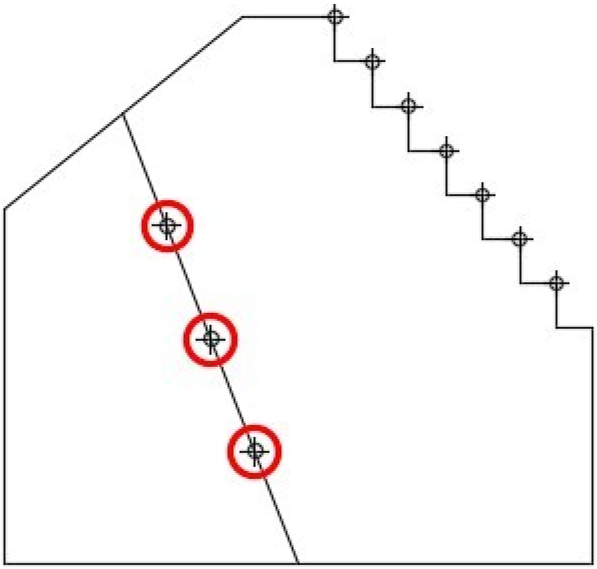


Figure Step 14

Step 15

Set layer: Construction as the current layer.

Step 16

Draw a construction line from the top corner to the bottom corner as shown in the figure. Ensure that you snap to the endpoints of the lines. (Figure Step 16)

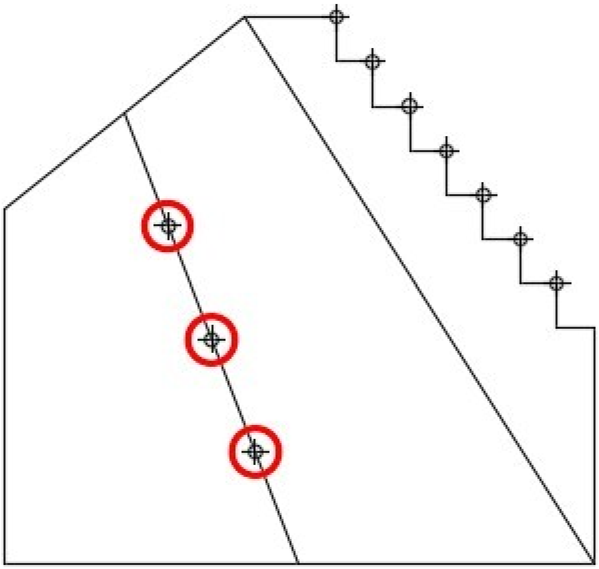


Figure Step 16

Step 17

Draw a 0.5 radius construction circle with its center located at the top corner as shown in the figure. Trim the line. (Figure Step 17A and 17B)

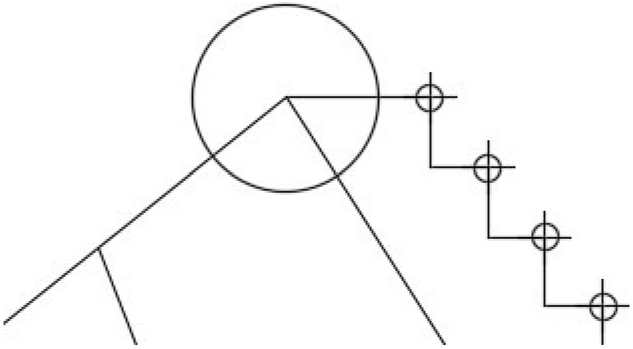


Figure Step 17A

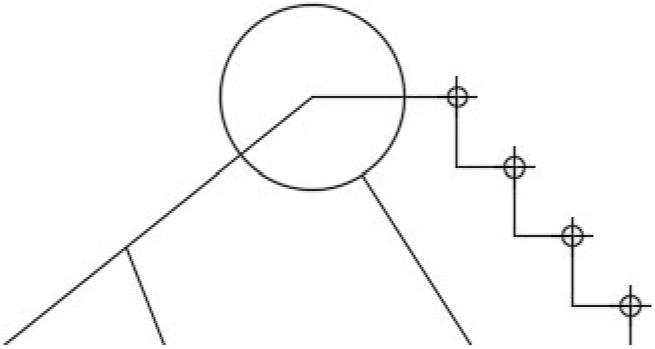


Figure Step 17B

Step 18

Enter the MEASURE command, as shown below, to insert points every 0.4325 units along the line. (Figure Step 18)

Command: **MEASURE**

Select object to measure: **P2**

Specify length of segment or [Block]: **0.4325**

Command:

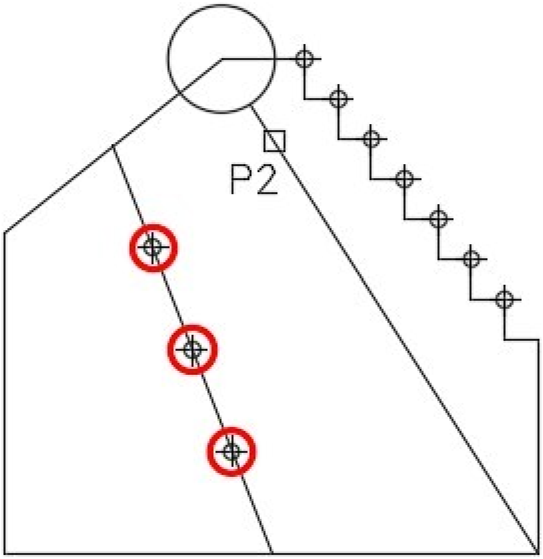


Figure Step 18

AUTHOR'S COMMENTS: Ensure that you select the line (P2) closest to the end of the line that you want to start the measurement from.

Step 19

The drawing should now appear as shown in the figure. To demonstrate how the point size relative to the screen works, zoom in on your drawing until it fills the screen. Then execute the REGEN command. Notice how the point style size for the points changes. Zoom back out and execute the REGEN command again. (Figure Step 19)

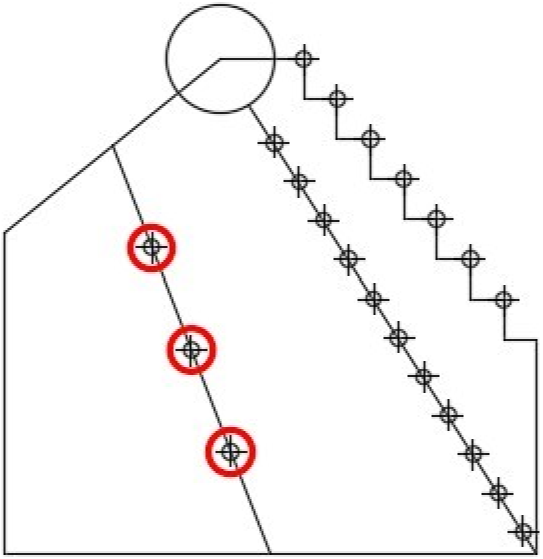


Figure Step 19

Step 20

Set layer: Object as the current layer. Insert 12 – 0.125 diameter circles. One with its center located at the end of the line and the others with their centers located on the points that you inserted with the MEASURE command in Step 18. (Figure Step 20)

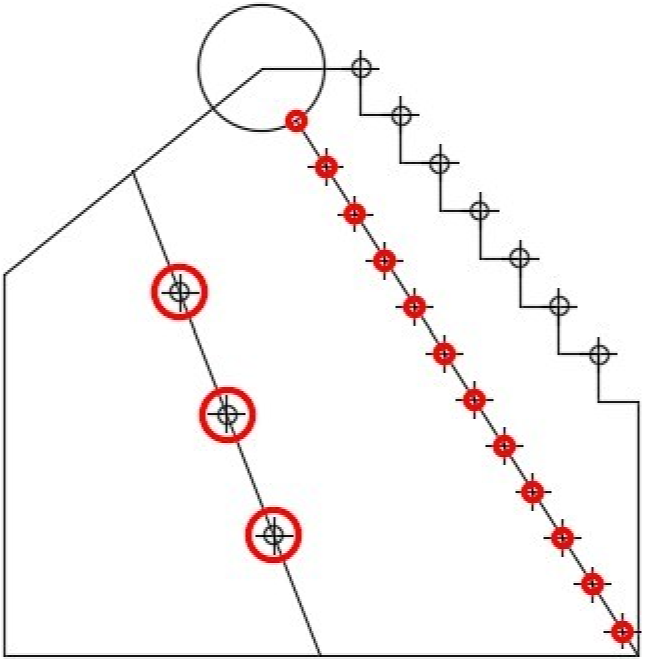


Figure Step 20

AUTHOR'S COMMENTS: It helps if you zoom in close to insert the circles.

Step 21

Set layer: Construction as the current layer. Enter the ID command, as shown below, and snap to the endpoint of the bottom line to establish a reference point. (Figure Step 21)

Command: **ID**

Specify point: *(end)* **P3**

X = 1.7500 Y = 4.0000 Z = 0.0000

Command:

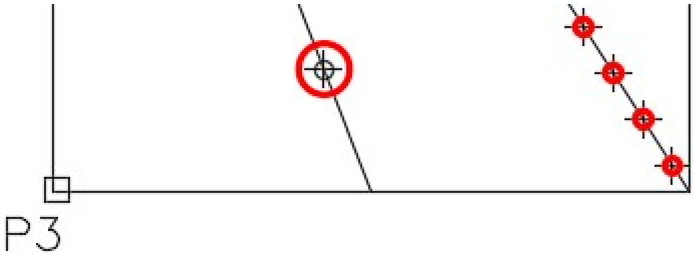


Figure Step 21

AUTHOR'S COMMENTS: I used the ID command in Step 21 to establish a reference point. I used that reference point in Step 22 to insert the first point. I inserted it using the @ (the ID reference point or lastpoint) location and measured from it. Ensure that you understand how the reference point was used. Note that it must used in the next command entered after the ID command is executed.

Step 22

Enter the POINT command, as shown below, to draw 3 points at the locations of the center of polygons. (Figure Step 22)

Command: **POINT**

Current point modes: PDMODE=34 PDSIZE=-3.0000 Specify a point: **@.75,1.5**

Command: **POINT**

Current point modes: PDMODE=34 PDSIZE=-3.0000 Specify a point: **@0,-1**

Command: **POINT**

Current point modes: PDMODE=34 PDSIZE=-3.0000

Specify a point: @2.25,0

Command:

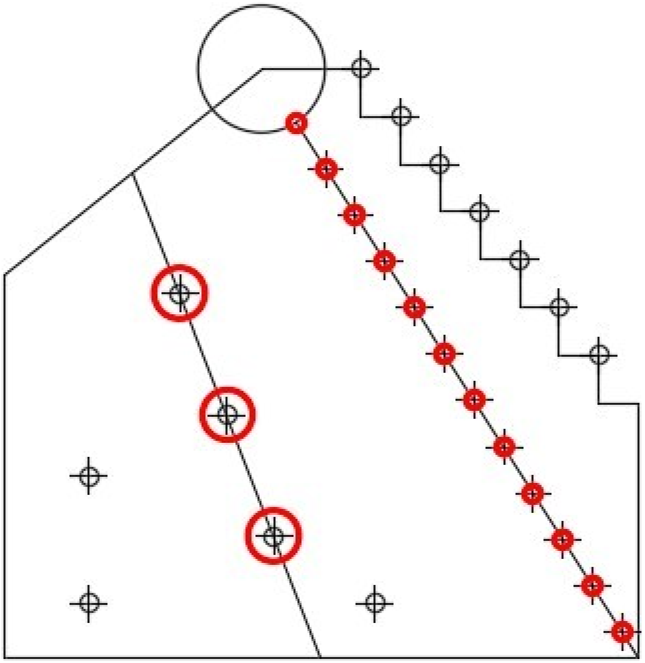


Figure Step 22

Step 23

Draw three 0.5 diameter circles using the points as the location of the center of the circles. Ensure that you snap to the node when locating the center of each circle. (Figure Step 23)

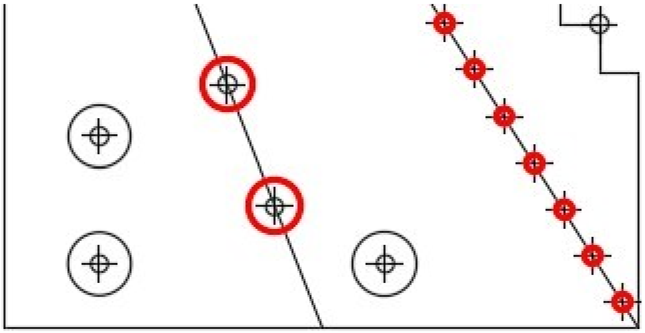


Figure Step 23

Step 24

Using the DIVIDE command, divide the top left circle into 6 segments. (Figure Step 24)

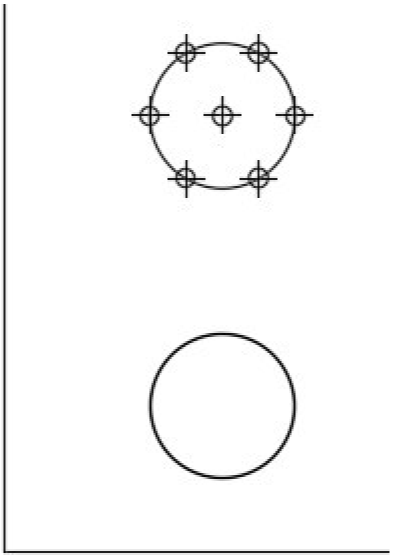


Figure Step 24

AUTHOR'S COMMENTS: Note how the first point is located along the X axis of the cursor. Six segments was used since a hexagon has six sides.

Step 25

Enter the SNAPANG system variable as shown below to change the angle of the cursor to 90 degrees.

```
Command: SNAPANG

Enter new value for SNAPANG <0>: 90

Command:
```

Step 26

Enter the DIVIDE command and divide the bottom left circle into 6 segments. (Figure Step 26)

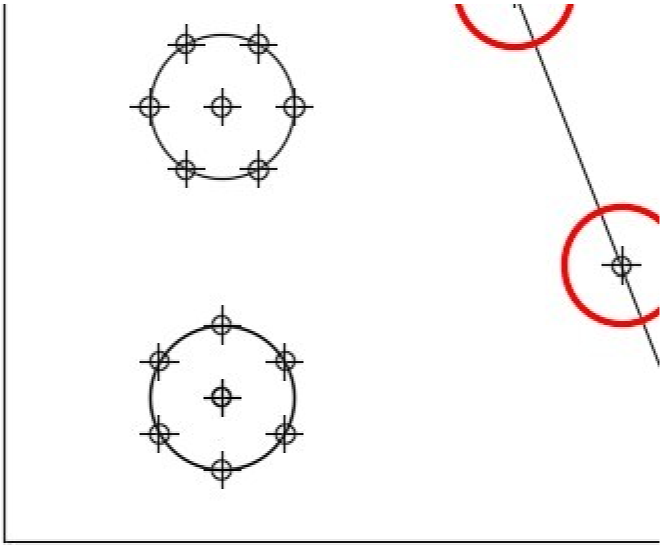


Figure Step 26

AUTHOR'S COMMENTS: Note how the first point is located along the X axis that has been rotated 90 degrees.

Step 27

Enter the system variable SNAPANG as shown below to change the angle of the crosshairs to 22.5 degrees.

```
Command: SNAPANG

Enter new value for SNAPANG <90>: 22.5

Command:
```

Step 28

Enter the DIVIDE command and divide the bottom right circle into 8 segments. (Figure Step 28A and 28B)

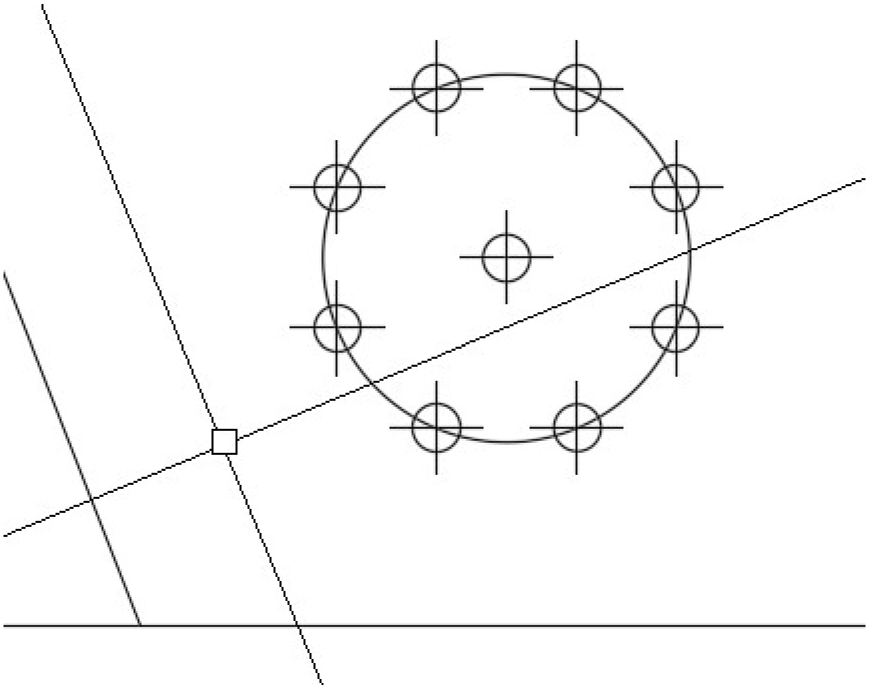


Figure Step 28A

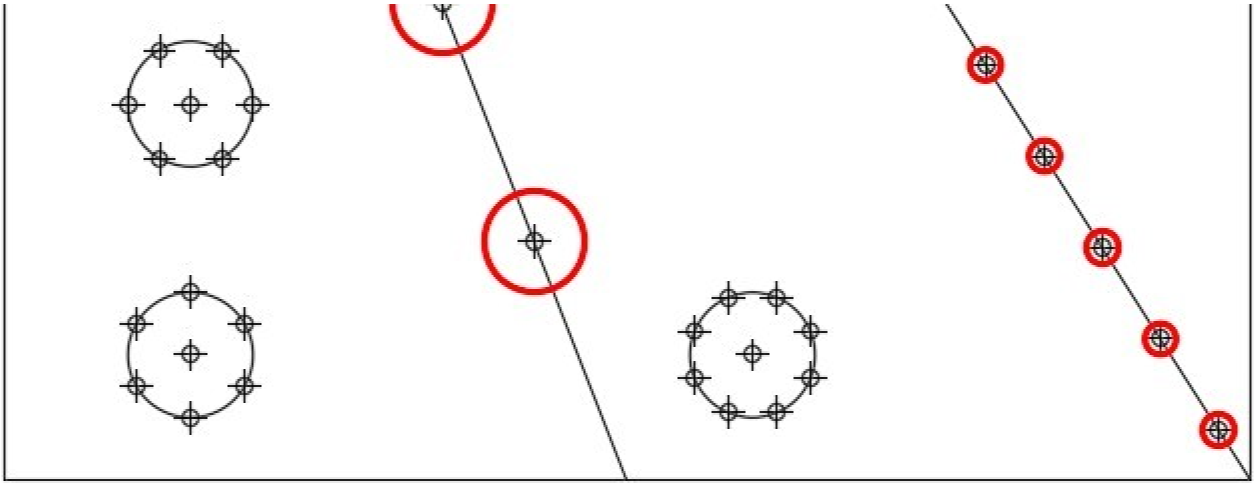


Figure Step 28B

AUTHOR'S COMMENTS: Note how the first point is located along the X axis that has been rotated 22.5 degrees. 22.5 degrees is one-half of 45 degrees which is the angle between the corners of an octagon. Eight segments were used since an octagon has 8 sides.

Step 29

Use the SNAPANG system variable, as shown below, to change the angle of the crosshairs to 0 degrees.

Command: **SNAPANG**

Enter new value for SNAPANG

<22.5>: **0**

Command:

AUTHOR'S COMMENTS: Ensure that you always set the SNAPANG system variable back to zero when you have finished working with it.

Step 30

On layer Object, draw the lines for the hexagons and the octagon by snapping from point to point (node to node) as shown in the figure. (Figure Step 30)

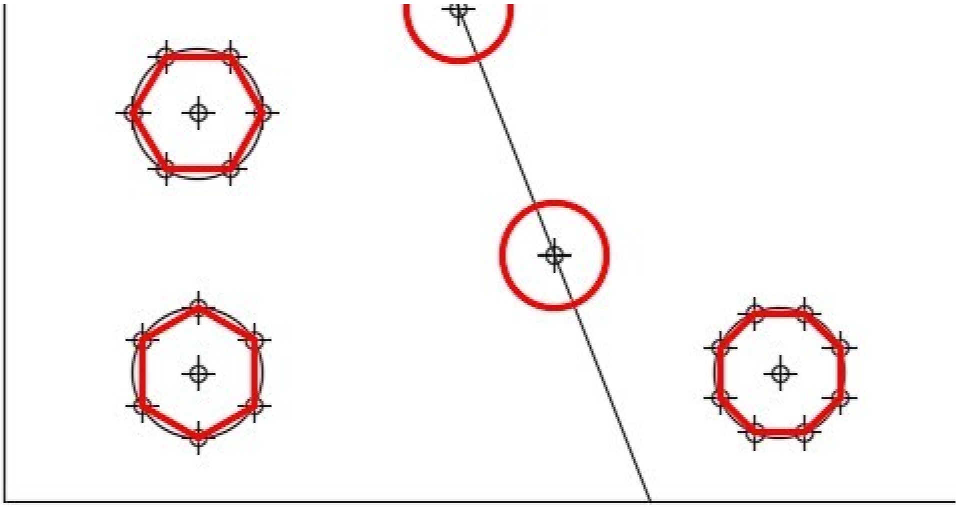


Figure Step 30

Step 31

Change the layer of any objects that you want to reside on layer Object. Freeze layer Construction. Your completed drawing should appear as shown in the figure. (Figure Step 31)

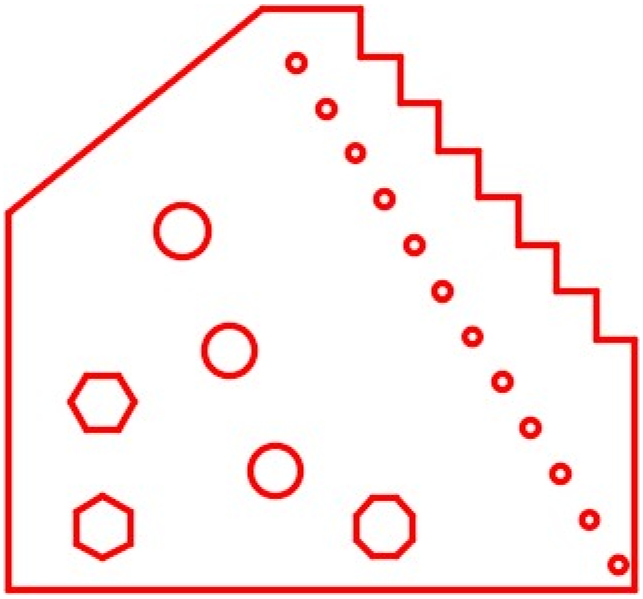


Figure Step 31

Step 32

Save and close the drawing.

> **MUST KNOW:** The MEASURE command inserts points at a specified distance. It starts measuring at the end of the object nearest where it is selected. It may not end up with an equal distance at the other end of the object. Make sure that the current layer is Construction before you execute this command to ensure that the points will be easily to isolate from the object it is measuring.

Making Inquiries

The DIST and PROPERTIES commands are used to measure distances, find sizes, and locations of drawing objects. Before using these commands, it is important to understand the importance of the UNITS command.

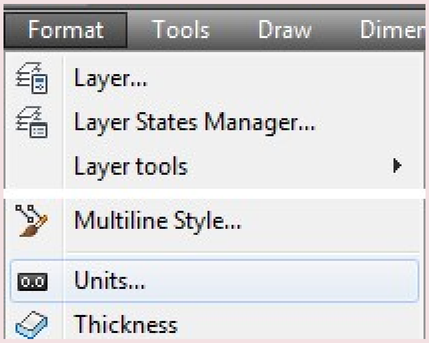
Displaying Units

AutoCAD stores the drawing object’s properties and locations to a very high degree of accuracy. When the drawing object’s properties are displayed, either on the Command Line or in the Properties window, AutoCAD uses the setting in the Drawing Units dialogue box to display these properties rounded off as per the current settings. The answers are always displayed in drawing units but the precision and format of the answers is controlled by the settings in the Drawing Units dialogue box.

AutoCAD Command: UNITS

The UNITS command is used to open the Drawing Units dialogue box. It is used to set the way AutoCAD displays inquiries or drawing object properties.

Shortcut: **UN**

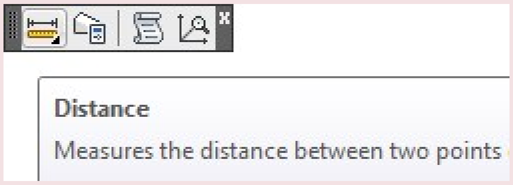

Format
Tools
Draw
Dimer
Layer...
Layer States Manager...
Layer tools
Multiline Style...
0.0
Units...
Thickness

Format Pull-down

AutoCAD Command: DIST

The DIST command is used to measure distances between two XY coordinate locations.

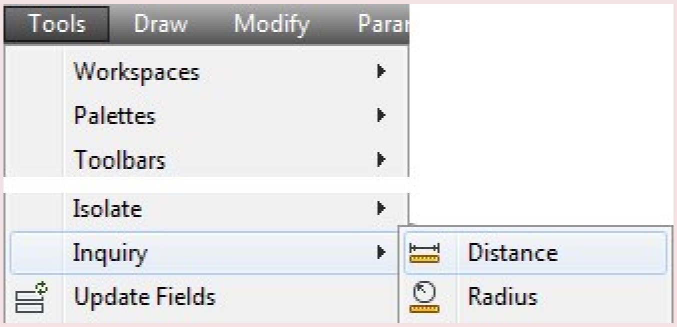
Shortcut: **DI**



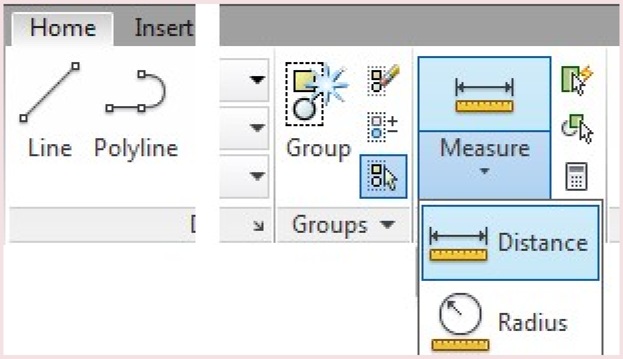
Distance

Measures the distance between two points

Inquiry Toolbar


Tools
Draw
Modify
Para
Workspaces
Palettes
Toolbars
Isolate
Inquiry
Update Fields
Distance
Radius

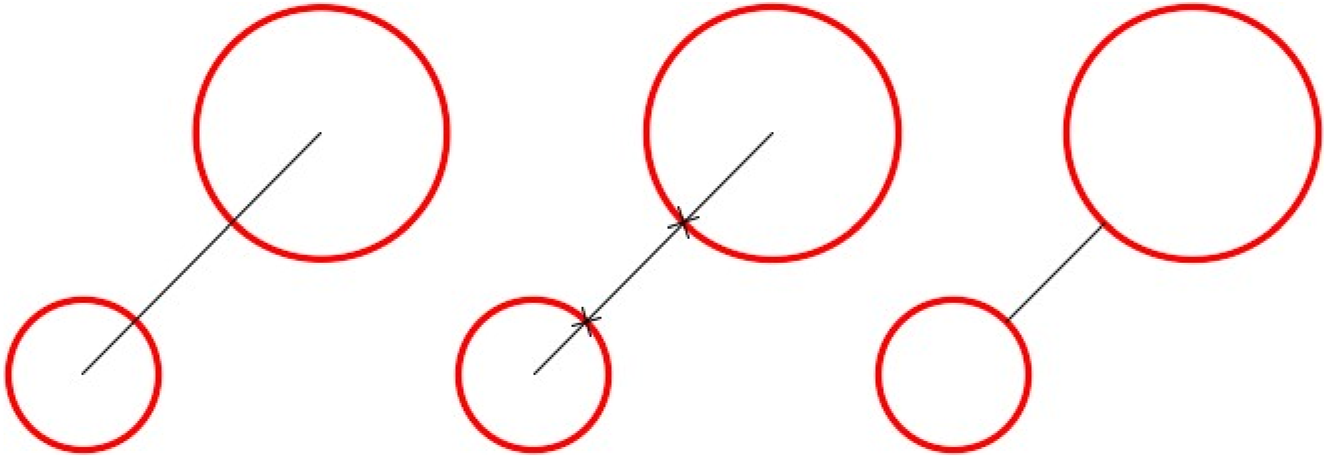
Tool Pull-down



Home Ribbon

Geometry Lesson: Finding the Shortest Distance Between the Circumference of Two Circles

To find the shortest distance between the circumference of two circles, draw a construction line from the center of one circle to the center of the other as shown in Step 1. Measure the distance between the intersections of the lines and circles as shown in Step 2. If the line is trimmed as shown in Step 3, the length of the line is the shortest distance between the circle’s circumference. See Figure 17-9.



Step 1 *Step 2* *Step 3*

Figure 17-9

Finding the Shortest Distance Between the Circumference of two Circles

WORKALONG: Using Inquiry Commands

Step 1

Open the drawing: AutoCAD 2D Workalong 17-1 . Using the SAVEAS command, save the drawing with the name: AutoCAD 2D Workalong 17-2. (Figure Step 1)

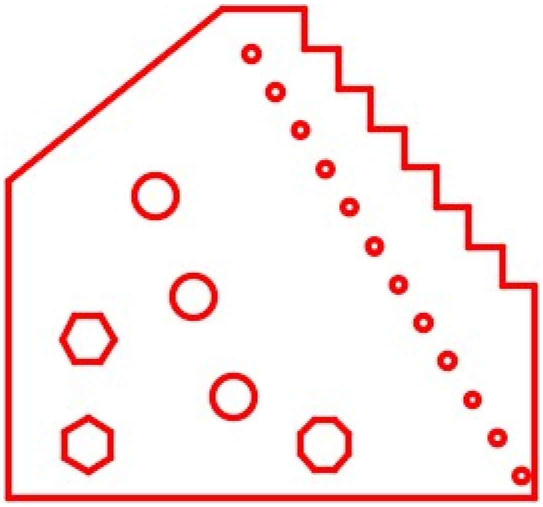
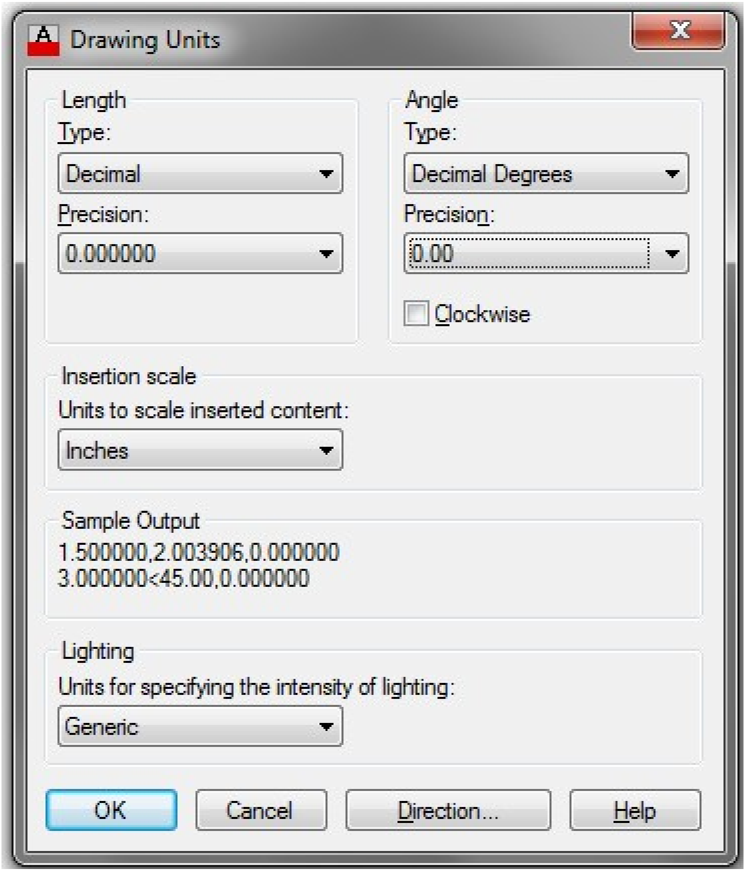


Figure Step 1

Step 2

Enter the UNITS command. It will open the Drawing Units dialogue box. Set the dialogue box to match the figure. (Figure Step 2)



Drawing Units

Length

Type: Decimal

Precision: 0.000000

Angle

Type: Decimal Degrees

Precision: 0.00

☐ Clockwise

Insertion scale

Units to scale inserted content: Inches

Sample Output

1.500000,2.003906,0.000000
3.000000<45.00,0.000000

Lighting

Units for specifying the intensity of lighting: Generic

Figure Step 2

Step 3

Enter the DIST command, as shown below, to measure the distance between the center of the circle and the end of the line as shown in the figure. Ensure that you snap to the center of the circle and the endpoint of the line. (Figure Step 3)

Command: **DIST**

Specify first point: *(cen)* **P1**

Specify second point: *(end)* **P2**

Command:

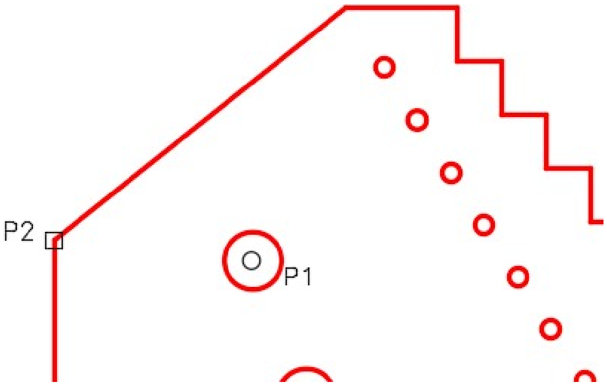
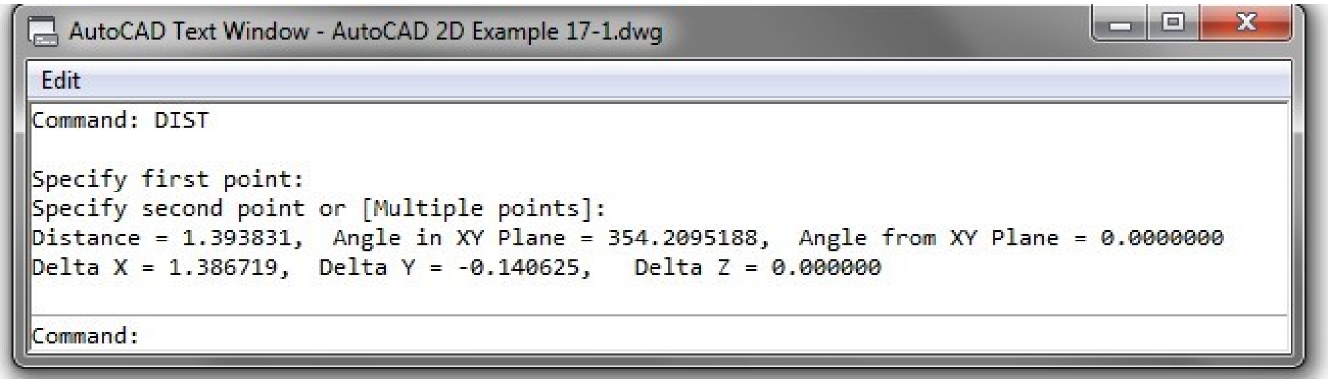


Figure Step 3

Step 4

Press F2 to open the Text window. The answer to the DIST command will display as shown in the figure. (Figure Step 4)



```
Command: DIST

Specify first point:
Specify second point or [Multiple points]:
Distance = 1.393831,  Angle in XY Plane = 354.2095188,  Angle from XY Plane = 0.0000000
Delta X = 1.386719,  Delta Y = -0.140625,   Delta Z = 0.000000

Command:
```

Figure Step 4

AUTHOR'S COMMENTS: The measured distance is 1.393831 units. The angle of the line is 174.21 degrees. For an explanation of delta X and delta Y, see the Geometry Lesson: A Right Triangle.

AUTHOR'S COMMENTS: Note that AutoCAD displays the answers to the DIST command to 6 decimal places. That is the precision that was set in the Length Precision in the Drawing Units dialogue box in Step 2.

Step 5

Press F2 to close the Text window.

Step 6

Draw a construction line from the center of circle to center of circle as shown in the figure. Ensure that you snap to the centers of the circle at both ends. Enter the DIST command as shown below to measure the shortest distance between circumferences of the circles by snapping to intersection of the construction line and the circles. (Figure Step 6)

```
Command: DIST

Specify first point: (int) P3

Specify second point: (int) P4

Command:
```

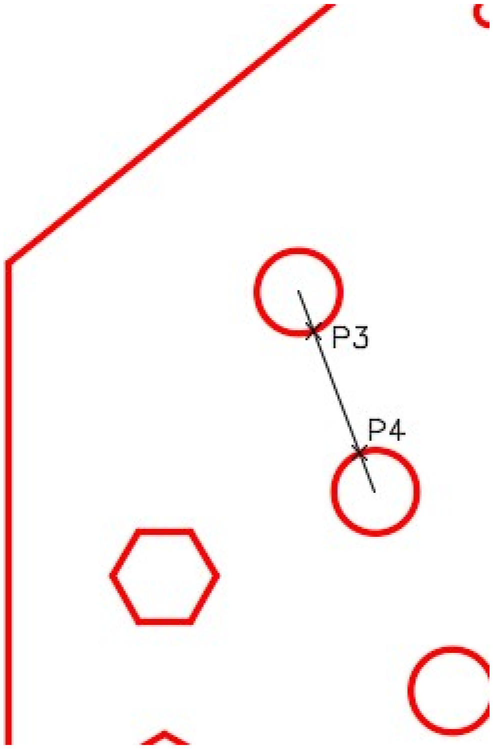
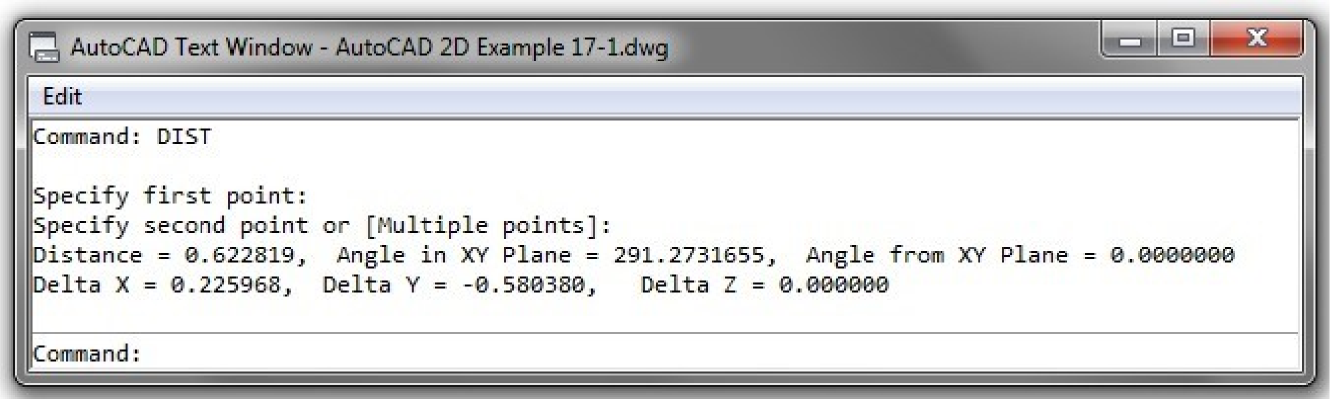


Figure Step 6

Step 7

Press F2 to open the Text window. The answer to the DIST command will display as shown in the figure. (Figure Step 7)



```
Command: DIST

Specify first point:
Specify second point or [Multiple points]:
Distance = 0.622819,  Angle in XY Plane = 291.2731655,  Angle from XY Plane = 0.0000000
Delta X = 0.225968,  Delta Y = -0.580380,   Delta Z = 0.000000

Command:
```

Figure Step 7

Step 8

Trim the construction line as shown in the figure. (Figure Step 8)

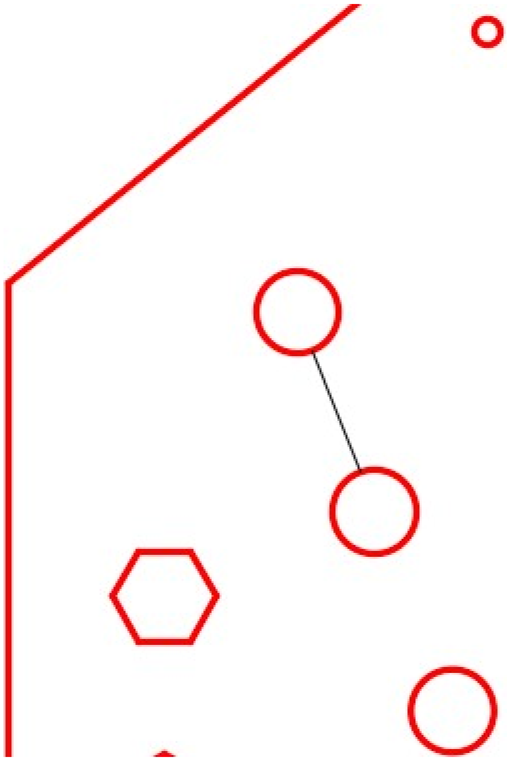
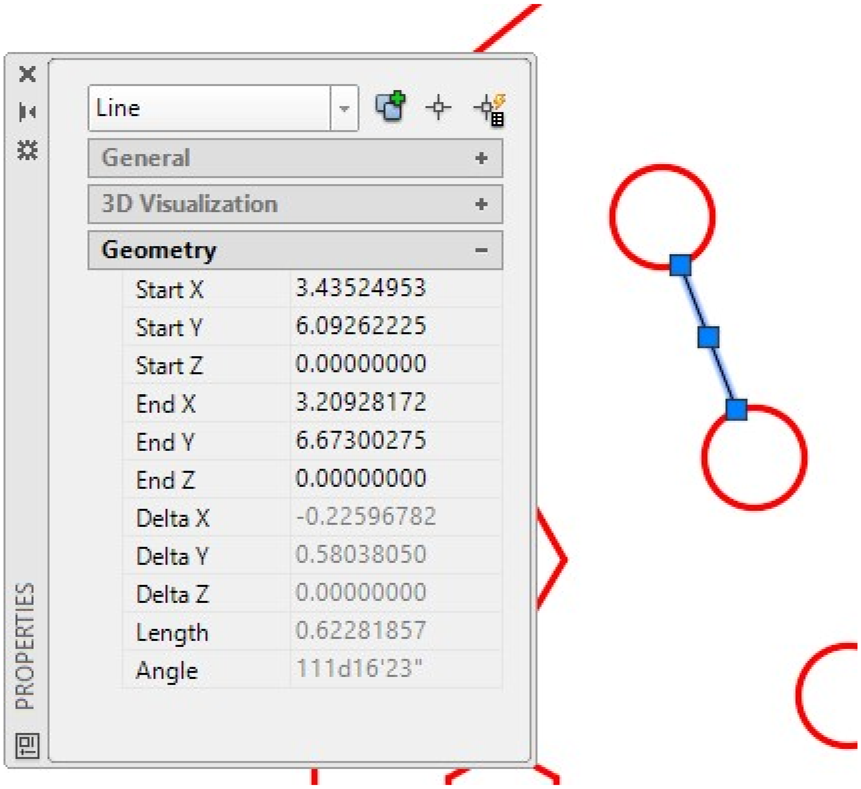


Figure Step 8

MUST KNOW: AutoCAD stores the accuracy of the drawing object's properties to a very high degree. The UNITS command is used to set the format and precision which instructs AutoCAD how to display answers in the Properties window and to the users inquiry commands.

Step 9

Open the Properties window. Without entering a command, select the line. The Properties window will display the length of the line. (Figure Step 9)



Line

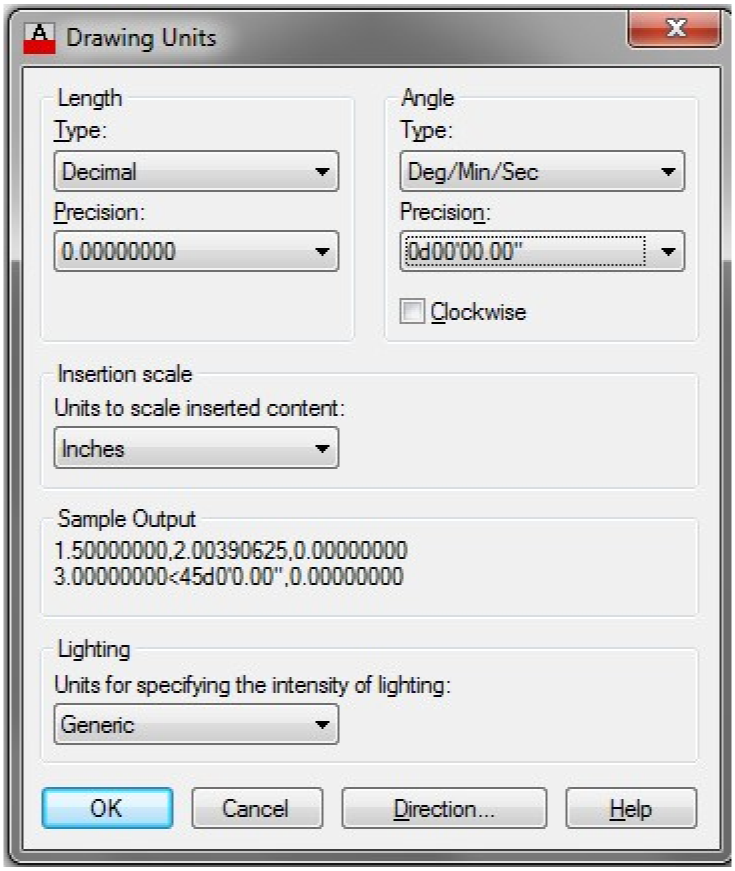
Start X	3.43524953
Start Y	6.09262225
Start Z	0.00000000
End X	3.20928172
End Y	6.67300275
End Z	0.00000000
Delta X	-0.22596782
Delta Y	0.58038050
Delta Z	0.00000000
Length	0.62281857
Angle	111d16'23"

Figure Step 9

AUTHOR'S COMMENTS: The length of the line (0.622819) and the angle of the line (111.27 degrees or 111.27 + 180.00 = 291.27) should match the answers in the DIST command in Step 7.

Step 10

Set the Drawing Units dialogue box, as shown in the figure. (Figure Step 10)



Length
Type: Decimal
Precision: 0.00000000

Angle
Type: Deg/Min/Sec
Precision: 0d00'00.00"
☐ Clockwise

Insertion scale
Units to scale inserted content: Inches

Sample Output
1.50000000,2.00390625,0.00000000
3.00000000<45d0'0.00",0.00000000

Lighting
Units for specifying the intensity of lighting: Generic

Figure Step 10

AUTHOR'S COMMENTS: Note that the Length Precision is set to 8 decimals and the Angle Type is set to display in Degree/Minutes/Seconds with a Precision of no decimals.

Step 11

Draw a line from the center of the circle to the vertex on the hexagon as shown in the figure. (Figure Step 11)

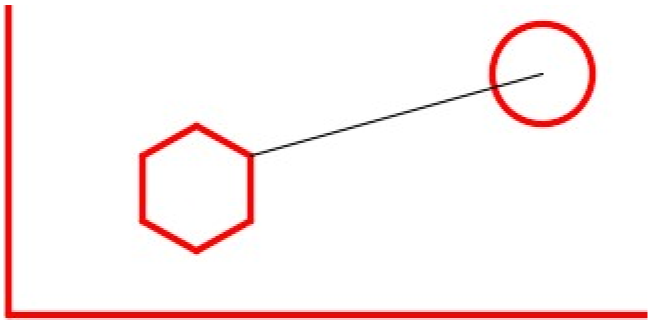
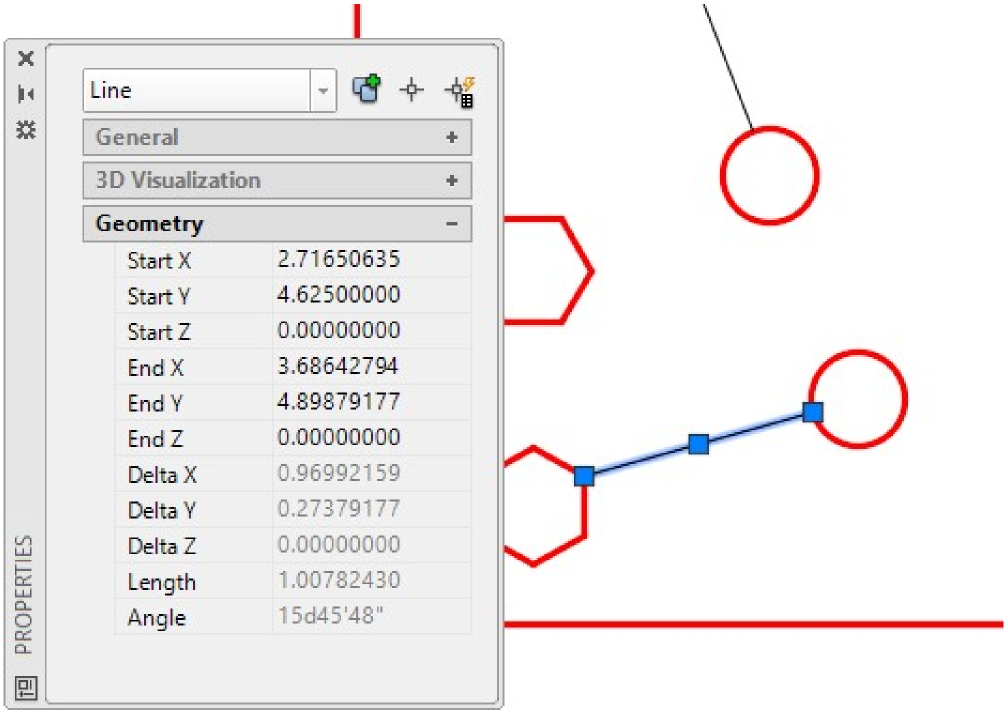


Figure Step 11

Step 12

After trimming the line, select it. Check its length and angle in the Properties window. (Figure Step 12)



Start X	2.71650635
Start Y	4.62500000
Start Z	0.00000000
End X	3.68642794
End Y	4.89879177
End Z	0.00000000
Delta X	0.96992159
Delta Y	0.27379177
Delta Z	0.00000000
Length	1.00782430
Angle	15d45'48"

Figure Step 12

AUTHOR’S COMMENTS: Your answers and the ones in the figure should match. Note the angle is now displayed in the Degree/Minutes/Seconds. If you drew the line from the vertex of the hexagon to the center of the circle, the angle will be 15d45’48”. Both answers are correct since each line always has two angles depending which direction the line was drawn.

Step 13

Turn layer: Construction off. Enter the ID command and snap to the center of the circle. Press F2 to open the Text window. (Figure Step 13A and 13B)

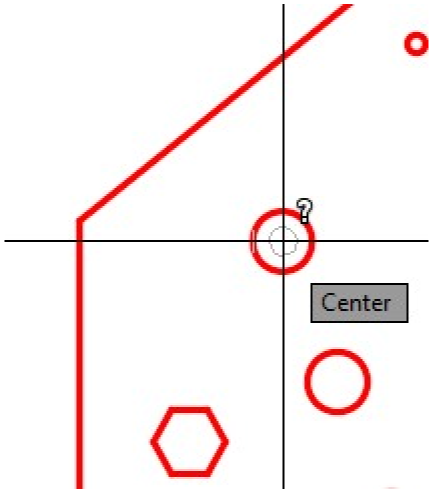
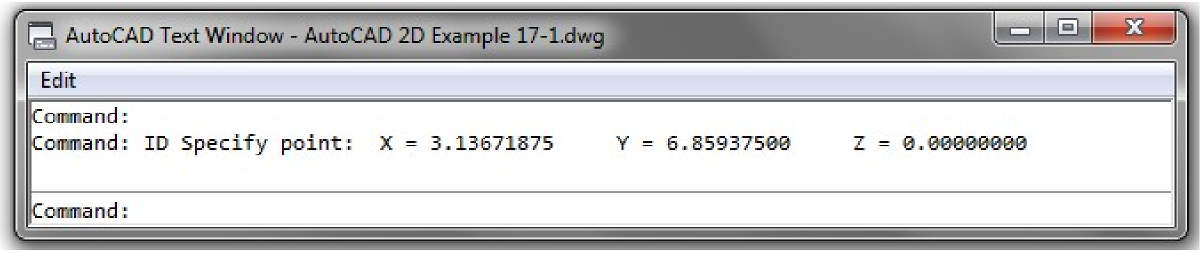


Figure Step 13A

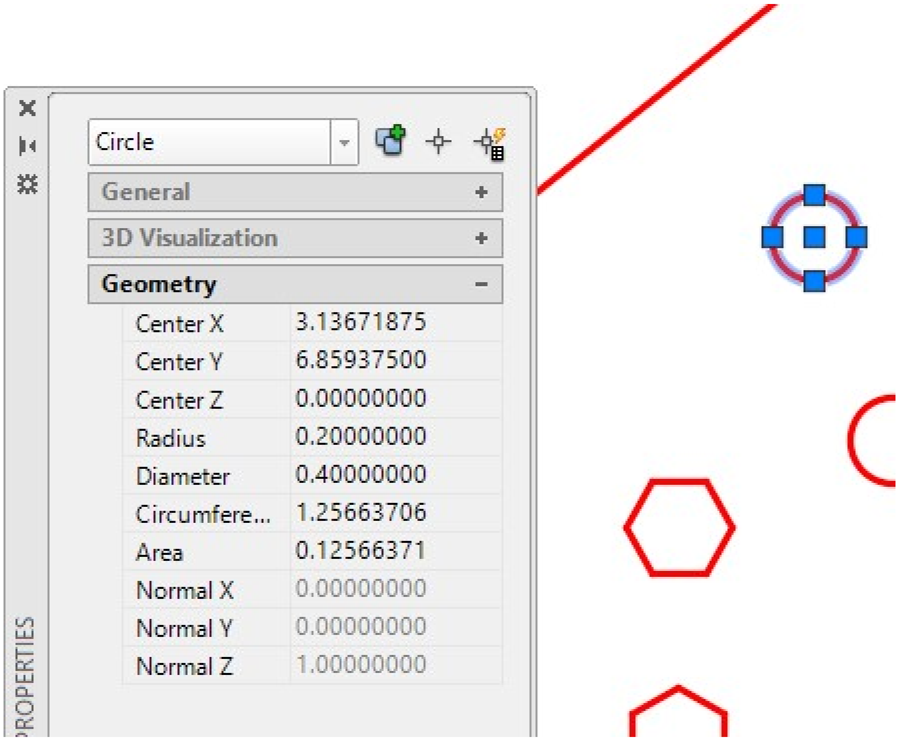

AutoCAD Text Window - AutoCAD 2D Example 17-1.dwg
Edit

```
Command:
Command: ID Specify point:  X = 3.13671875     Y = 6.85937500     Z = 0.00000000

Command:
```

Figure Step 13B

Step 14

Select the circle as shown in the figure. Take note that the Properties window displays the XY center location plus the radius, diameter, circumference, and area of the circle. (Figure Step 14)


Circle
General
3D Visualization
Geometry
ROPERTIES

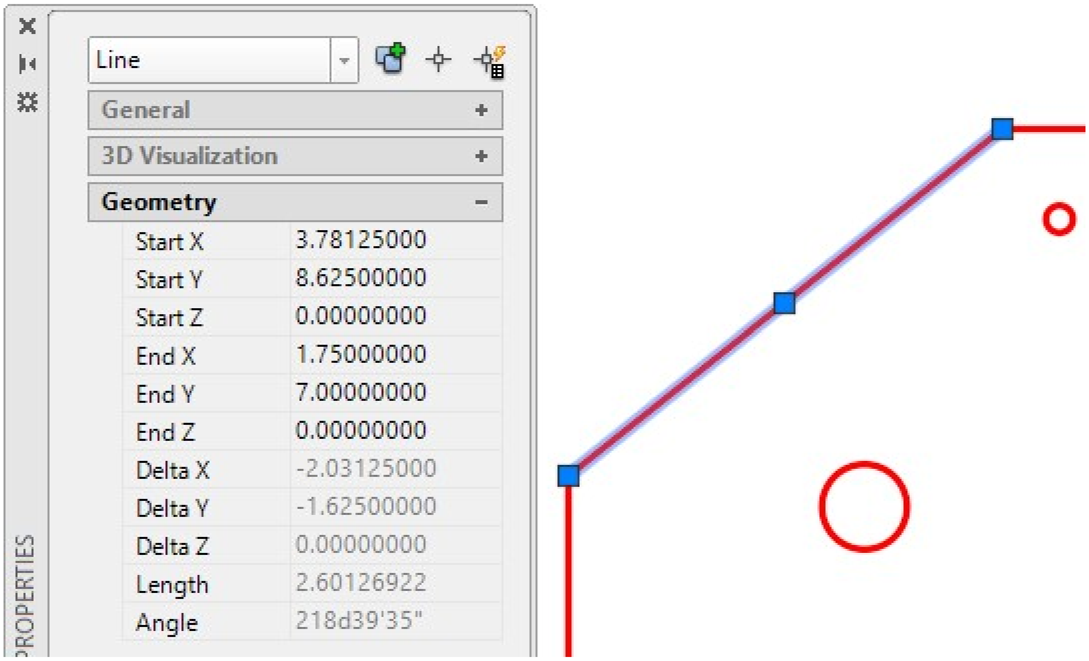
Center X	3.13671875
Center Y	6.85937500
Center Z	0.00000000
Radius	0.20000000
Diameter	0.40000000
Circumfere...	1.25663706
Area	0.12566371
Normal X	0.00000000
Normal Y	0.00000000
Normal Z	1.00000000

Figure Step 14

AUTHOR’S COMMENTS: Note that the Length Precision is set to 8 decimals.

Step 15

With the Properties window open, select the line as shown in the figure. Figure Step 15)



Start X	3.78125000
Start Y	8.62500000
Start Z	0.00000000
End X	1.75000000
End Y	7.00000000
End Z	0.00000000
Delta X	-2.03125000
Delta Y	-1.62500000
Delta Z	0.00000000
Length	2.60126922
Angle	218d39'35"

Figure Step 15

Step 16

Save and close the drawing.

Geometry Lesson: Finding the Centroid of any Triangle

To find the centroid (geometrical center) of any triangle, draw construction lines from each vertex to the midpoint of the opposite line of the triangle. The centroid of the triangle is the intersection of the three lines. See Figure 17-9.

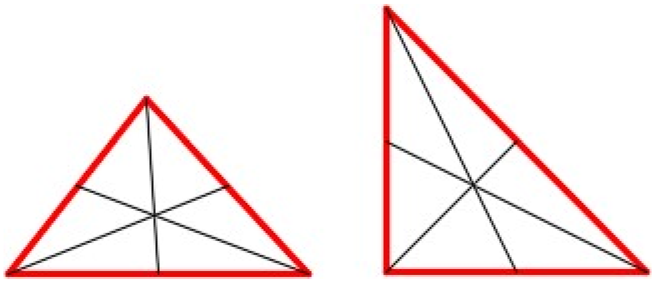


Figure 17-9
Finding the Center of a Triangle

Key Principles

Key Principles in Module 17

1. A point is an AutoCAD drawing object that is one pixel in size.
2. A node is the object snap mode for a point.
3. AutoCAD stores the accuracy of the drawing object's properties and locations to a very high degree. The UNITS command is used open the Drawing Units dialogue box where the you can set the format of how AutoCAD displays these properties and locations.
4. The DIVIDE command insert points on the object by dividing the object's length or perimeter into an equal number of segments specified by the user.
5. The MEASURE command inserts points on an object at a user specified distance.

Lab Exercise 17-1

Time allowed: 45 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 17-1	2D Metric	Millimeters

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Construction	Construction objects	253
Object	All objects	Red

Step 1

Draw the object shown in the figure using the layering scheme. (Figure Step 1A and 1B)

AUTHOR'S COMMENTS: Do not delete the construction lines. HOR = Horizontal and VER = Vertical.

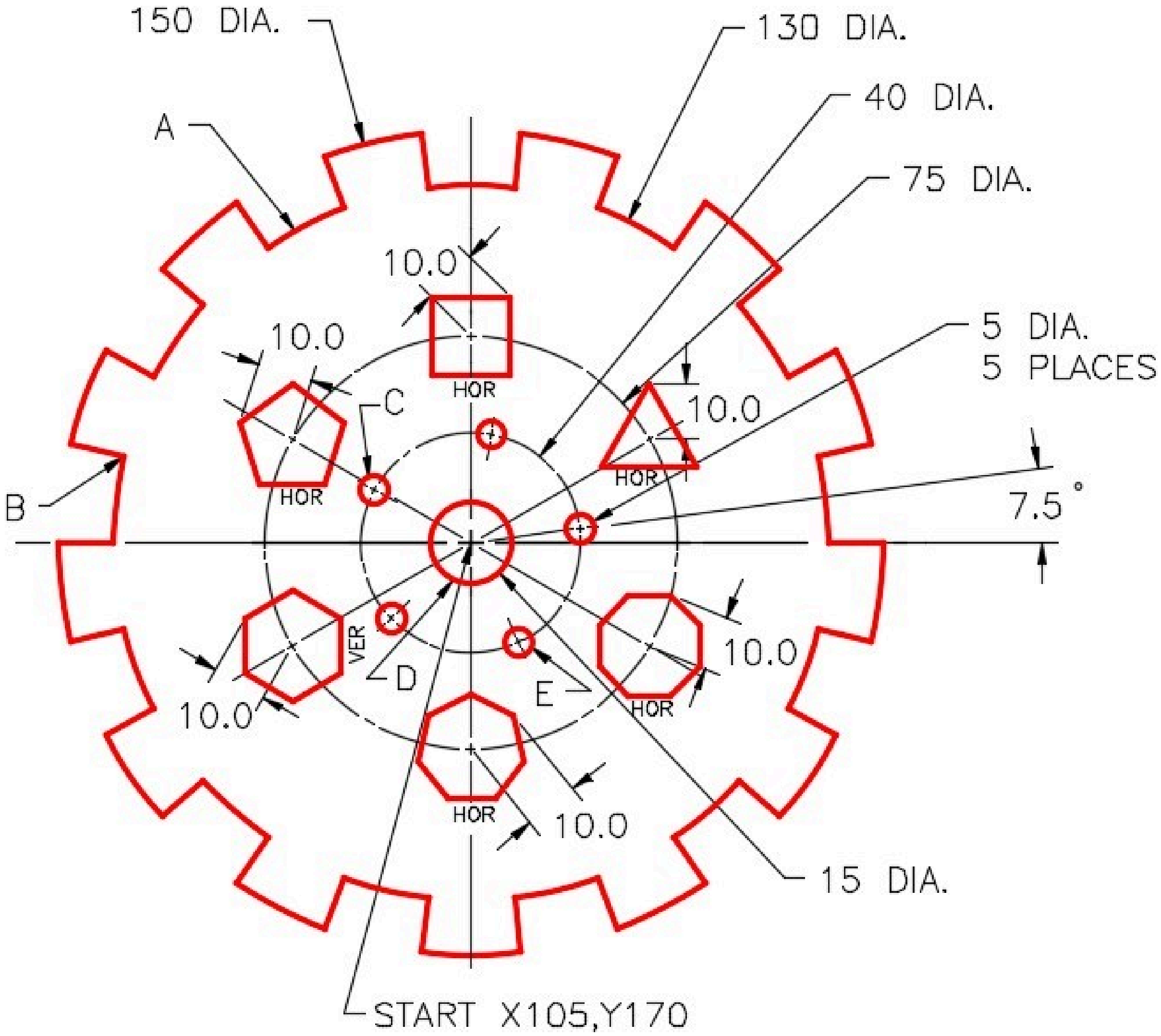

150 DIA.
130 DIA.
40 DIA.
75 DIA.
A
B
C
D
E
5 DIA.
5 PLACES
7.5 °
10.0
10.0
10.0
10.0
10.0
10.0
HOR
HOR
HOR
HOR
HOR
VER
15 DIA.
START X105,Y170

Figure Step 1A

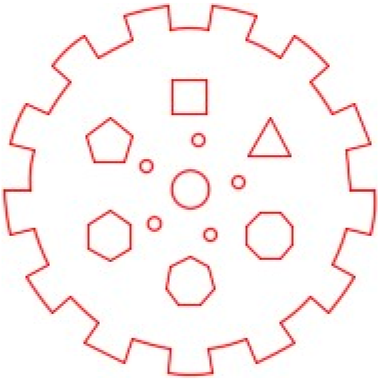


Figure Step 1B
Completed Drawing

Step 2

Set the insertion units and check your drawing with the key.

Step 3

Turn layer: Key off and freeze layer: Construction.

Step 4

Using either the DIST command or the Properties window, find the following answers with the precision of 8 decimal places:

A. Length of arc A: ______________________

B. Distance from the center of circle C to center of circle E. ______________________

C. Shortest distance from corner B to the circumference of circle C. ______________________

D. The circumference of circle D: ______________________

(Check your answers on the key)

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

Using the DIVIDE command, divide the large circle into 26. (Figure Hint 1)

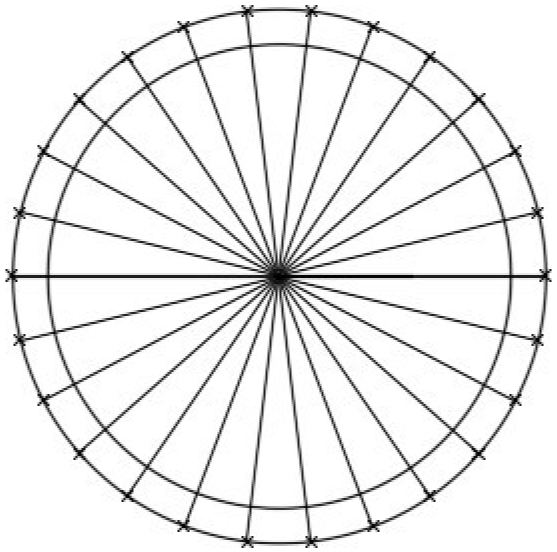


Figure Hint 1

Hint 2

Rotate the cursor 7.5 degrees and divide the 40 diameter circle into 5. (Figure Hint 2)

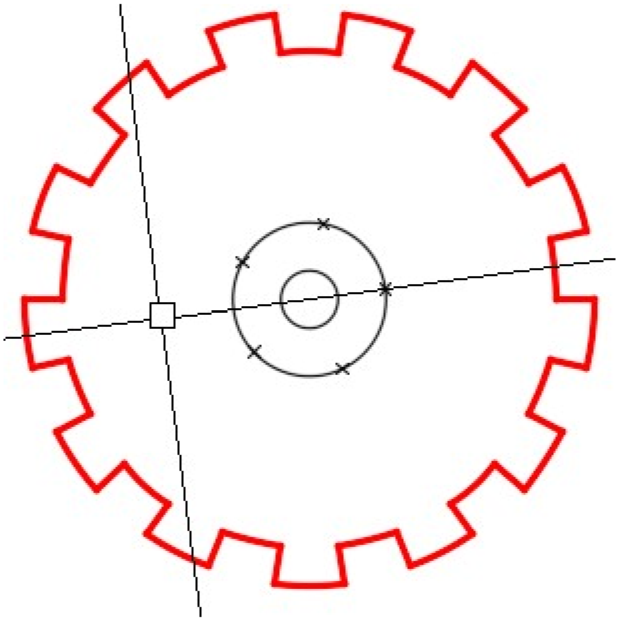


Figure Hint 2

Hint 3

Rotate the cursor 30 degrees and divide the 75 diameter circle into 6. (Figure Hint 3)

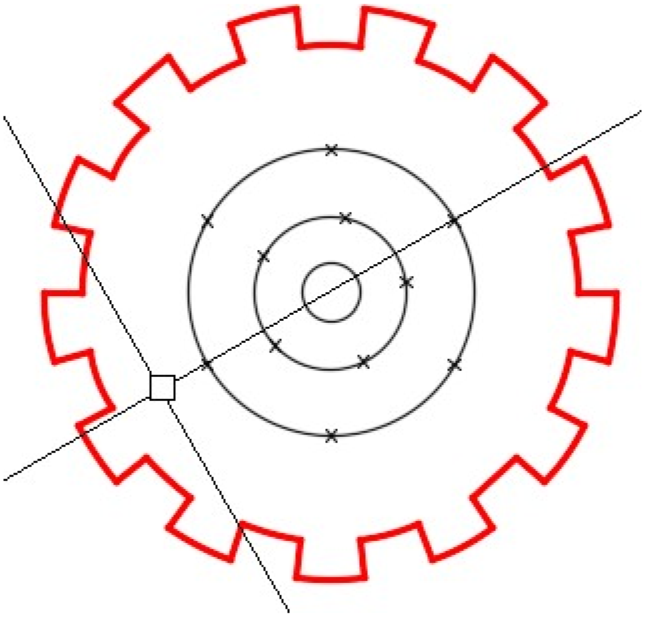


Figure Hint 3

Hint 4

Draw 6 – 20 diameter circles locating their centers on the points. (Figure Hint 4)

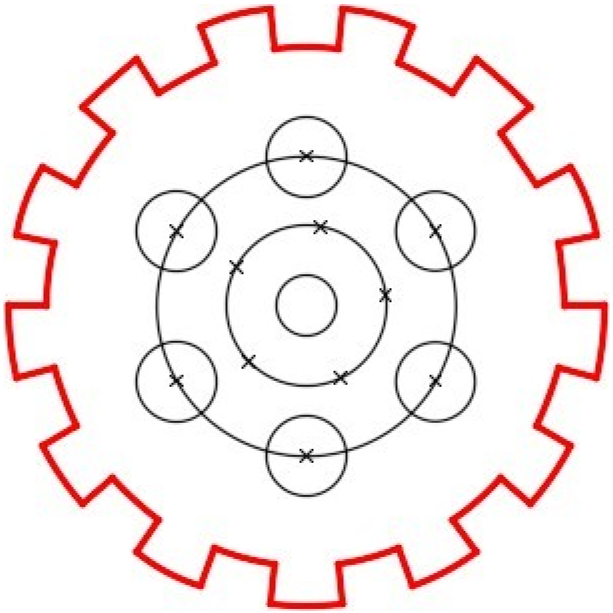


Figure Hint 4

Hint 5

To draw the polygons, rotate the cursor to the appropriate angle as shown in the list below. (Figure Hint 5A and 5B)

Triangle = 90 degrees

Square = 45 degrees

Pentagon = 90 degrees

Hexagon = 90 degrees

Heptagon = 90 degrees

Octagon = 22.5 degrees

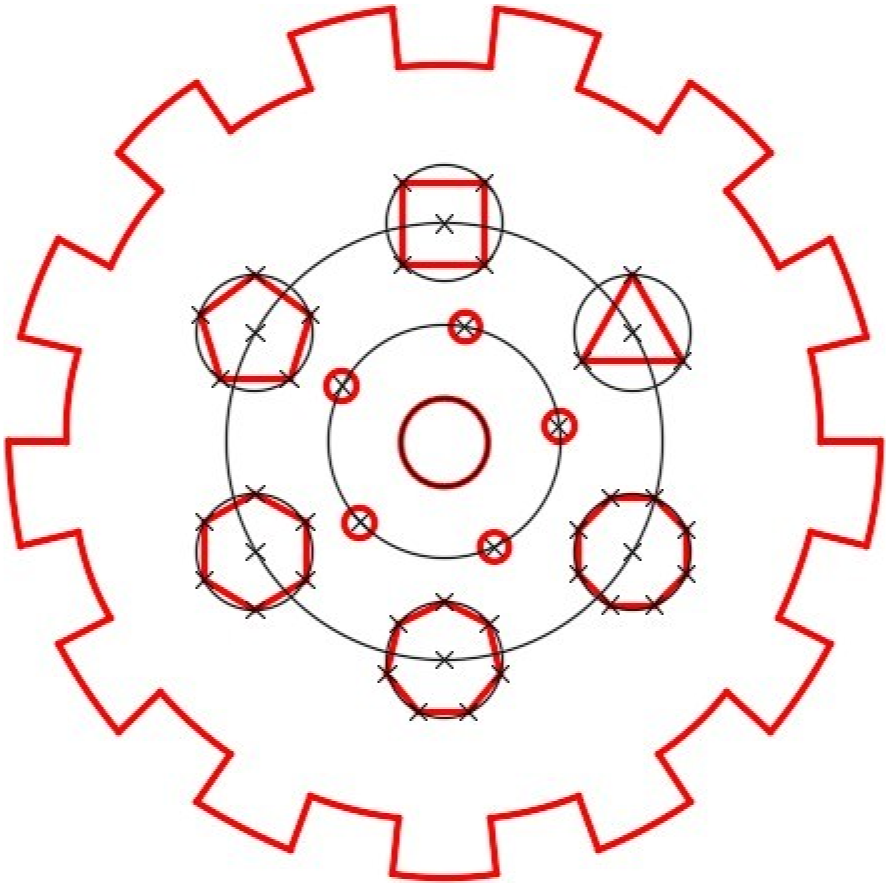


Figure Hint 5A

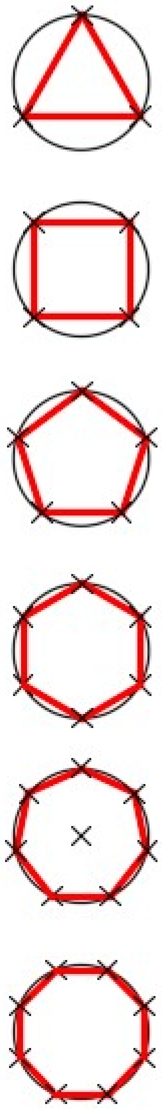


Figure Hint 5B

Lab Exercise 17-2

Time allowed: 50 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 17-2	2D English	Inches

Layering Scheme

Layer Name	Objects on Layer	Color
Construction	Construction objects	253
Object	All objects	Red

Step 1

Draw the object shown in the figure using the layering scheme. (Figure Step 1A and 1B)

Step 2

Set the insertion units and check your drawing with the key.

Step 3

Turn layer: Key off and freeze layer: Construction.

AUTHOR'S COMMENTS: Do not delete the construction lines.

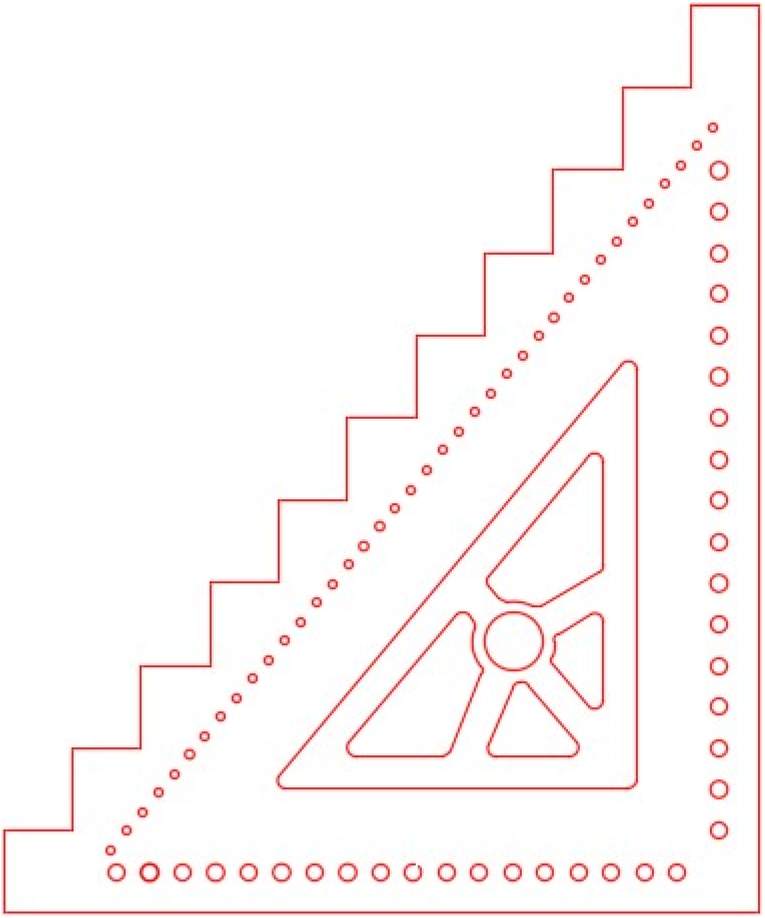


Figure Step 1A
Completed Drawing

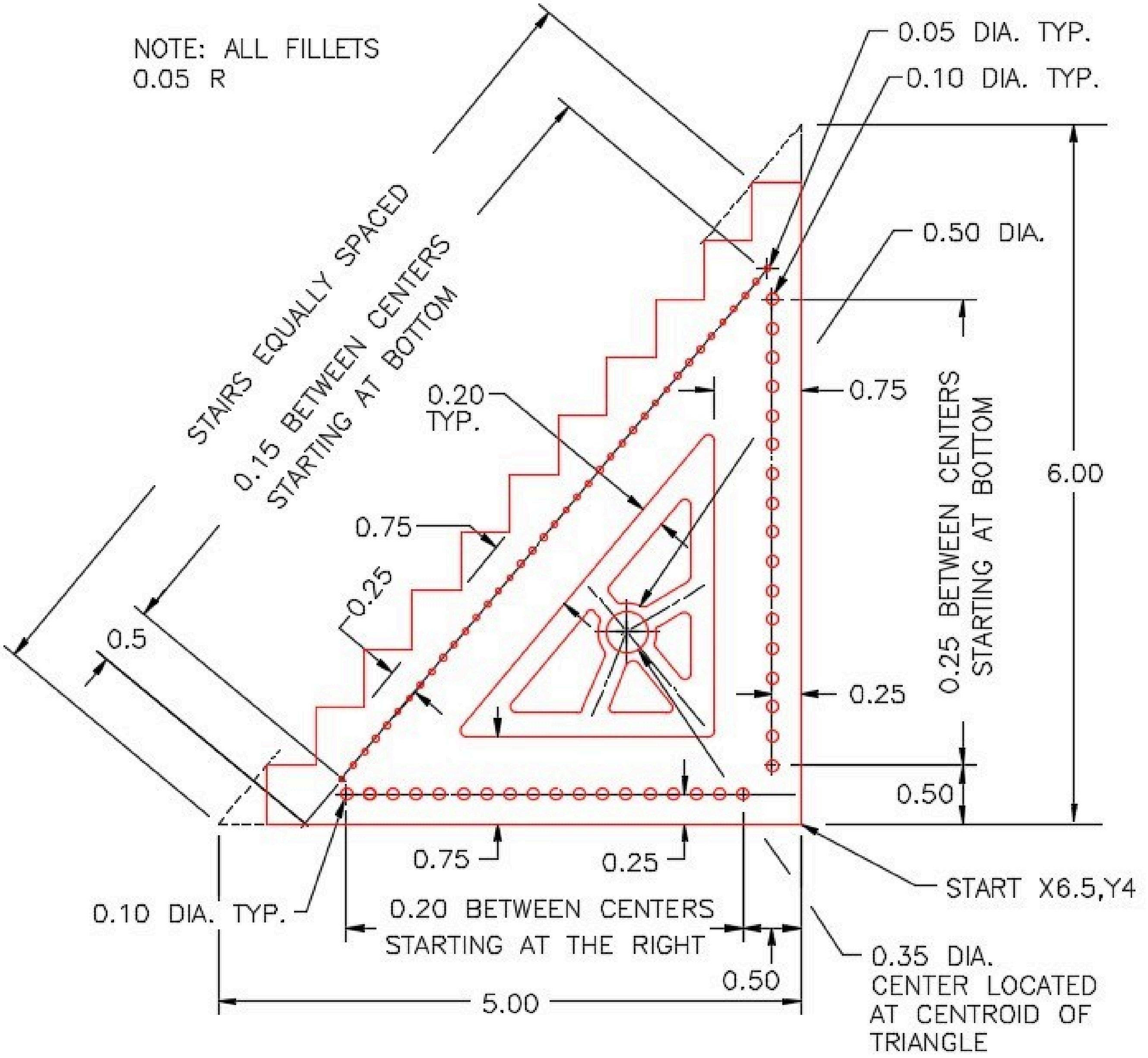

NOTE: ALL FILLETS
0.05 R
STAIRS EQUALLY SPACED
0.15 BETWEEN CENTERS
STARTING AT BOTTOM
0.05 DIA. TYP.
0.10 DIA. TYP.
0.50 DIA.
0.20
TYP.
0.75
0.75
0.25
0.5
0.25
0.25
0.25 BETWEEN CENTERS
STARTING AT BOTTOM
6.00
0.50
0.75
0.25
START X6.5,Y4
0.10 DIA. TYP.
0.20 BETWEEN CENTERS
STARTING AT THE RIGHT
0.50
5.00
0.35 DIA.
CENTER LOCATED
AT CENTROID OF
TRIANGLE

Figure Step 1B

AUTHOR’S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

Using the DIVIDE command, divide the inclined line into 12. (Figure Hint 1)

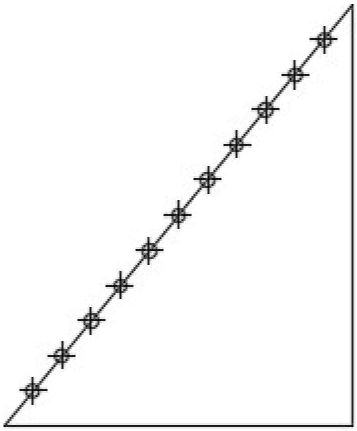


Figure Hint 1

Hint 2

Construct the steps using ortho mode. (Figure Hint 2)

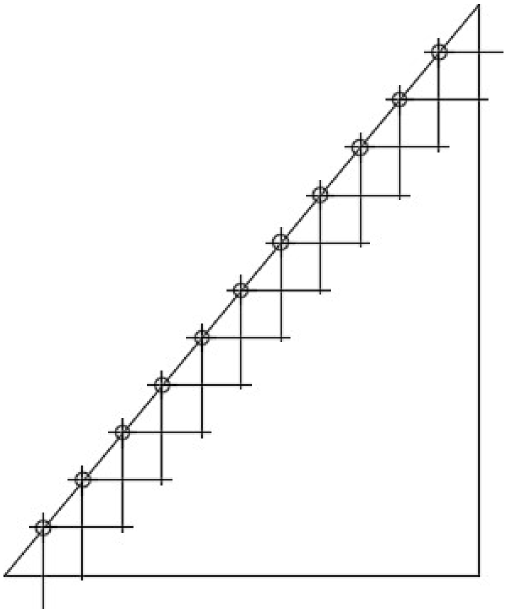


Figure Hint 2

Hint 3

Trim the steps and change the layer they reside on to layer: Object. Offset the inclined line and draw a circle to measure the distance along the inclined line. (Figure Hint 3)

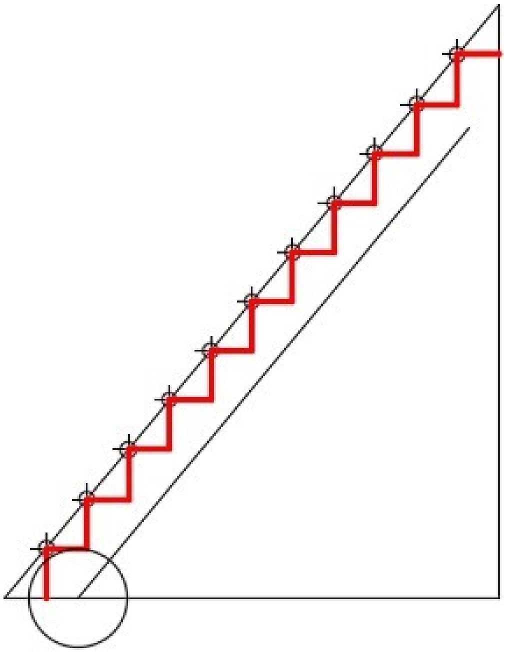


Figure Hint 3

Hint 4

Draw the triangle by offsetting the three outsides 0.75 inches as shown in the dimensioned drawing. Using the Geometry Lesson: Finding the Centroid of any Triangle, find the centroid of the triangle and draw a 0.35 diameter circle using the intersection of the lines to locate its center. (Figure Hint 4)

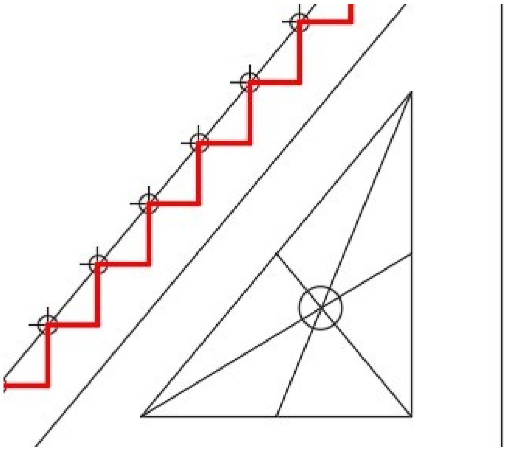


Figure Hint 4

Hint 5

Offset the outside lines of the triangle 0.20 inches towards the center. (Figure Hint 5)

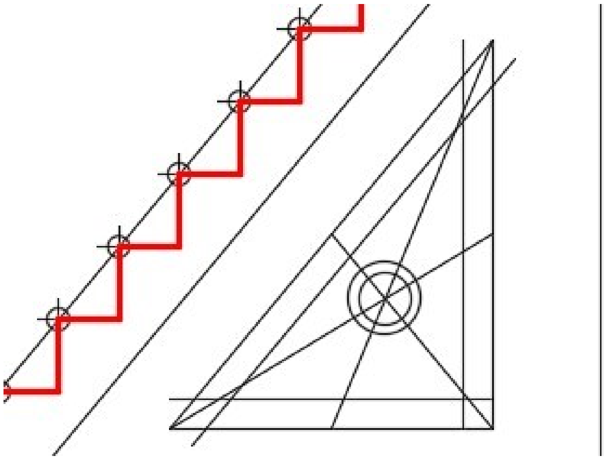


Figure Hint 5

Hint 6

Offset the inside lines 0.10 inches on each side. (Figure Hint 6)

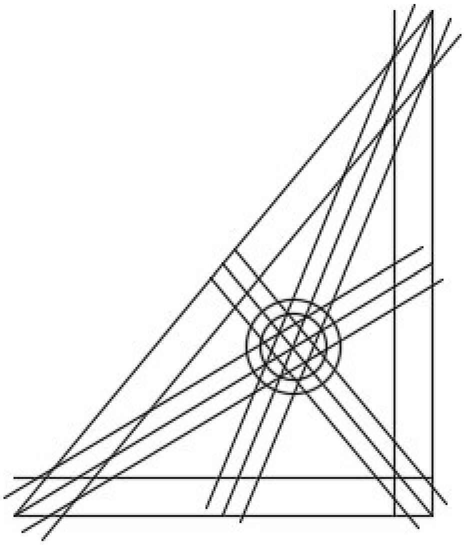


Figure Hint 6

Hint 7

Trim all lines and the circle. Fillet all corners using a radius of 0.05 inches. The fillets should always be inserted last. (Figure Hint 7)

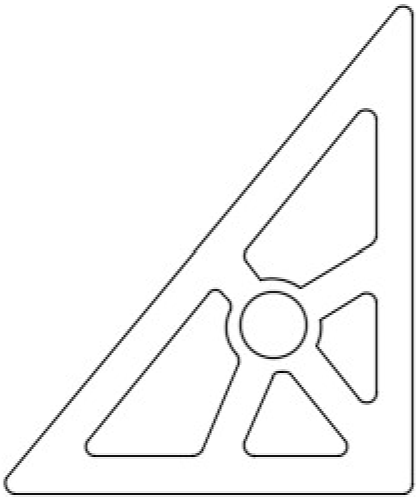


Figure Hint 7

Hint 8

Change the layer of all objects to layer: Object. (Figure Hint 8)

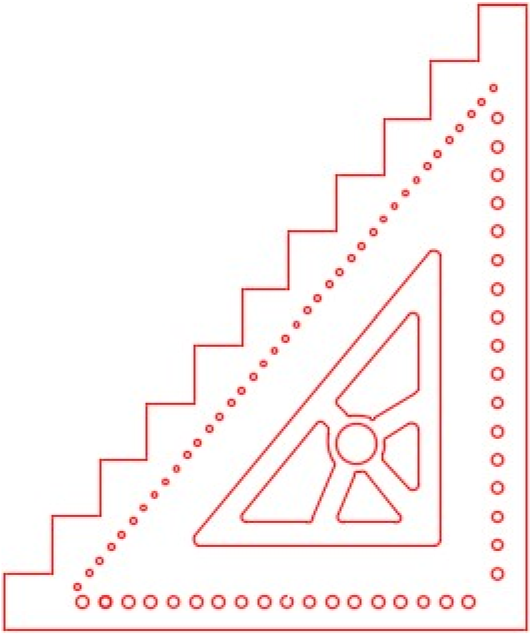


Figure Hint 8

Module 18: Layouts – Part 1

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe Model space and Paper space and explain how they are used in AutoCAD.
2. Describe a Layout and explain how they are used in an AutoCAD drawing.
3. Apply the MSPACE and PSPACE commands to change the working space.
4. Describe a viewport and explain how to create, edit, lock, and set the viewport scale.
5. Apply the MVIEW command to create viewports on layout drawings.

Model and Paper Space

Up to this point in the book, you have been drawing Model space. In this module, Paper space and layouts will be introduced. The ideal way to work in AutoCAD is to draw all real objects in Model space and draw all objects that not real, in Paper space.

Model space is a limitless three-dimensional space where the real objects of the model are drawn. For example, if you are drawing a house, everything that is part of the house such as the walls, windows, doors, etc. are drawn in Model space. All parts of the house would be drawn at full scale or full size, exactly as they exist in real life.

Paper space is a virtual two-dimensional space where all objects that are related to the plotted piece of paper, called a layout, are drawn. Objects drawn in Paper space are things like the titleblock, border, dimensions, notes, labels, and bills of material. Paper space is also drawn at full size but the units are the same units as the paper, such as inches or millimeters. The Model space units and the Paper space units do not have to be the same in a drawing.

Layouts

A *layout* represents a sheet of paper on which one or more scaled views of the model is created complete with such thing as dimensions, notes, border, and titleblock. Layouts are drawn in Paper space. They are an important and essential AutoCAD feature and should be used to plot all AutoCAD drawings.

As many layouts as required can be created in a drawing. The AutoCAD 2D book supplies pre- drawn layouts in two template files named ‘ 2D Layout English.dwt ‘ and ‘ 2D Layout Metric.dwt ‘. You can create your own layouts and save them in a drawing or template file. Creating your own templates is taught in the AutoCAD 2D Advanced book.

Model and Layout Tabs

The Model and Layout tabs are located along the bottom of the Graphic window as shown in Figure 18-1 and 18-2. There is only one Model tab in each drawing but there can be an unlimited number of Layout tabs. You can create your own layouts tabs. This taught in the AutoCAD 2D Advanced book.

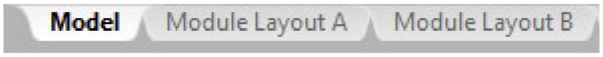


Figure 18-1
Model – Enabled

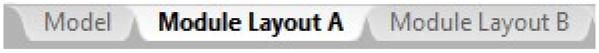


Figure 18-2
Module Layout A Layout – Enabled

When the Model tab is enabled, all drawing objects drawn by you will be Model space objects and exist in Model space. When one of the Layout tabs is enabled, as shown in Figure 18-2, the layout drawing will be displayed. In a Layout tab, you can work in either Paper space or Model space.

The Working Space

The current space, which is either Model or Paper, is called the *working space*. When the MODEL icon is enabled, the working space is Model and when the PAPER icon is enabled, the working space is Paper.

When a Layout tab is enabled, the working space can be changed back and forth between Model or Paper space by toggling the PAPER/MODEL icon in the Status bar. See Figures 18-3 and 18-4. Click the MODEL icon to change the working space to Paper space or click the PAPER icon to change the working space to Model space.

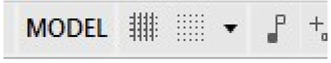


Figure 18-3
Model Space is Currently the Working Space

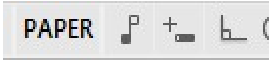


Figure 18-4
Paper Space is Currently the Working Space

The Paper Space Icon

When a Layout tab is enabled and the working space is Paper, the Paper space icon, as shown in Figure

18-5, will display. When the icon is displayed, it is indicating to you that Paper space is the working space and all objects drawn will be Paper space objects and exist in Paper space only.

If this icon does not display, even when a layout is displayed, all objects drawn will be Model space objects and exist in Model space only.

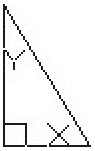


Figure 18-5 The Paper Space Icon

AutoCAD Command: MSPACE

The MSPACE command is used to change the current working space to Model space.

Shortcut: **MS**

AutoCAD Command: PSPACE

The PSPACE command is used to change the current working space to Paper space.

Shortcut: **PS**

Viewports

A *viewport* is a rectangular or square hole cut through the sheet of drawing paper (the layout) so that the model can be viewed through it. See Figure 18-6. It is a 2D Paper space drawing object. The view of the model can be orientated and scaled. A layout can have an unlimited number of viewports cut through it, all viewing the same model at different scales and orientations.

Each viewport will display all visible objects that exist in Model space. If the model is changed after a viewport is created, the viewport will automatically display the changes since it is merely viewing the model. The orientation and scale of a viewport can be adjusted at any time. After a viewport is set, the display can be locked so that it cannot be accidentally changed. If changes are required, the viewport display must be unlocked before they can be made. It is best to insert the viewport on its own layer and turn that layer off before the drawing is plotted to avoid plotting the viewport drawing object.

Paper space objects like dimensions, notes, titleblocks, borders, and text are then drawn in Paper space on the sheet of paper. They are drawn full size using the same units as the paper is measured.

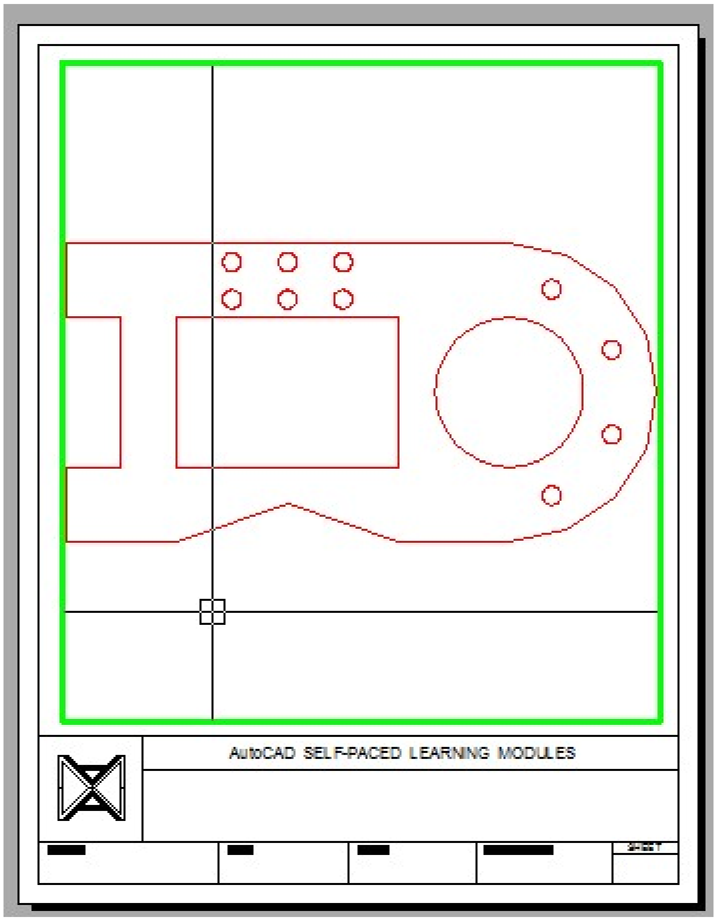


Figure 18-6
A Viewport

AutoCAD Command: MVIEW

The MVIEW command is used to create a viewport on a layout. It is a Paper space command and the viewport it creates is a Paper space drawing object.

Shortcut: **MV**

WORKALONG: Working With Layouts

Step 1

Start a new drawing using the template: 2D Layout English.

AUTHOR’S COMMENTS: Ensure that you select the correct template. The templates that you have being using to this point in the book will not be used from this module on. The new templates that you will be using are named: 2D Layout English and 2D Layout Metric.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 18-1.

Step 3

Using what you learned to this point in the book, draw the object shown in the figure on layer: Object. Draw any construction objects on layer: Construction. (Figure Step 3)

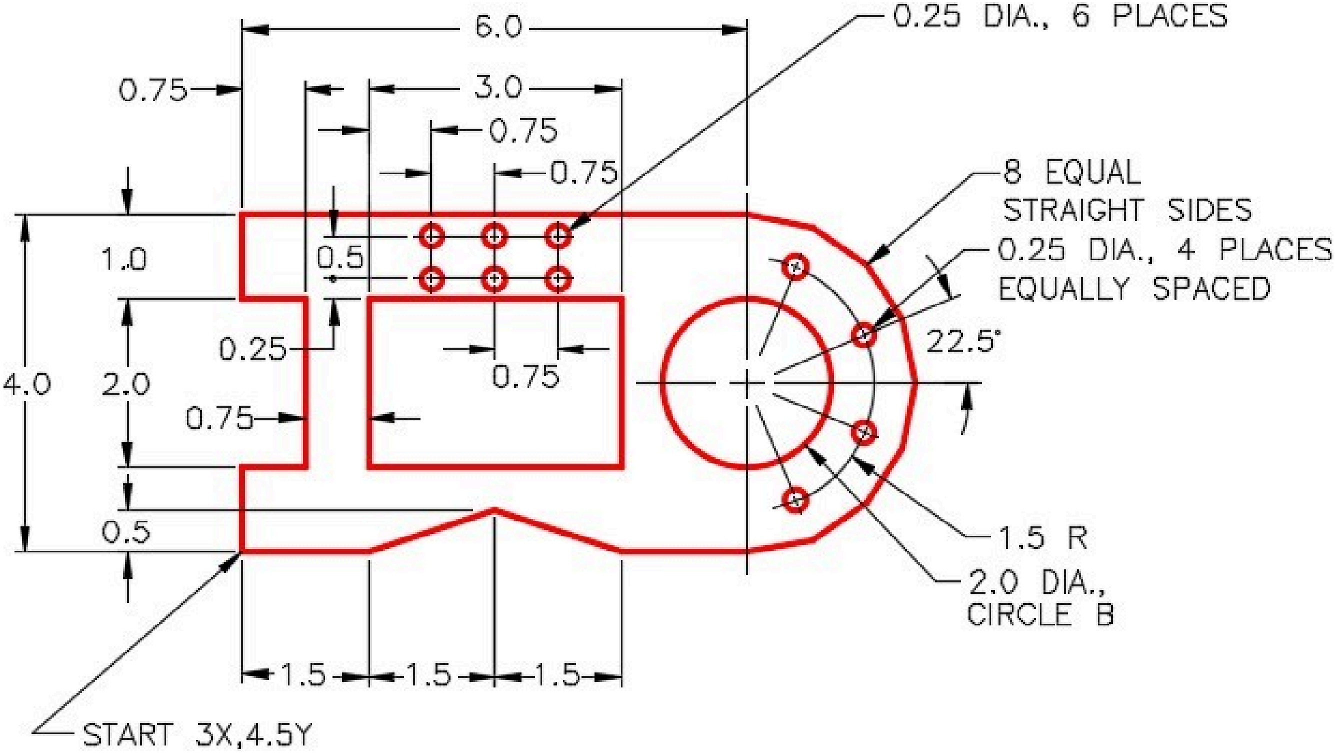

6.0
0.25 DIA., 6 PLACES
0.75
3.0
0.75
0.75
8 EQUAL
STRAIGHT SIDES
0.25 DIA., 4 PLACES
EQUALLY SPACED
1.0
0.5
22.5°
0.25
4.0
2.0
0.75
0.75
0.5
1.5 R
2.0 DIA.,
CIRCLE B
1.5
1.5
1.5
START 3X,4.5Y

Figure Step 3

AUTHOR'S COMMENTS: You will notice that the drawing already contains the layers that you require to complete the lab exercises. They were created in the template file and will save you creating most layers in the drawings in the remaining drawings in this book.

Step 4

Freeze layer: Construction. The drawing should appears as shown in the figure. (Figure Step 4)

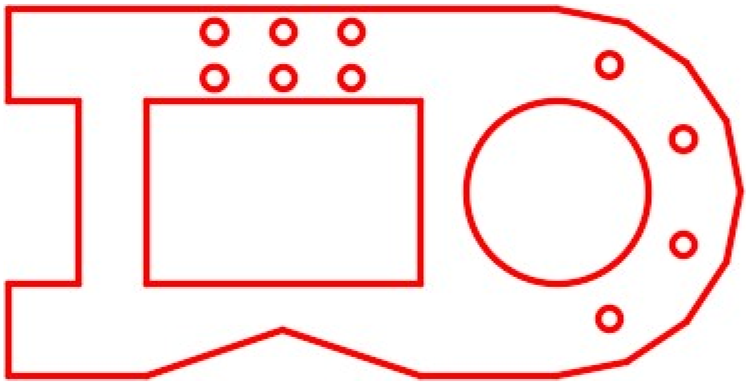
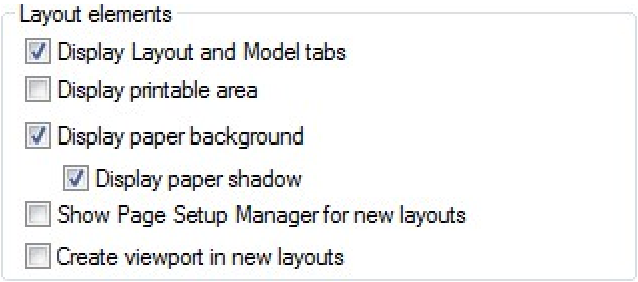


Figure Step 4

Step 5

Enter the OPTIONS command to open the Options dialogue box. In the Display tab, compare the settings in the Layout elements area and change them, if necessary, to match the figure. Click OK to close the dialogue box. (Figure Step 5)



Layout elements

☑ Display Layout and Model tabs
☐ Display printable area
☑ Display paper background
  ☑ Display paper shadow
☐ Show Page Setup Manager for new layouts
☐ Create viewport in new layouts

Figure Step 5

MUST KNOW: Model space is a limitless three-dimensional space where the real objects of the model are drawn full scale while Paper space is a virtual two-dimensional space where all objects related to the plotted paper drawing are drawn.

Step 6

Enable layout: Module Layout A by clicking it. Your Graphic window should match the figure. (Figure Step 6)

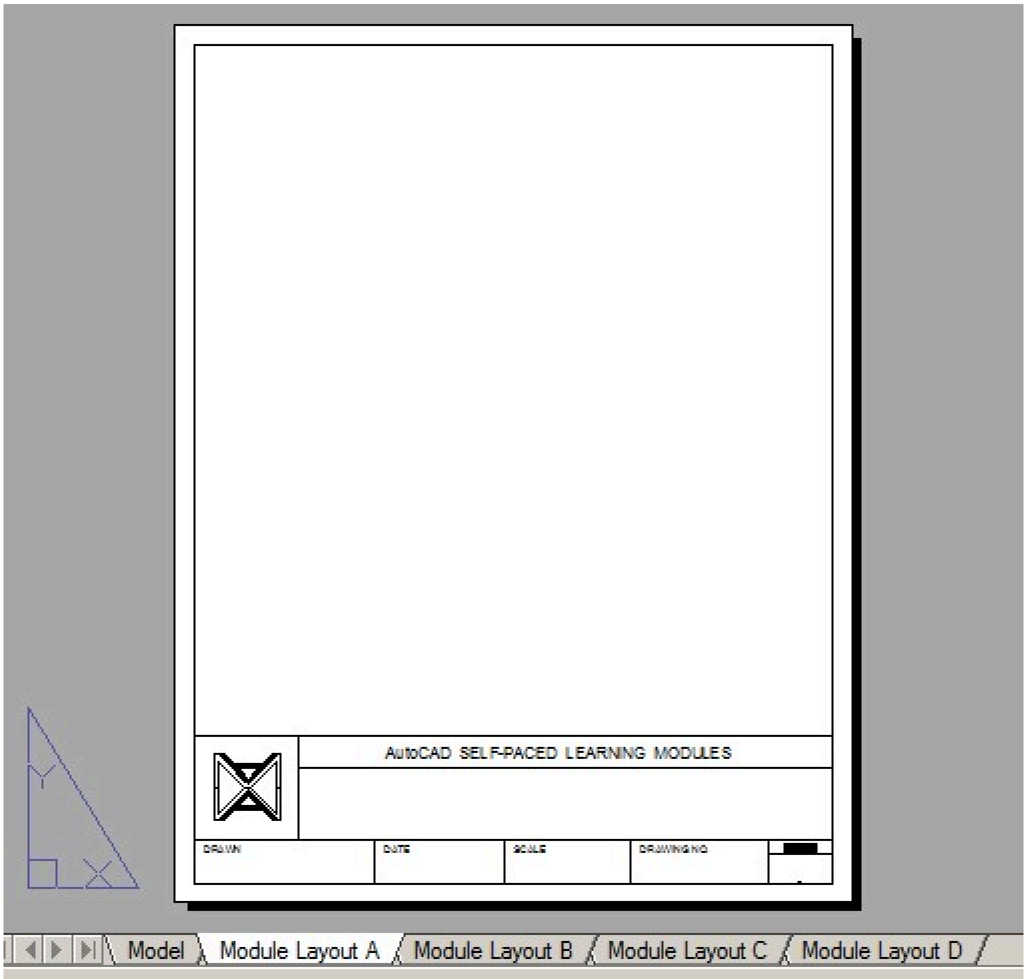

AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO
Model
Module Layout A
Module Layout B
Module Layout C
Module Layout D

Figure Step 6

AUTHOR'S COMMENTS: A layout is a piece of paper that can be plotted. The model is located behind the paper but cannot be seen until a hole (viewport) is cut in the paper using the MVIEW command. The paper size in Module Layout A is A size or 8 1/2X11 inches.

Step 7

Paper space should be the working space. To check that, ensure that the PAPER icon is enabled on the Status bar. (Figure Step 7)

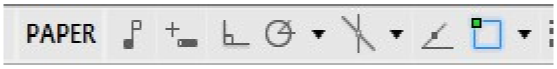


Figure Step 7

Step 8

Set layer: Viewport as the current layer. Enter the MVIEW command, as shown below, to create a viewport on the layout. (Figure Step 8)

Command: **MVIEW**

Specify corner of viewport or [ON/OFF/Fit/Shadeplot/Lock/Object/Polygonal/Restore/LAyer/2/3/4] <Fit>: **P1**

Specify opposite corner: **P2**

Command:

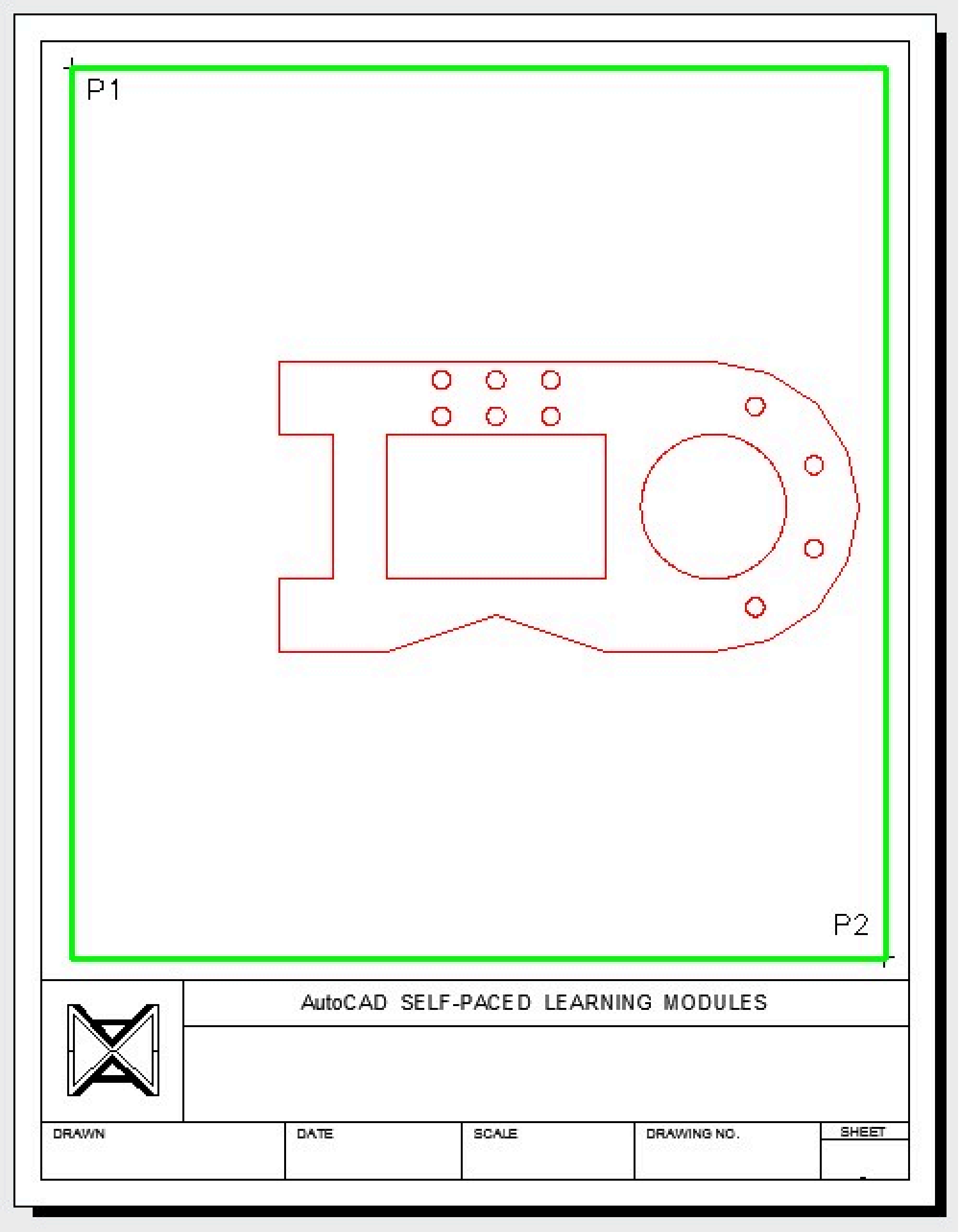

P1
P2
AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO.
SHEET

Figure Step 8

AUTHOR'S COMMENTS: Selecting two opposite corners in the MVIEW command creates a viewport. The locations for P1 and P2 can be selected by eye. Their exact locations are not important. Imagine the viewport as cutting a hole in the piece of paper to view the model located behind the paper. Since a viewport is a drawing object, you can delete it and recreate it if you want to change its size or location. Ensure that you are in Paper space to delete a viewport. The AutoCAD 2D Advanced book teaches you how to resize viewports.

Step 9

Change the working space to Model by clicking the PAPER icon on the Status bar. After you do that, it will display MODEL as shown in the figure. (Figure Step 9A and 9B)

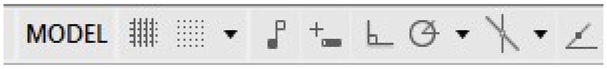


Figure Step 9A

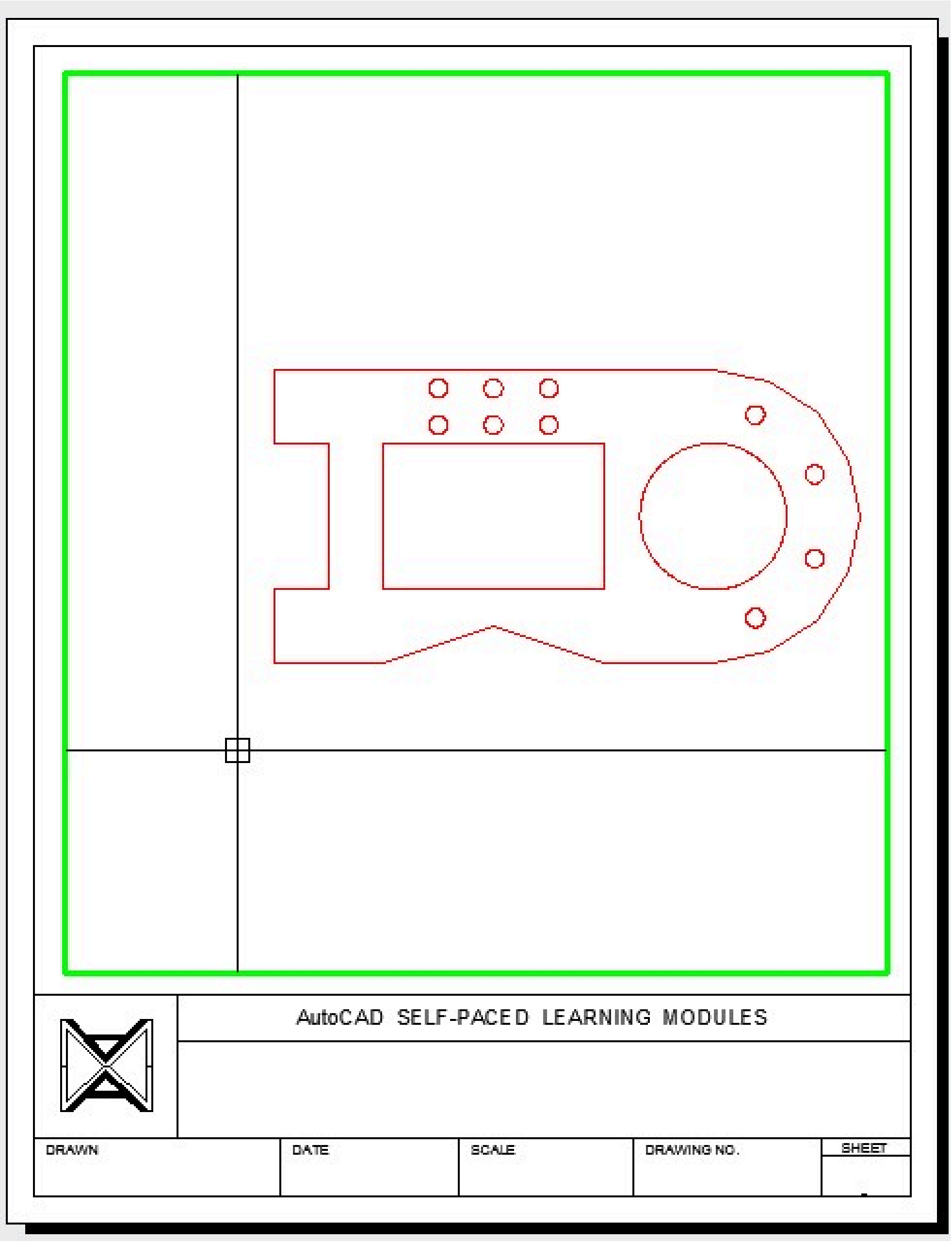

AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO.
SHEET

Figure Step 9B

AUTHOR’S COMMENTS: Note that when the working space is Model, the crosshairs will only display inside the viewport (the hole in the paper).

Step 10

Enter the ZOOM Extents command to display all model objects in the viewport. (Figure Step 10)

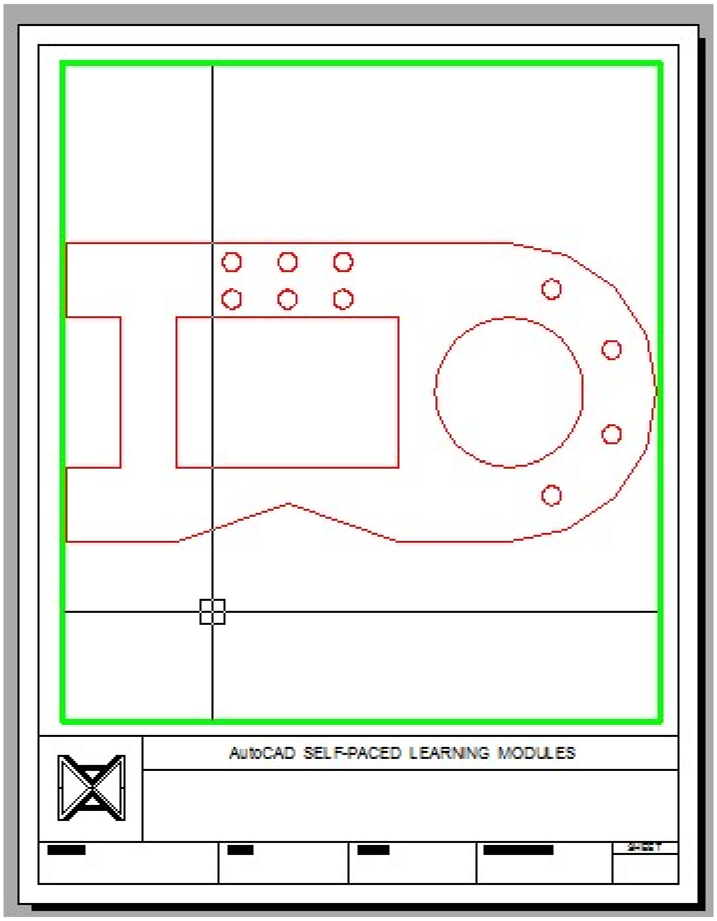

AutoCAD SELF-PACED LEARNING MODULES
SHEET

Figure Step 10

Step 11

Change the working space to Paper space. (Figure Step 11)

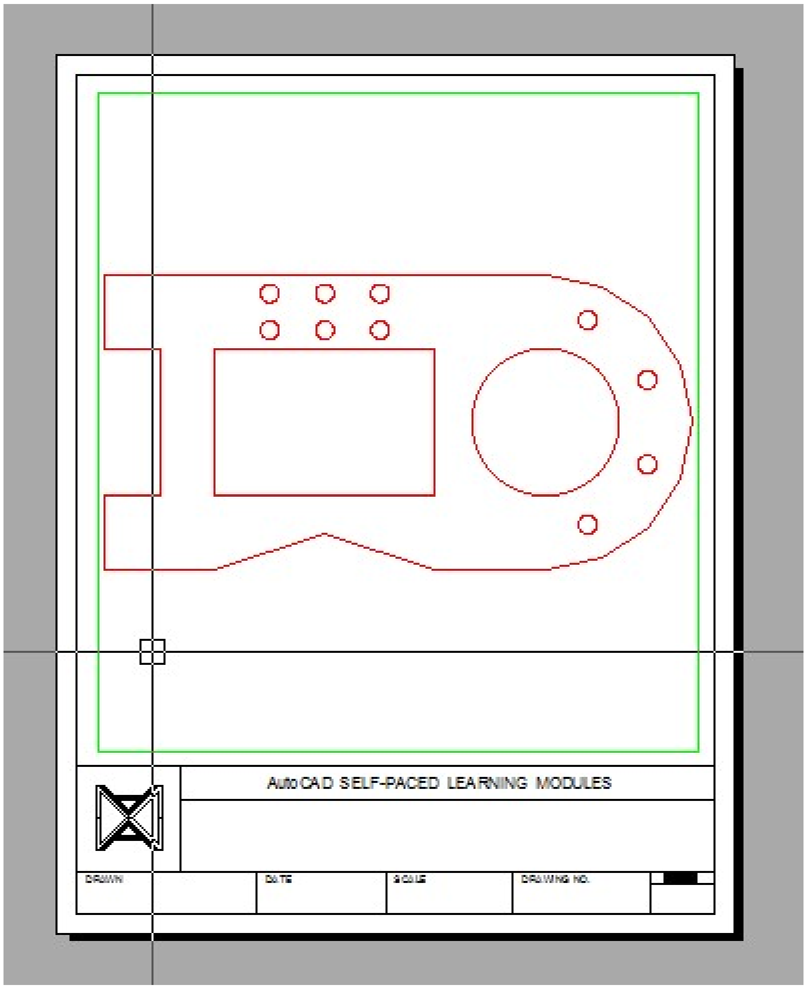
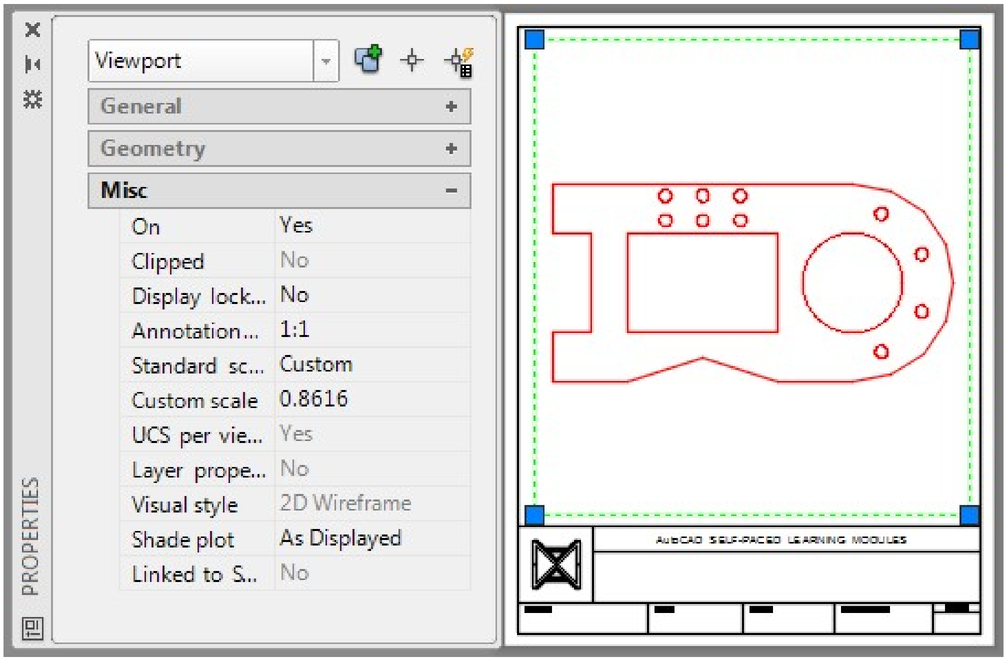

AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO.

Figure Step 11

AUTHOR’S COMMENTS: Note that when the working space is Paper, the crosshairs will display to cover the complete Graphic window. This means that you can draw anywhere in the Graphic window. All objects drawn, while the working space is Paper, will be Paper space objects.

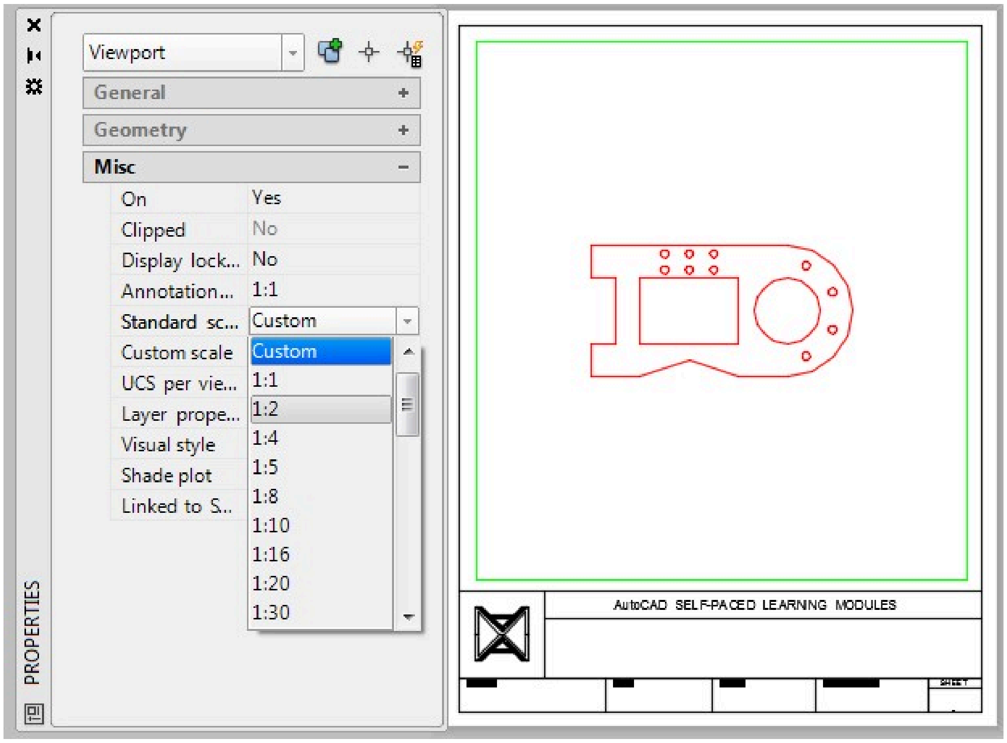
Step 12

Open the Properties window. Without entering a command, select the viewport. It will highlight, as shown in the figure, and grips will display on the four corners. While the viewport is selected, pull down the Standard scale property and set it to 1:2. (Figure Step 12A and 12B)


PROPERTIES
Viewport
General
Geometry
Misc
AutoCAD SELF-PACED LEARNING MODULES

On	Yes
Clipped	No
Display lock...	No
Annotation...	1:1
Standard sc...	Custom
Custom scale	0.8616
UCS per vie...	Yes
Layer prope...	No
Visual style	2D Wireframe
Shade plot	As Displayed
Linked to S...	No

Figure Step 12A

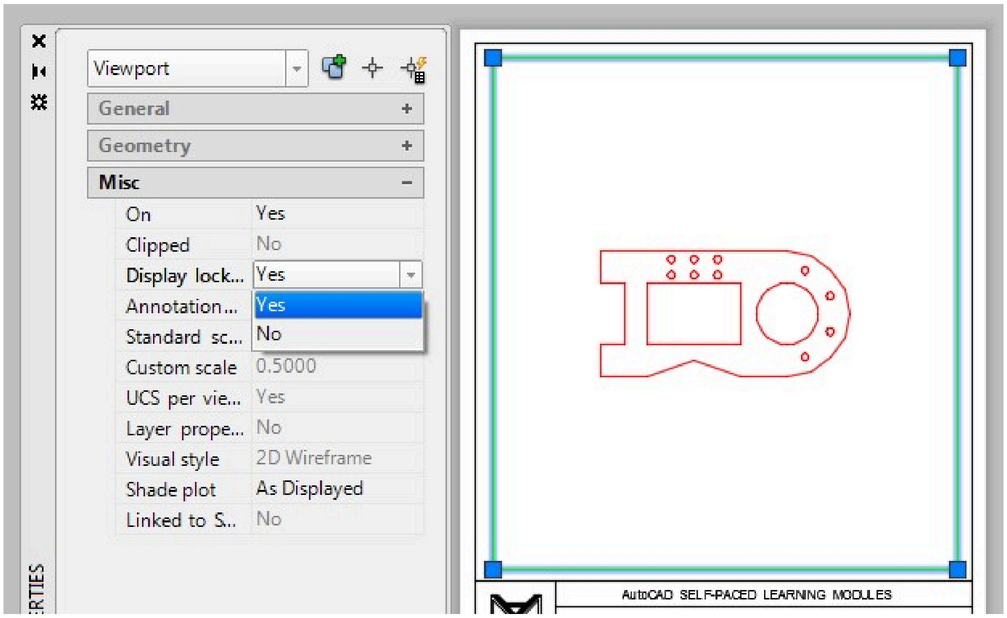

PROPERTIES
Viewport
General
Geometry
Misc
Custom
1:1
1:2
1:4
1:5
1:8
1:10
1:16
1:20
1:30
AutoCAD SELF-PACED LEARNING MODULES

On	Yes
Clipped	No
Display lock...	No
Annotation...	1:1
Standard sc...	Custom
Custom scale	
UCS per vie...	
Layer prope...	
Visual style	
Shade plot	
Linked to S...	

Figure Step 12B

Step 13

While the viewport is selected, pull down the Display locked property and set it to Yes. (Figure Step 13)



Viewport

General

Geometry

Misc

On	Yes
Clipped	No
Display lock...	Yes
Annotation...	
Standard sc...	
Custom scale	0.5000
UCS per vie...	Yes
Layer prope...	No
Visual style	2D Wireframe
Shade plot	As Displayed
Linked to S...	No

Figure Step 13

AUTHOR’S COMMENTS: When the Display locked property is set to Yes, the viewport cannot be altered by zooming and panning. That means that the scale cannot be changed. It is best to always lock the viewport display after it is set to protect it from accidentally being changed.

Step 14

Your completed layout should appear as shown in the figure. (Figure Step 14)

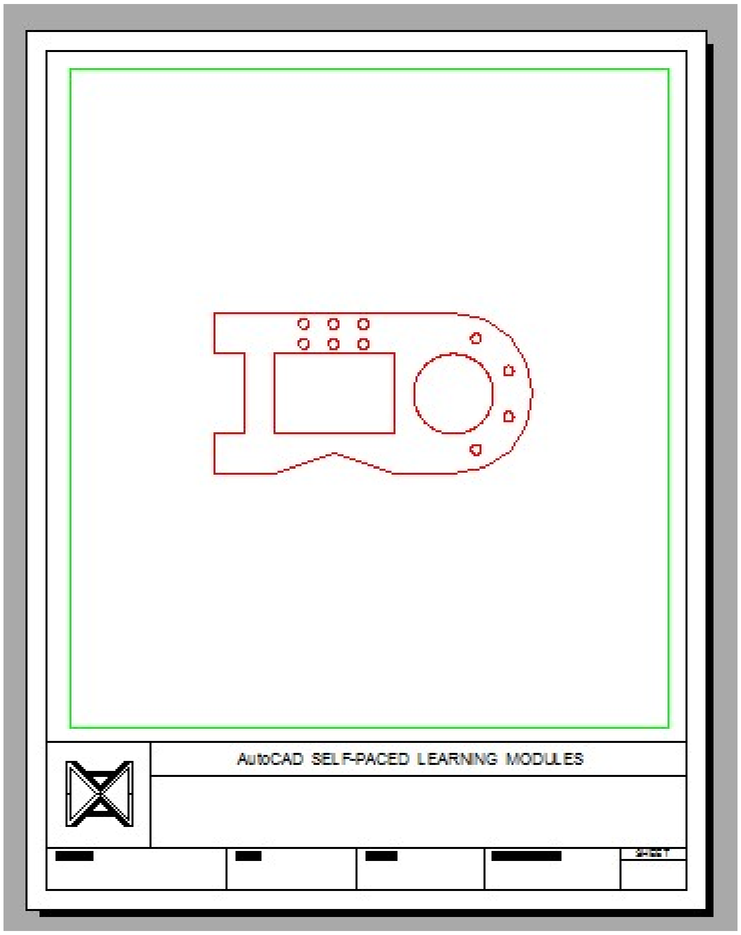

AutoCAD SELF-PACED LEARNING MODULES
SHEET

Figure Step 14

AUTHOR'S COMMENTS: The model in this viewport is set to a scale of 1:2 or one-half size of the original size of the model.

Step 15

Enable the display of layout: Module Layout B. (Figure Step 15)

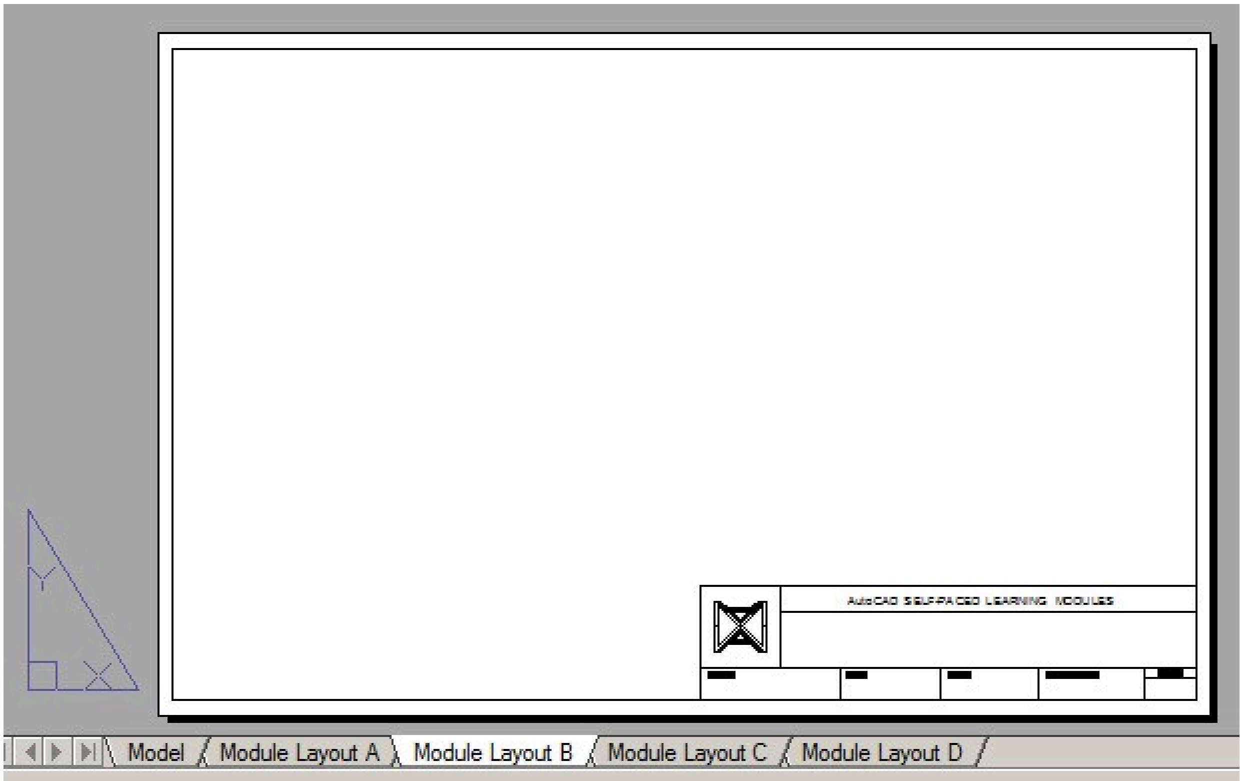

AutoCAD SELF-PACED LEARNING MODULES
Model
Module Layout A
Module Layout B
Module Layout C
Module Layout D

Figure Step 15

AUTHOR'S COMMENTS: The paper size in Module Layout B is B size or 11X17 inches.

Step 16

Using what was taught in Step 8, enter the MVIEW command, two times, to create two viewports. After creating the viewports, change the working space to Model and click inside the left viewport. Enter the ZOOM Extents command. Click inside the viewport on the right and enter the ZOOM Extents command again. Change the working space to Paper and the layout should appear similar to the figure. (Figure Step 16)\

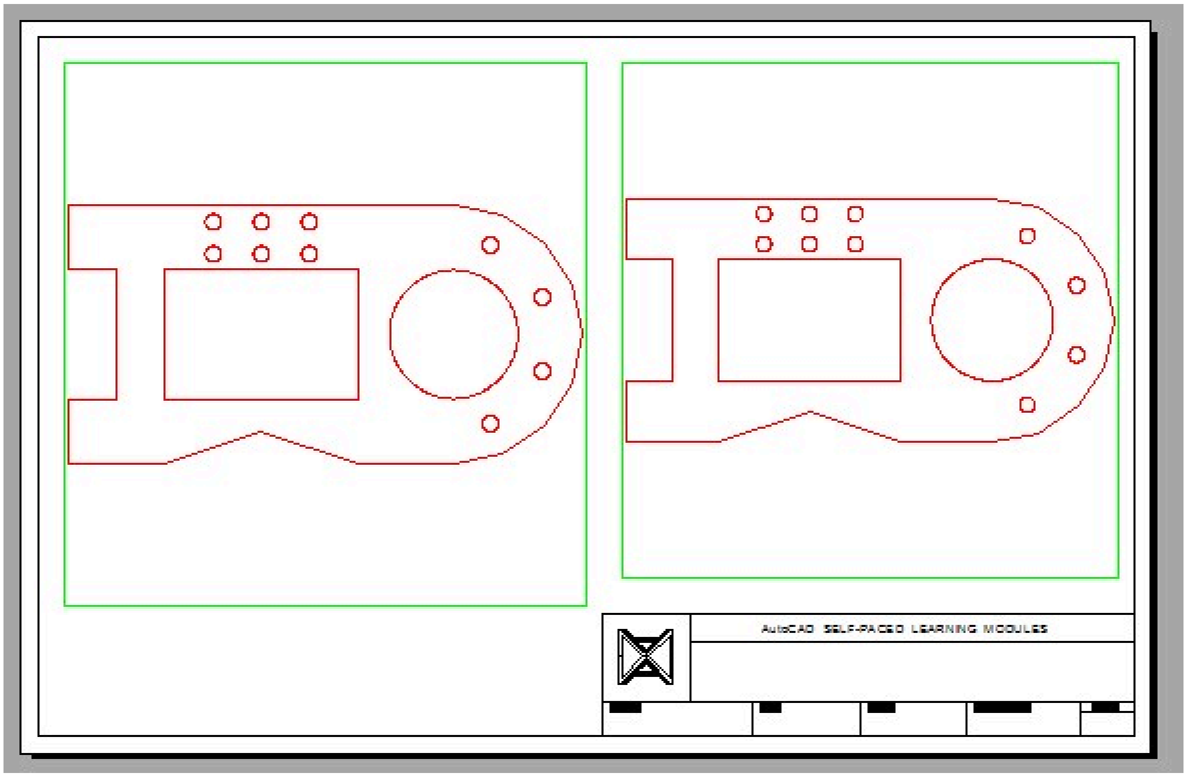
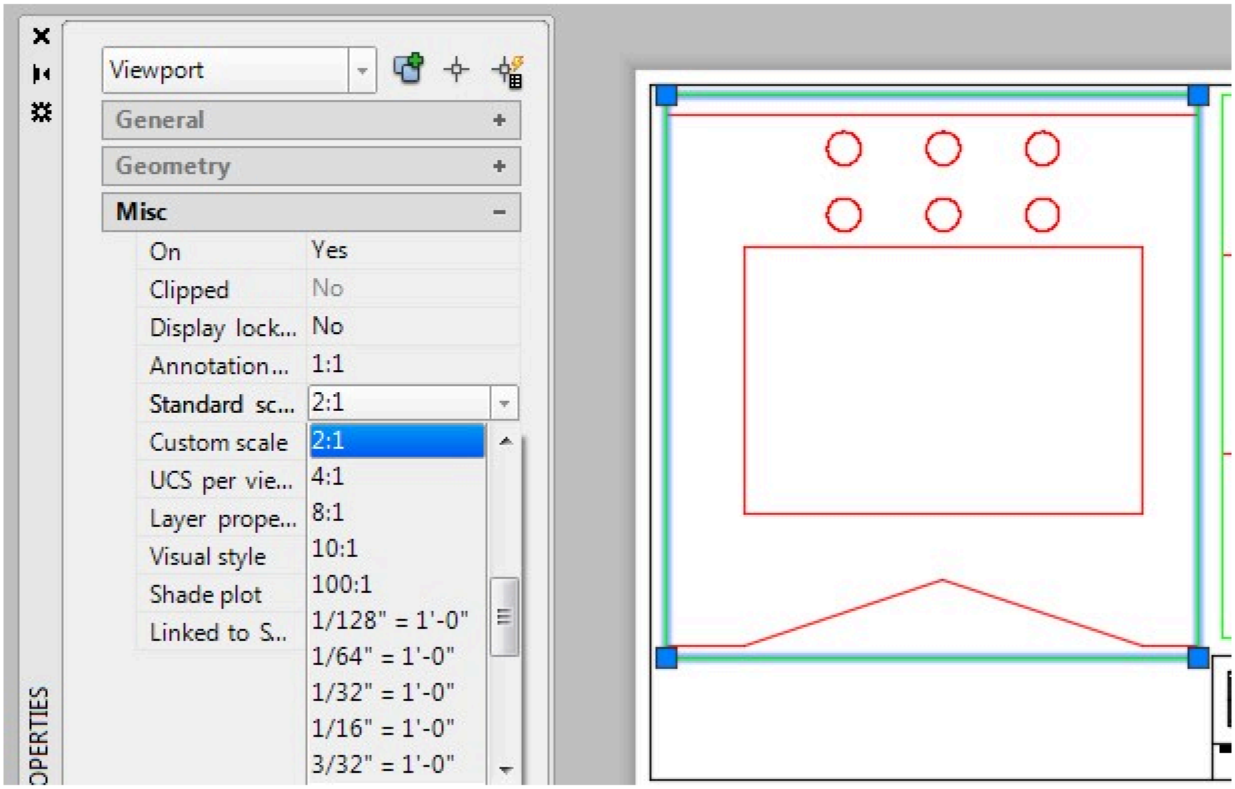


Figure Step 16

Step 17

Using what was taught earlier in the workalong, select the viewport on the left and set the scale to 2:1. (Figure Step17)



Viewport

General

Geometry

Misc

On: Yes
Clipped: No
Display lock...: No
Annotation...: 1:1
Standard sc...: 2:1
Custom scale
UCS per vie...
Layer prope...
Visual style
Shade plot
Linked to S...

2:1
4:1
8:1
10:1
100:1
1/128" = 1'-0"
1/64" = 1'-0"
1/32" = 1'-0"
1/16" = 1'-0"
3/32" = 1'-0"
OPERTIES

AUTHOR'S COMMENTS: Paper space must be the working space to select the viewport.

Step 18

Change the working space to Model and click inside viewport on the left. (Figure Step 18)

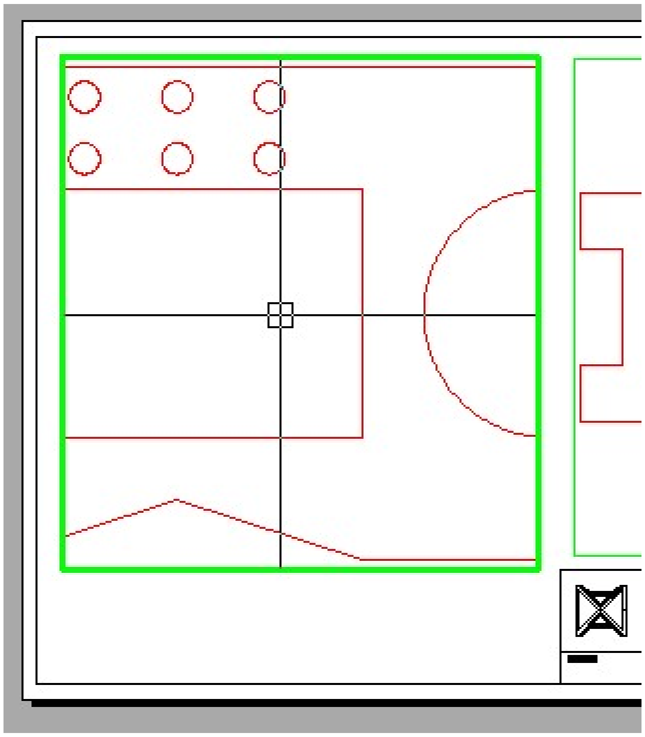


Figure Step 18

Step 19

Without changing the zoom, pan of the model as shown in the figure. Change to Paper space. (Figure Step 19)

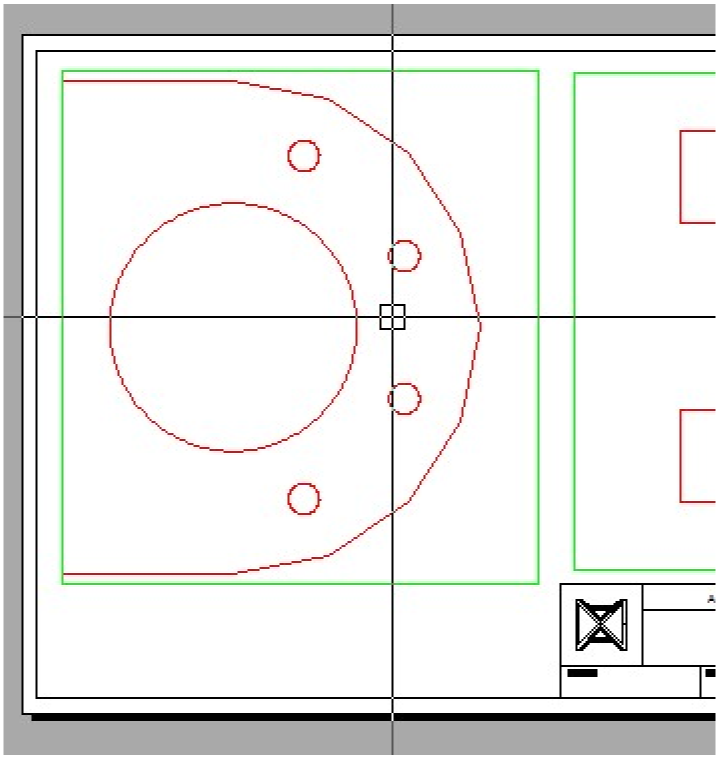
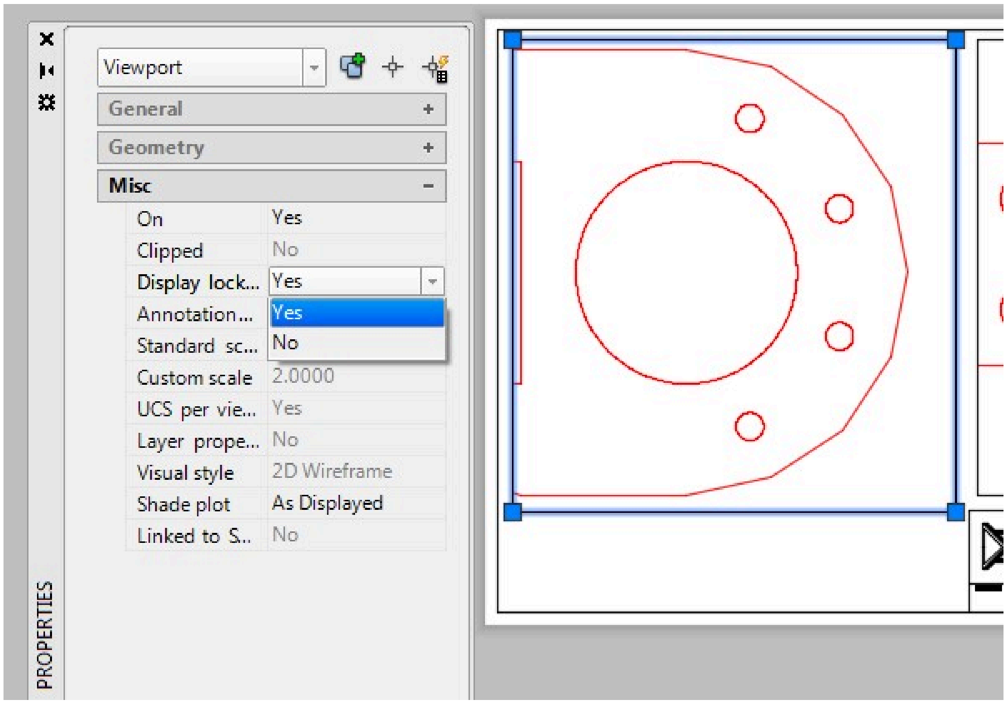


Figure Step 19

Step 20

Using what was taught earlier in the workalong, use the Properties window and check to ensure that the scale is still set to 2:1. Lock the display. (Figure Step 20)



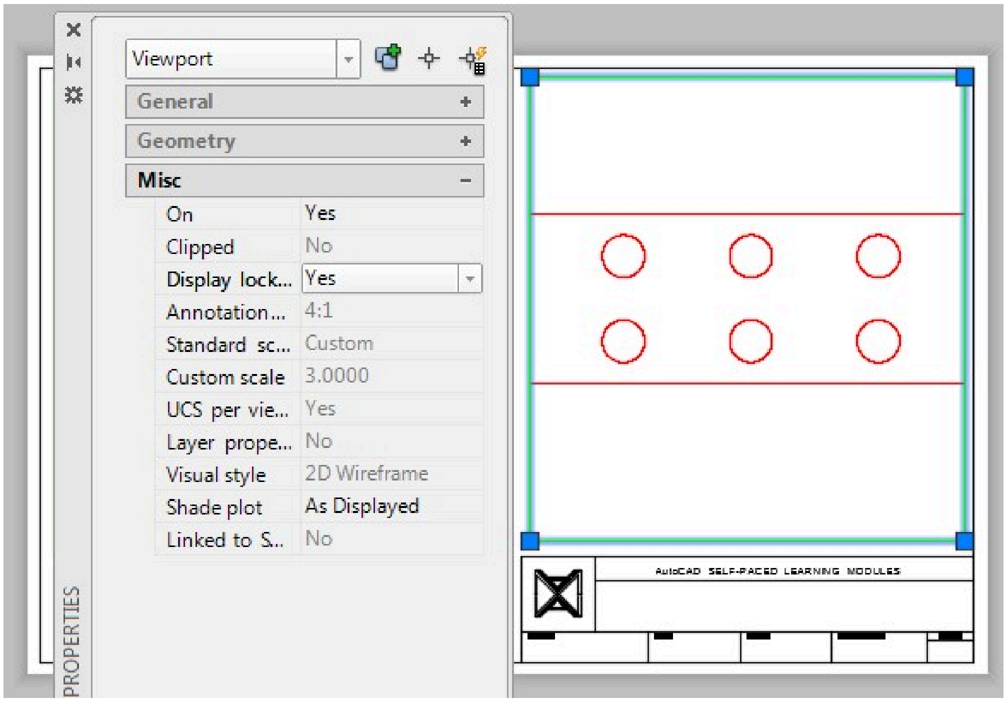
Property	Value
On	Yes
Clipped	No
Display lock...	Yes
Annotation...	Yes No
Standard sc...	
Custom scale	2.0000
UCS per vie...	Yes
Layer prope...	No
Visual style	2D Wireframe
Shade plot	As Displayed
Linked to S...	No

Figure Step 20

AUTHOR'S COMMENTS: The model in this layout is set to a scale of 2:1 or two times the size of the original model.

Step 21

Using what was taught earlier in the workalong, zoom and pan the model in the right viewport. Set the Custom scale to 3 and lock the display. (Figure Step 21A and 21B)



Viewport

General

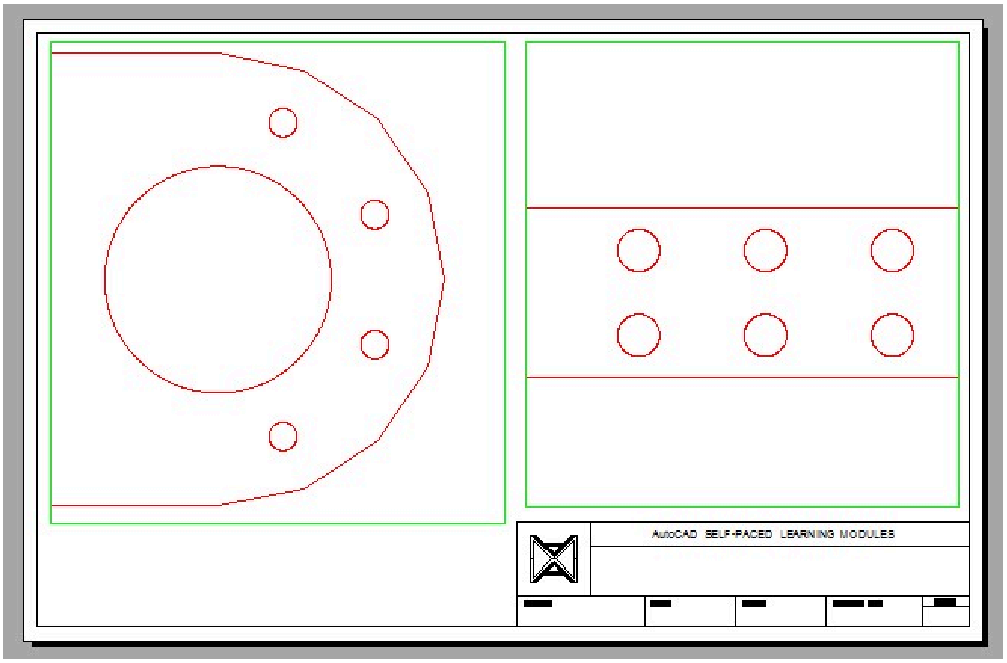
Geometry

Misc

On	Yes
Clipped	No
Display lock...	Yes
Annotation...	4:1
Standard sc...	Custom
Custom scale	3.0000
UCS per vie...	Yes
Layer prope...	No
Visual style	2D Wireframe
Shade plot	As Displayed
Linked to S...	No

PROPERTIES
AutoCAD SELF-PACED LEARNING MODULES

Figure Step 21A



AutoCAD SELF-PACED LEARNING MODULES

Figure Step 21B

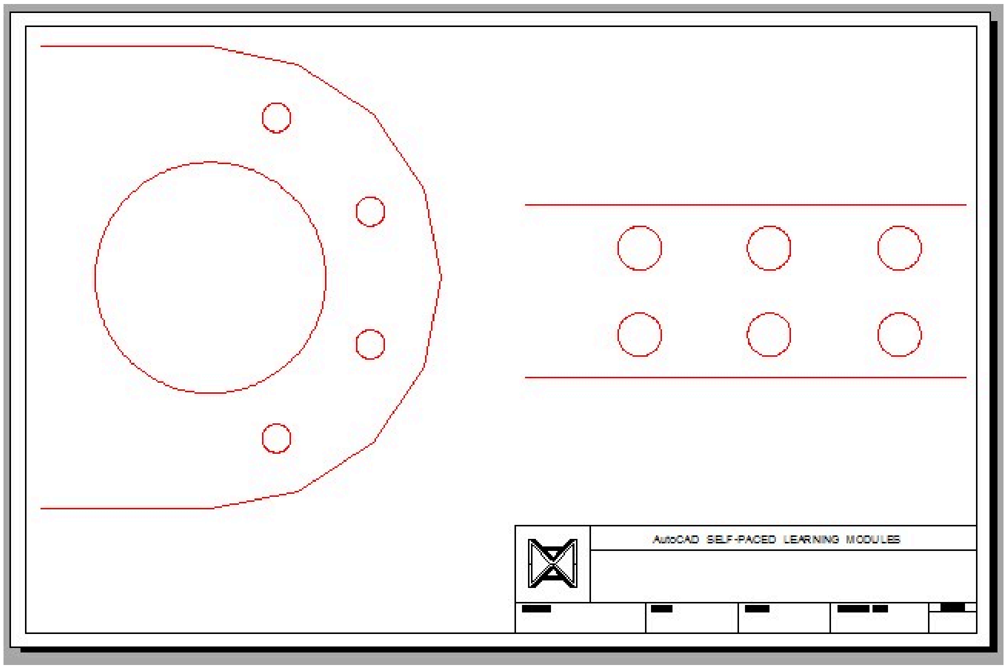

AutoCAD SELF-PACED LEARNING MODULES

Figure Step 21C

AUTHOR’S COMMENTS: The model in this layout is set to a scale of 3:1 or three times the size of the original model.

Step 22

Turn layer: Viewport off. Your layout should appears as shown in the figure. (Figure Step 22)

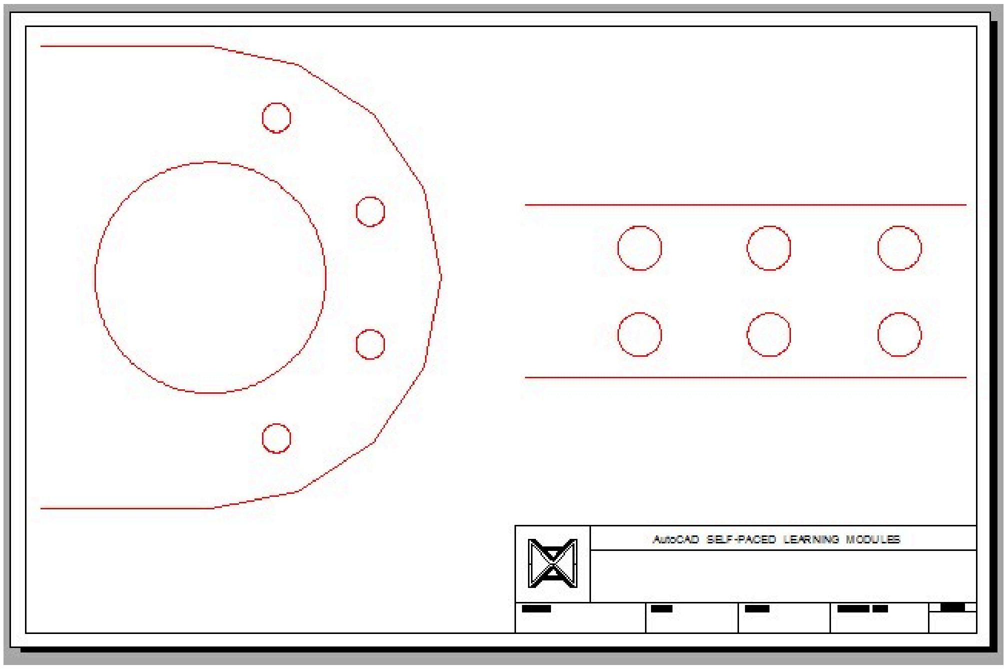
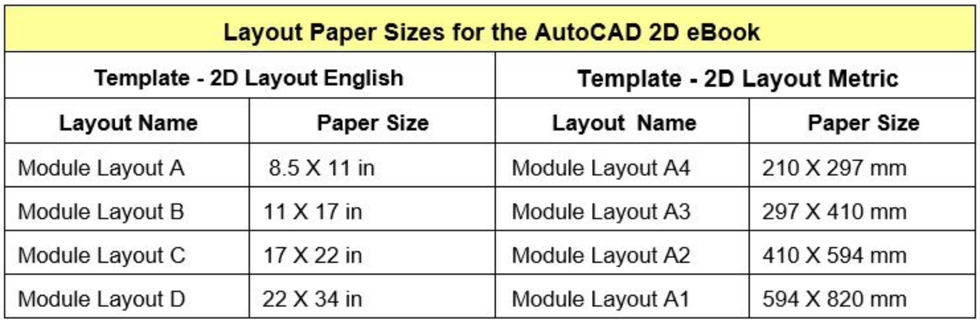


Figure Step 22

Step 23

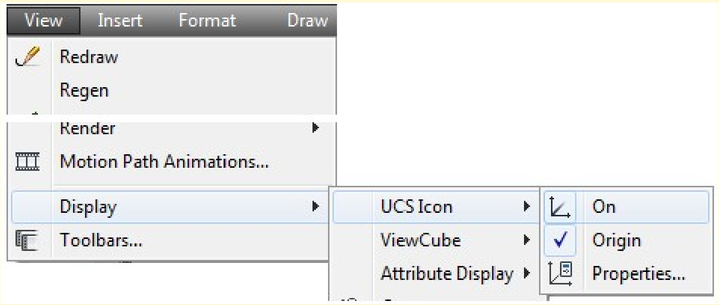
Save and close the drawing.



Layout Paper Sizes for the AutoCAD 2D eBook			
Template - 2D Layout English		**Template - 2D Layout Metric**	
Layout Name	**Paper Size**	**Layout Name**	**Paper Size**
Module Layout A	8.5 X 11 in	Module Layout A4	210 X 297 mm
Module Layout B	11 X 17 in	Module Layout A3	297 X 410 mm
Module Layout C	17 X 22 in	Module Layout A2	410 X 594 mm
Module Layout D	22 X 34 in	Module Layout A1	594 X 820 mm

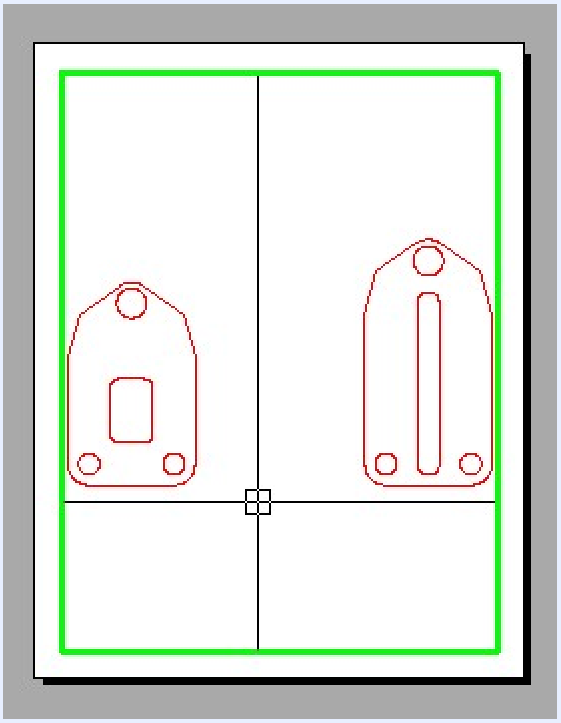
Figure 18-7
Layout Paper Sizes

USER TIP: Type your textbox content here.

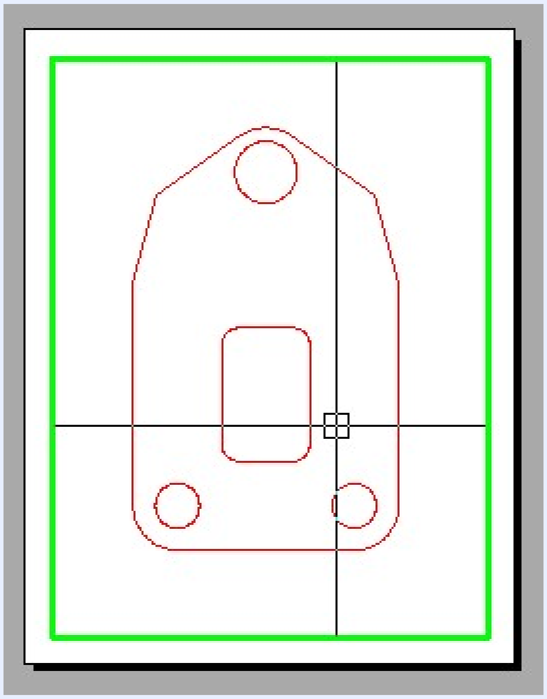

View
Insert
Format
Draw
Redraw
Regen
Render
Motion Path Animations...
Display
Toolbars...
UCS Icon
ViewCube
Attribute Display
On
Origin
Properties...

MUST KNOW: There are two completely different spaces to work in when you are drawing in AutoCAD. They are called Model space and Paper space. The real objects that you are drawing should be drawn in Model space and all other objects that are part of the description of the object, are drawn in Paper space.Objects created in Model space can only be modified when the working space is Model and should alway be drawn at full size using the appropriate model drawing units. Objects drawn in Paper space can only be modified while the working space is Paper. The paper units may differ from the model units.

MUST KNOW: After you use the MVIEW command to create a viewport in the layout, you should always execute the ZOOM Extents command immediately after. Before you enter the ZOOM Extents command, change the working space to Model. After executing the command, you will be able to see all the visible model objects in the viewport. You can then zoom and pan the model to set the desired view.



Zoomed to the Extents



View zoomed and Panned

Key Principles

Key Principles in Module 18

1. Model space is a limitless three-dimensional space where the real objects of the model are drawn.
2. Paper space is a virtual two-dimensional space where all objects related to the piece of paper, called a layout, are drawn. Objects drawn in Paper space are things like the titleblock, border, dimensions, notes, labels, and bills of material.
3. A viewport is a rectangular or square hole cut through a sheet of drawing paper (layout) so that the model can be viewed at a specified scale and orientation.
4. The current space, which is either Model or Paper, is called the working space. Objects drawn in Model space are model space drawing objects and objects drawn in Paper space are paper space drawing objects.

Lab Exercise 18-1

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 18-1	2D Metric	Millimeters

Layering Scheme

Layer Name	Objects on Layer	Color
Gasket	All objects	Red

Step 1

Draw the top view of the gasket shown in the figure using the layering scheme. (Figure Step 1A, 1B, and 1C)

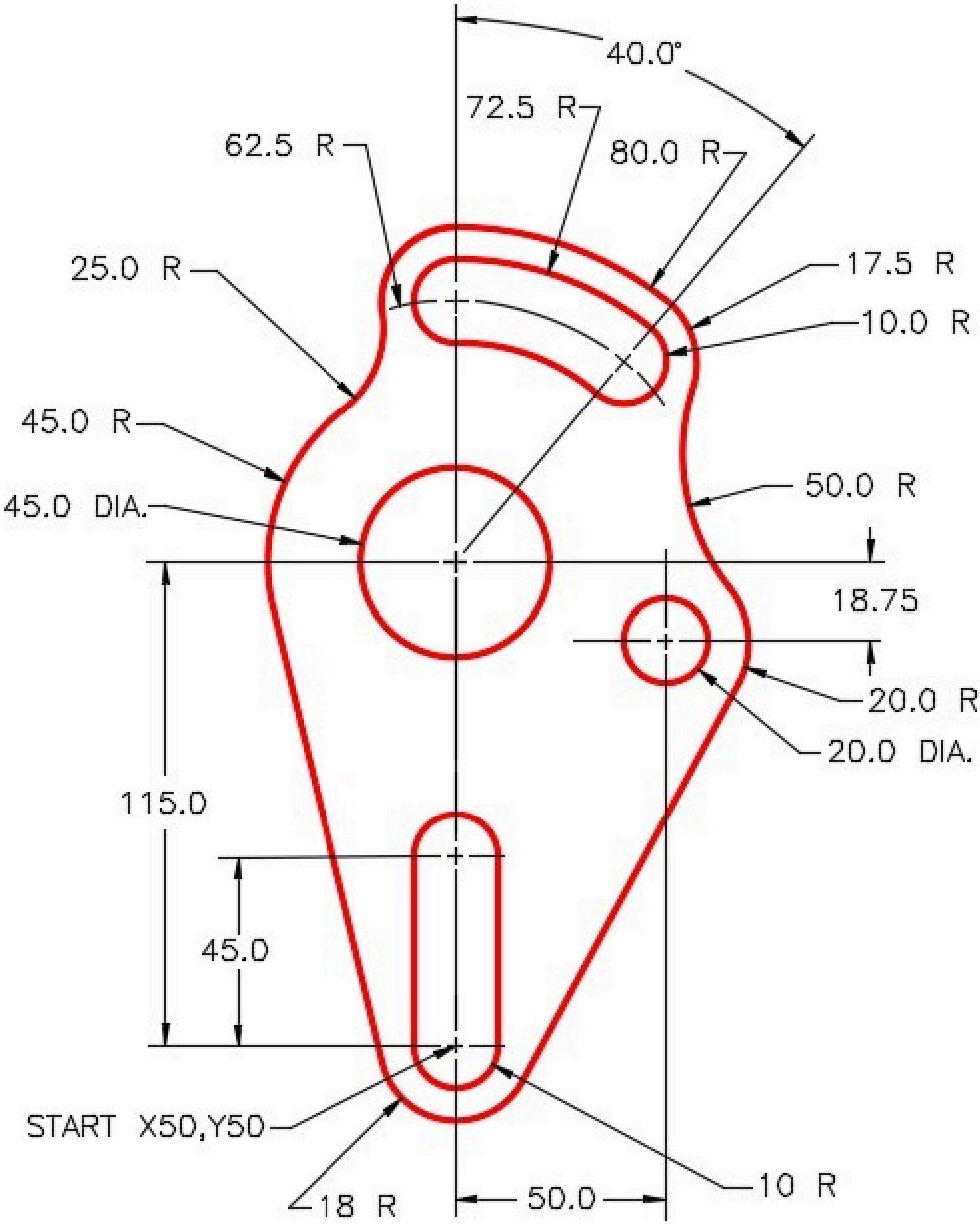

40.0°
72.5 R
62.5 R
80.0 R
25.0 R
17.5 R
10.0 R
45.0 R
45.0 DIA.
50.0 R
18.75
20.0 R
20.0 DIA.
115.0
45.0
START X50,Y50
18 R
50.0
10 R

Figure Step 1A

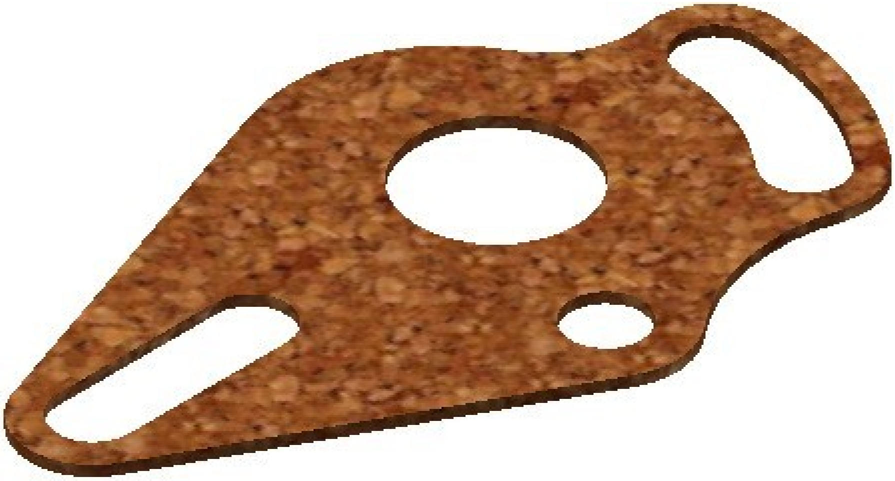


Figure Step 1B
Gasket

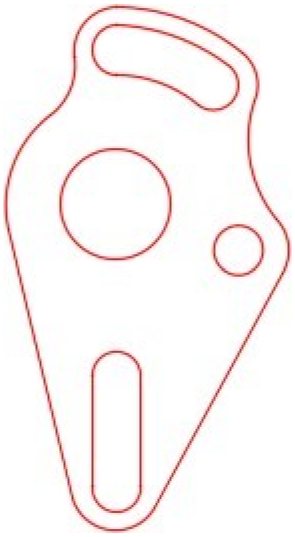


Figure Step 1C
Completed Drawing

Step 2

Set the insertion units and check your drawing with the key. Turn layer: Key off and freeze layer: Construction.

Step 3

In layout: Module Layout A4, on layer: Viewport, create a viewport using the MVIEW command. Pan the model as shown in the figure. Set the scale to 1:1 and lock the display. (Figure Step 3)

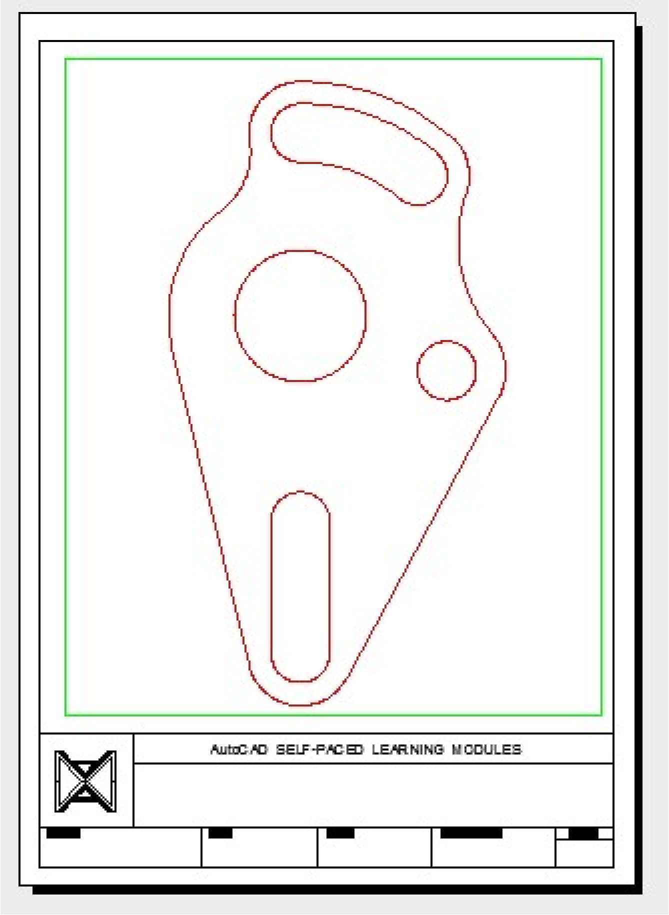

AutoCAD SELF-PACED LEARNING MODULES

Figure Step 3

Step 4

In layout: Module Layout A1, on layer Viewport, create three viewports using the MVIEW command. Pan the model as shown in the figure and set the custom scale of the top left viewport to 1:1, the bottom left viewport to 4:1, and the one on the right to 6:1. Lock the display of all three viewports. (Figure Step 4)

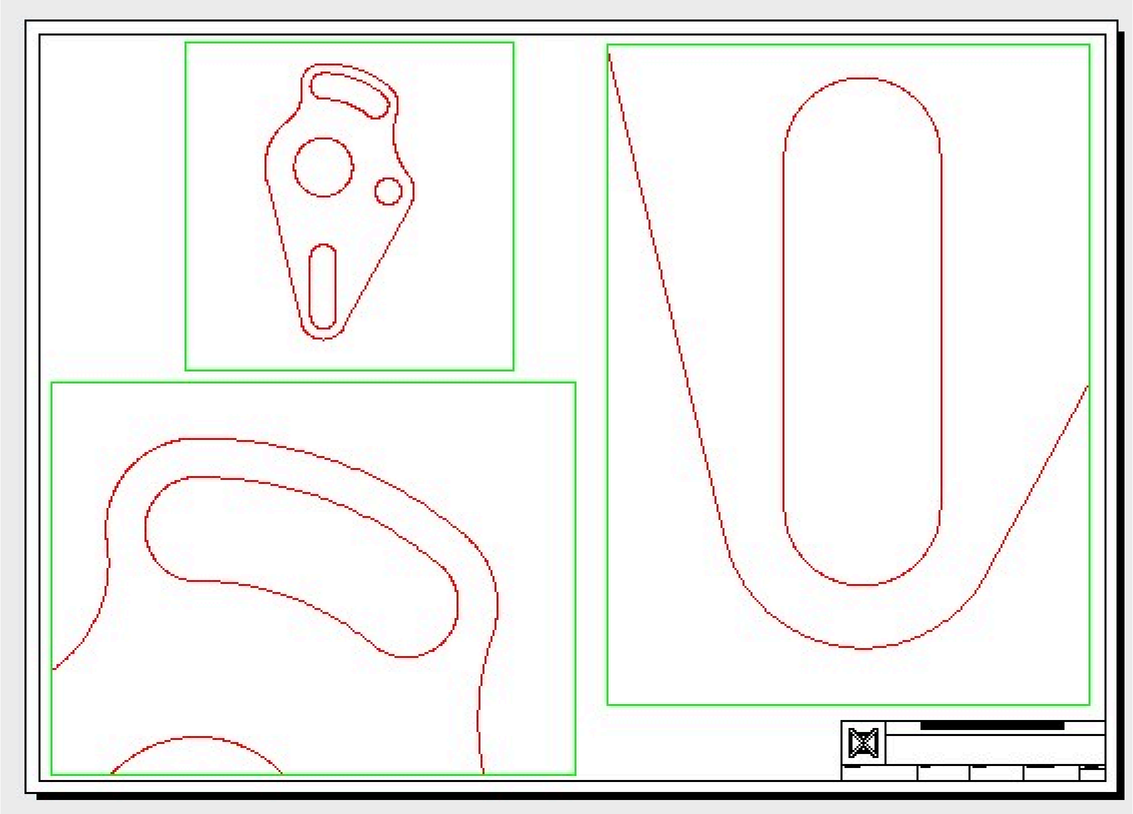


Figure Step 4

Step 5

Turn layer: Viewport off. (Figure Step 5)

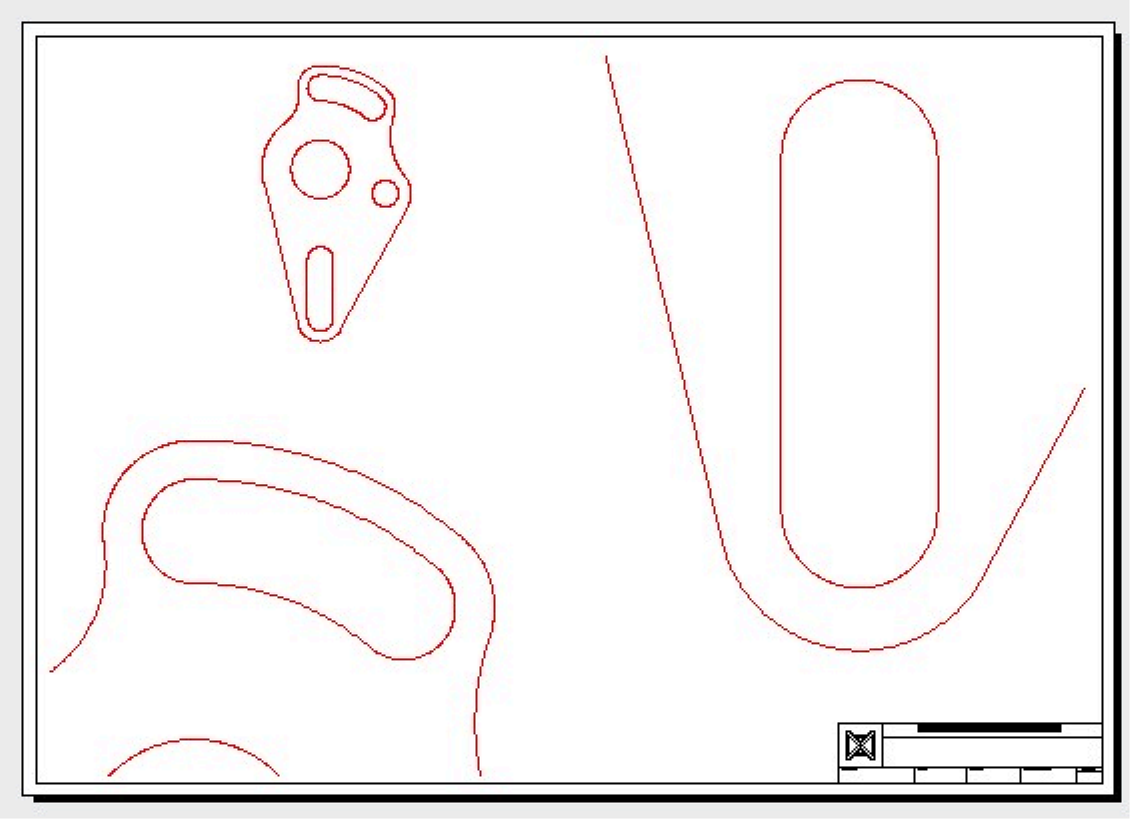
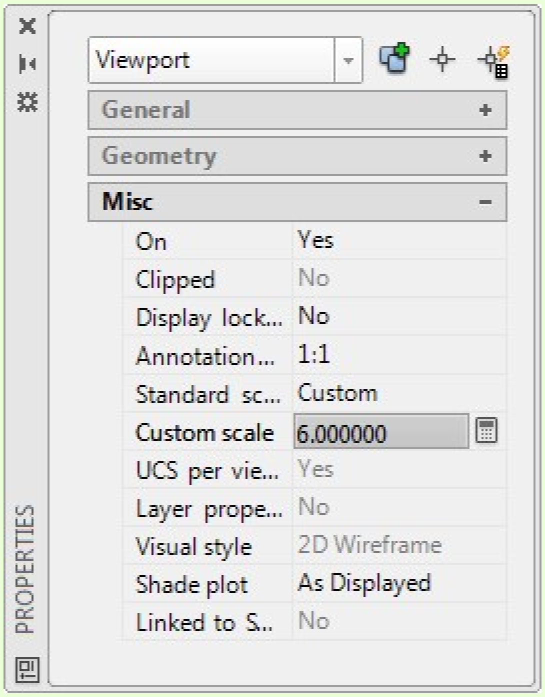


Figure Step 5

AUTHOR’S HINTS: When you are setting the viewport scale and a standard scale is not available to you, simply type in the scale number in the Custom scale property as shown in the figure below. In this example the scale of 6:1 is entered as 6. If the scale you want to enter is 1:4, then you enter 0.25 (1 divided by 4). Setting custom scale is taught in a future module aand creating standard scales is taught in the AutoCAD 2D Advanced book.



Viewport

General

Geometry

Misc

On	Yes
Clipped	No
Display lock...	No
Annotation...	1:1
Standard sc...	Custom
Custom scale	6.000000
UCS per vie...	Yes
Layer prope...	No
Visual style	2D Wireframe
Shade plot	As Displayed
Linked to S...	No

Lab Exercise 18-2

Time allowed: 40 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 18-2	2D Metric	Millimeters

Layering Scheme

Layer Name	**Objects on Layer**	**Color**
Wrench	All objects	Blue

Step 1

Draw the top view of the wrench shown in the figure using the layering scheme. (Figure Step 1A, 1B, and 1C)

Figure Step 1A
Open End Wrench

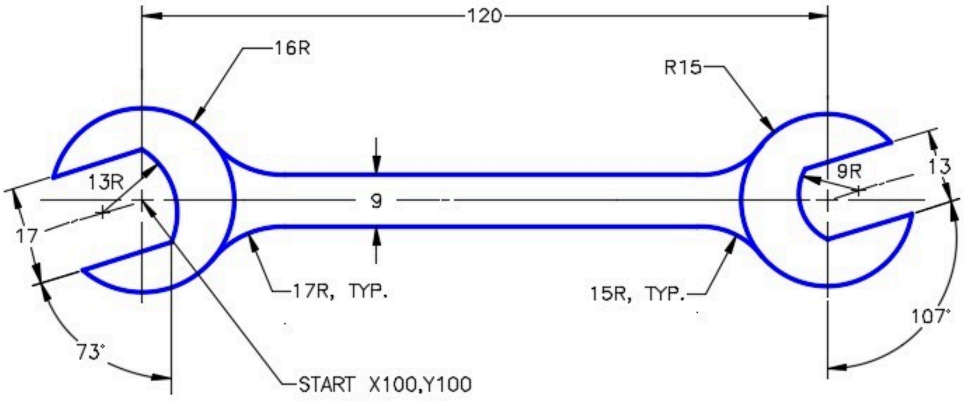

120
16R
R15
13R
9R
13
17
9
17R, TYP.
15R, TYP.
107°
73°
START X100,Y100

Figure Step 1B

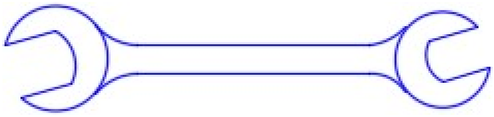


Figure Step 1C
Completed Drawing

Step 2

Set the insertion units and check your drawing with the key. Turn layer: Key off and freeze layer: Construction.

Step 3

In layout: Module Layout A4, on layer: Viewport, create a viewport using the MVIEW command. Pan the model as shown in the figure. Set the scale to 1:1 and lock the display. (Figure Step 3)

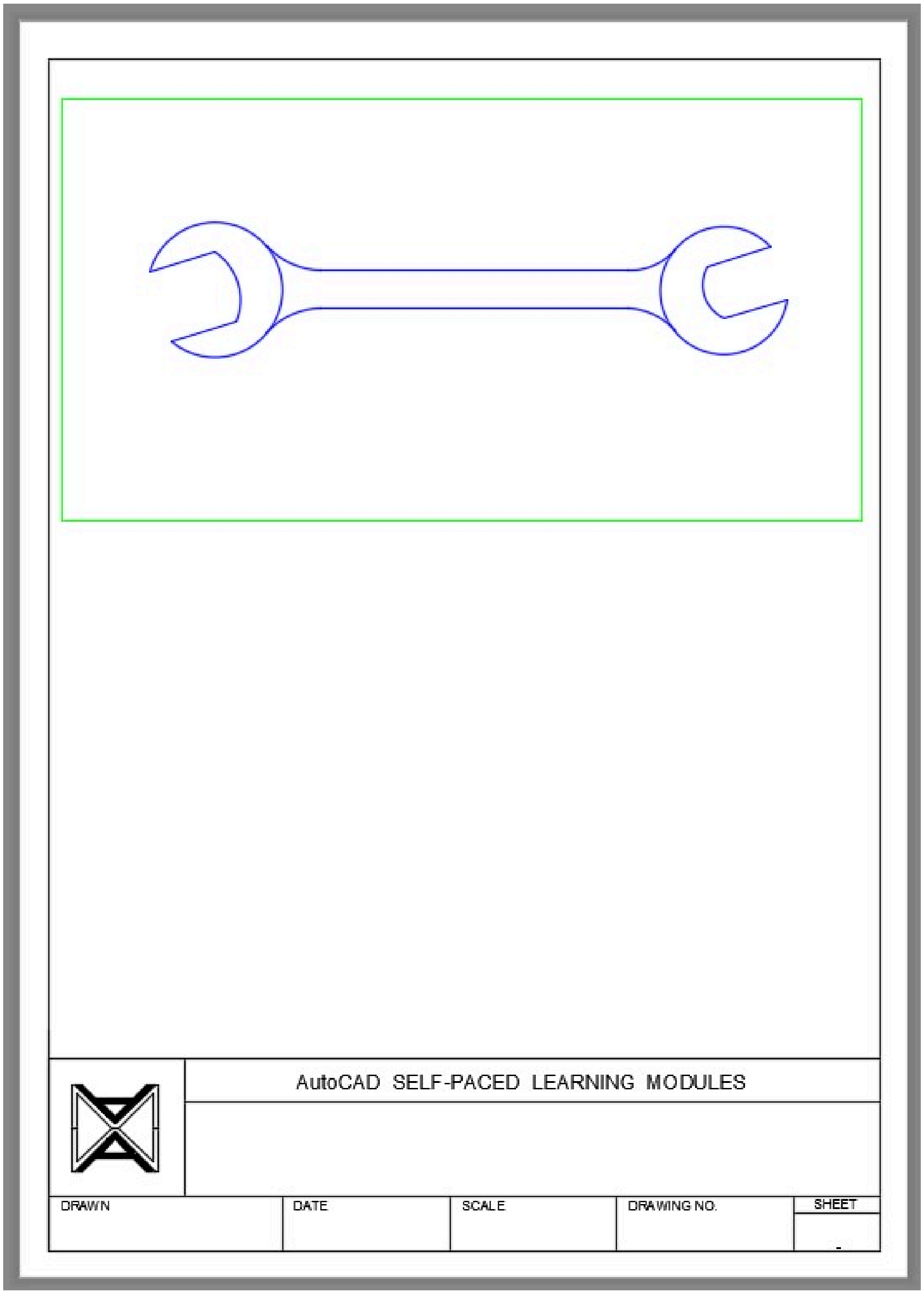

AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO.
SHEET

Figure Step 3

Step 4

In layout: Module Layout A4, on layer: Viewport, create two additional viewports using the MVIEW command. Pan the model as shown in the figure. Set the scale to 2:1 and lock the display. (Figure Step 4)

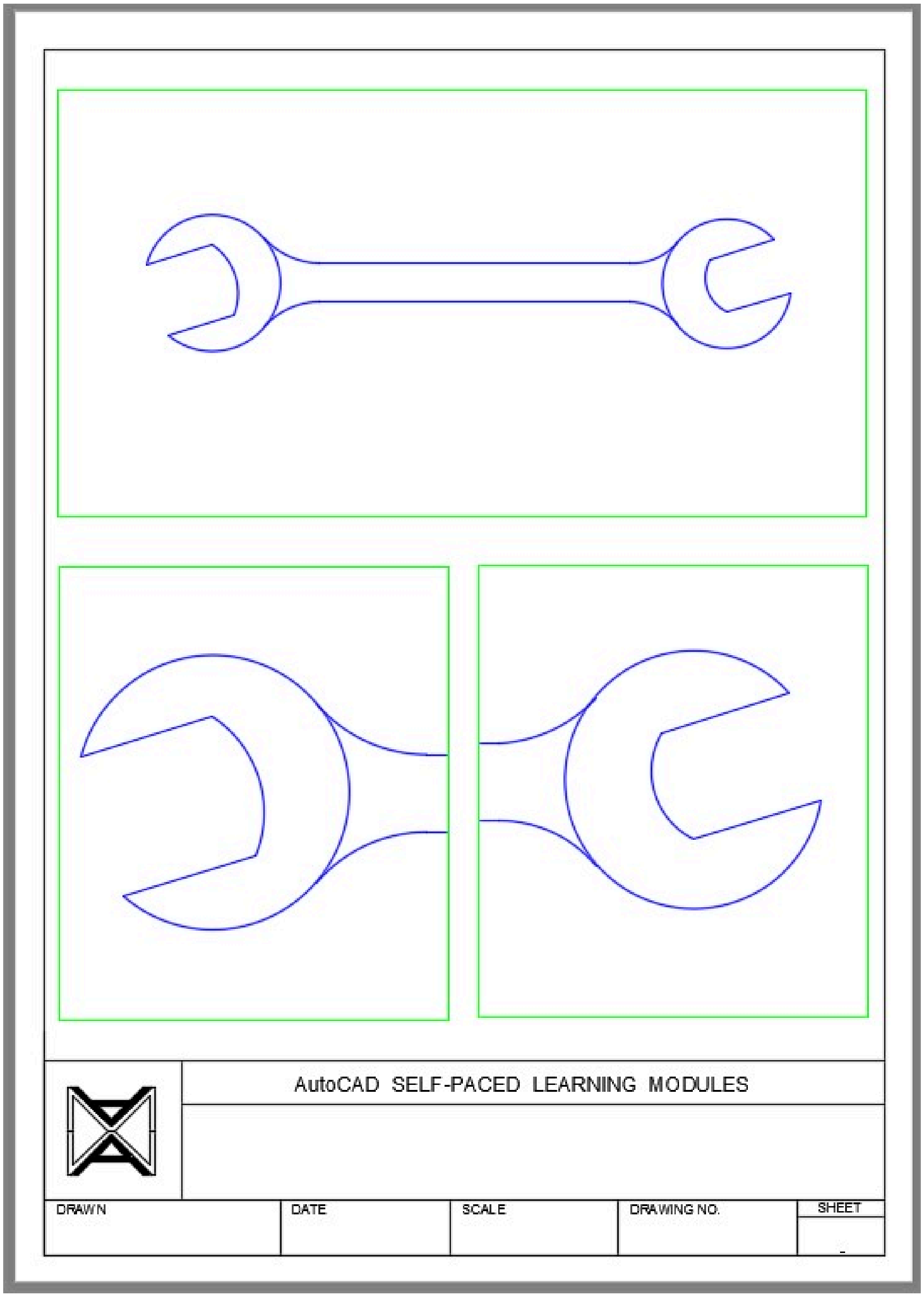

AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO.
SHEET

Figure Step 4

AUTHOR'S HINT: The following steps show how one end of the wrench is constructed: Use the same principles to draw the other end.

Step 1

Draw the construction lines for the circle and center lines. I am showing only one end in the following figures.

Step 2

Offset the inclined center line on each side. On the left side of the wrench the offset distance is 6.5 mm.

Step 3

Extend the top offset line to the vertical center line.

Step 4

Using the end of the line that you just extended as the center, draw a circle with radius of the small arc. On the left side of the wrench it is 13 mm. I drew it in red to help you identify it.

Step 5

Using the intersection of the circle you just drew and the inclined line as the center, draw a circle with the radius of the small arc. On the left side of the wrench it is 13 mm. I drew it in blue to help you identify it.

Step 6

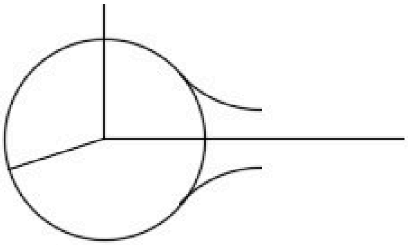
Extend the bottom offset line to the circle.

Step 7

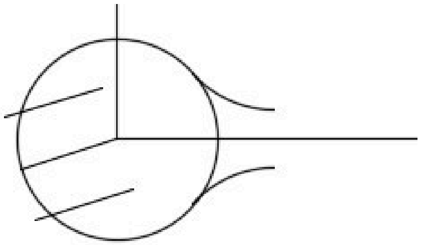
Trim the blue circle. I removed the red circle to help you. You can leave it in your drawing as a construction line.

Step 8

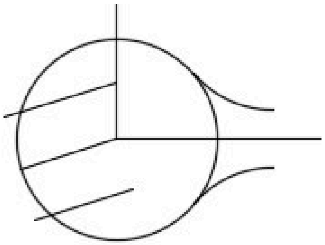
Finish the trim to complete the left side of the wrench.



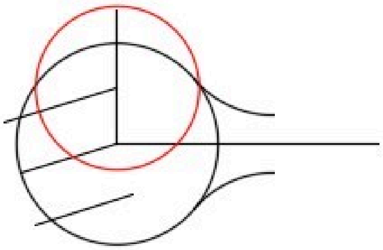
Step 1



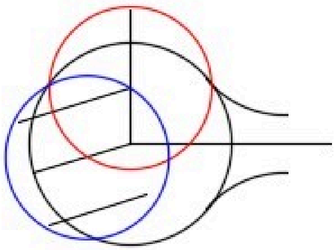
Step 2



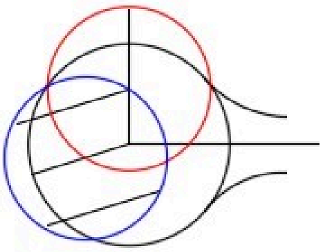
Step 3



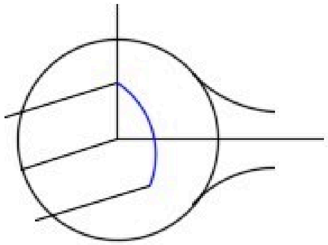
Step 4



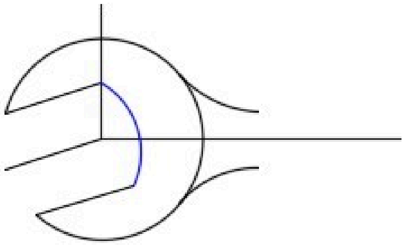
Step 5



Step 6



Step 7



Step 8

Module 19: Text – Part 1

Learning Outcomes

When you have completed this module, you will be able to:

1 Describe single line text, text style, and text justification.

2 Apply the STYLE, TEXT, and DDEDIT commands to format, insert, and edit single line text.

Text

In this module, formatting, inserting, and editing single line text is taught. *Single line text* is text that is inserted as a single line, is edited one line at a time, and each line is a individual drawing object. Module 25 teaches multiline text.

Text Style

The STYLE command is used to create and format text styles plus used to set the current text style. A *text style* contains text properties, is user created, named, and saved in the drawing. As many text styles as required can be created and saved in each drawing. The *current text style* is the text style that will be assigned to text that is inserted in the drawing using the TEXT command.

A good example to explain text style is to use the text font. Font is a text property that is set in a text style. If you want to change the font of an existing text object on the drawing, the style property of the text must be changed. This is done using Properties window. Simply find the text style assigned to the selected text and then change the font property of that style. All existing text, with that style, will change to reflect the new font assigned. If you only want to change the font of the selected text, defined a new text style with the font required and change the style assigned to the selected text.

All AutoCAD drawings contain a text style named Standard. It is special style that cannot be deleted or purged from the drawing. It should not be used as it may cause problems in the drawing sometime in the future. The reason for this will be taught later in the book. For now, create your own text style or use ones that already exist.

Text Justification

Text justification is a property attached to text objects that locates the text in relation to its insert point selected by the you in the TEXT command. There are many ways to justify text and it is important for you to know how, where, and when to use each one. In this module, the *left*, *middle*, and *right* justified text are taught. Figure 19-1 shows how a text object would justify itself if inserted at the selected locations P1, P2, and P3. If you do not specify a justification when inserting text, the default justification is left.

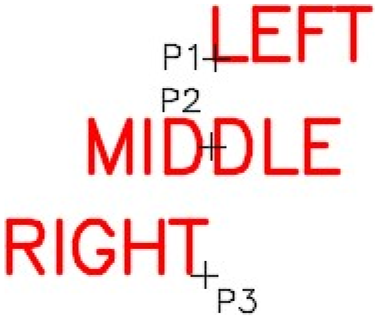

P1
LEFT
P2
MIDDLE
RIGHT
P3

Figure 19-1 Text Justification

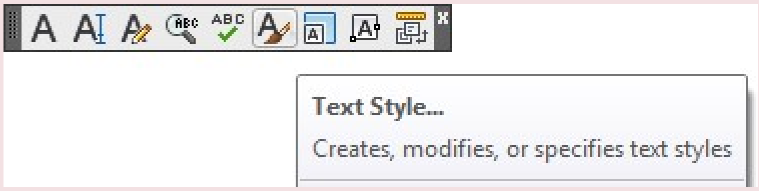
Text Height

Text height is a property assigned to the text drawing object and not to the text style. In almost all cases, the text height property, in the text style, should be set to zero. When it is set to zero, AutoCAD allows you to change the text height of text inserted with that style. It also allows the text style to automatic scale text in a dimension.

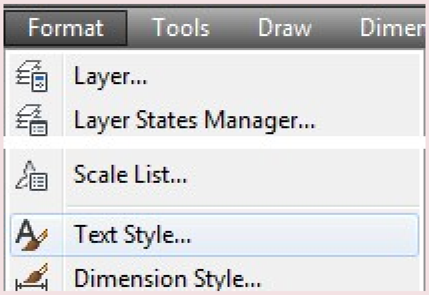
AutoCAD Command: STYLE

The STYLE command is used to create, format, and edit text styles. It also is used to set the current text style.

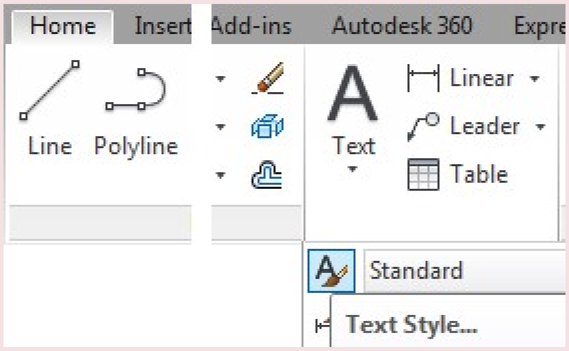
Shortcut: **ST**


Text Style...
Creates, modifies, or specifies text styles

Text Toolbar


Format
Tools
Draw
Dimen
Layer...
Layer States Manager...
Scale List...
Text Style...
Dimension Style...

Format Pull-down

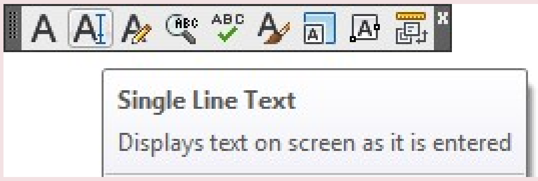

Home
Insert
Add-ins
Autodesk 360
Expre
Line
Polyline
Text
Linear
Leader
Table
Standard
Text Style...

Home Ribbon

AutoCAD Command: TEXT

The TEXT command is used to insert single line text.

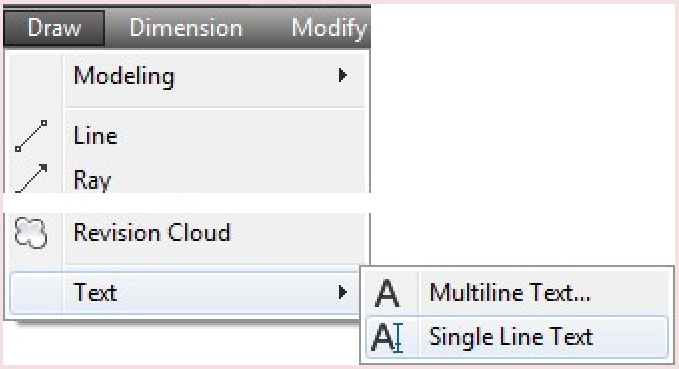
Shortcut: **TE**



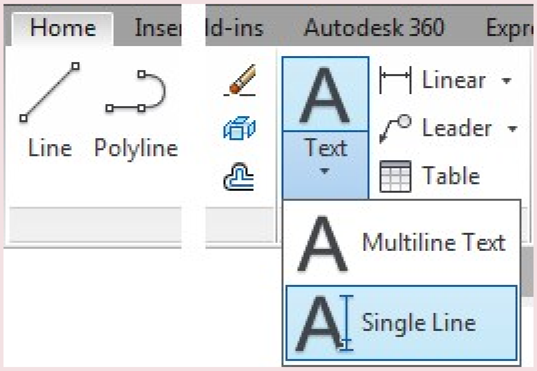
Single Line Text

Displays text on screen as it is entered

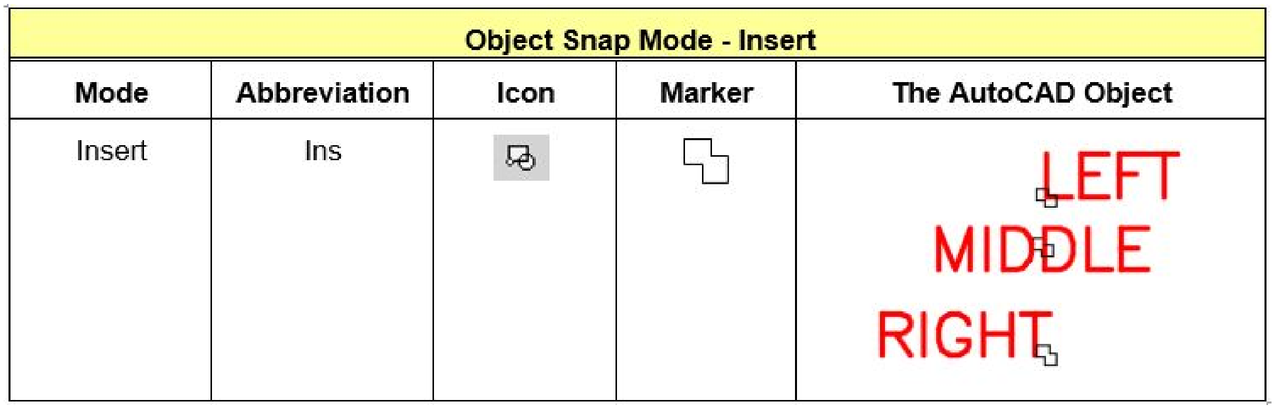
Text Toolbar


Draw
Dimension
Modify
Modeling
Line
Ray
Revision Cloud
Text
Multiline Text...
Single Line Text

Draw Pull-down


Home
Inse
ld-ins
Autodesk 360
Expr
Line
Polyline
Text
Linear
Leader
Table
Multiline Text
Single Line

Home Ribbon



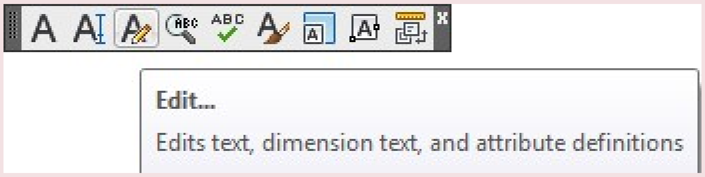
Object Snap Mode - Insert				
Mode	**Abbreviation**	**Icon**	**Marker**	**The AutoCAD Object**
Insert	Ins			LEFT MIDDLE RIGHT

Figure 19-2
Object Snap Mode – Insert

AutoCAD Command: DDEDIT

The DDEDIT command is used to edit existing text objects.

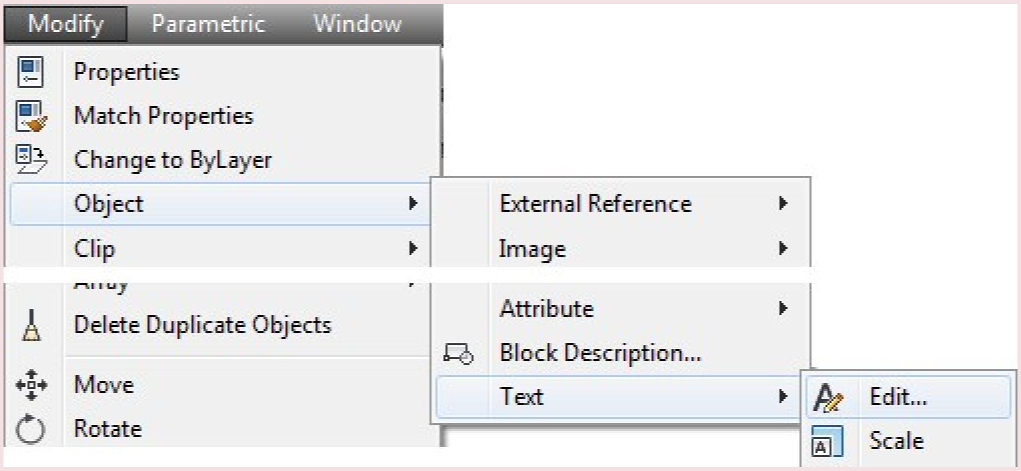
Shortcut: **Double click the text**.



Edit...

Edits text, dimension text, and attribute definitions

Text Toolbar


Modify
Parametric
Window
Properties
Match Properties
Change to ByLayer
Object
Clip
Array
Delete Duplicate Objects
Move
Rotate
External Reference
Image
Attribute
Block Description...
Text
Edit...
Scale

Modify Pull-down

USER TIP: If a text object is double clicked, the Edit Text dialogue box will open automatically with the text selected, ready to be edited. This is a fast way to edit single line text and almost eliminates the use of the DDEDIT command. Text contents can also be edited in the Properties window.

WORKALONG: Creating Text Styles and Inserting Single Line Text

Step 1

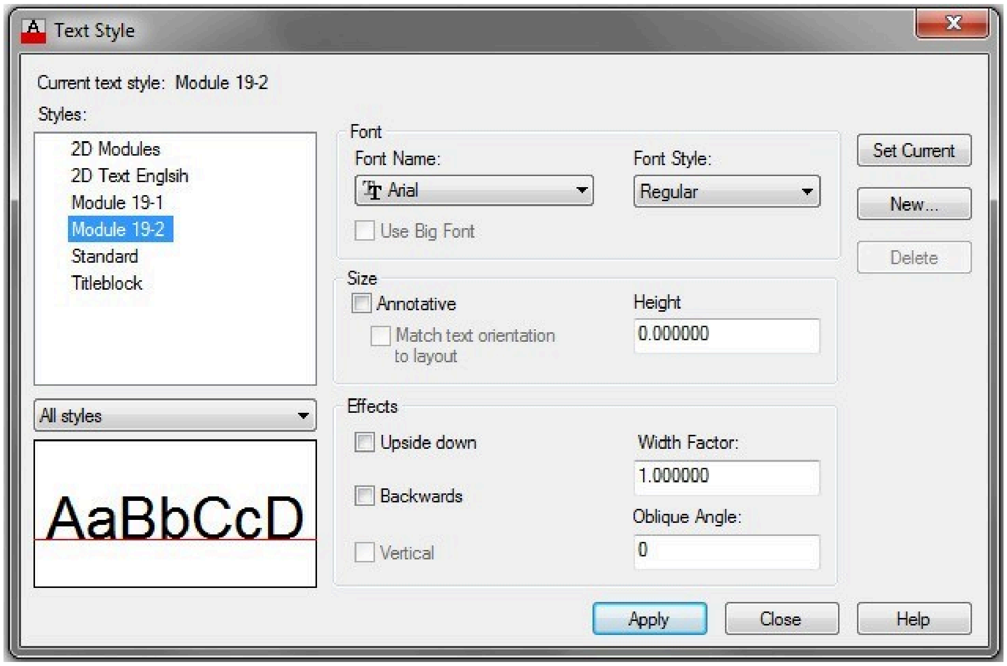
Start a new drawing using the template: 2D Layout English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 19-1.

Step 3

Enter the STYLE command. It will open the Text Style dialogue box. (Figure Step 3)



Text Style

Current text style: Module 19-2

Styles:
- 2D Modules
- 2D Text Englsih
- Module 19-1
- Module 19-2
- Standard
- Titleblock

Font Name: Arial

Font Style: Regular

☐ Use Big Font

☐ Annotative

☐ Match text orientation to layout

Height: 0.000000

☐ Upside down

☐ Backwards

☐ Vertical

Width Factor: 1.000000

Oblique Angle: 0

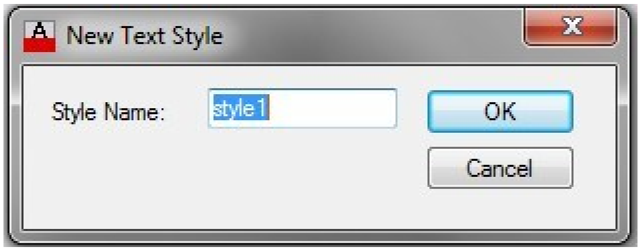
Styles:
All styles
AaBbCcD
Font
Size
Effects
Set Current
New...
Delete
Apply
Close
Help

Figure Step 3

AUTHOR'S COMMENTS: The Standard text style is a special style that all drawings contain. It cannot be deleted or purged from the drawing. I suggest that you do NOT use it. It is better to create your own styles.

Step 4

Click the New button to open the New Text Style dialogue box. (Figure Step 4)



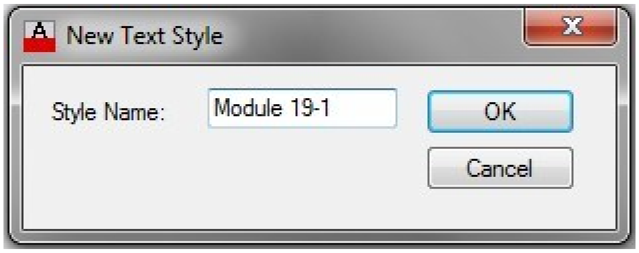
New Text Style

Style Name: style1

Figure Step 4

Step 5

Enter the name: Module 19-1 in the Style Name box as shown in the figure. Click OK to save and close the dialogue box. (Figure Step 5)

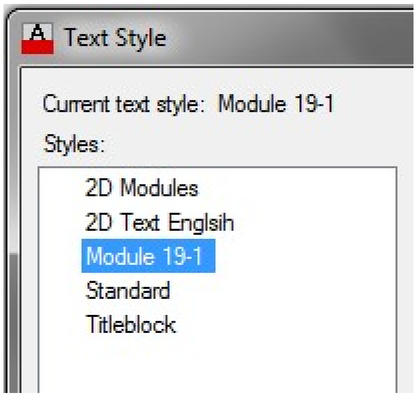

New Text Style
Style Name:
OK
Cancel

Module 19-1

Figure Step 5

Step 6

Check to ensure that the Current text style is set to: Module 19-1. (Figure Step 6)

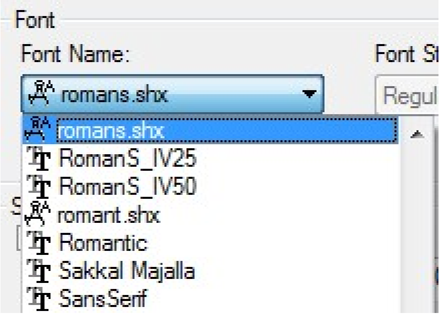

Text Style
Styles:
2D Modules
2D Text Englsih
Module 19-1
Standard
Titleblock

Current text style: Module 19-1

Figure Step 6

Step 7

In the Font Name: box, pull down the list and select the font: romans.shx. (Figure Step 7)


Font
Font St
Regul
romans.shx
RomanS_IV25
RomanS_IV50
romant.shx
Romantic
Sakkal Majalla
SansSerif

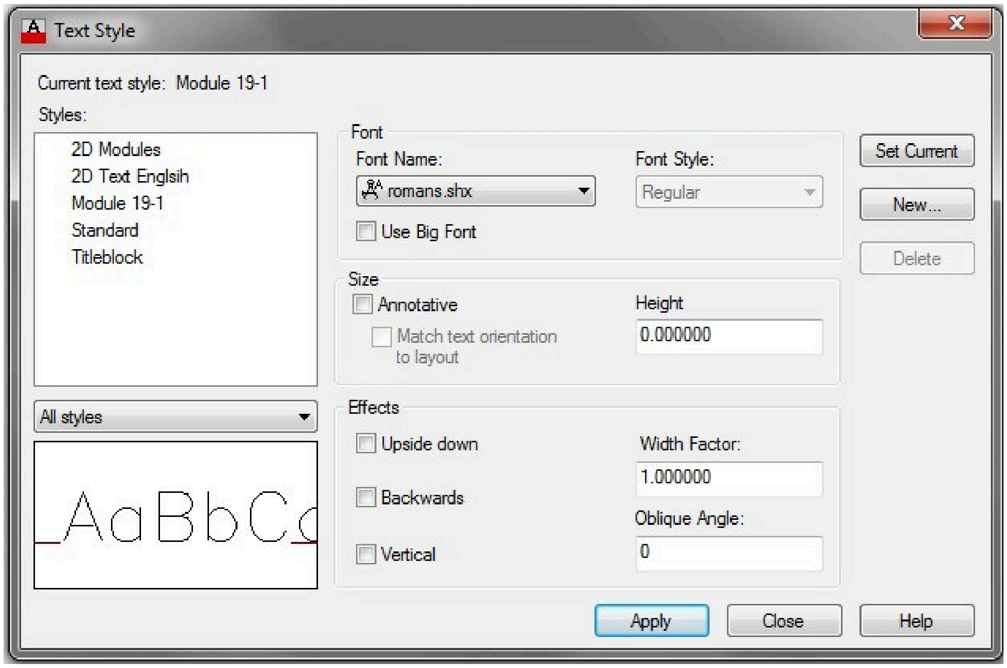
Font Name: romans.shx

Figure Step 7

AUTHOR’S COMMENTS: Once you pull down the font name list, type the first letter of the name of the font you are looking for. In this case, R. That will force the list to jump directly to the font names that begin with the letter R.

Step 8

Do not change any other items. The Text Style dialogue box should now appear as shown in the figure. Click the Close button to save and close the dialogue box. (Figure Step 8)



Current text style: Module 19-1

Styles:
2D Modules
2D Text Englsih
Module 19-1
Standard
Titleblock

All styles

Font
Font Name: romans.shx
Font Style: Regular
☐ Use Big Font

Size
☐ Annotative
☐ Match text orientation to layout
Height: 0.000000

Effects
☐ Upside down
☐ Backwards
☐ Vertical
Width Factor: 1.000000
Oblique Angle: 0

Figure Step 8

AUTHOR'S COMMENTS: In almost all cases, the text height in the Text Style dialogue box should be set to zero. When it is set to zero, the text height for any text assigned that style can be set and changed to any height you desire. If it is set to a value other then zero, the text height of all text assigned that text style property will not be allowed to be changed.

Step 9

Enable layout: Module Layout A. (Figure Step 9)

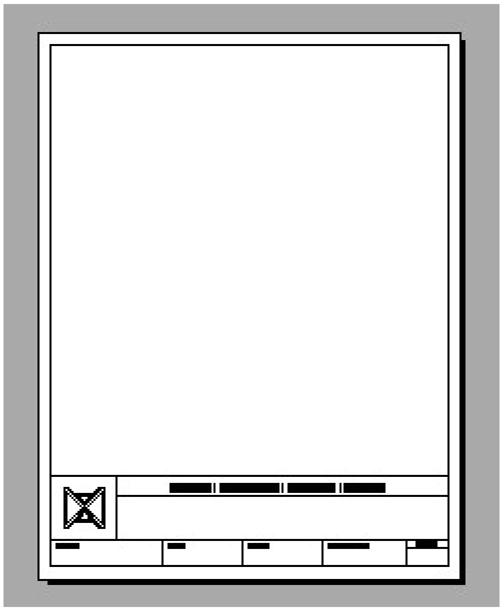


Figure Step 9

Step 10

Set layer: Text as the current layer.

Step 11

Enter the TEXT command, as shown below, to insert text on the drawing. (Figure Step 11)

Command: **TEXT**

Current text style: “Module 19-1” Text height: 0.2000

(The defaults will display. Ensure that Module 19-1 is the current default style.)

Specify start point of text or [Justify/Style]: **1,9**

(This is the insert point. By default, the text will be left justified)

Specify height <0.2000>: **0.35**

Specify rotation angle of text <0>:

(Press Enter to leave the rotation angle at 0 degrees.)

THIS IS MODULE 19

(Type the text and press Enter)

(Press Enter again)

Command:

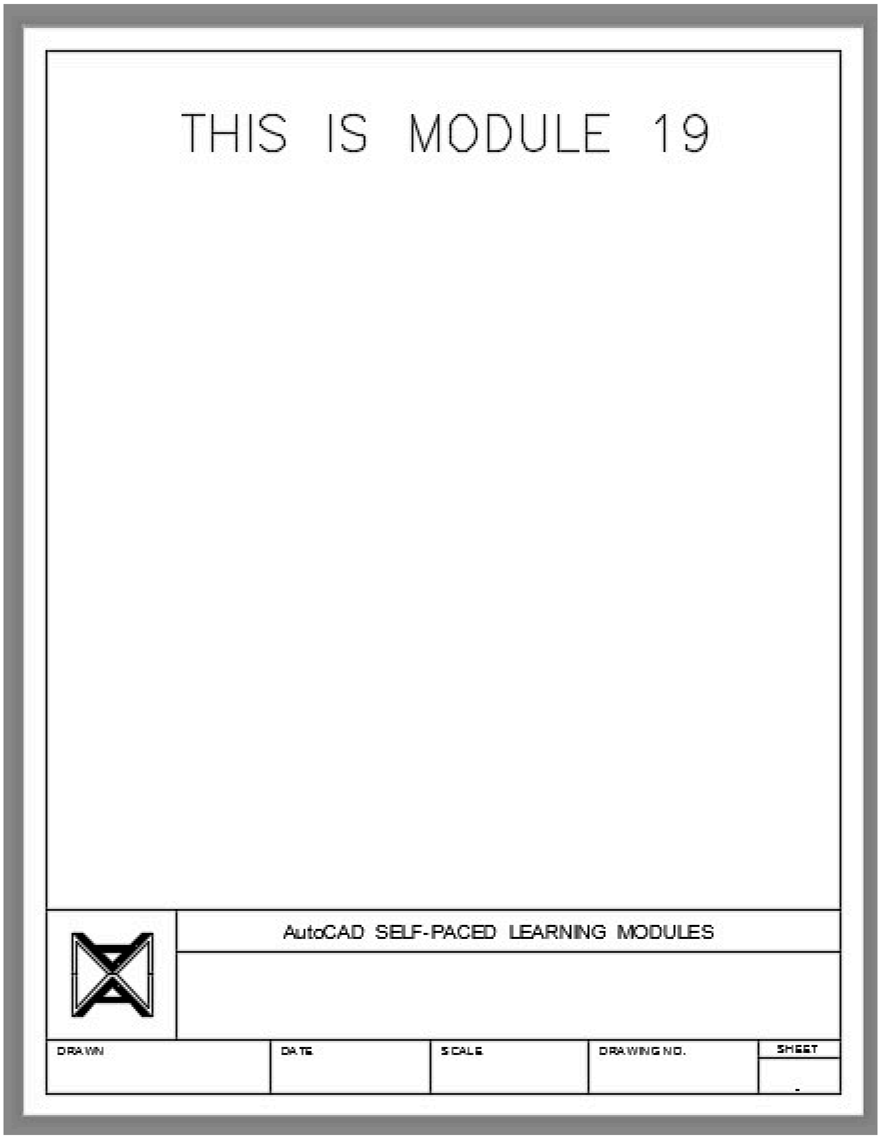

THIS IS MODULE 19
AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO.
SHEET

Figure Step 11

Step 12

Enter the TEXT command, as shown below, to insert text on the drawing. (Figure Step 12)

Command: **TE**

Current text style: “Module 19-1” Text height: 0.3500 Specify start point of text or [Justify/Style]: **0.5,3.5** Specify height <0.3500>: **0.25**

Specify rotation angle of text <0>: **45**

THIS IS STYLE MODULES 19-1

(Press Enter)

INSERTED AT 0.25 TEXT HEIGHT

(Press Enter)

IT IS ROTATED AT 45 DEGREES

(Press Enter)

Command:

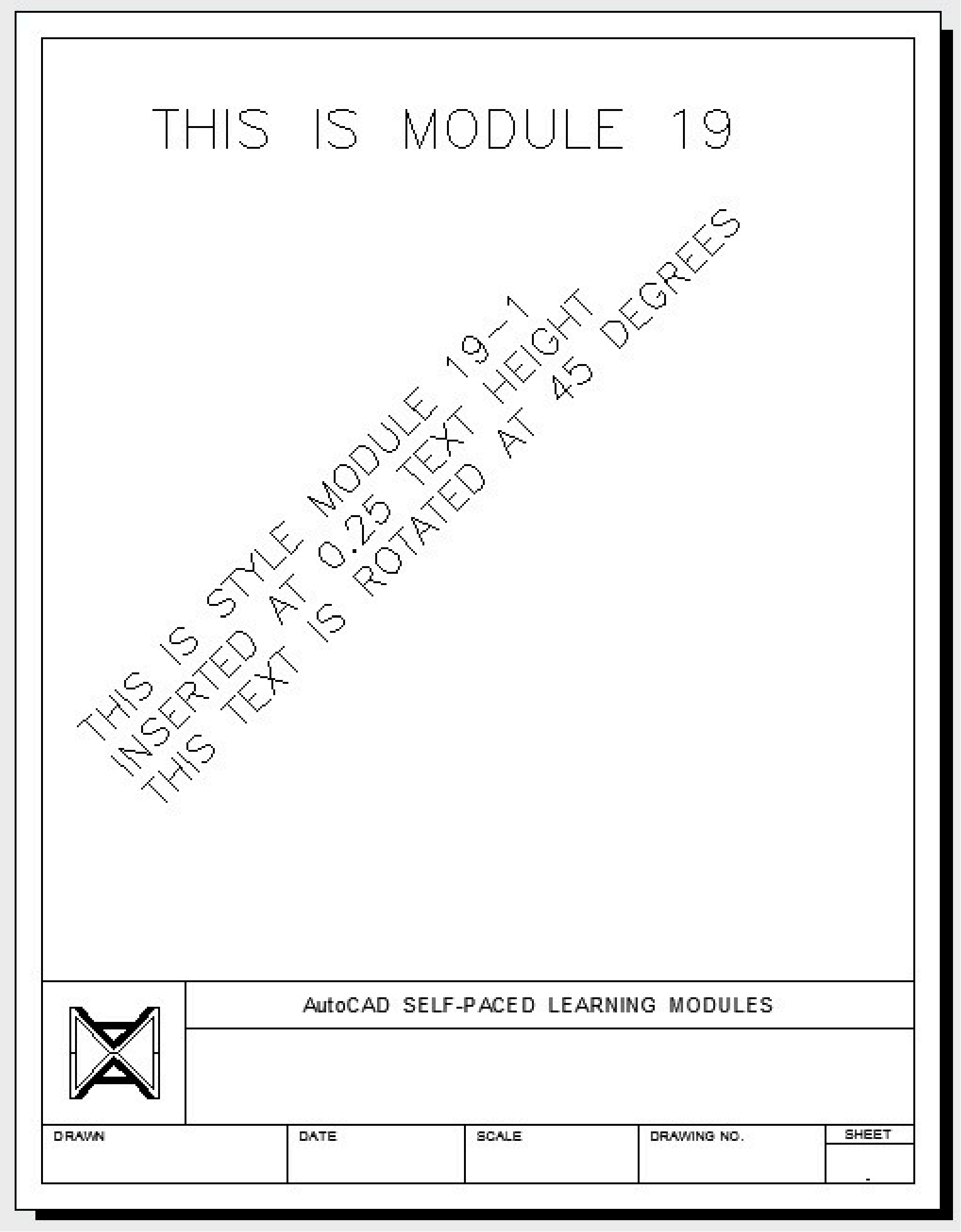

THIS IS MODULE 19
THIS IS STYLE MODULE 19–1
INSERTED AT 0.25 TEXT HEIGHT
THIS TEXT IS ROTATED AT 45 DEGREES
AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO.
SHEET

Figure Step 12

Step 13

Enter the TEXT command, as shown below, to insert text on the drawing. (Figure Step 13)

Command: **TE**

Current text style: “Module 19-1” Text height: 0.0250

Specify start point of text or [Justify/Style]: **J**

Enter an option [Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]: **R**

(Setting a right justification.)

Specify right endpoint of text baseline: **5,2**

Specify height <0.2500>: **0.175**

Specify rotation angle of text <45>: **0**

THIS TEXT IS RIGHT JUSTIFIED. *(Press Enter)*

Command:

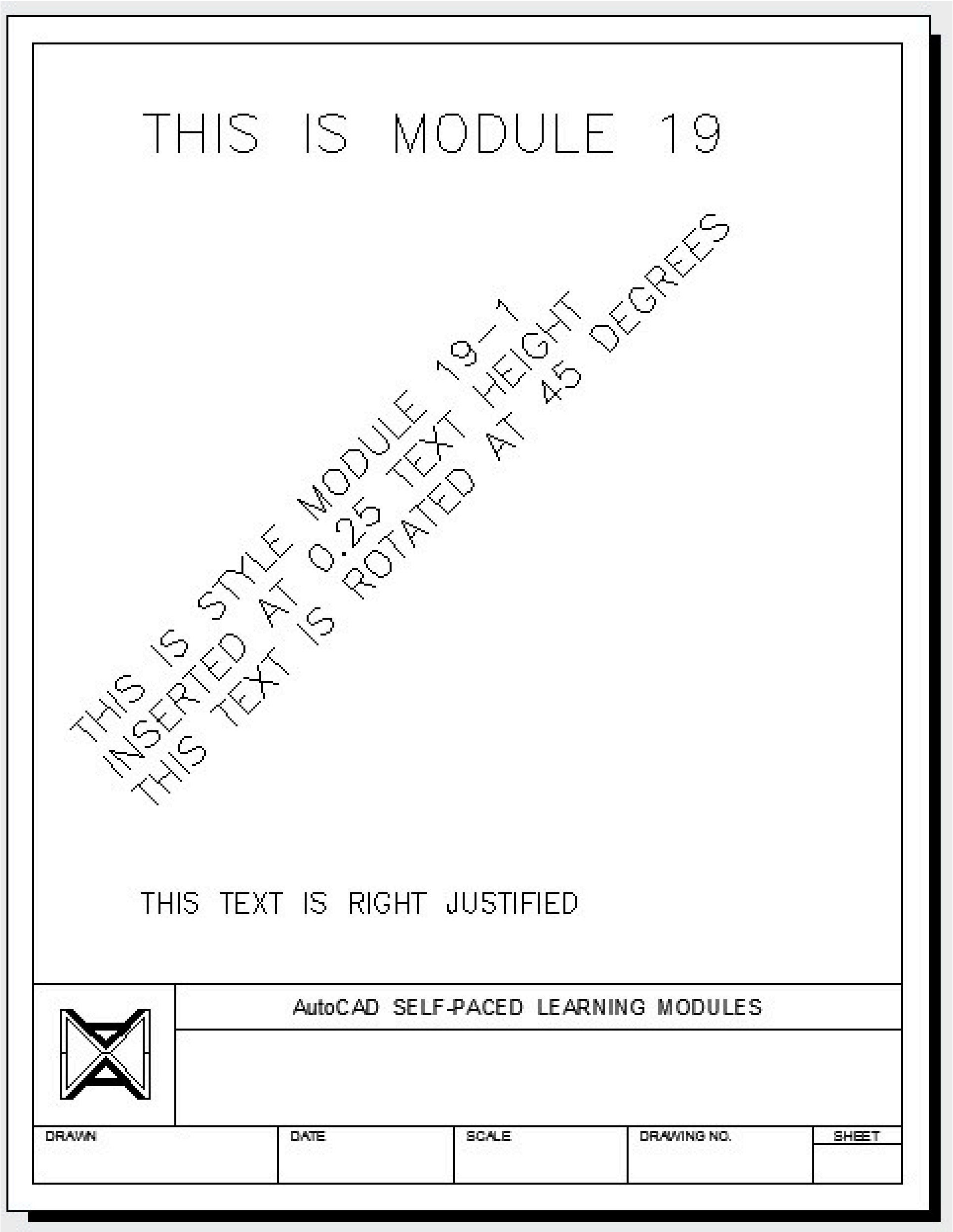

THIS IS MODULE 19
THIS IS STYLE MODULE 19—1
INSERTED AT 0.25 TEXT HEIGHT
THIS TEXT IS ROTATED AT 45 DEGREES
THIS TEXT IS RIGHT JUSTIFIED
AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO.
SHEET

Figure Step 13

Step 14

Save and close the drawing.

WORKALONG: Editing Single Line Text

Step 1

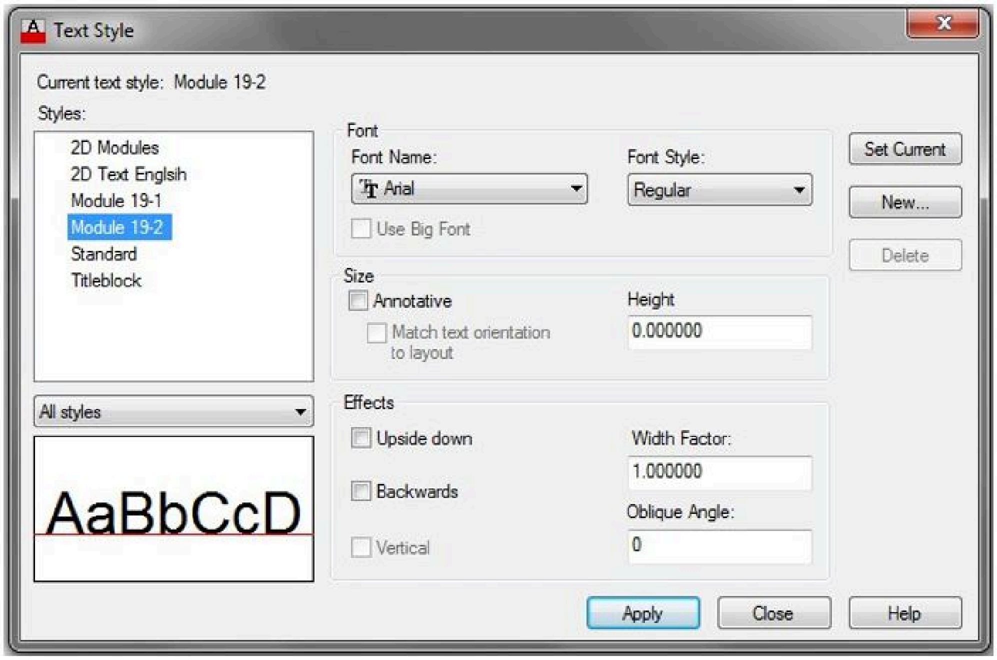
Open the drawing: AutoCAD 2D Workalong 19-1.

Step 2

Using the SAVEAS command, save it with the name: AutoCAD 2D Workalong 19-2.

Step 3

Using what you learned in the first workalong, enter the STYLE command and create a new text style named: Module 19-2. Set the text style to match the figure. (Figure Step 3)


Current text style: Module 19-2

Font Name: Arial

Font Style: Regular

Height: 0.000000

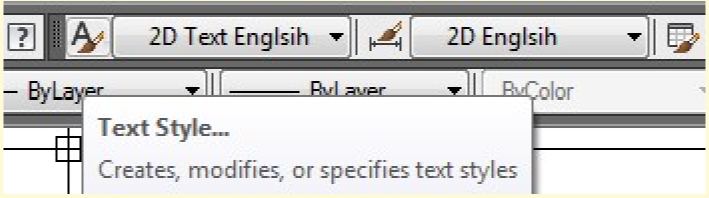
Width Factor: 1.000000

Oblique Angle: 0

Text Style
Styles:
2D Modules
2D Text Englsih
Module 19-1
Module 19-2
Standard
Titleblock
All styles
AaBbCcD
Font
Use Big Font
Size
Annotative
Match text orientation to layout
Effects
Upside down
Backwards
Vertical
Set Current
New...
Delete
Apply
Close
Help

Figure Step 3

USER TIP: The Style toolbar at the top of you Graphic window is very handy to execute the STYLE command or set the current text style.


Text Style...

Creates, modifies, or specifies text styles

2D Text Englsih
2D Englsih
ByLayer
ByLayer
ByColor

Step 4

Enable the display of layout: Module Layout A. (Figure Step 4)

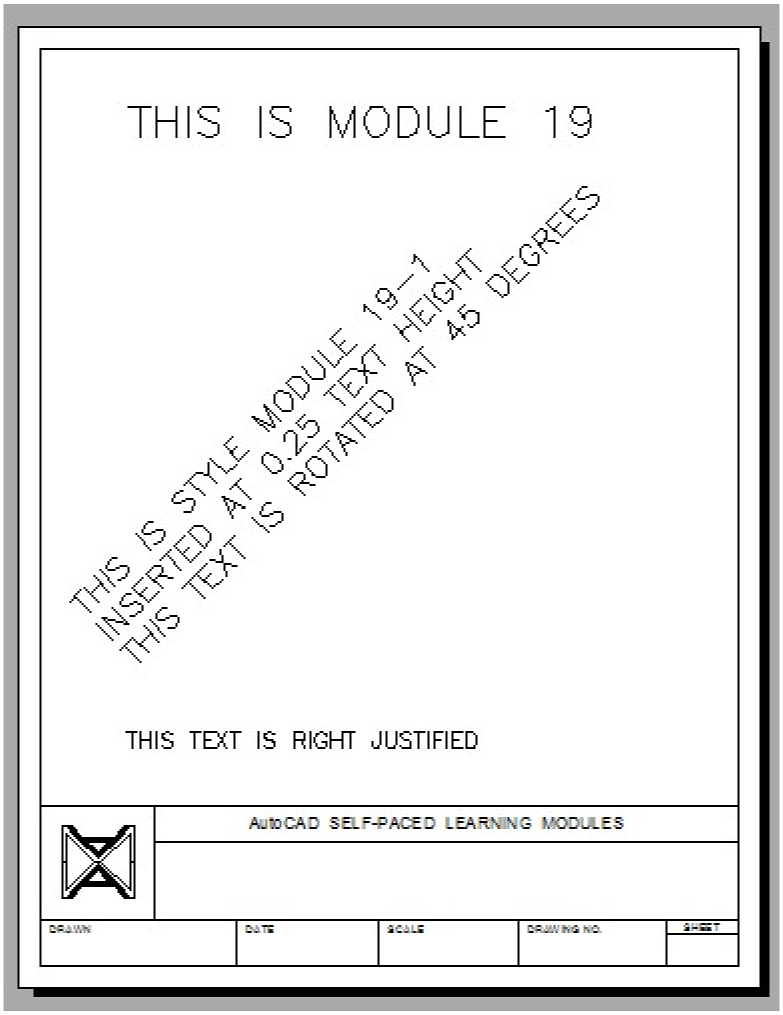

THIS IS MODULE 19
THIS IS STYLE MODULE 19–1
INSERTED AT 0.25 TEXT HEIGHT
THIS TEXT IS ROTATED AT 45 DEGREES
THIS TEXT IS RIGHT JUSTIFIED
AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO.
SHEET

Figure Step 4

Step 5

Double click the text at the top of the drawing to enter edit mode. (Figure Step 5)

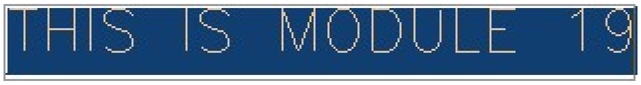

THIS IS MODULE 19

Figure Step 5

Step 6

With the text in edit mode, change the text to read THIS IS EDITED TEXT. (Figure Step 6)

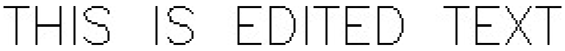

THIS IS EDITED TEXT

Figure Step 6

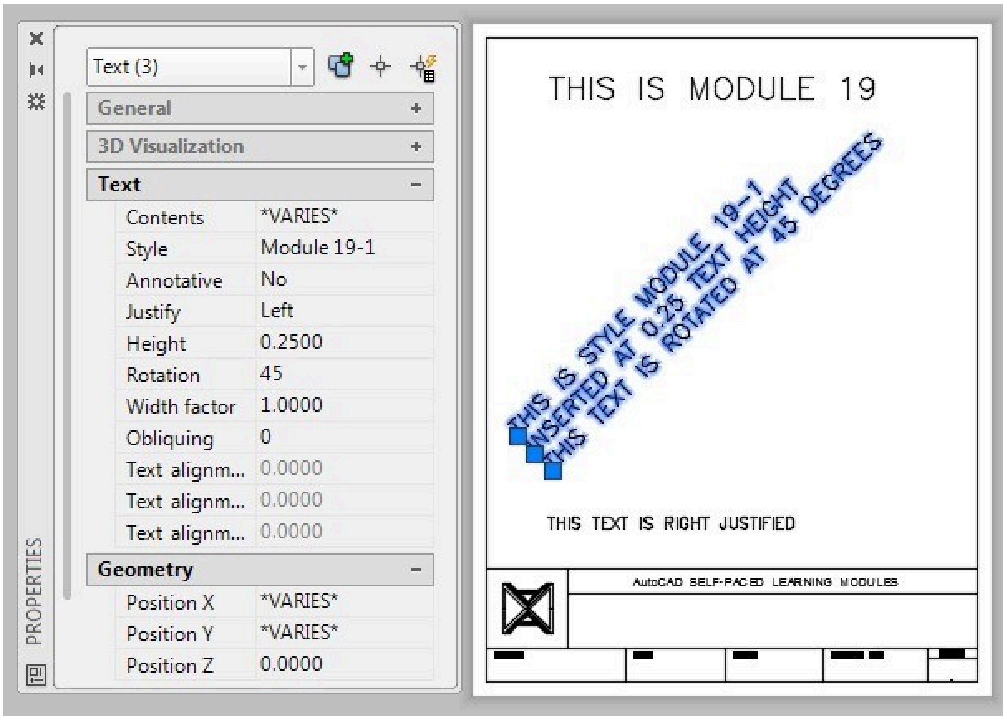
USER TIP: If the text editor does not open when you double click the text object, enter the DBLCLKEDIT system variable, as shown below, and set to ON.

```
Command: DBLCLKEDIT
Enter new value for DBLCLKEDIT <OFF>: ON
Command:
```

MUST KNOW: In the Text Style dialogue box, the text height should almost always be set to zero. When it is set to zero, AutoCAD allows you to change the text height of text assigned that style. It also allows the text style to automatic scale text in a dimension.

Step 7

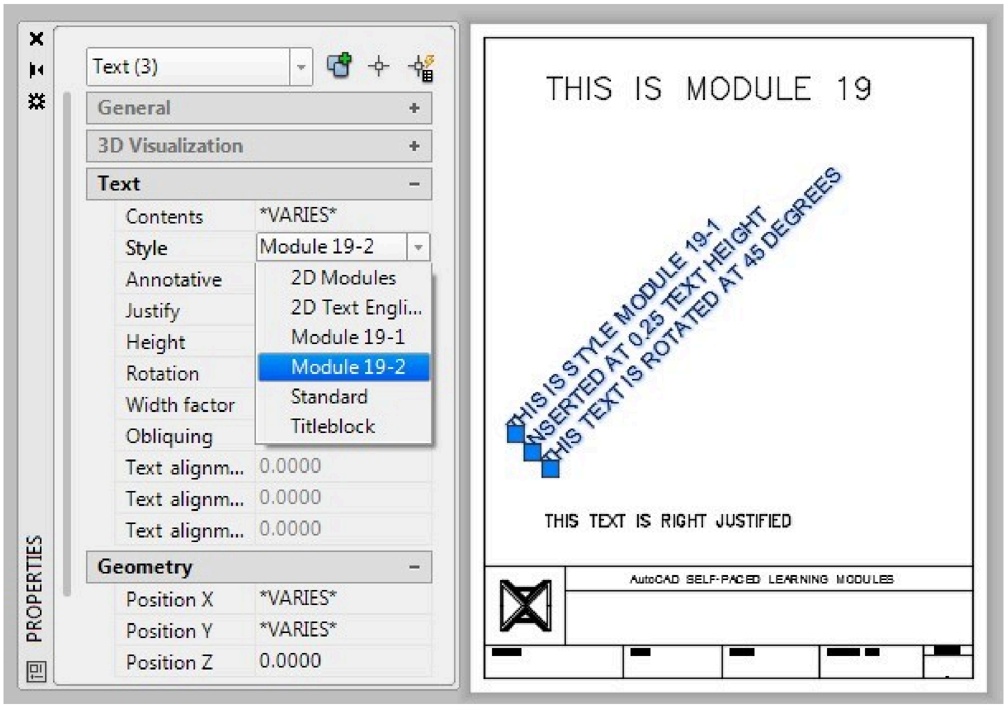
Open the Properties window. Select the three lines of text as shown in the figure. While the text is selected, pull-down the Style property list and select layout: Module 19-2. Press Enter and then Esc to clear the selected text. (Figure Step 7A and 7B)


PROPERTIES
General
3D Visualization
THIS IS MODULE 19
THIS IS STYLE MODULE 19-1
INSERTED AT 0.25 TEXT HEIGHT
THIS TEXT IS ROTATED AT 45 DEGREES
THIS TEXT IS RIGHT JUSTIFIED
AutoCAD SELF-PACED LEARNING MODULES

Text (3)

Text	
Contents	*VARIES*
Style	Module 19-1
Annotative	No
Justify	Left
Height	0.2500
Rotation	45
Width factor	1.0000
Obliquing	0
Text alignm...	0.0000
Text alignm...	0.0000
Text alignm...	0.0000
Geometry	
Position X	*VARIES*
Position Y	*VARIES*
Position Z	0.0000

Figure Step 7A



Text (3)

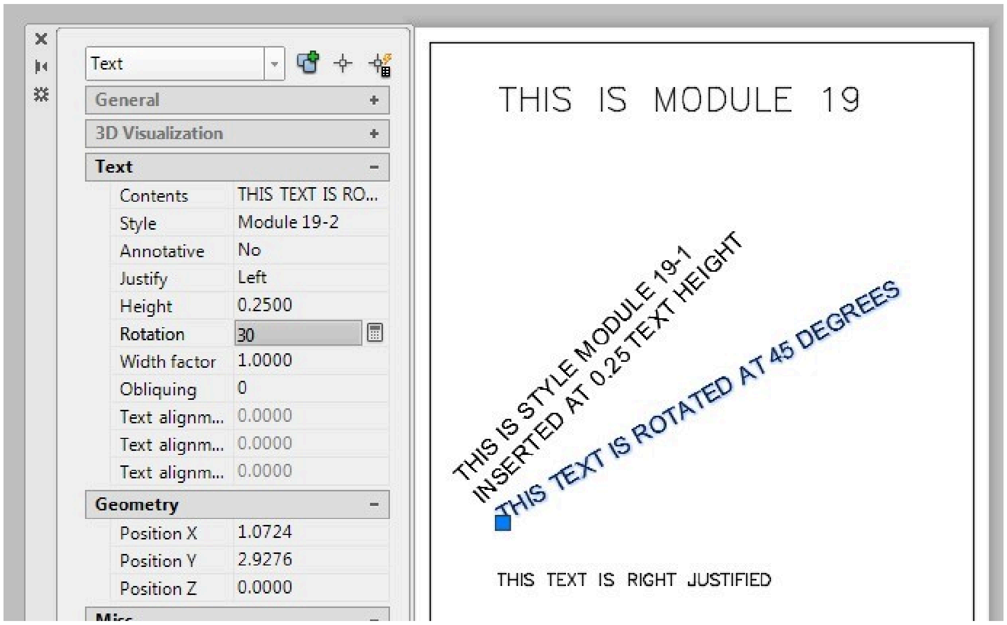
Text	
Contents	*VARIES*
Style	Module 19-2
Annotative	
Justify	
Height	
Rotation	
Width factor	
Obliquing	
Text alignm...	0.0000
Text alignm...	0.0000
Text alignm...	0.0000
Geometry	
Position X	*VARIES*
Position Y	*VARIES*
Position Z	0.0000

PROPERTIES
General
3D Visualization
2D Modules
2D Text Engli...
Module 19-1
Module 19-2
Standard
Titleblock
THIS IS MODULE 19
THIS IS STYLE MODULE 19-1
INSERTED AT 0.25 TEXT HEIGHT
THIS TEXT IS ROTATED AT 45 DEGREES
THIS TEXT IS RIGHT JUSTIFIED
AutoCAD SELF-PACED LEARNING MODULES

Figure Step 7B

Step 8

Select the bottom line of the rotated text as shown in the figure. In the Properties window, change the Rotation angle property to 30. (Figure Step 8)



Text

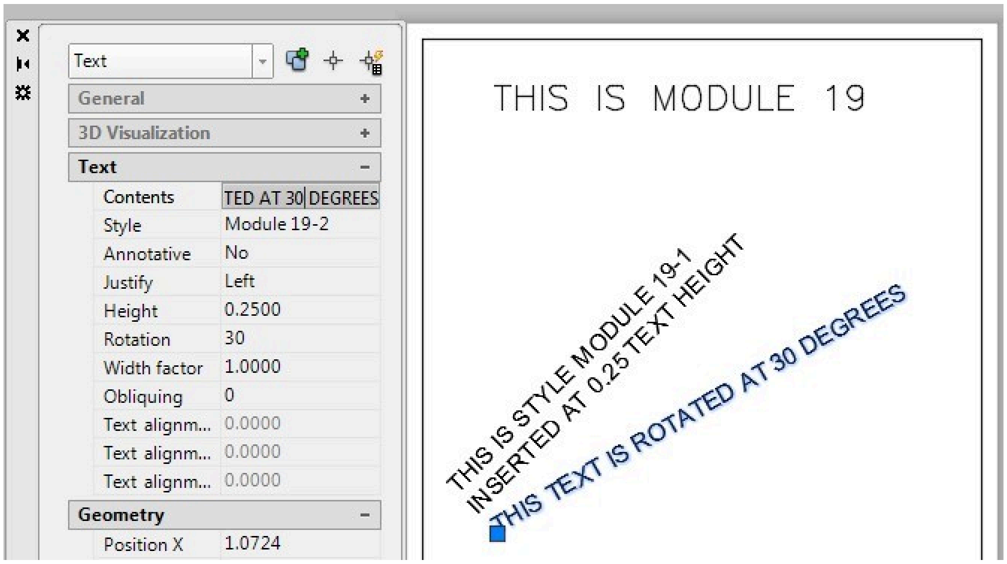
Text	
Contents	THIS TEXT IS RO...
Style	Module 19-2
Annotative	No
Justify	Left
Height	0.2500
Rotation	30
Width factor	1.0000
Obliquing	0
Text alignm...	0.0000
Text alignm...	0.0000
Text alignm...	0.0000
Geometry	
Position X	1.0724
Position Y	2.9276
Position Z	0.0000

General
3D Visualization
Misc
THIS IS MODULE 19
THIS IS STYLE MODULE 19-1
INSERTED AT 0.25 TEXT HEIGHT
THIS TEXT IS ROTATED AT 45 DEGREES
THIS TEXT IS RIGHT JUSTIFIED

Figure Step 8

Step 9

While the text is selected, change the Contents property as shown in the figure. (Figure Step 9)

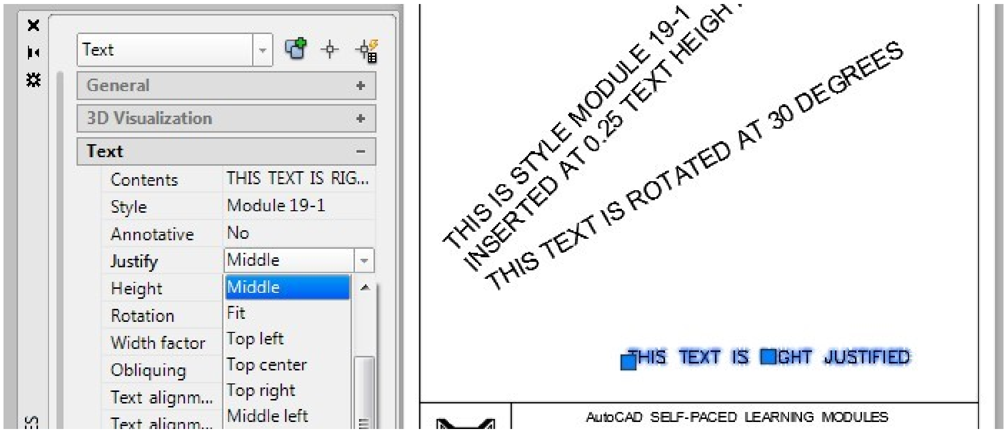

Text
General
3D Visualization
Text
Geometry
THIS IS MODULE 19
THIS IS STYLE MODULE 19-1
INSERTED AT 0.25 TEXT HEIGHT
THIS TEXT IS ROTATED AT 30 DEGREES

Contents TED AT 30 DEGREES
Style Module 19-2
Annotative No
Justify Left
Height 0.2500
Rotation 30
Width factor 1.0000
Obliquing 0
Text alignm... 0.0000
Text alignm... 0.0000
Text alignm... 0.0000
Position X 1.0724

Figure Step 9

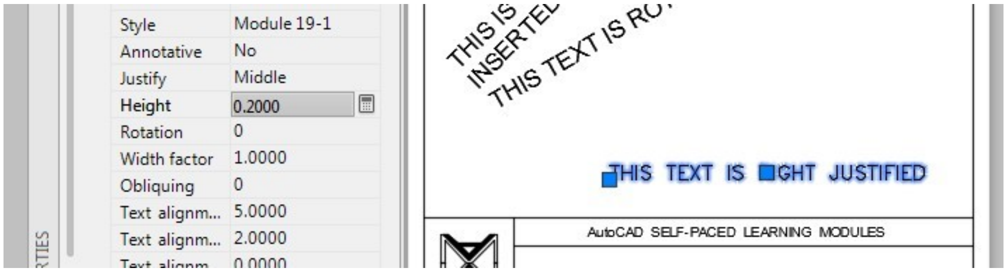
Step 10

Select the bottom text line. In the Properties window, change the Justification property to Middle, the Height property to 0.2, and the Contents property as shown in the figure. (Figure Step 10A, 10B, and 10C)


Text
General
3D Visualization
Text
Middle
Fit
Top left
Top center
Top right
Middle left
THIS IS STYLE MODULE 19-1
INSERTED AT 0.25 TEXT HEIGH
THIS TEXT IS ROTATED AT 30 DEGREES
THIS TEXT IS RIGHT JUSTIFIED
AutoCAD SELF-PACED LEARNING MODULES

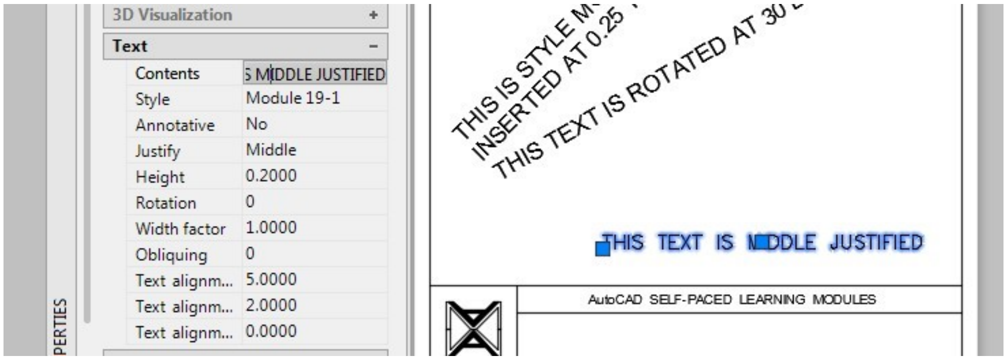
Contents THIS TEXT IS RIG...
Style Module 19-1
Annotative No
Justify Middle
Height
Rotation
Width factor
Obliquing
Text alignm...

Figure Step 10A


THIS IS
INSERTED
THIS TEXT IS RO
THIS TEXT IS RIGHT JUSTIFIED
AutoCAD SELF-PACED LEARNING MODULES

Style Module 19-1
Annotative No
Justify Middle
Height 0.2000
Rotation 0
Width factor 1.0000
Obliquing 0
Text alignm... 5.0000
Text alignm... 2.0000
Text alignm... 0.0000

Figure Step 10B



3D Visualization
Text
PERTIES
THIS IS STYLE M
INSERTED AT 0.25
THIS TEXT IS ROTATED AT 30
THIS TEXT IS MIDDLE JUSTIFIED
AutoCAD SELF-PACED LEARNING MODULES

Contents	S MIDDLE JUSTIFIED
Style	Module 19-1
Annotative	No
Justify	Middle
Height	0.2000
Rotation	0
Width factor	1.0000
Obliquing	0
Text alignm...	5.0000
Text alignm...	2.0000
Text alignm...	0.0000

Figure Step 10C

Step 11

Double click the text and edit it as shown in the figure. (Figure Step 11)

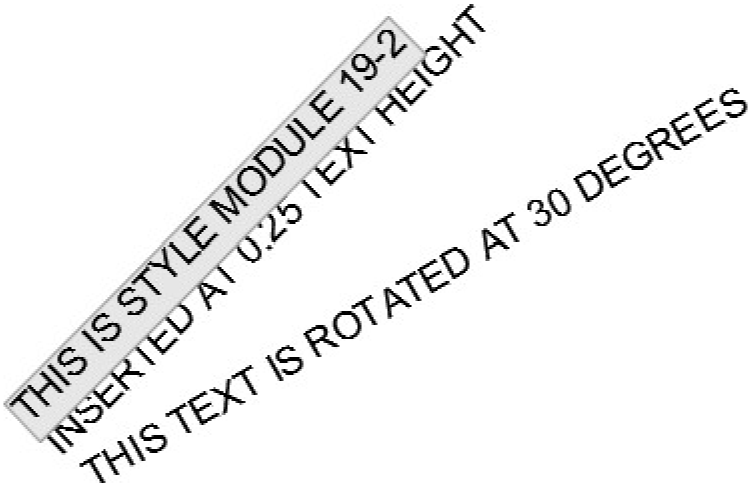

THIS IS STYLE MODULE 19-2
INSERTED AT 0.25 TEXT HEIGHT
THIS TEXT IS ROTATED AT 30 DEGREES

Figure Step 11

Step 12

Your completed layout should appear as shown in the figure. (Figure Step 12)

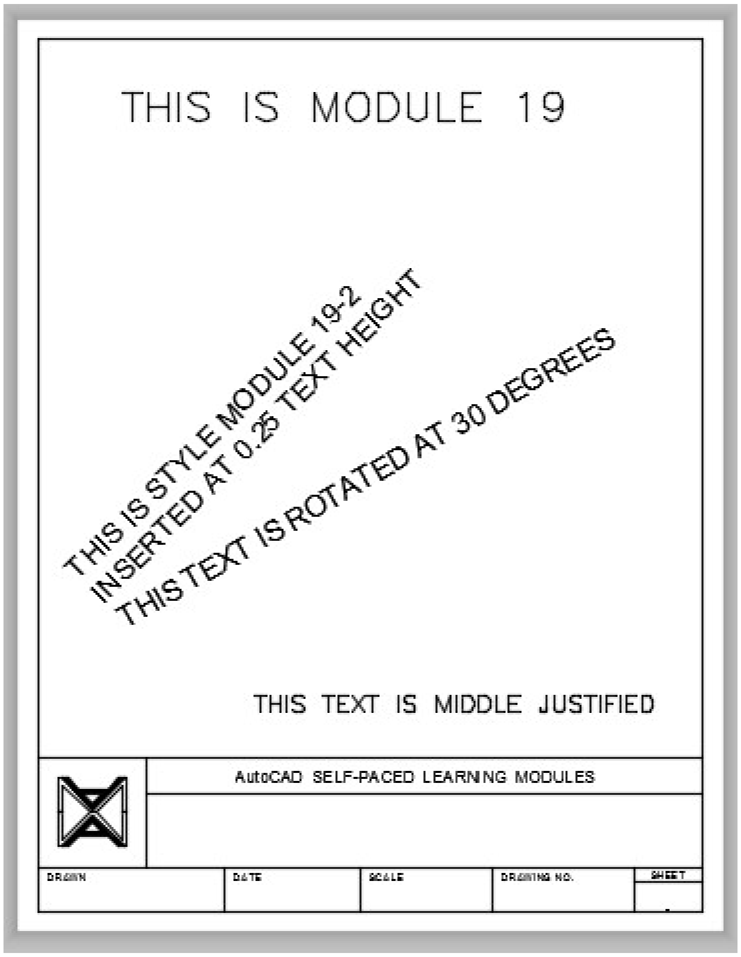
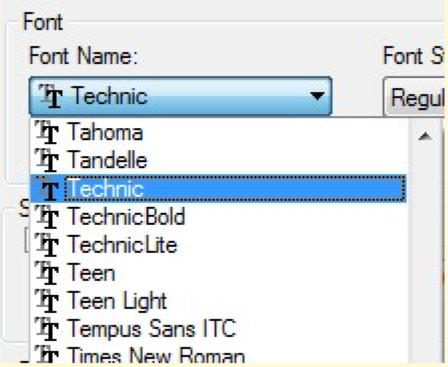

THIS IS MODULE 19
THIS IS STYLE MODULE 19-2
INSERTED AT 0.25 TEXT HEIGHT
THIS TEXT IS ROTATED AT 30 DEGREES
THIS TEXT IS MIDDLE JUSTIFIED
AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO.
SHEET

Figure Step 12

Step 13

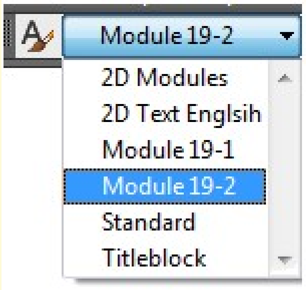
Save and close the drawing.

USER TIP: When using a pull down window with lists, such as the font list in the Text Style dialogue box, and you know the name of the font you want to use, enter the first letter of the font after you pullthe list down. The list will jump to the font names that start with that letter. This should work for all software written to Window’s standards.

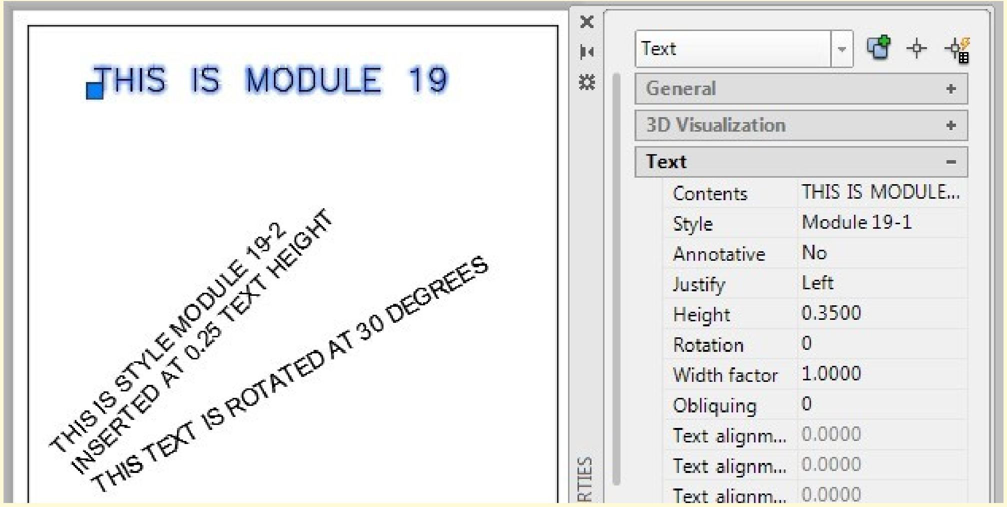

Font
Font Name:
Font S
Technic
Regul
Tahoma
Tandelle
Technic
TechnicBold
TechnicLite
Teen
Teen Light
Tempus Sans ITC
Times New Roman

MUST KNOW: The Standard text style is a special style and cannot be deleted or purged out of the drawing. It is best NOT to use this text style in your drawings. It is much better to create your own styles.

USER TIP: The Styles toolbar has a Pull-down menu that you can use to set the current text style. This is a great feature to quickly change the current text style to speed drawing production.


Module 19-2
2D Modules
2D Text Englsih
Module 19-1
Module 19-2
Standard
Titleblock

USER TIP: The text height, style, rotation angle, content, justification, width factor, and obliquing angle are text properties and can be changed in the Properties window. The font is controlled by the style property. If you want to change the font of an existing text object, change its style. Simply open the Properties window and single click the text to edit its properties.


THIS IS MODULE 19
THIS IS STYLE MODULE 19-2
INSERTED AT 0.25 TEXT HEIGHT
THIS TEXT IS ROTATED AT 30 DEGREES
RTIES
Text
General
3D Visualization
Text

Contents	THIS IS MODULE...
Style	Module 19-1
Annotative	No
Justify	Left
Height	0.3500
Rotation	0
Width factor	1.0000
Obliquing	0
Text alignm...	0.0000
Text alignm...	0.0000
Text alignm...	0.0000

Key Principles

Key Principles in Module 19

1. Single line text is text that is inserted as a single line, can be edited one line at a time, and each line is its own drawing object.
2. The text style defines the text properties, like the font and width of the text. To change the font of a text, its style property must be changed.
3. The text height should almost always be set to zero in the Text Style dialogue box.
4. The Standard text style is a special text style that is contained in every drawing. It cannot be deleted or purged. It is best not to use this text style in your drawings.
5. The text height, style, rotation angle, content, justification, width factor, and obliquing angle are text properties and can be changed in the Properties window.
6. To edit text, double click it. This will automatically open the Edit Text dialogue box with the selected text, ready to be edited. This is a very fast way to edit single line text and almost eliminates the use of the DDEDIT command. Text contents can also be edited in the Properties window.

Lab Exercise 19-1

Time allowed: 60 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 19-1	2D Layout English	Inches

Step 1

On layer: Object, draw the object shown in the figure. (Figure Step 1A and 1B)

AUTHOR’S COMMENTS: Do not delete the construction objects.

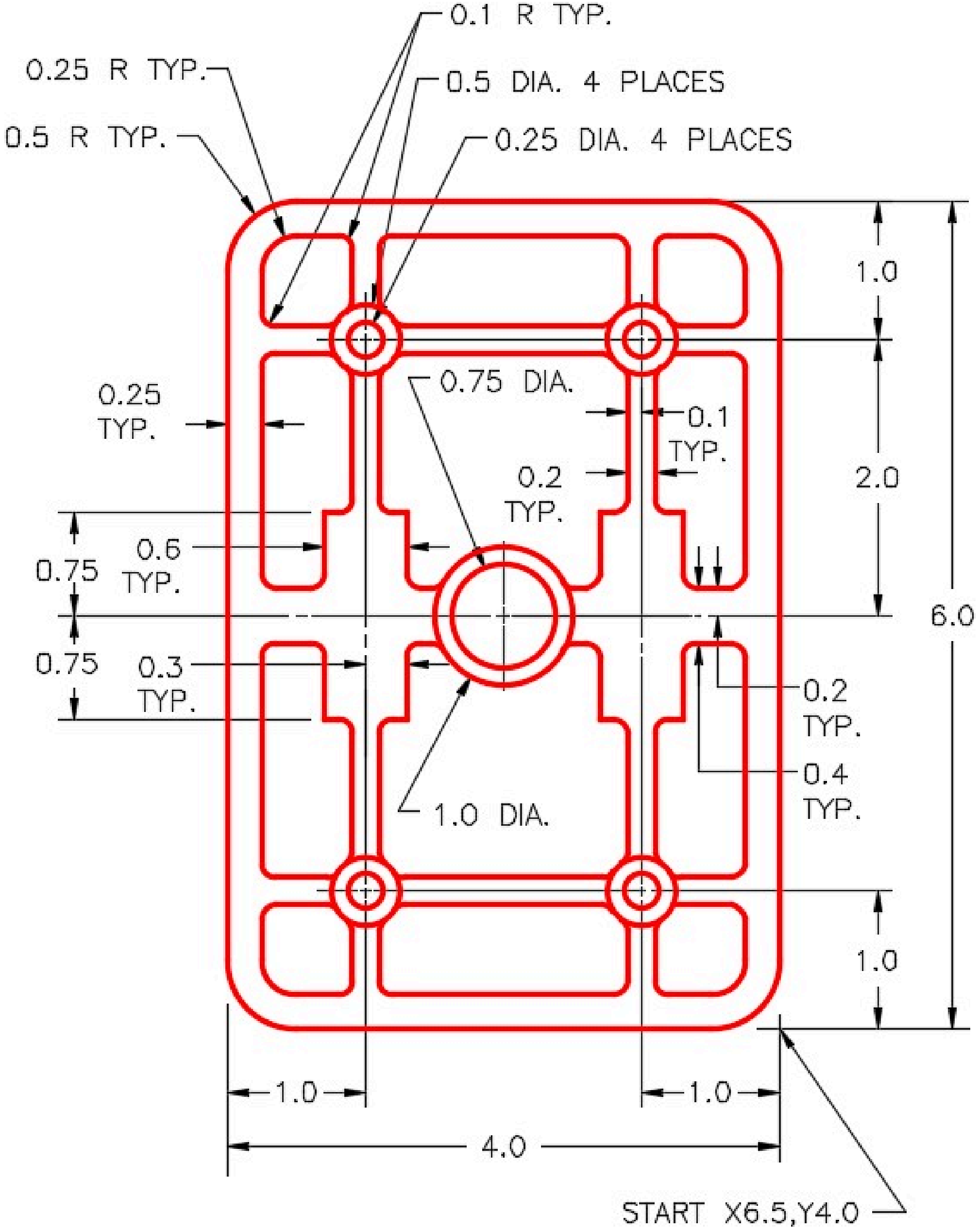

0.1 R TYP.
0.25 R TYP.
0.5 DIA. 4 PLACES
0.5 R TYP.
0.25 DIA. 4 PLACES
1.0
0.25
TYP.
0.75 DIA.
0.1
TYP.
0.2
TYP.
2.0
0.75
0.6
TYP.
6.0
0.75
0.3
TYP.
0.2
TYP.
0.4
TYP.
1.0 DIA.
1.0
1.0
1.0
4.0
START X6.5,Y4.0

Figure Step 1A

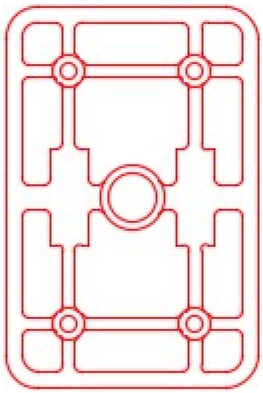


Figure Step 1B
Completed Drawing

Step 2

Draw all construction objects on layer: Construction.

Step 3

Set the insertion units and insert the key. Check the drawing for accuracy.

Step 4

Turn layer: Key off and freeze layer: Construction.

Step 5

In layout: Module Layout A, on layer: Viewport, create a viewport using the MVIEW command. Pan the model as shown in the figure. Set the scale to 1:1 and lock the display. (Figure Step 5)

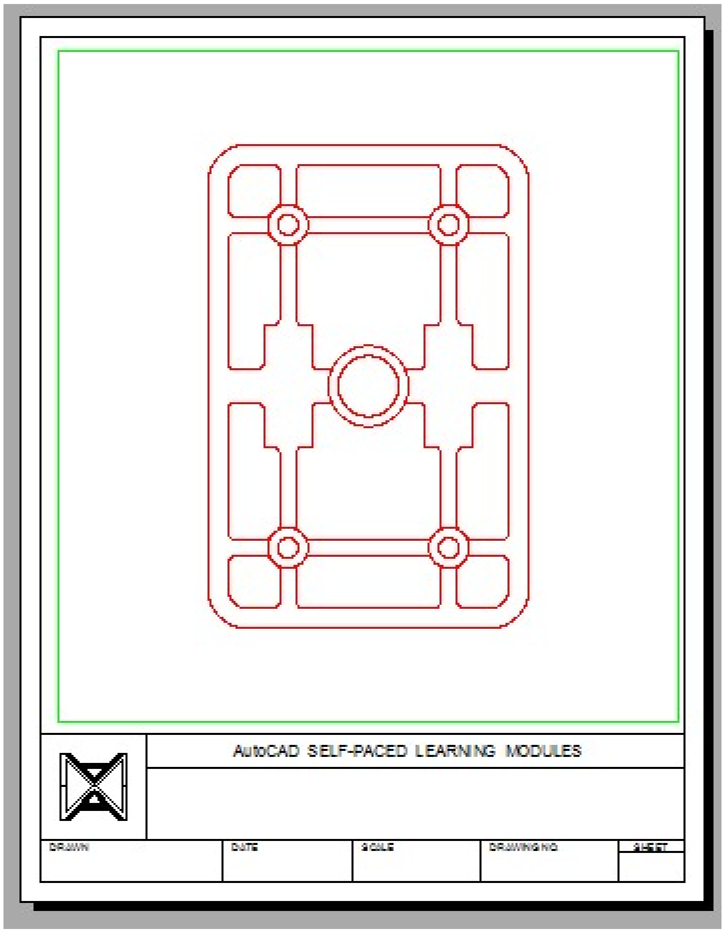

AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO
SHEET

Figure Step 5

Step 6

In layout: Module Layout B, on layer: Viewport, create two viewports using the MVIEW command. Pan the model as shown in the figure and set the custom scale of the left viewport to 1.5:1 and the one on the right to 4:1. Lock the display of both viewports. (Figure Step 6)

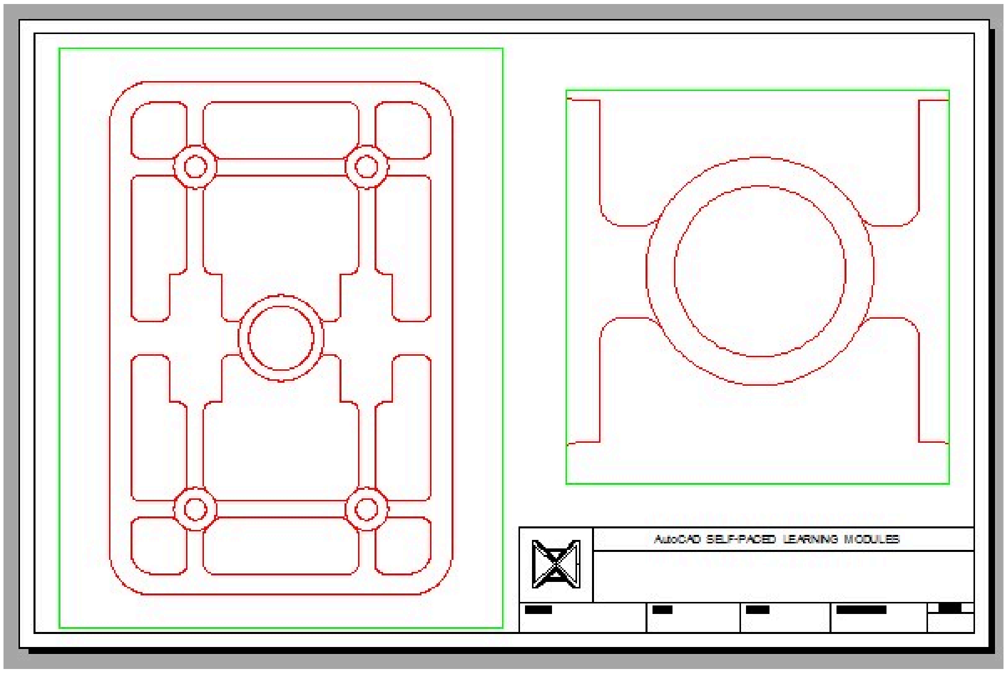

AutoCAD SELF-PACED LEARNING MODULES

Figure Step 6

Step 7

Turn layer: Viewport off. (Figure Step 7)

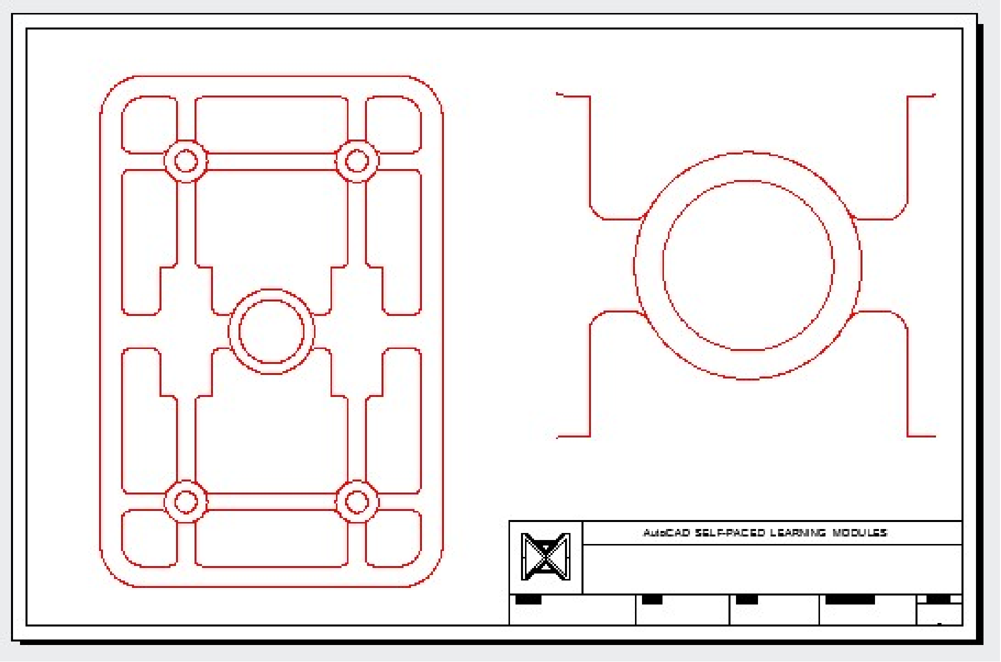
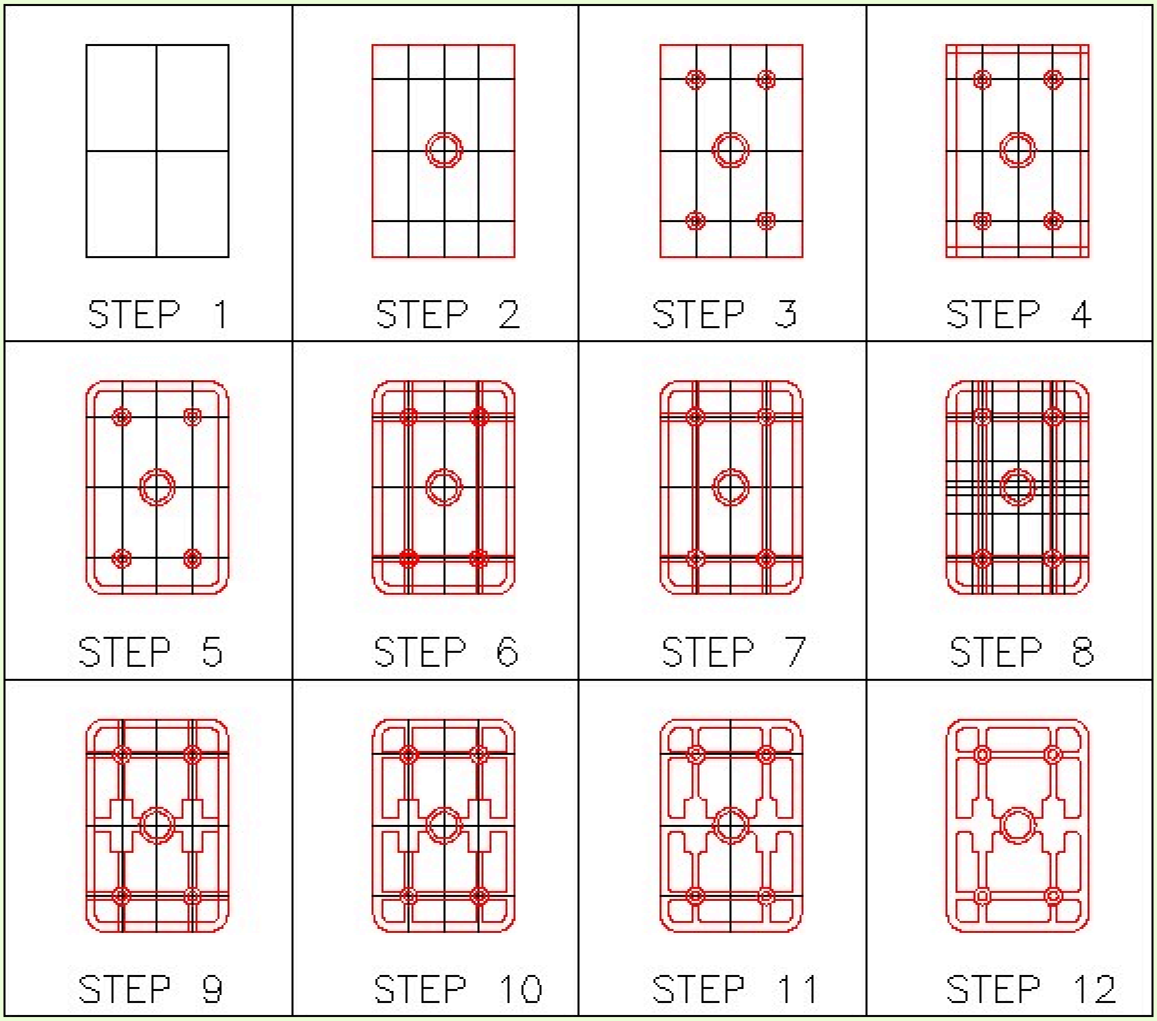


Figure Step 7

Step 8

Save and close the drawing.

AUTHOR'S CONSTRUCTION TECHNIQUES: The following steps are the construction technique suggested by the author to help you learn how to construct objects using AutoCAD. It is only the suggested method and if you can complete the drawing accurately using a different construction technique, that is what is important. You may want to compare your construction technique with the authors.


STEP 1
STEP 2
STEP 3
STEP 4
STEP 5
STEP 6
STEP 7
STEP 8
STEP 9
STEP 10
STEP 11
STEP 12

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

Once you have drawn the outer boundary and the inner construction lines (Step 3 above), use the OFFSET command to construct most of the remaining drawing.

Hint 2

Before you insert the fillets, trim the lines as shown in the figure. If you do not trim them first, you will lose portions of the lines when you insert the fillets. Always insert the fillets last. (Figure Hint 2)

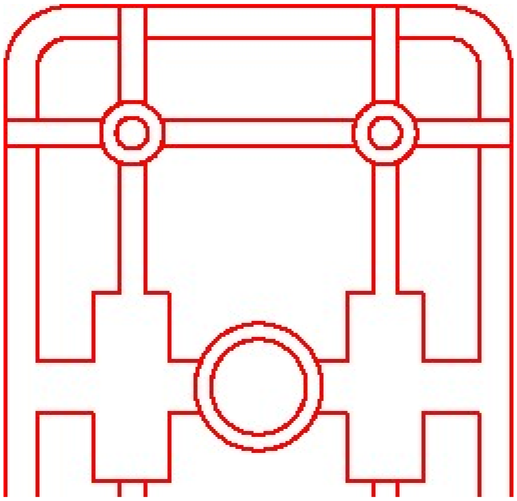


Figure Hint 2

Lab Exercise 19-2

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 19-2	2D Layout English	Inches

Step 1

On layer: Object, draw the object shown in the figure. (Figure Step 1A and 1B)

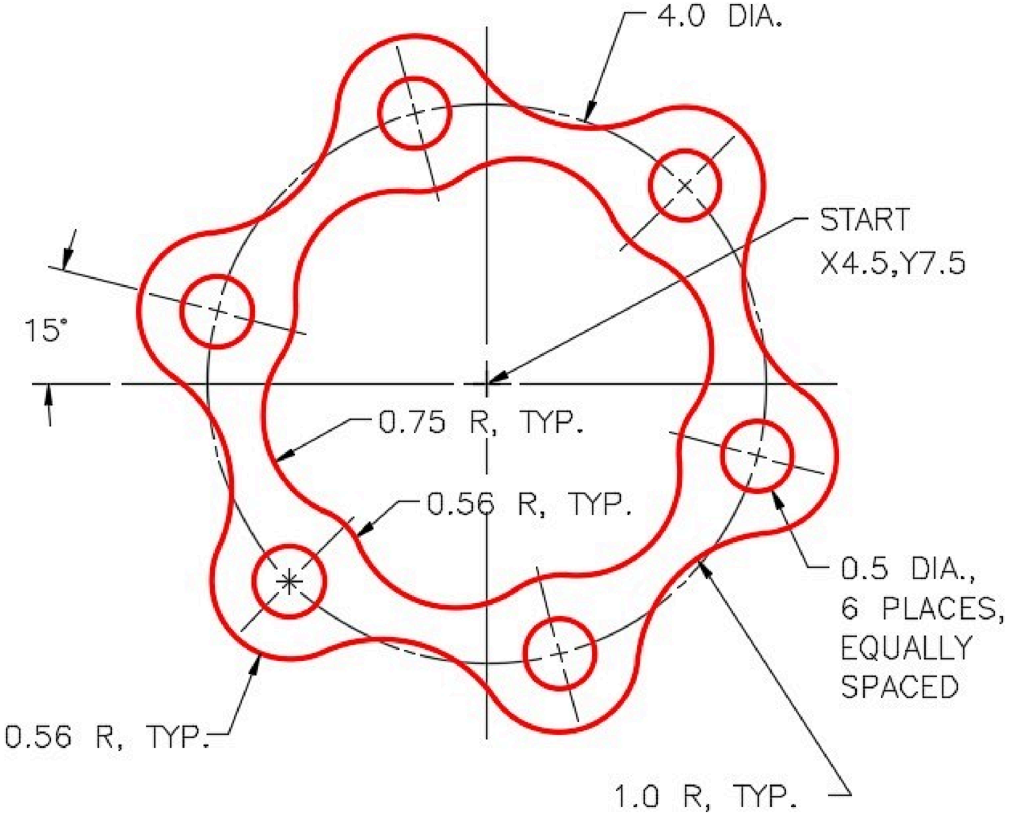

4.0 DIA.
START
X4.5,Y7.5
15°
0.75 R, TYP.
0.56 R, TYP.
0.5 DIA.,
6 PLACES,
EQUALLY
SPACED
0.56 R, TYP.
1.0 R, TYP.

Figure Step 1A

AUTHOR'S COMMENTS: Do not delete the construction objects.

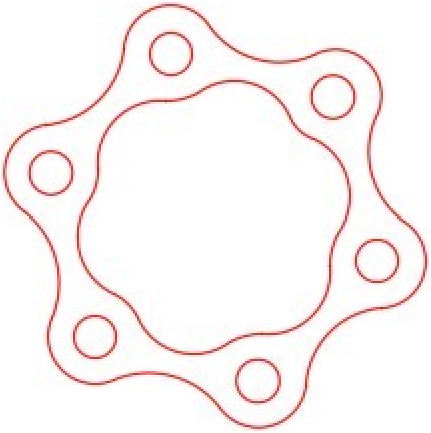


Figure Step 1B Completed Drawing

Step 2

Draw all construction objects on layer: Construction.

Step 3

Set the insertion units and insert the key. Check the drawing for accuracy.

Step 4

Turn layer Key off and freeze layer: Construction.

Step 5

In layout: Module Layout C, on layer: Viewport, create two viewports using the MVIEW command. Pan the model as shown in the figure. Set the custom scale of the left viewport to 2:1 and the one on the right to 3:1. Lock the display of both viewports. (Figure Step 5)

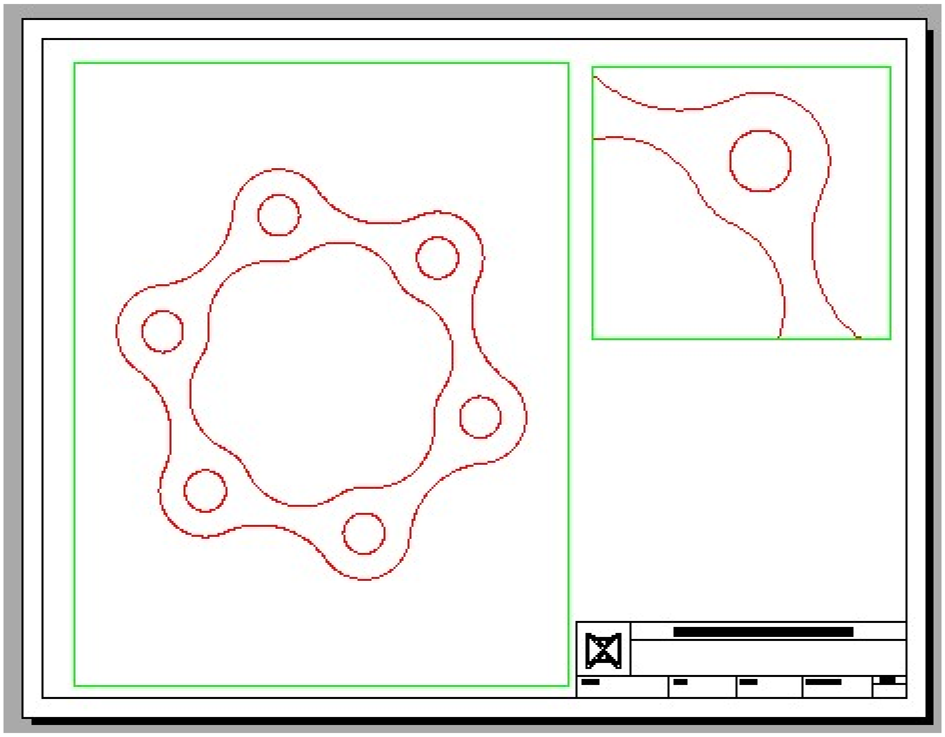


Figure Step 5

Step 6

Turn layer: Viewport off. (Figure Step 6)

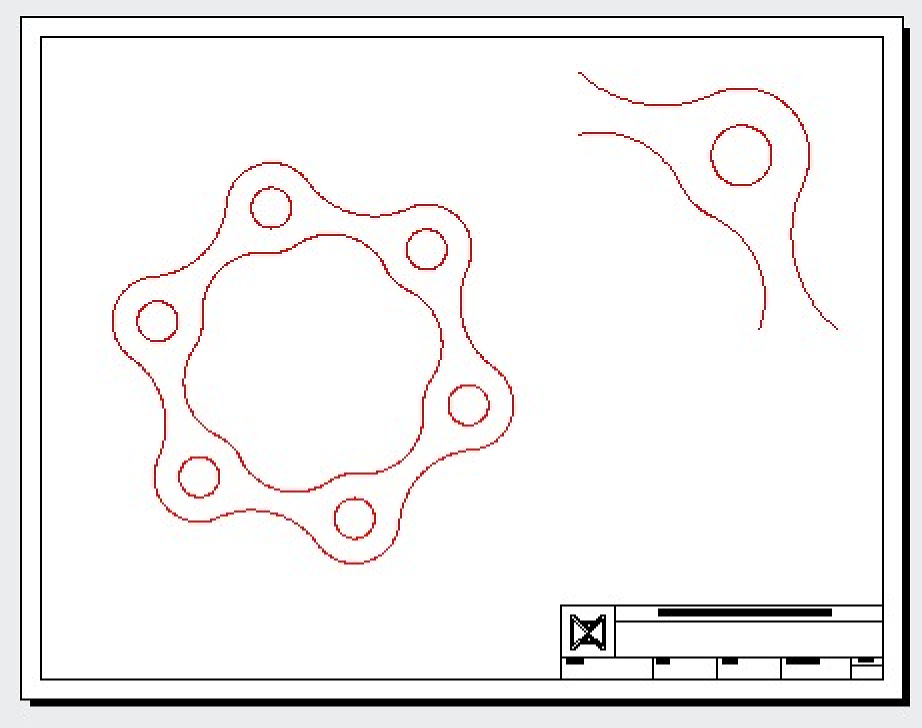
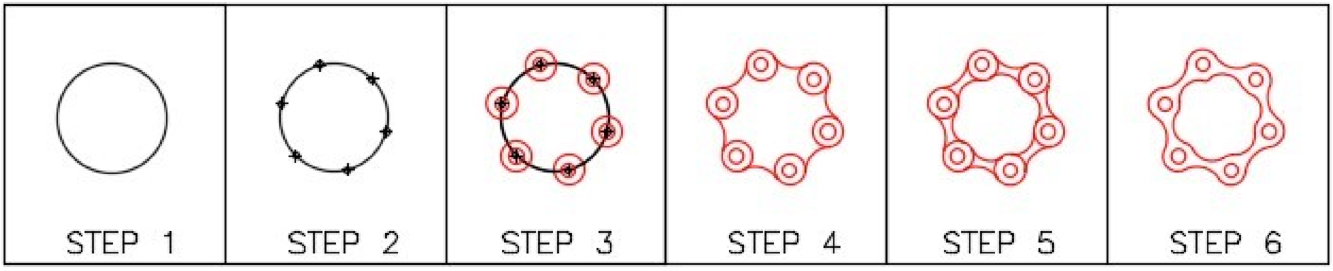


Figure Step 6

Step 7

Save and close the drawing.

AUTHOR'S CONSTRUCTION TECHNIQUES: The following steps are the construction technique suggested by the author to help you learn how to construct objects using AutoCAD. It is only the suggested method and if you can complete the drawing accurately using a different construction technique, that is what is important. You may want to compare your construction technique with the authors.



AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

Rotate the cursor to -15 degrees before dividing the circle. Draw six 0.5 diameter and 1.12 diameter circles using the points as their center locations. (Figure Hint 1A and 1B)

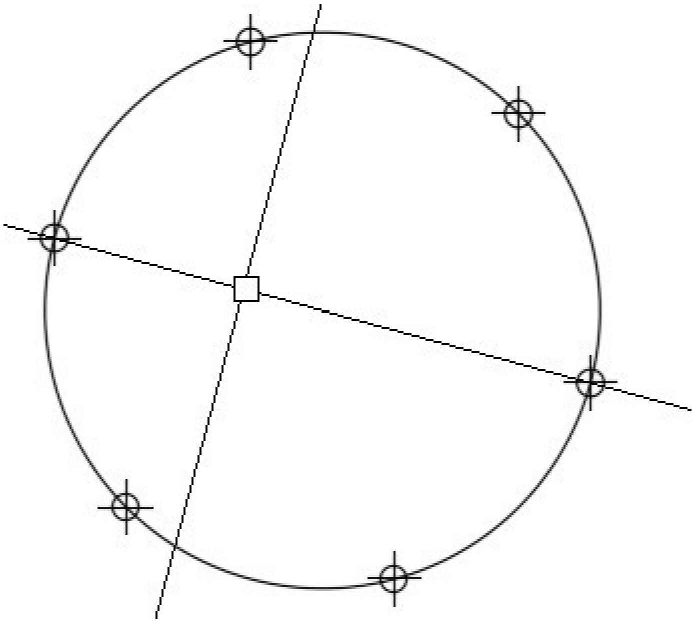


Figure Hint 1A

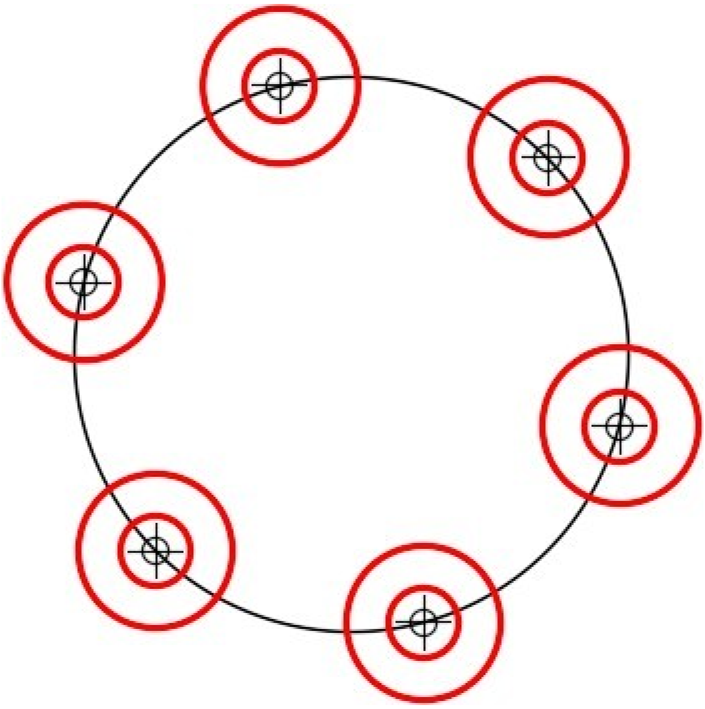


Figure Hint 1B

Hint 2

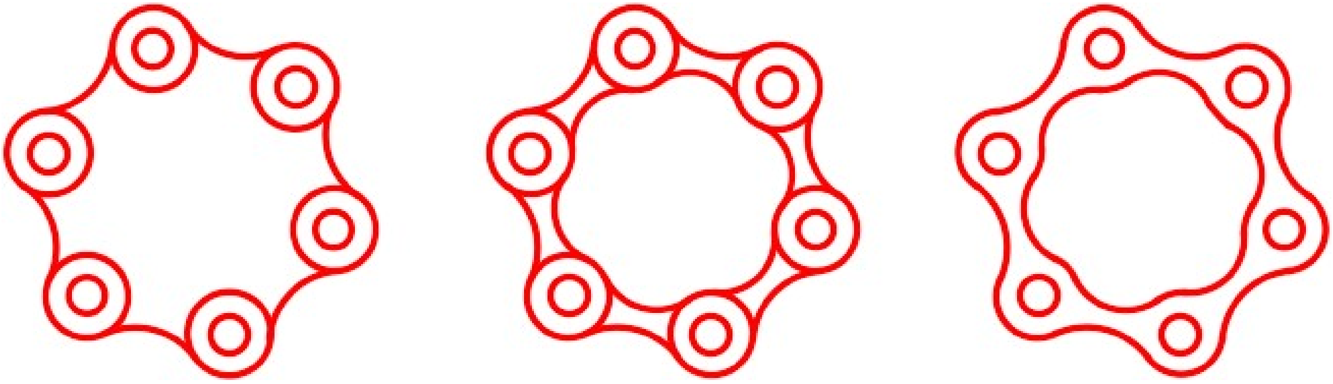
See the following steps:

Step 1 – Insert 1.0 radius fillets.

Step 2 – Insert 0.75 radius fillets.

Step 3 – Trim the circles.

(Figure Step Hint 2)



Step 1 *Step 2* *Step 3*

Figure Hint 2

Module 20: Moving and Copying

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe how to move and copy objects in the current drawing and from one drawing to another.
2. Apply the MOVE, COPY, COPYCLIP, PASTECLIP, and COPYBASE commands to move and copy objects in the current drawing plus copy objects from one drawing to another.
3. Apply the FILETAB and FILETABCLOSE commands to enable the display of the open drawing file tabs above the Graphic window.
4. Describe how to construct a tangent arc that has one end tangent to a line and other end going through a point.
5. Fill out titleblocks in Paper space using the AutoCAD 2D book Standards.

Moving and Copying Objects

One of the real benefits of using a CAD program is increasing your design and drawing productivity. Using the MOVE and COPY commands are a big part of that productivity. In AutoCAD, objects only have to be drawn once and then they can be used over and over again even if they are in a different drawing or at a different size or orientation. If an object has to be drawn that is similar to one that was previously drawn, simply copy it and change it. Using these principles in the your day-to-day work will noticeably increase your productivity.

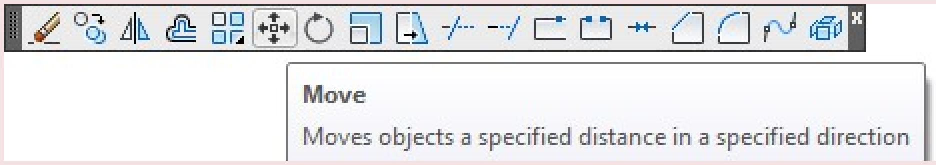
MUST KNOW: When an object is copied, the copies retain all of the properties of the original object even the layer it resides on. This happens regardless of the current layer when the COPY command is executed.

MUST KNOW: Drawing objects only have to be drawn once in AutoCAD and then you can use them over and over. Their size and orientation are not important as they can easily be changed as you will see in future modules. If you have to draw an object that is similar to one you or someone else has drawn before, simply make a copy of it and make your changes. Drawing objects can be copied from one drawing to another.

AutoCAD Command: MOVE

The MOVE command is used to move drawing objects from one location to another on the current drawing.

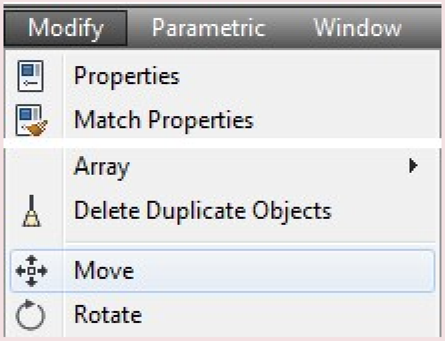
Shortcut: **M**



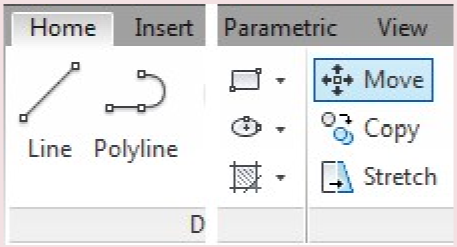
Move

Moves objects a specified distance in a specified direction

Modify Toolbar


Modify
Parametric
Window
Properties
Match Properties
Array
Delete Duplicate Objects
Move
Rotate

Modify Pull-down

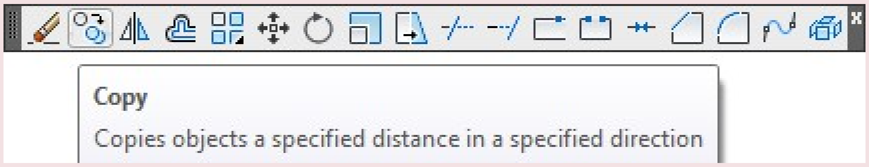

Home
Insert
Parametric
View
Line
Polyline
Move
Copy
Stretch
D

Home Ribbon

AutoCAD Command: COPY

The COPY command is used to copy existing drawing objects in the current drawing.

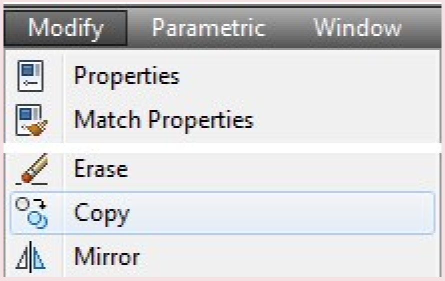
Shortcut: **CO**



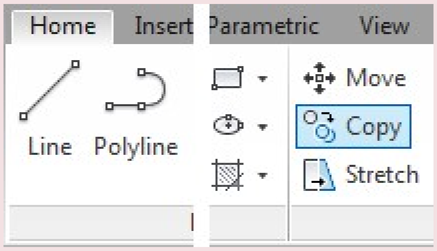
Copy

Copies objects a specified distance in a specified direction

Modify Toolbar


Modify
Parametric
Window
Properties
Match Properties
Erase
Copy
Mirror

Modify Pull-down

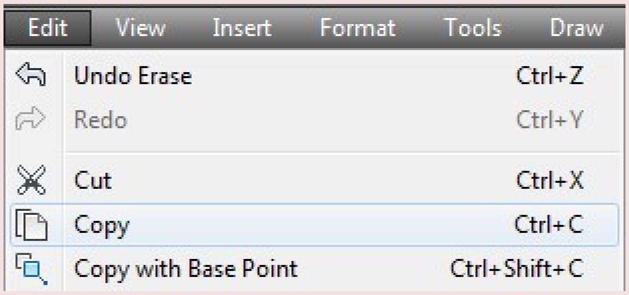

Home
Insert
Parametric
View
Line
Polyline
Move
Copy
Stretch

Home Ribbon

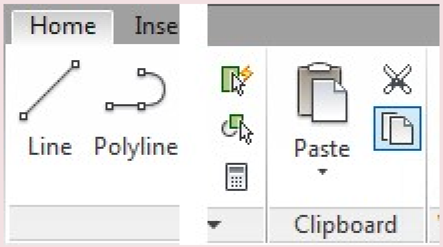
AutoCAD Command: COPYCLIP

The COPYCLIP command is used to copy drawing objects from the current drawing onto the clipboard.

Shortcut: **CTRL+C**


Edit
View
Insert
Format
Tools
Draw
Undo Erase Ctrl+Z
Redo Ctrl+Y
Cut Ctrl+X
Copy Ctrl+C
Copy with Base Point Ctrl+Shift+C

Edit Pull-down

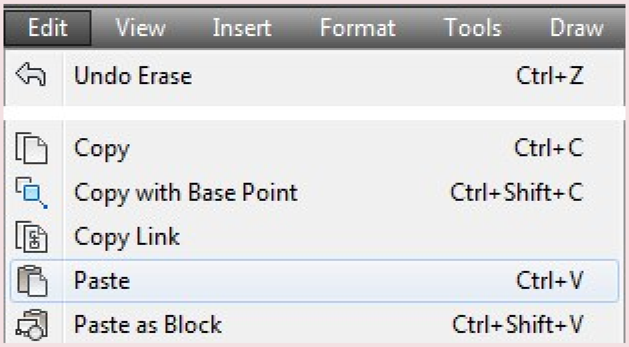

Home
Inse
Line
Polyline
Paste
Clipboard

Home Ribbon

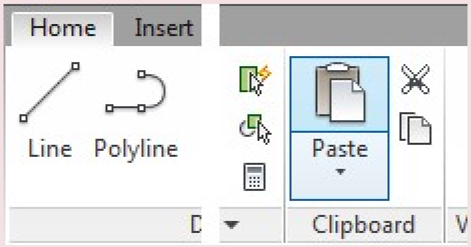
AutoCAD Command: PASTECLIP

The PASTECLIP command is used to paste drawing objects from the clipboard into the current drawing.

Shortcut: **CTRL+V**


Edit
View
Insert
Format
Tools
Draw
Undo Erase Ctrl+Z
Copy Ctrl+C
Copy with Base Point Ctrl+Shift+C
Copy Link
Paste Ctrl+V
Paste as Block Ctrl+Shift+V

Edit Pull-down

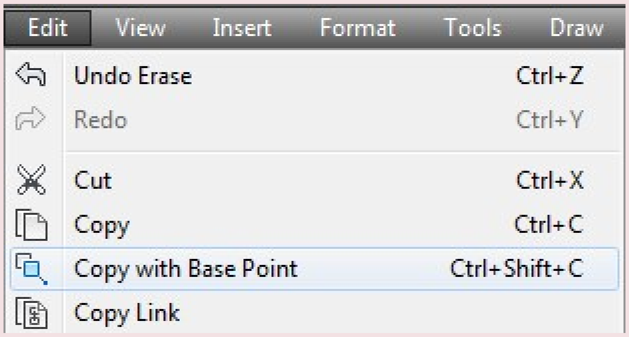

Home
Insert
Line
Polyline
Paste
Clipboard

Home Ribbon

AutoCAD Command: COPYBASE

The COPYBASE command is used to copy drawing objects in the current drawing onto the clipboard with a known basepoint.

Shortcut: **CTRL+SHIFT+C**


Edit
View
Insert
Format
Tools
Draw
Undo Erase Ctrl+Z
Redo Ctrl+Y
Cut Ctrl+X
Copy Ctrl+C
Copy with Base Point Ctrl+Shift+C
Copy Link

Edit Pull-down

WORKALONG: Moving and Copying Objects

Step 1

Start a new drawing using the template: 2D Layout English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 20-1.

Step 3

On layer: Object, draw the rectangle shown in the figure. (Figure Step 3)

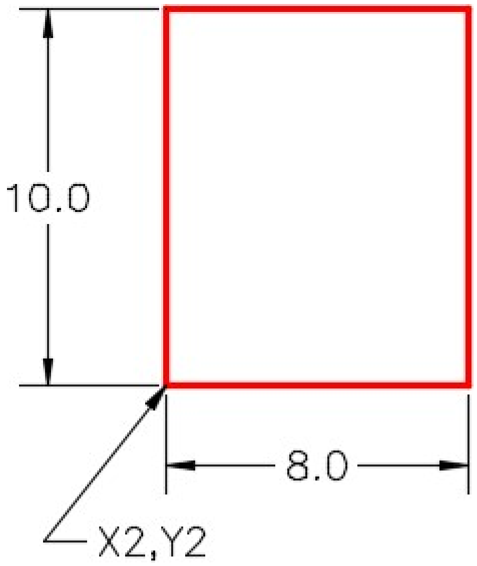


Figure Step 3

Step 4

On layer: Object, draw a circle and the inscribed hexagon as shown in the figure. Draw it anywhere outside of the rectangle. (Figure Step 4A and 4B)

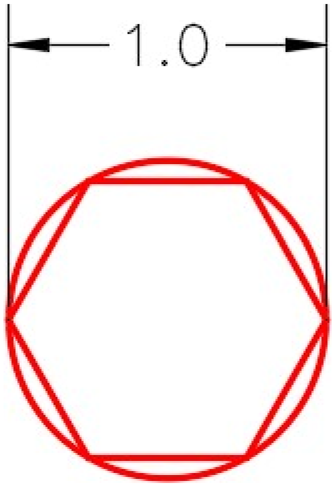


Figure Step 4A

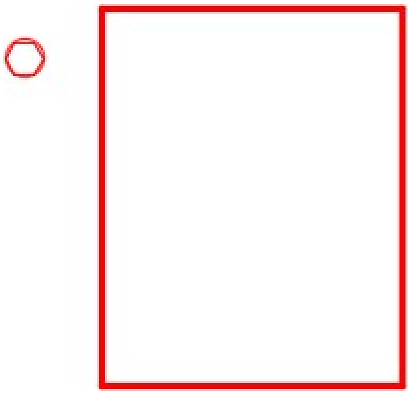


Figure Step 4B

AUTHOR'S COMMENTS: Use the DIVIDE command to create the points to draw the hexagon.

Step 5 IMPORTANT STEP

Unless otherwise instructed, when starting each lab exercise, while working on workalongs and lab exercises in the AutoCAD 2D book, ensure that all items on the Status bar are disabled. (Figure Step 5)

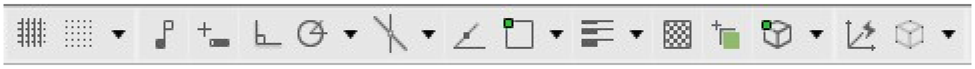


Figure Step 5

Step 6

Enter the MOVE command, as shown below, to move the circle and hexagon inside the rectangle to the absolute coordinates X3.5,Y11. (Figure Step 6A, 6B, and 6C)

Command: **MOVE**

Select objects:

Specify opposite corner:

(Selecting the object using a window.)

Select objects:

(Press Enter)

Specify base point or [Displacement] <Displacement>: *(cen)*

(Snap to the center of the circle.)

Specify second point or <use first point as displacement>: **3.5,11**

Command:

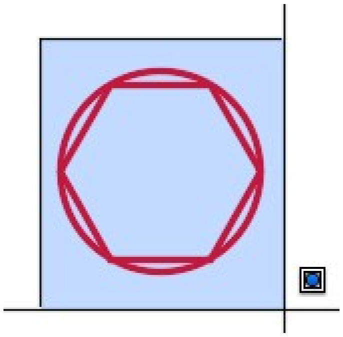


Figure Step 6A

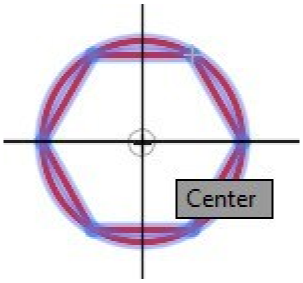


Figure Step 6B

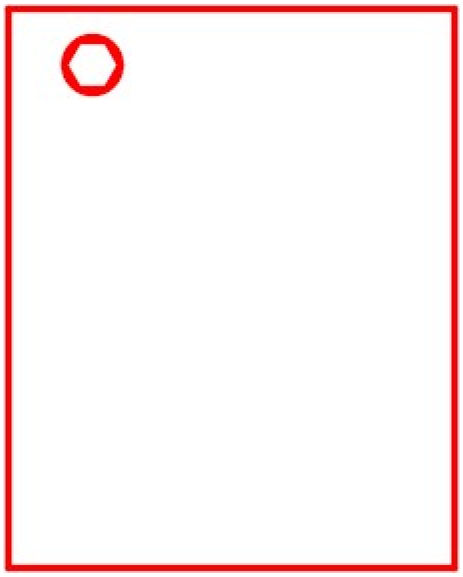


Figure Step 6C

AUTHOR'S COMMENTS: The circle and hexagon were moved to locate the center of the circle at the absolute coordinate location X3.5,Y11.

Step 7

Using the OFFSET command, offset the top line of the rectangle 1 inch down. Change the layer of the line you just created to layer: Construction. (Figure Step 7)

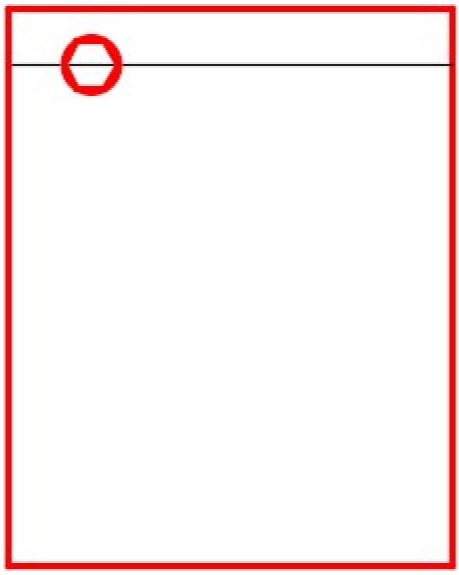


Figure Step 7

Step 8

Using the OFFSET command, offset the top construction line 8 times at a distance of 1 inch as shown in the figure. (Figure Step 8)

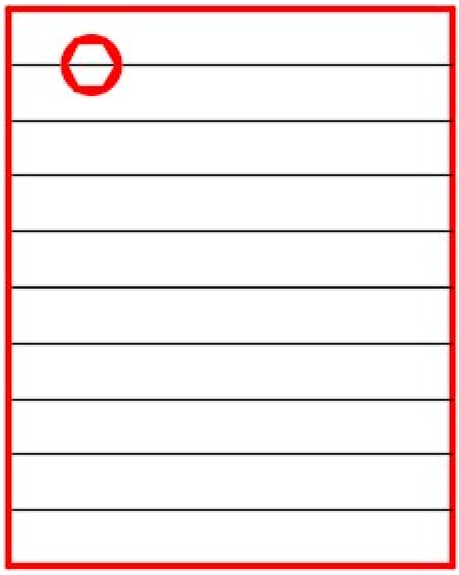


Figure Step 8

AUTHOR'S COMMENTS: There should be a total of 9 construction lines, 1 inch apart.

Step 9

Using the OFFSET command, offset the left line of the rectangle 1.5 inches to the right. Change the layer of the line you just created to layer: Construction. (Figure Step 9)

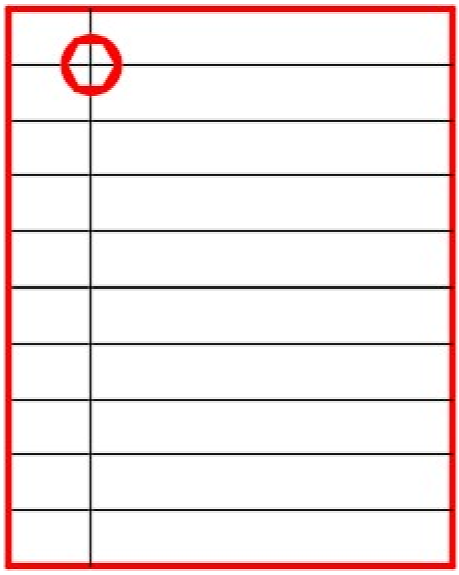


Figure Step 9

Step 10

Enter the COPY command, as shown below, to copy the circle and hexagon 4 times. (Figure Step 10A, 10B, 10C, and 10D)

Command: **COPY**

Select objects:

Specify opposite corner: 7 found

(Use a window to select the objects)

Select objects:

Current settings: Copy mode = Multiple

Specify base point or [Displacement/mOde] <Displacement>:

or <use first point as displacement>: *(int)*

(Snap to the intersection of the lines for the base point)

Specify second point or [Exit/Undo] <Exit>: *(int)*

Specify second point or [Exit/Undo] <Exit>: *(int)*

Specify second point or [Exit/Undo] <Exit>: *(int)*

Specify second point or [Exit/Undo] <Exit>: *(int)*

(Snap to the intersection of the construction lines)

Command:

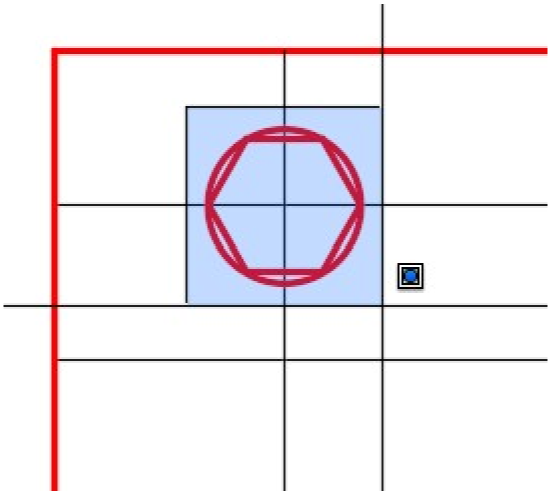


Figure Step 10A

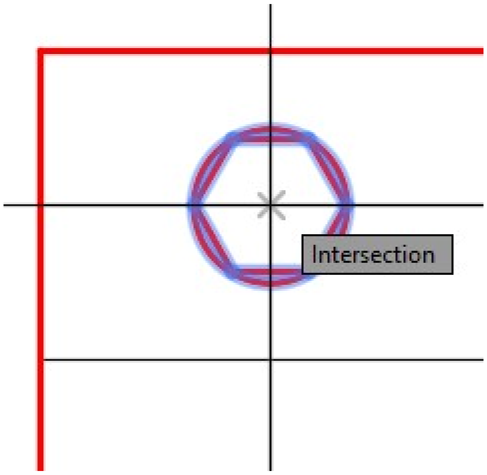


Figure Step 10B

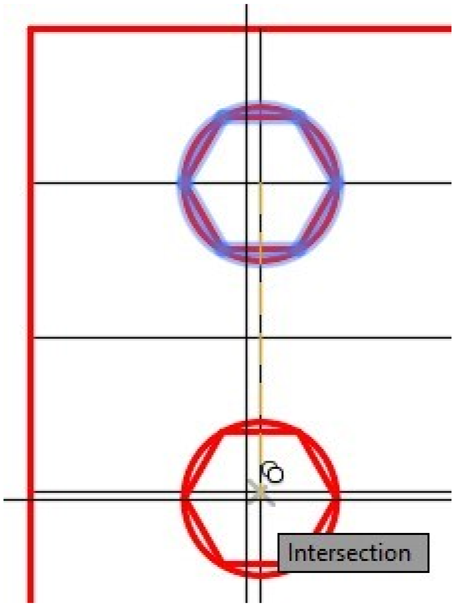

Intersection

Figure Step 10C

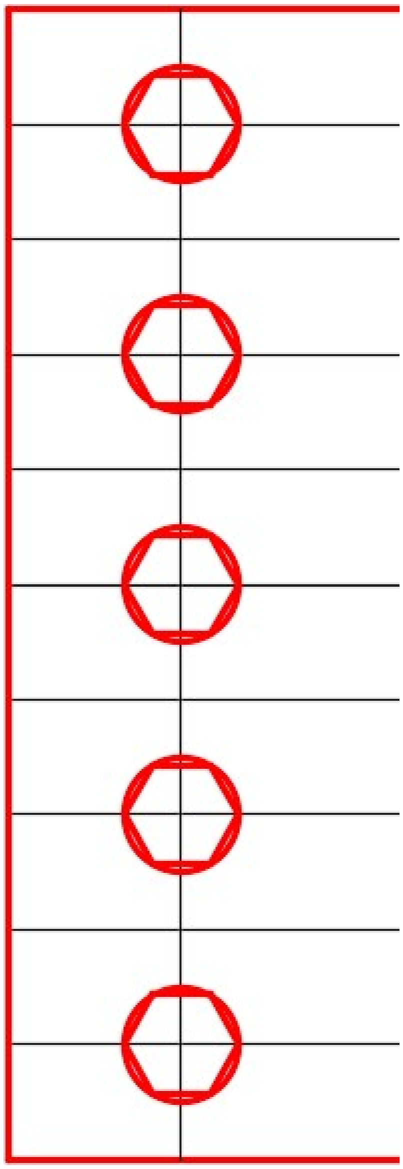


Figure Step 10D

Step 11

Enter the COPY command, as shown below, to copy all circles and hexagons 4 inches in the positive X direction. (Figure Step 11A and 11B)

Command: **COPY**

Select objects: Specify opposite corner: 35 found

Select objects:

Current settings: Copy mode = Multiple

Specify base point or [Displacement/mOde] <Displacement>: **0,0**

(The base point)

Specify second point or <use first point as displacement>: **@4,0**

(The second point is set 4 inches in the positive X direction.)

Specify second point or [Exit/Undo] <Exit>:

Command:

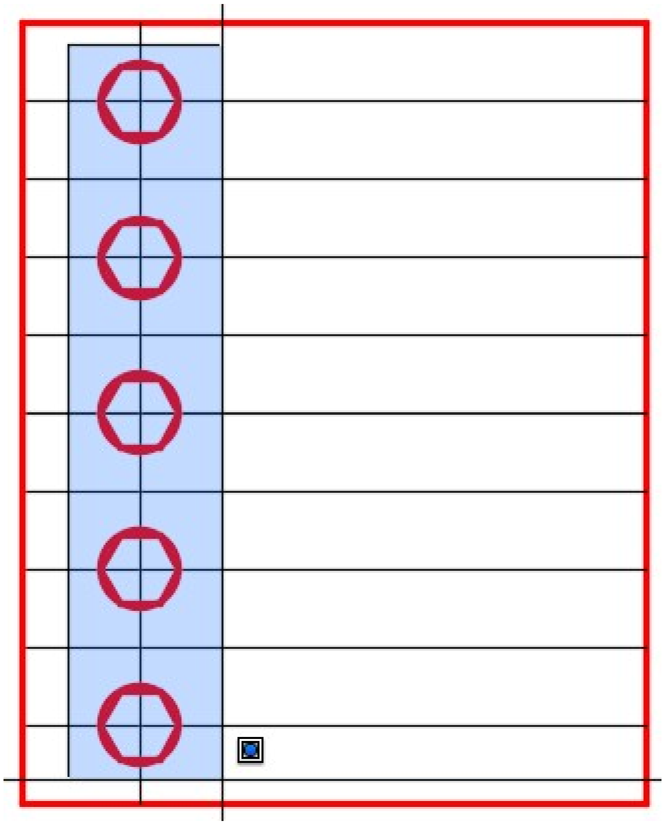


Figure Step 11A

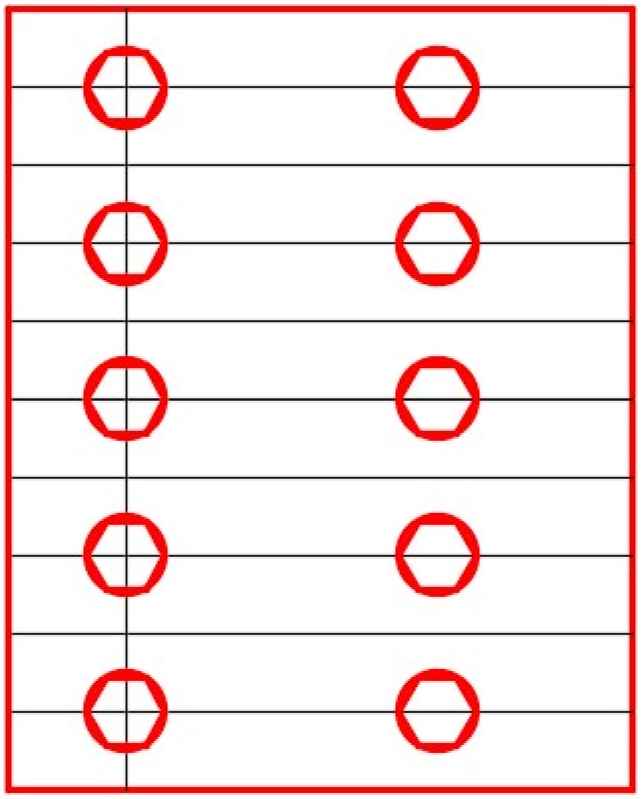


Figure Step 11B

AUTHOR'S COMMENTS: I used the base point 0,0. Any XY location could have been used. To keep it simple, I use 0,0.

Step 12

Enter the MOVE command, as shown below, to move all the circles and hexagons 0.5 inches in the positive X direction. (Figure Step 12A and 12B)

Command: **MOVE**

Select objects:

Specify opposite corner: 70 found

(Use a window to select the objects.)

Select objects:

Specify base point or [Displacement] <Displacement>: **0,0**

Specify second point or <use first point as displacement>: **@0.5,0**

Command:

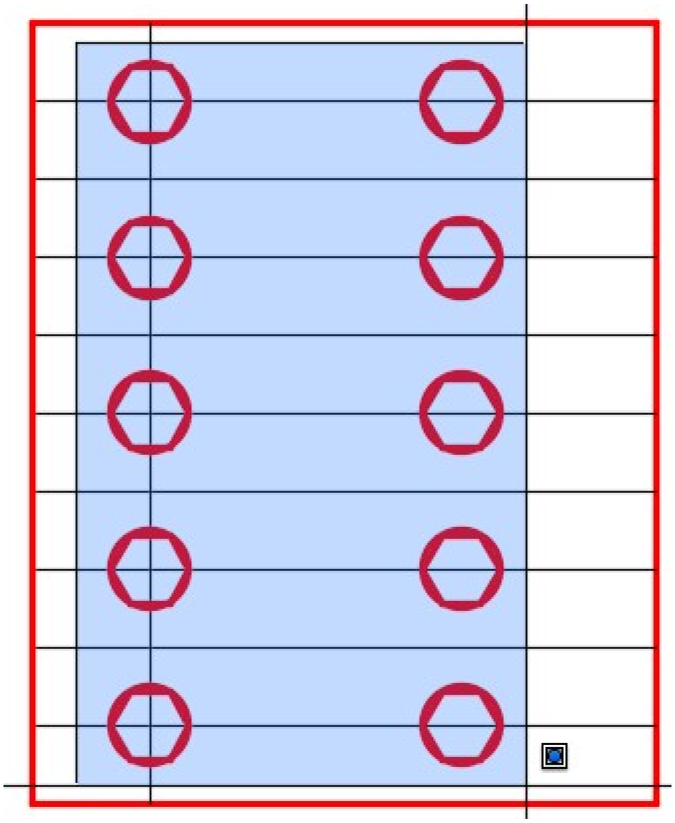


Figure Step 12A

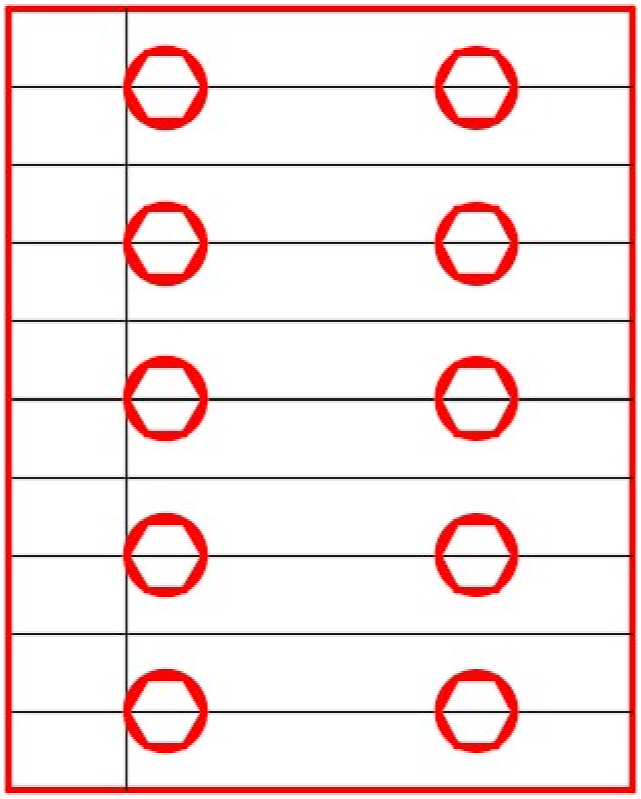


Figure Step 12B

Step 13

Freeze layer: Construction. (Figure Step 13)

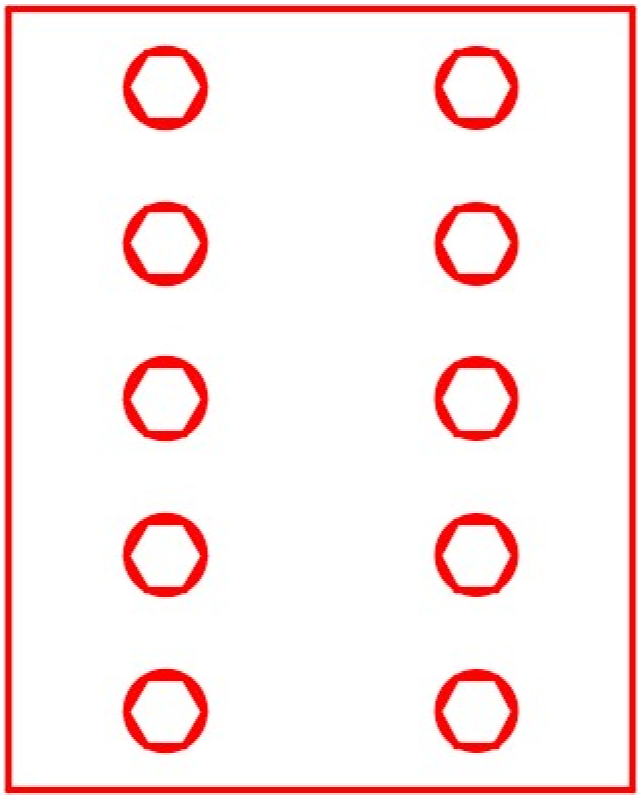


Figure Step 13

Step 14

Enter the COPY command, as shown below, to copy all objects. (Figure Step 14A, 14B, and 14C)

Command: **COPY**

Select objects:

Specify opposite corner: 74 found

(Select all objects using a crossing window.)

Select objects:

Current settings: Copy mode = Multiple

Specify base point or [Displacement/mOde] <Displacement>:

(end)

(Snap to the end of the line.)

Specify second point: *(end)*

(Snap to the other end of the line.)

Command:

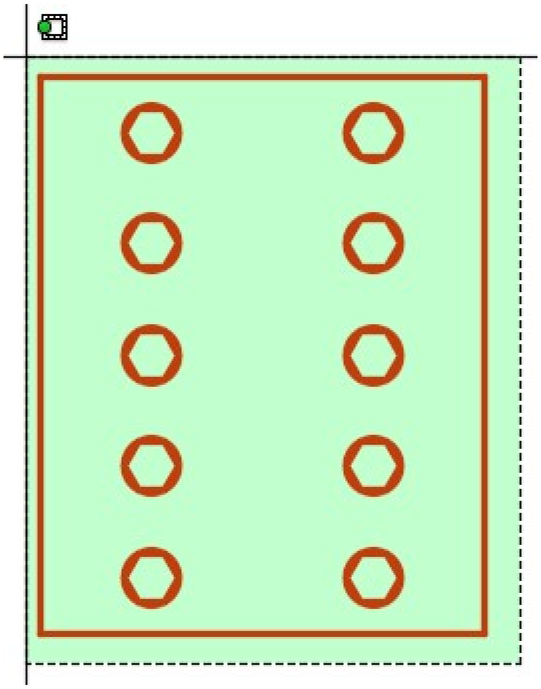


Figure Step 14A

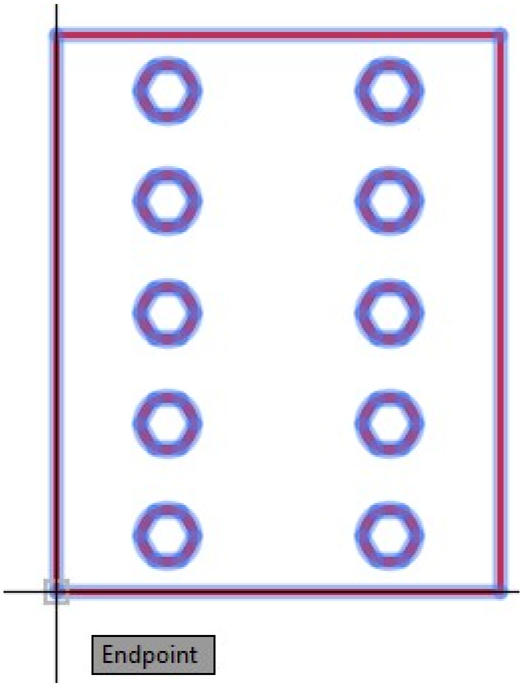


Figure Step 14B

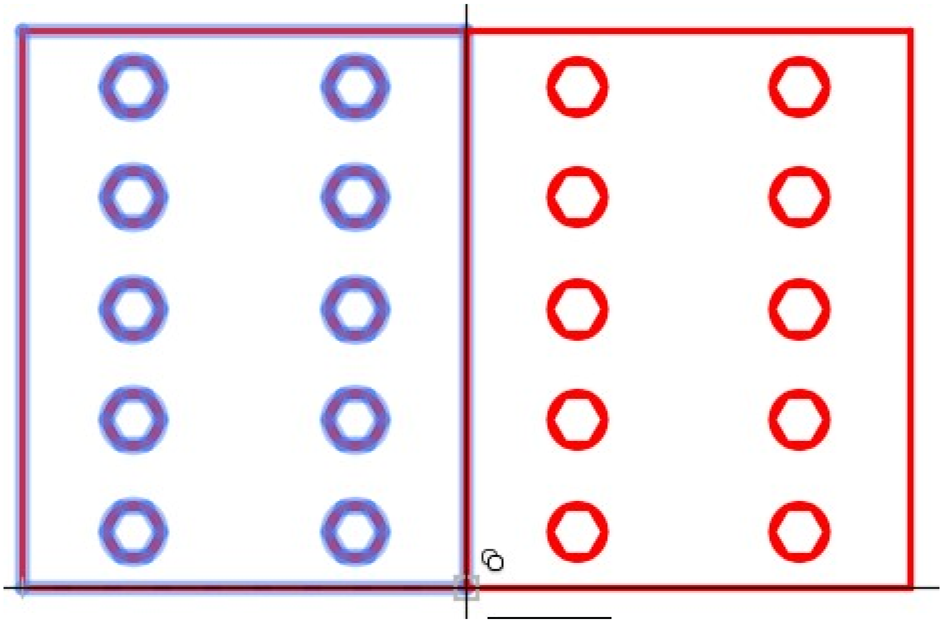


Figure Step 14C

Step 15

Delete the line in the center by picking it and pressing the Delete key. (Figure Step 15)

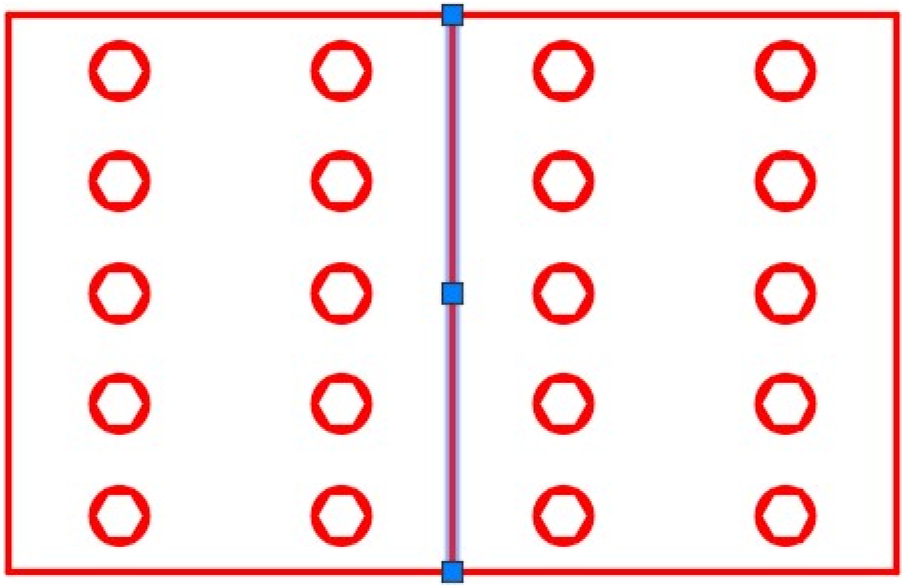


Figure Step 15

AUTHOR’S COMMENTS: When you copied the objects, the line in the center was duplicated. More then one object at the same location will cause problems in a drawing. When you know that you have duplicated an objects, correct it by deleting the duplicate as soon as you can. Use the Pick method to select the line since it will only select one object at a time. If you use a window of some kind, it would select all objects.

Step 16

Using what you just learned, copy all of the objects down to match the figure. (Figure Step 16)

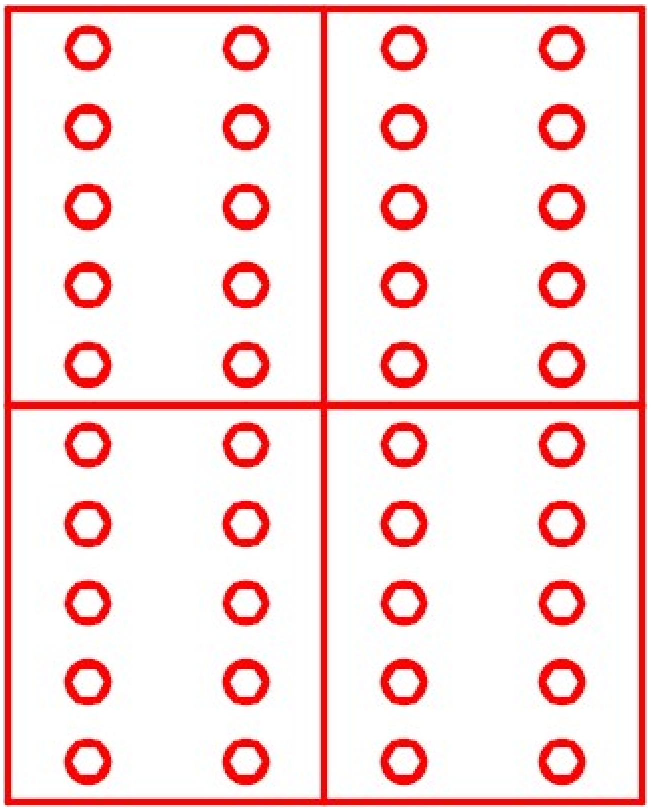


Figure Step 16

Step 17

Delete the lines that were duplicated.

AUTHOR'S COMMENTS: This time, you will have to delete two lines.

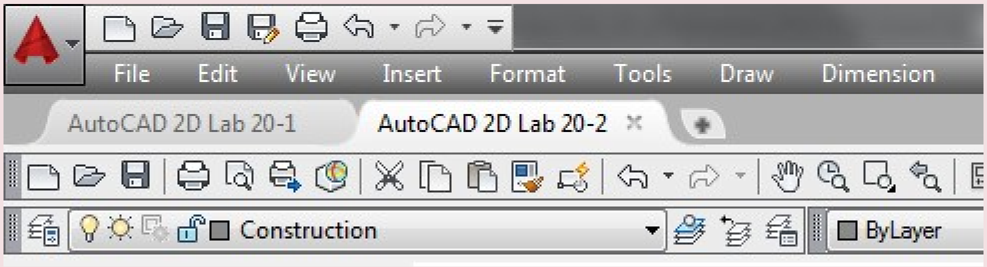
Step 18

Save and close the drawing.

AutoCAD Command: FILETAB

The FILETAB command enables the display of the open drawing file tabs above the Graphic window.

Shortcut: none


File
Edit
View
Insert
Format
Tools
Draw
Dimension
AutoCAD 2D Lab 20-1
AutoCAD 2D Lab 20-2
Construction
ByLayer

AutoCAD Command: FILETABCLOSE

The FILETABCLOSE command disables the display of the open drawing file tabs on top of the Graphic window.

Shortcut: **none**

WORKALONG: Using the COPYCLIP, PASTECLIP and COPYBASE Commands

Step 1

Start a NEW drawing using the template: 2D Layout English. Save and name the drawing: AutoCAD 2D Workalong 20-2.

Step 2

Using the OPEN command, open the drawing: AutoCAD 2D Workalong 20-1. (Figure Step 2)

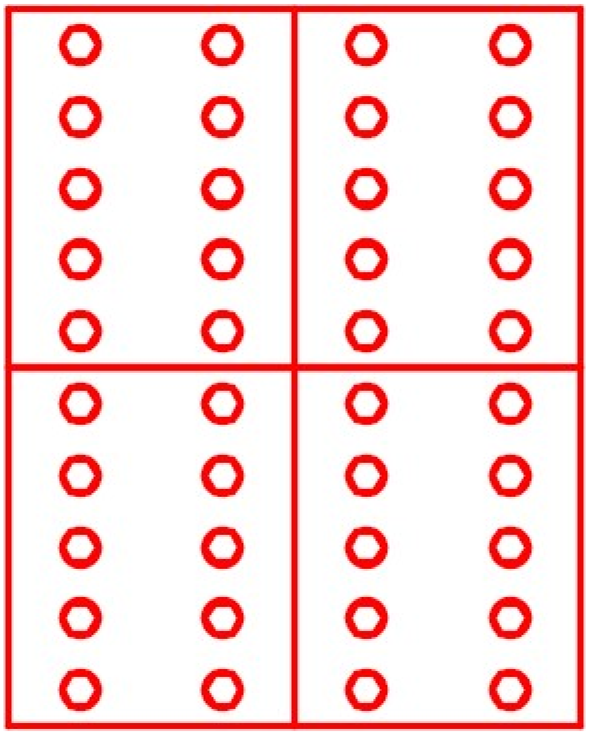


Figure Step 2

Step 3

If necessary, enter the FILETAB command to enable the display of the drawing file tabs on top of the Graphic window. You should see two tabs, one for each open drawing. Ensure that the drawing named: AutoCAD 2D Workalong 20-1 is the current drawing as shown in the figure. (Figure Step 3)

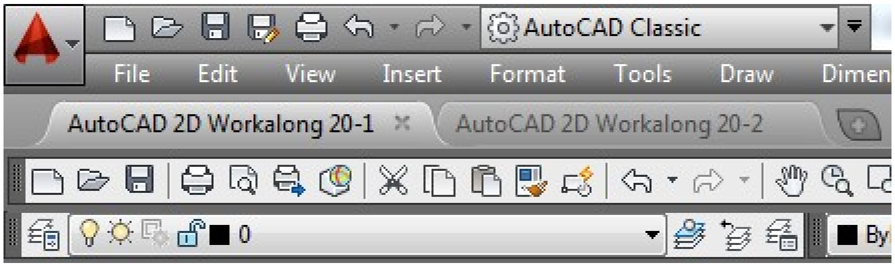

AutoCAD Classic
File
Edit
View
Insert
Format
Tools
Draw
Dimen
AutoCAD 2D Workalong 20-1
AutoCAD 2D Workalong 20-2
0
By

Figure Step 3

AUTHOR'S COMMENTS: You now have two drawings open at the same time in your current AutoCAD session. You can switch between them by selecting the current drawing tab.

Step 4

Without entering a command, use a window and select the objects as shown in the figure. (Figure Step 4)

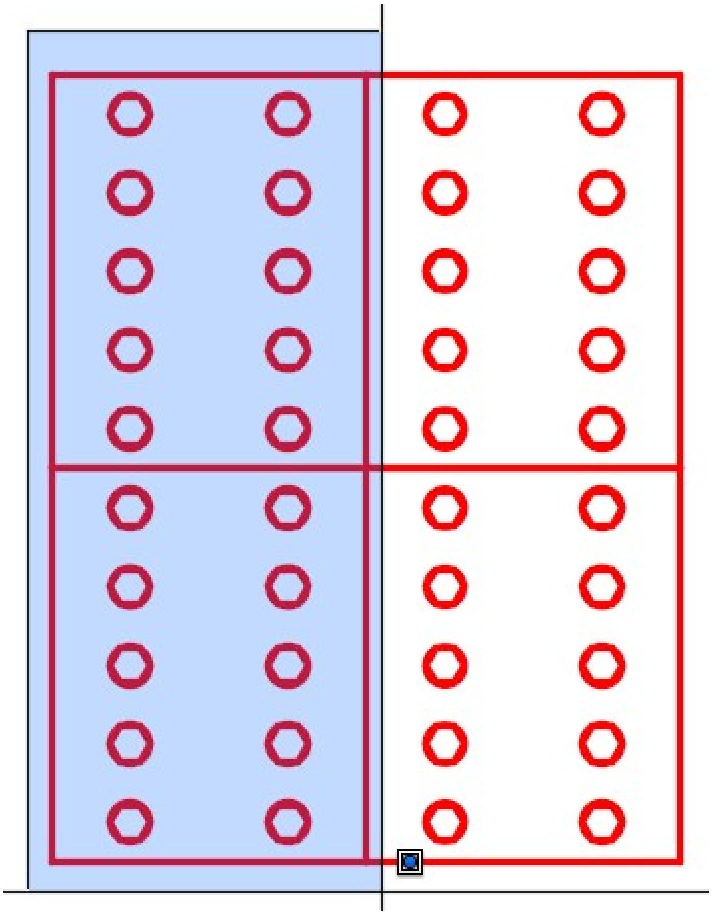


Figure Step 4

Step 5

Enter the COPYBASE command using the Pull-down menu or by entering in the command as shown below. Enter 0,0 for the base point. (Figure Step 5).

Command: **COPYBASE**

Specify base point: **0,0**

150 found

Command:

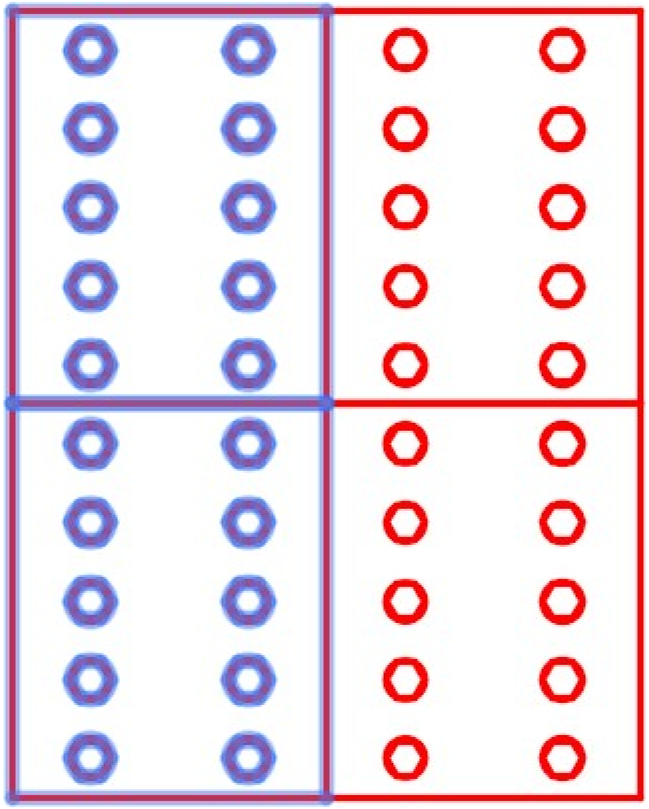


Figure Step 5

AUTHOR'S COMMENTS: The COPYBASE command copies the selected objects onto the clipboard.

Step 6

Click the tab to set the current drawing to: AutoCAD 2D Workalong 20-2. (Figure Step 6)

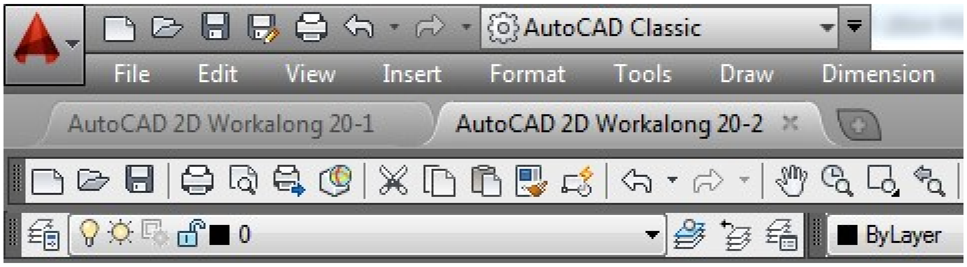

AutoCAD Classic
File
Edit
View
Insert
Format
Tools
Draw
Dimension
AutoCAD 2D Workalong 20-1
AutoCAD 2D Workalong 20-2
0
ByLayer

Figure Step 6

AUTHOR'S COMMENTS: The Graphic window should now appear blank.

Step 7

Enter the PASTECLIP command using the Pull-down menu or by entering the command as shown below. Enter 0,0 for the insertion point. (Figure Step 7).

Command: **PASTECLIP**

Specify insertion point: **0,0**

Command:

AUTHOR'S COMMENTS: By setting the insertion point at 0,0, the objects will be inserted in the exact location that they were in the drawing you are copying them from.

Step 8

Enter the ZOOM Extents command.

AUTHOR'S COMMENTS: By entering the ZOOM Extents command, all objects in the current drawing will be displayed in the Graphic window. By entering this command, you will ensure that you are viewing all on and thawed objects in the current drawing. It is important to keep this in mind while working on any AutoCAD drawing.

Step 9

Enter the COPYCLIP command and select all objects using a crossing window. (Figure Step 9A, 9B, and 9C)

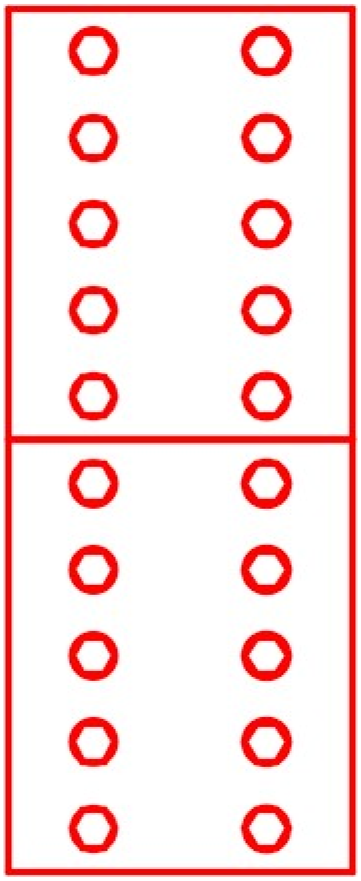


Figure Step 9A

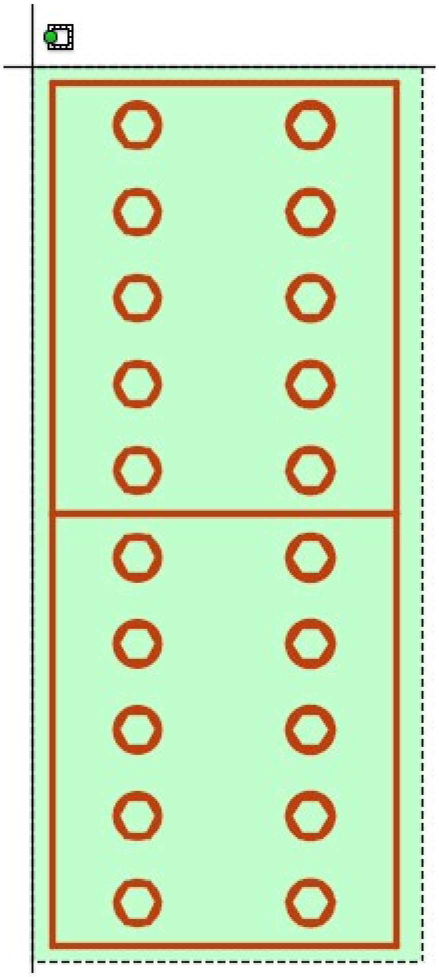


Figure Step 9B

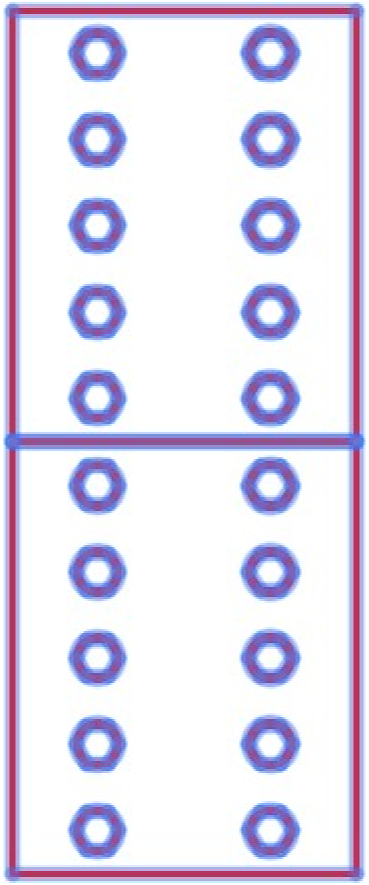


Figure Step 9C

AUTHOR'S COMMENTS: The COPYCLIP command copies the selected objects onto the clipboard.

Step 10

Enter the PASTECLIP command and when prompted for the Insertion point, select anywhere on the drawing. Match the location the best you can as shown in the figure. (Figure Step 10A, and 10B)

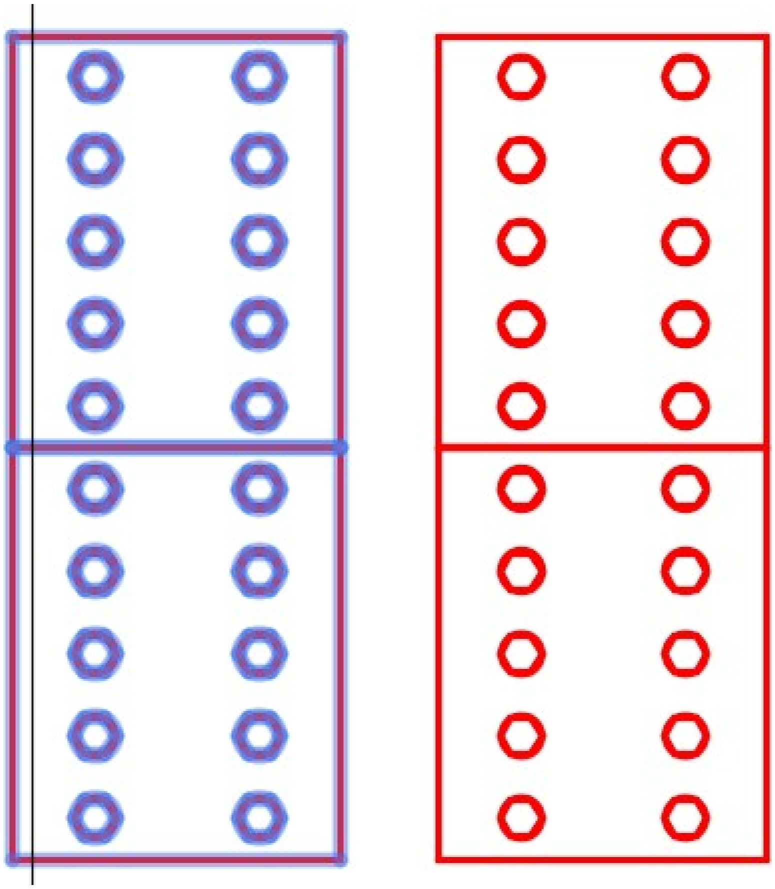


Figure Step 10A

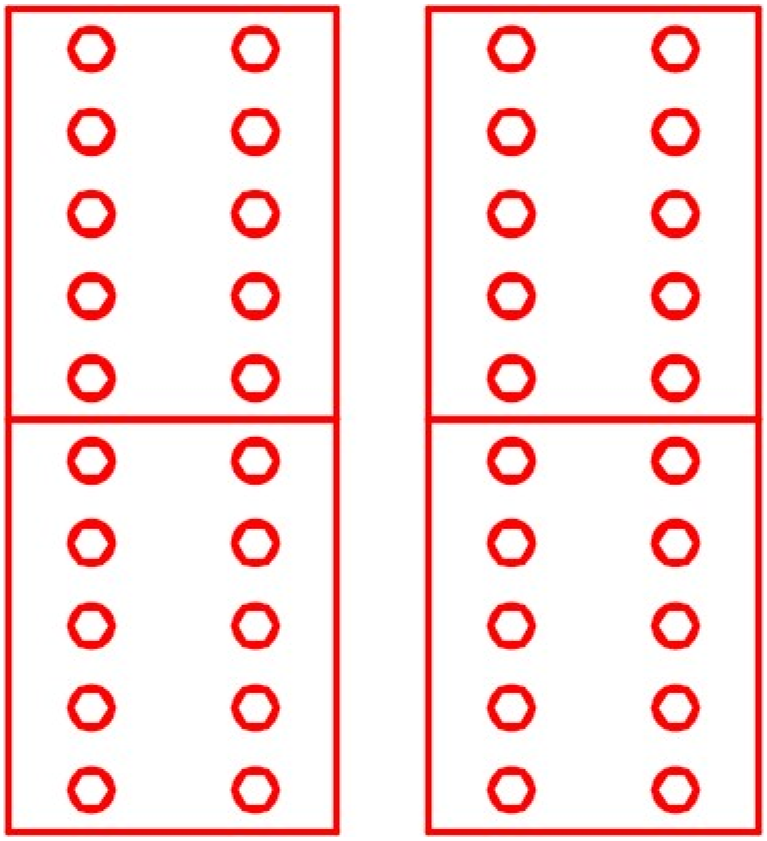


Figure Step 10B

Step 11

Using what you learned earlier in this module, move the objects on the right and snap the bottom left corner to the bottom right corner of the original objects. (Figure Step 11A, 11B, 11C, and 11D)

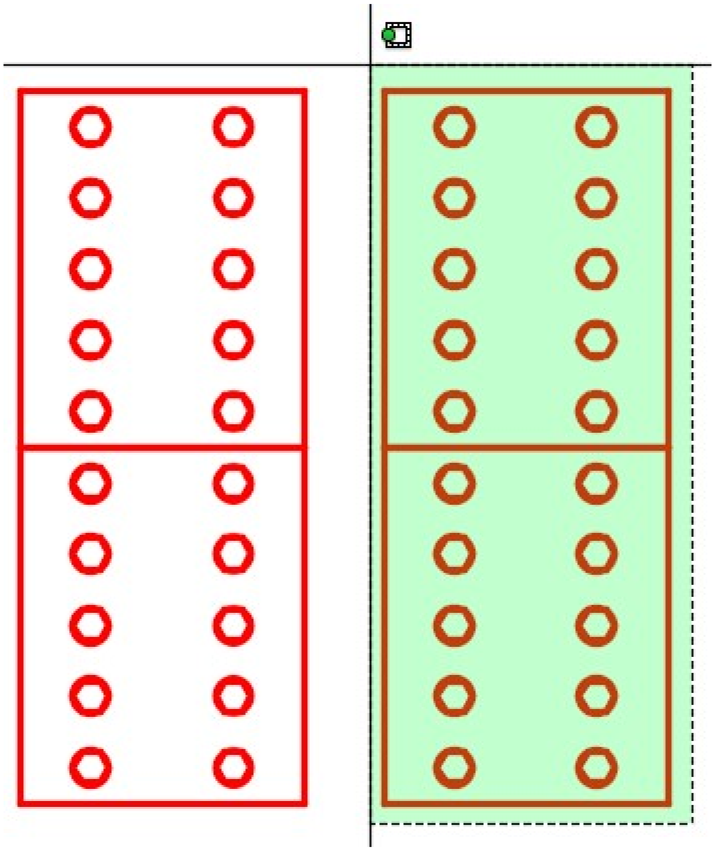


Figure Step 11A

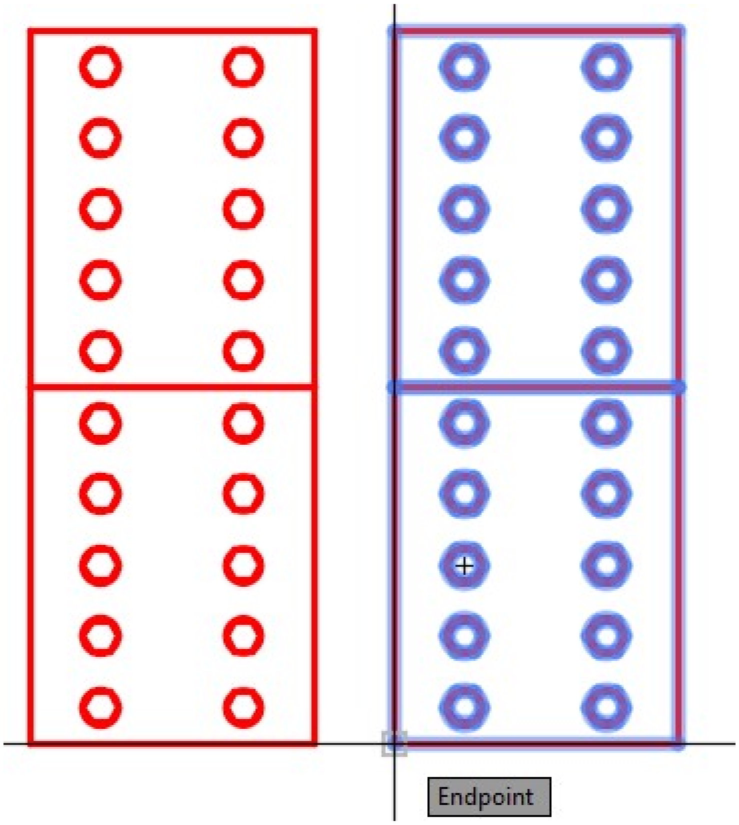


Figure Step 11B

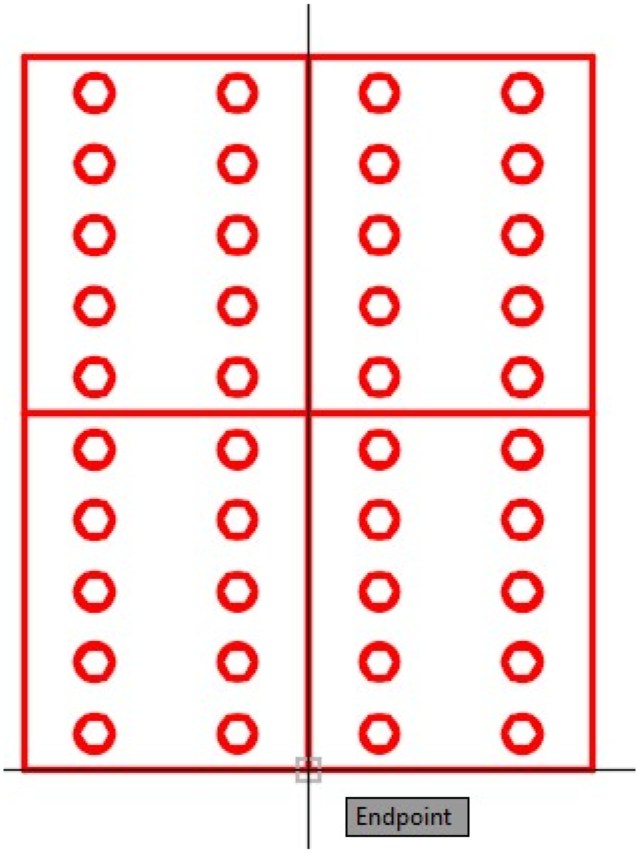


Figure Step 11C

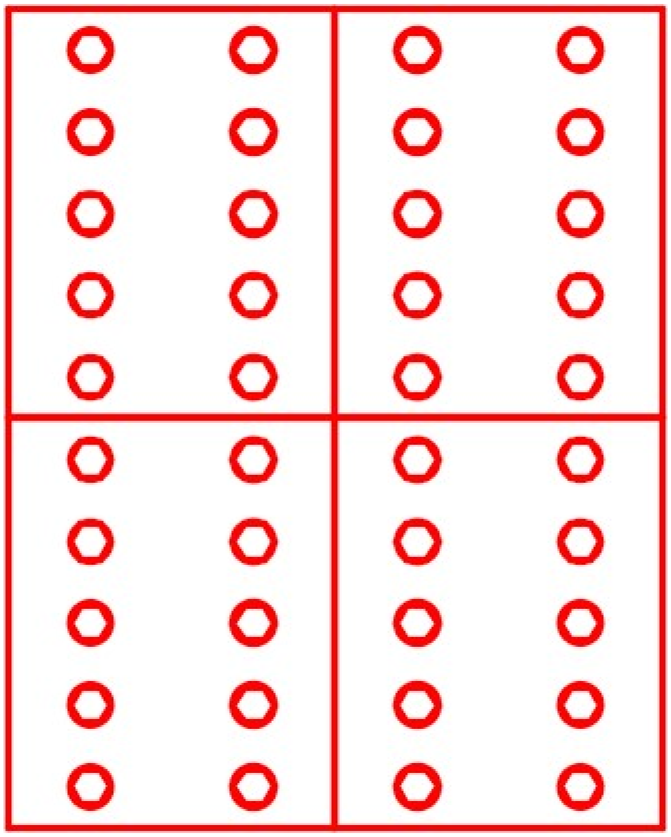


Figure Step 11D

Step 12

Delete the lines that were duplicated. (Figure step 12)

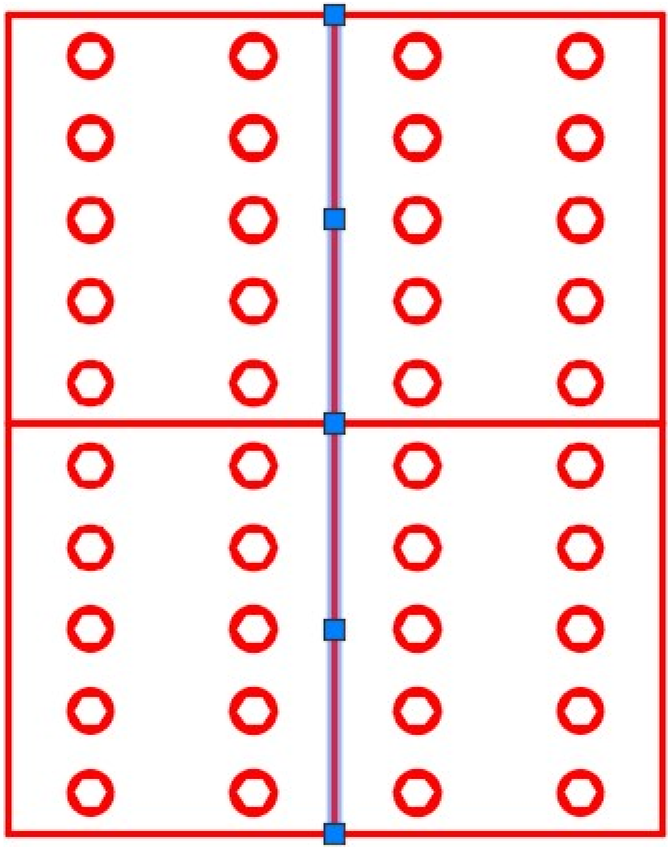


Figure Step 12

Step 13

Save and close the drawing.

MUST KNOW: The COPYBASE command is very important to the AutoCAD Operator who works in disciplines such as mapping and architectural. When copying objects from one drawing to another, the objects must be in the exact location as they were in the original drawing. Use the COPYBASE command with 0,0 as the base point and use the same coordinates when pasting it into the new drawing.

Geometry Lesson: Tangency – Part 2

When an arc must be constructed that has one end tangent to a line and other end going through a point, you cannot use the FILLET command. A fillet MUST be tangent at both ends. It must be drawn as an arc as follows:

Step 1

This is the arc to be drawn. (Figure Step 1)

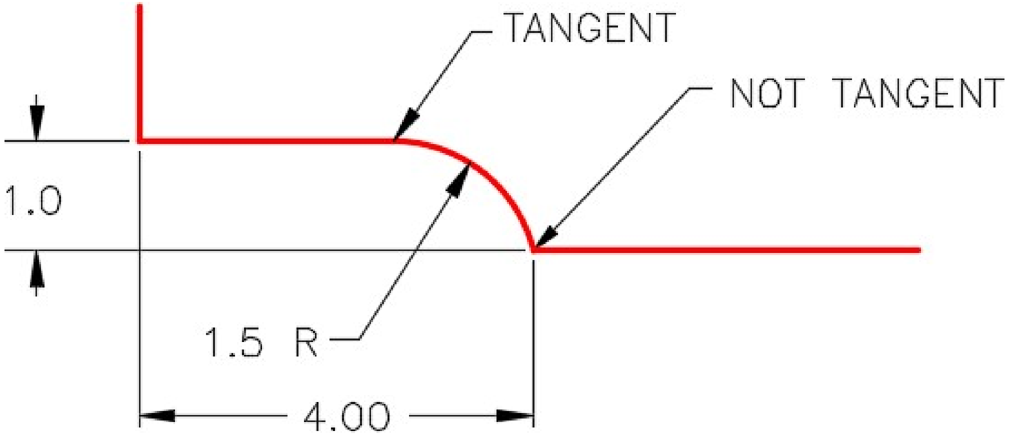

TANGENT
NOT TANGENT
1.0
1.5 R
4.00

Figure Step 1

Step 2

Draw the construction lines. (Figure Step 2)

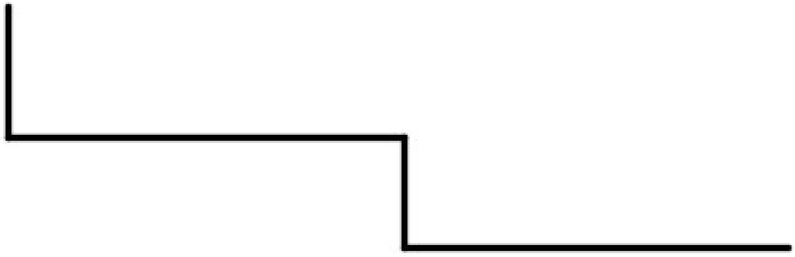


Figure Step 2

Step 3

Offset the line which is tangent to the arc at a distance of 1.5 units.1.5 is the radius of the arc being inserted. Draw a circle with the radius of 1.5 units with its center at the point where the arc ends. (Figure Step 3)

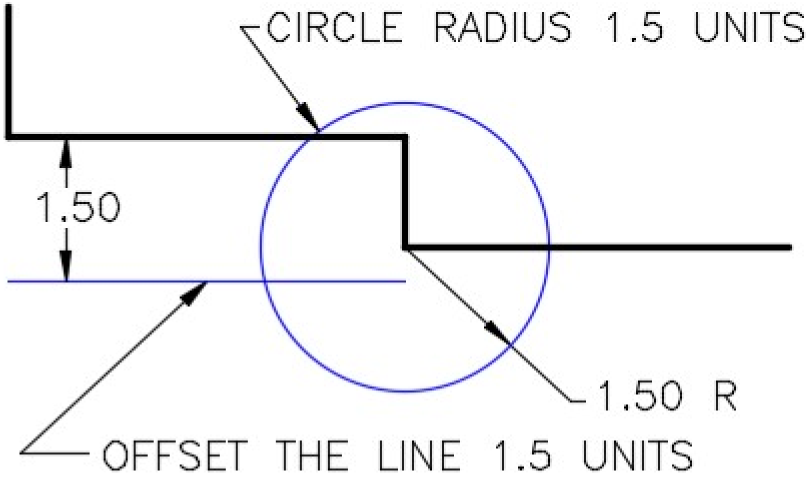

CIRCLE RADIUS 1.5 UNITS
1.50
1.50 R
OFFSET THE LINE 1.5 UNITS

Figure Step 3

Step 4

Draw the circle with a radius of 1.5. Its center is the intersection of the offset line and the circle. (Figure Step 4)

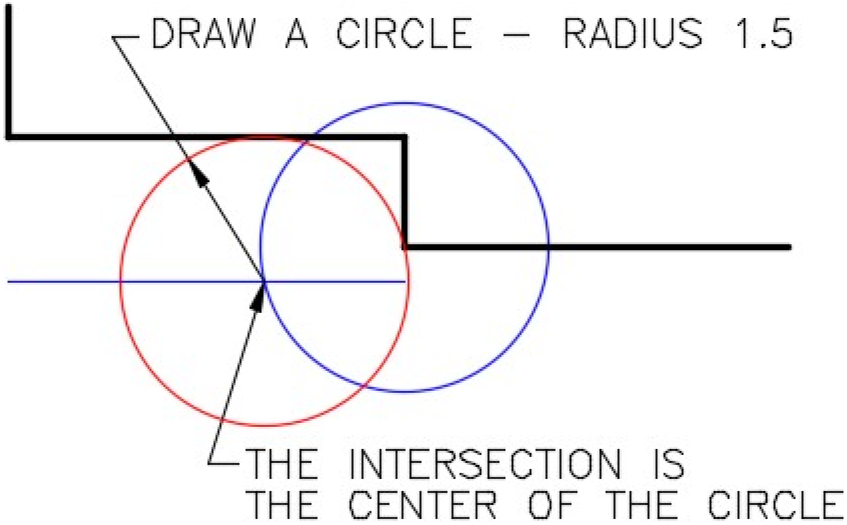

DRAW A CIRCLE – RADIUS 1.5
THE INTERSECTION IS
THE CENTER OF THE CIRCLE

Figure Step 4

Step 5

Trim the circle you drew in Step 4 and turn layer Construction off. (Figure Step 5)

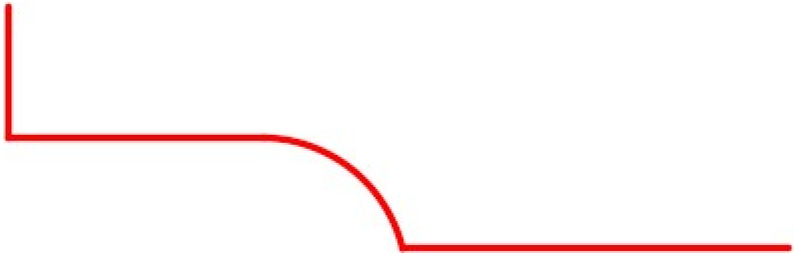


Figure Step 5

AutoCAD book Standards: Filling in the Titleblock

When filling in the titleblock in the workalongs and lab exercises in the AutoCAD 2D book, use the following standards:

1. Set Titleblock as the current text style.
2. Set layer Titleblock Text as the current layer.
3. Use a text height that matches the area being filling. You can pick a text height to use that looks pleasing to your eye. See the figures below for approximate text heights.
4. Center the text by eye. It helps to use middle justified. Use the MOVE and ORTHO commands to make the job easier.
5. Fill in the bottom row with your actual name, the actual date, scale, drawing number, and sheet number.
6. The format of the date is: Month-Day-Year, MM-DD-YYYY For example: 01-20-2014 (For January 20, 2014)

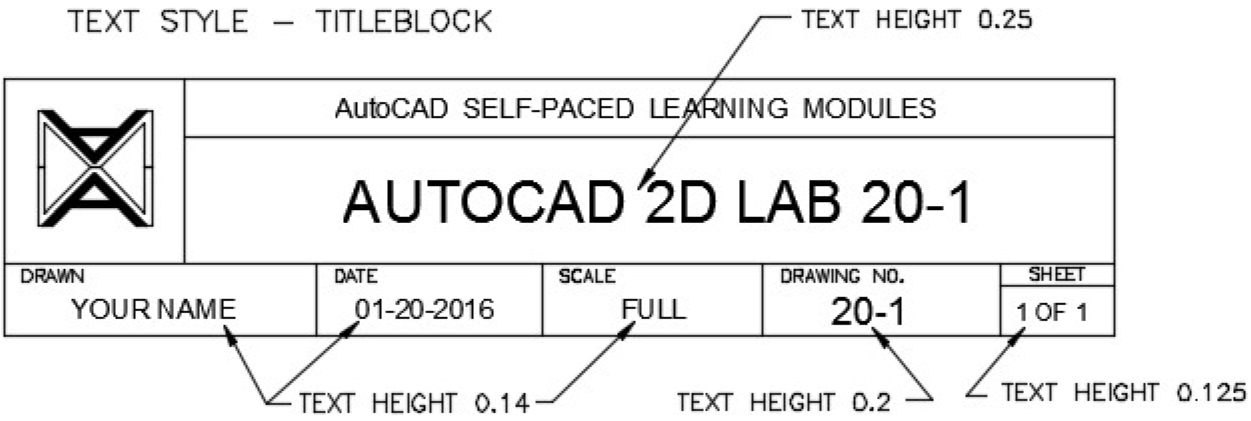

TEXT STYLE – TITLEBLOCK
TEXT HEIGHT 0.25
AutoCAD SELF-PACED LEARNING MODULES
AUTOCAD 2D LAB 20-1
DRAWN
YOUR NAME
DATE
01-20-2016
SCALE
FULL
DRAWING NO.
20-1
SHEET
1 OF 1
TEXT HEIGHT 0.14
TEXT HEIGHT 0.2
TEXT HEIGHT 0.125

Figure 20-1

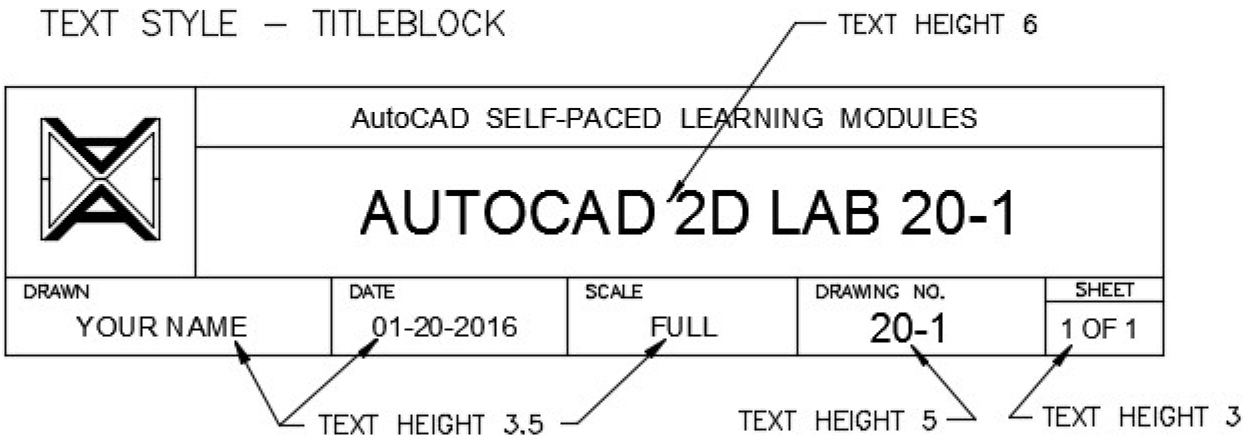

TEXT STYLE – TITLEBLOCK
TEXT HEIGHT 6
AutoCAD SELF-PACED LEARNING MODULES
AUTOCAD 2D LAB 20-1
DRAWN
YOUR NAME
DATE
01-20-2016
SCALE
FULL
DRAWING NO.
20-1
SHEET
1 OF 1
TEXT HEIGHT 3.5
TEXT HEIGHT 5
TEXT HEIGHT 3

Figure 20-2

WORKALONG: Construction Techniques

Step 1

Start a new drawing using the template: 2D Layout English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 20-3.

Step 3

Set layer: Construction as the current layer. Using the dimensioned drawing as a reference, draw the lines and circle to match the figure. (Figure Step 3A and 3B)

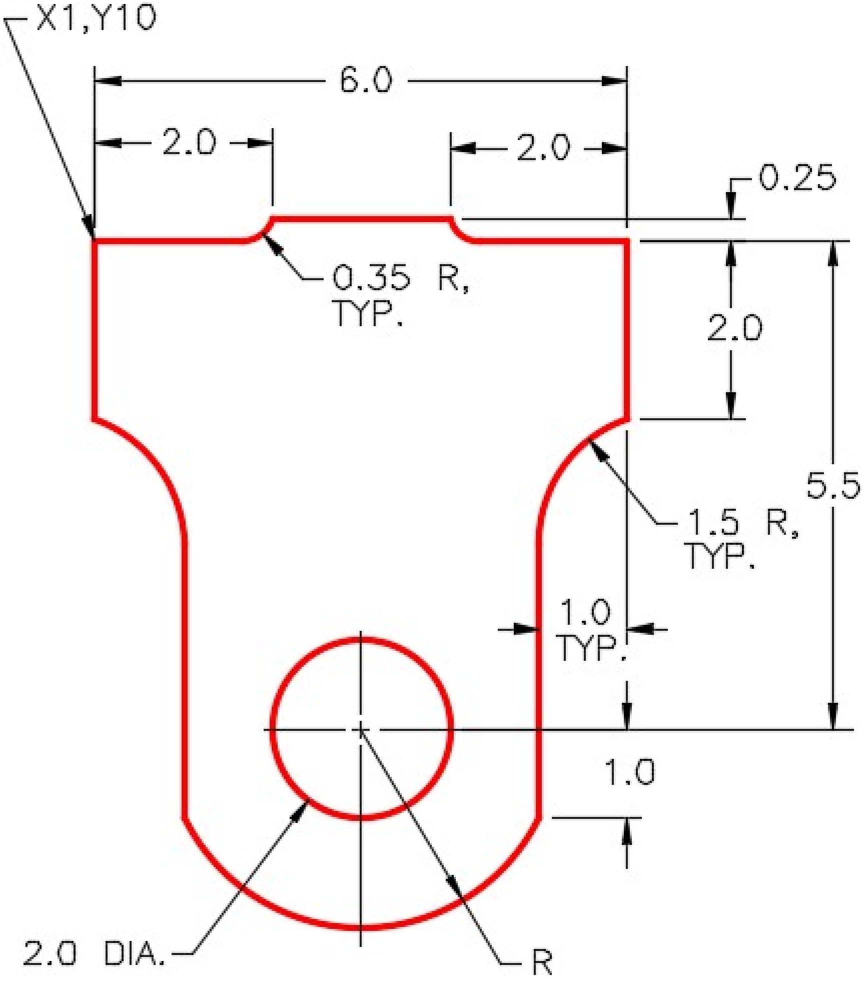

X1,Y10
6.0
2.0
2.0
0.25
0.35 R,
TYP.
2.0
5.5
1.5 R,
TYP.
1.0
TYP.
1.0
2.0 DIA.
R

Figure Step 3A

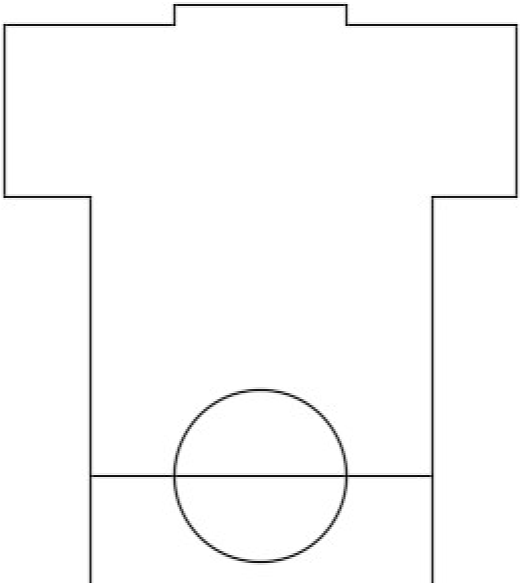


Figure Step 3B

Step 4

Change the layer of the object lines and the circle to layer: Object. (Figure Step 4)

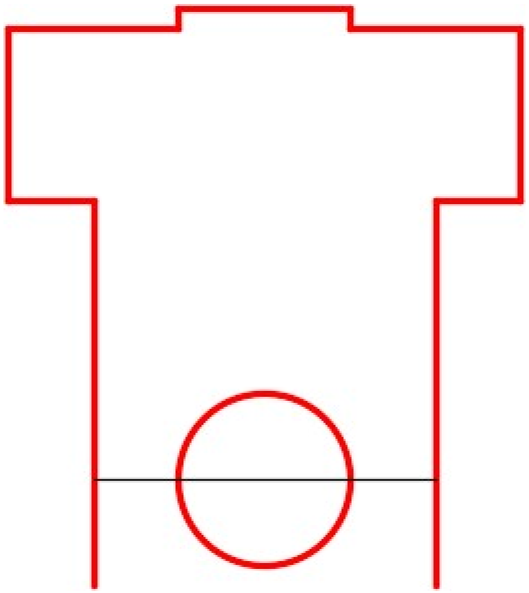


Figure Step 4

Step 5

Draw a circle snapping its center location to the center of the circle and snap to the end of the line to specify the radius. Trim the circle. (Figure Step 5A and 5B)

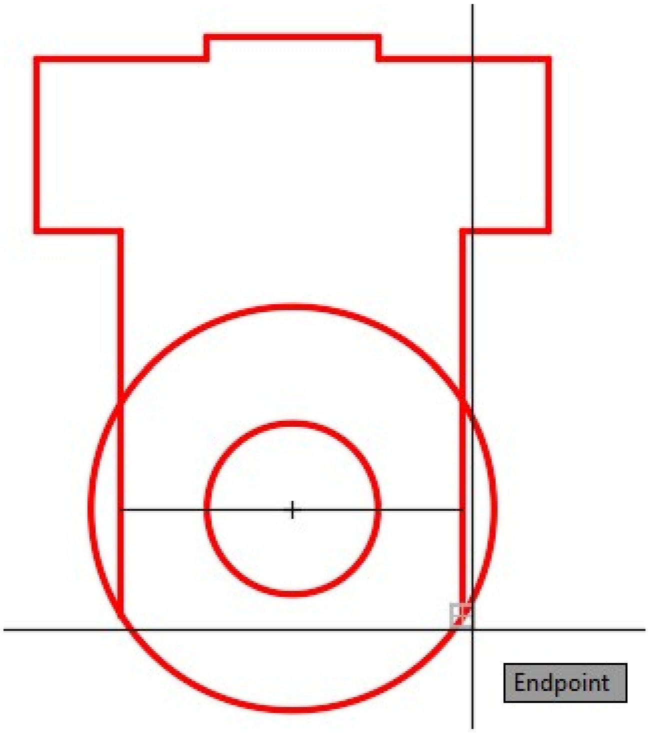


Figure Step 5A

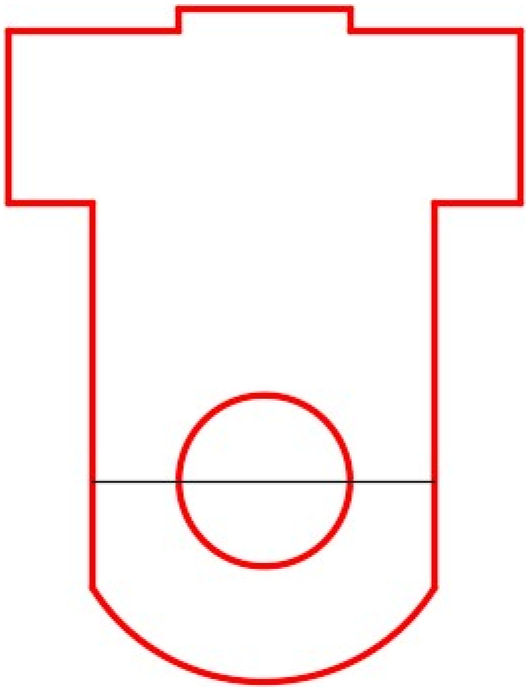


Figure Step 5B

AUTHOR'S COMMENTS: Since you do not know the actual radius of the arc, show AutoCAD the radius by snapping to the end of the line rather then entering a number.

Step 6

Offset the line, 1.5 inches, on the right side. Change the layer of the line to layer: Construction. On layer: Construction, draw a circle with a 1.5 radius and snap its center to the end of the line as shown in the figure. (Figure Step 6A, 6B, and 6C)

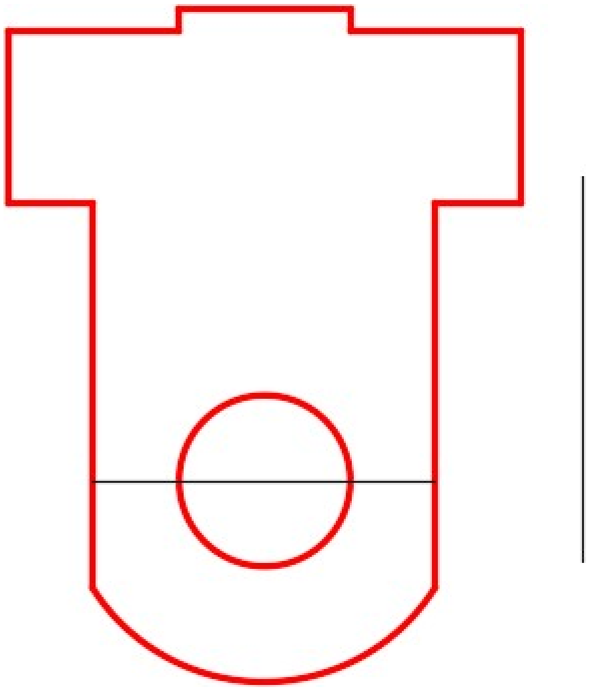


Figure Step 6A

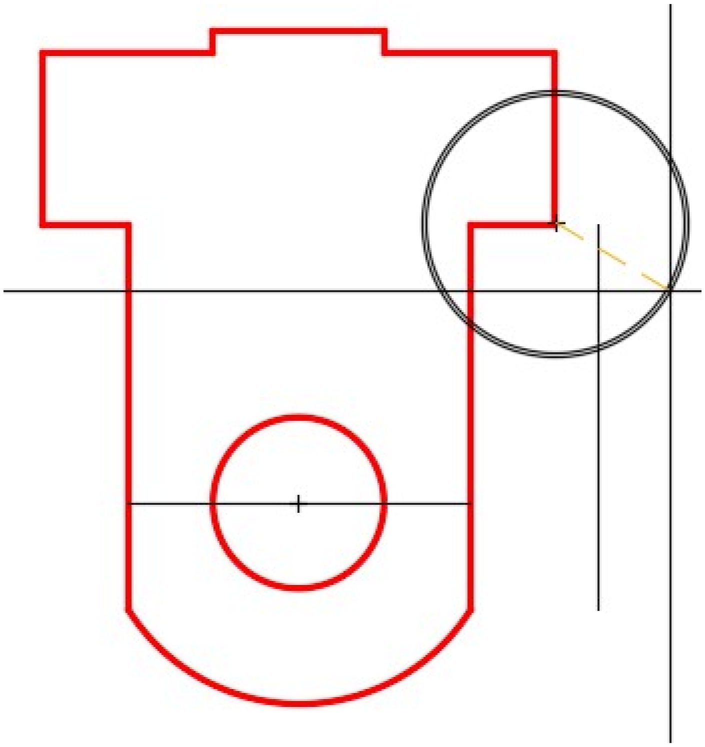


Figure Step 6B

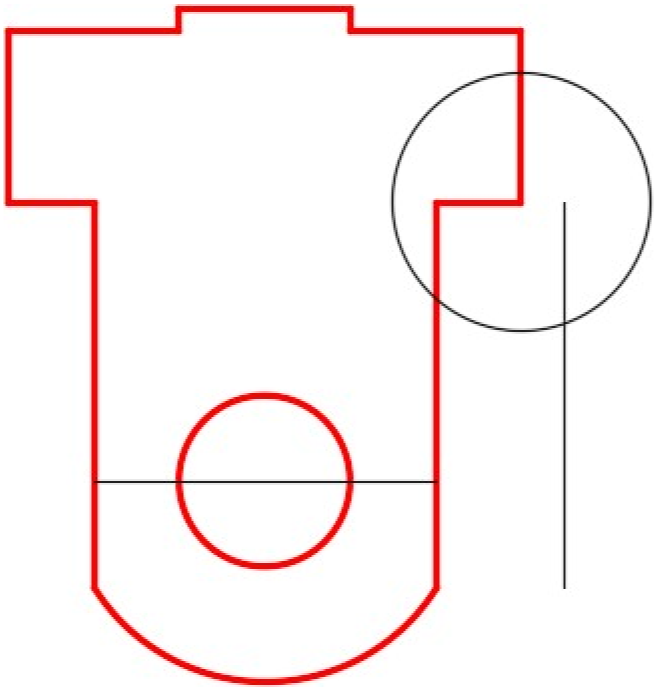


Figure Step 6C

AUTHOR'S COMMENTS: The offset distance and the radius of the circle inserted in Step 6 is set 1.5 inches since that is the radius of finished arc.

Step 7

On layer: Object, draw a circle by snapping its center to the intersection of the construction circle and offset line. Show AutoCAD the radius, of 1.5, by snapping to the end of the line. (Figure Step 7A and 7B)

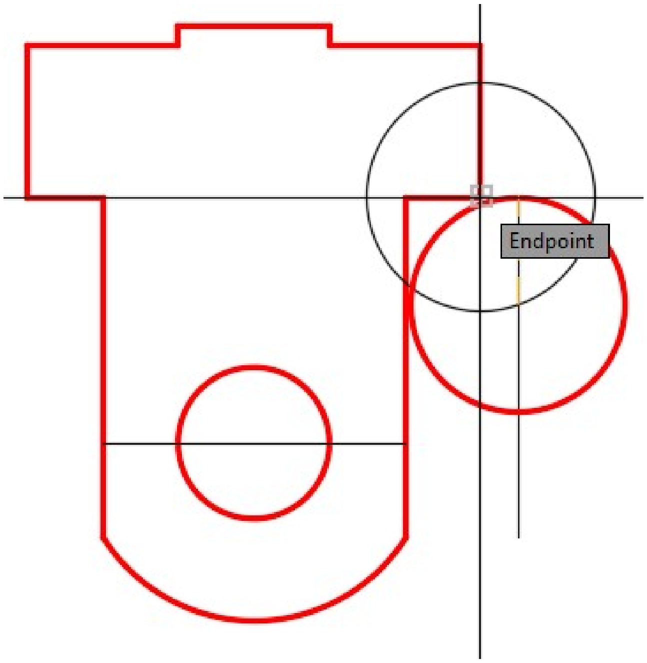


Figure Step 7A

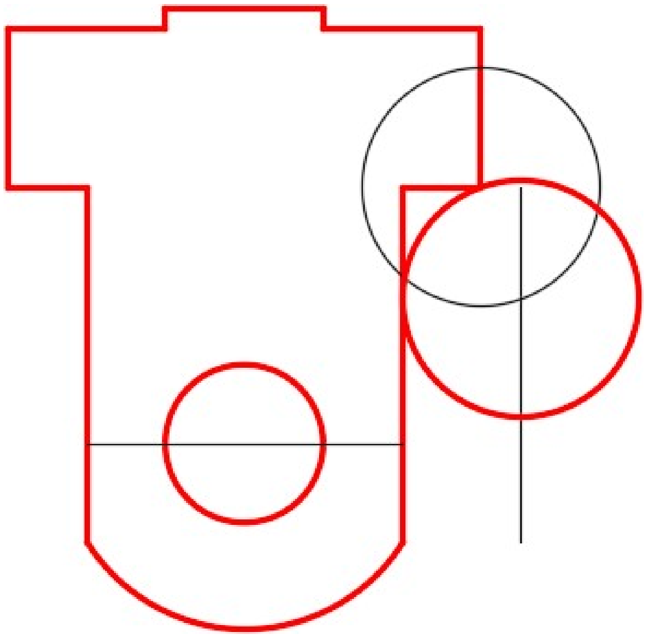


Figure Step 7B

Step 8

Trim the circle first. Delete one line and trim the other line. (Figure Step 8)

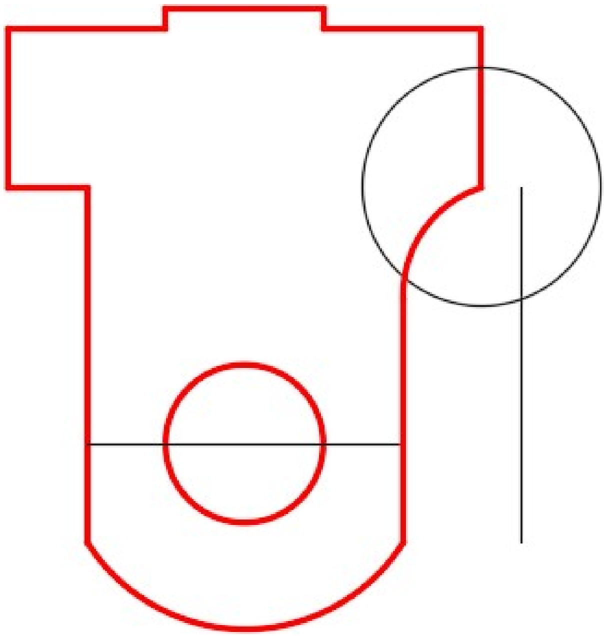


Figure Step 8

Step 9

Offset the line on the top 0.35 inches. Change the layer of the line to layer: Construction. On layer: Construction, draw a circle with a 0.35 inch radius. Snap its center to the end of the line as shown in the figure. (Figure Step 9A and 9B)

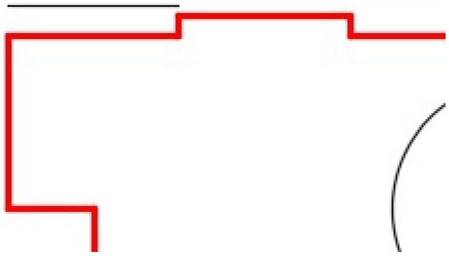


Figure Step 9A

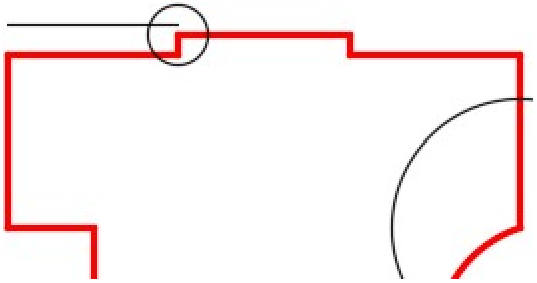


Figure Step 9B

Step 10

On layer: Object, draw a circle by snapping its center to the intersection of the construction circle and construction line. Show AutoCAD the radius, of 0.35, by snapping to the end of the line as shown in the figure. (Figure Step 10)

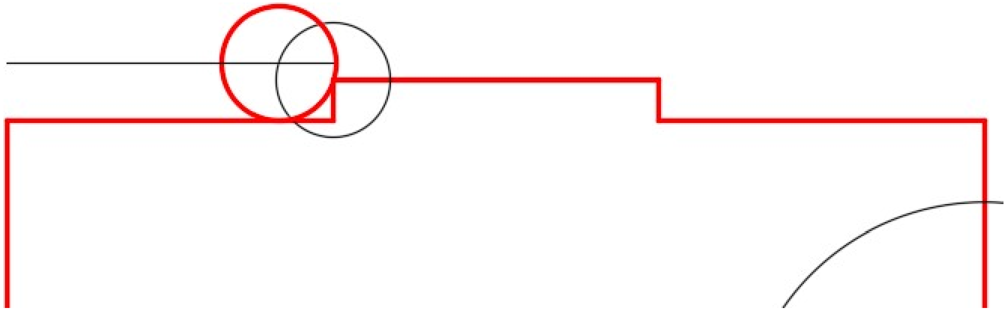


Figure Step 10

Step 11

Delete one line and trim the other line. Trim the circle that you drew in Step 10. (Figure Step 11)

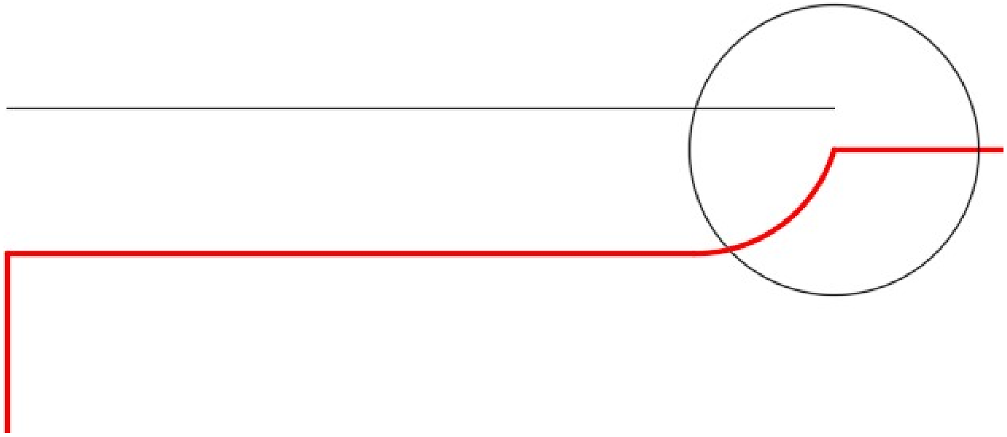


Figure Step 11

Step 12

Your drawing should match the figure. (Figure Step 12)

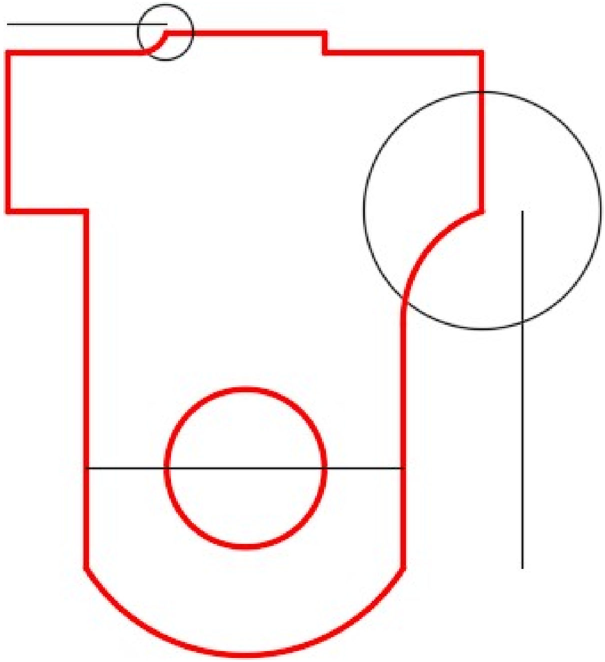


Figure Step 12

Step 13

Using what you just learned, draw the arcs on the opposite sides. (Figure Step 13)

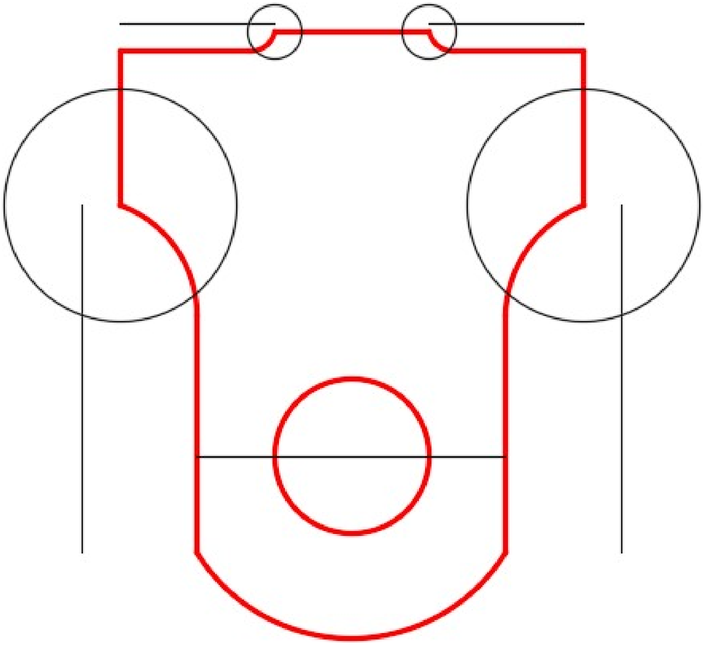


Figure Step 13

Step 14

Turn layer: Construction off. Your completed object should match the figure. (Figure Step 14)

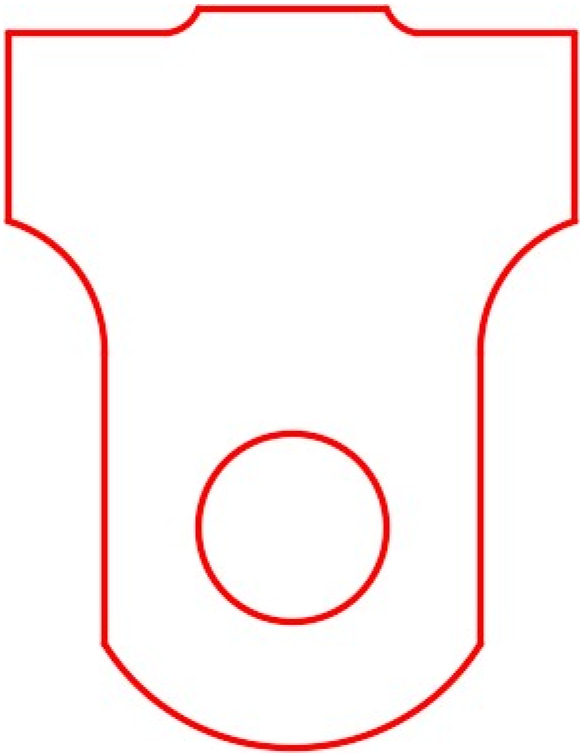


Figure Step 14

Step 15

Enable layout: Module Layout A. (Figure Step 15)

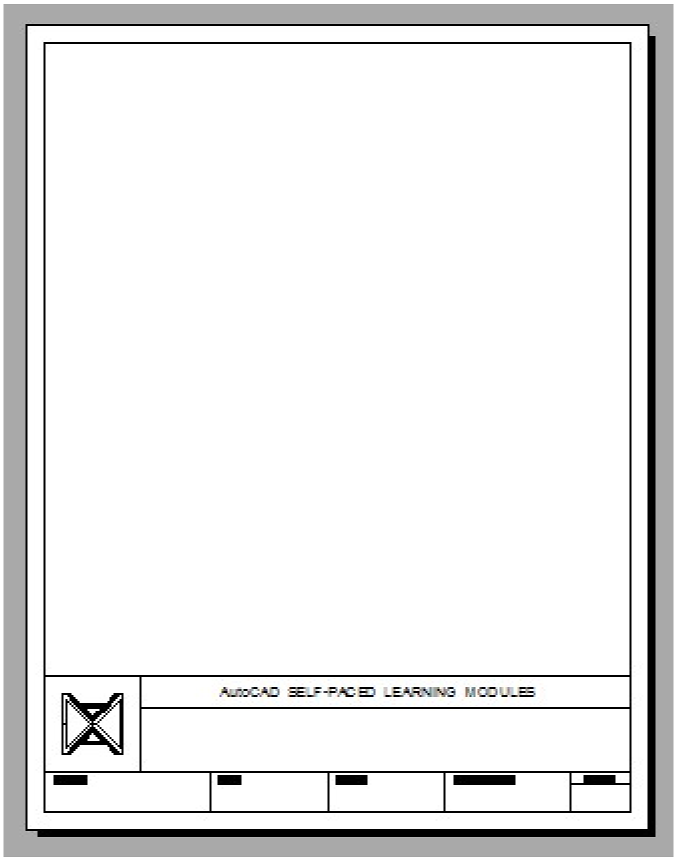

AutoCAD SELF-PACED LEARNING MODULES

Figure Step 15

Step 16

On layer: Viewport, use the MVIEW command and create a viewport as shown in the figure. Switch to Model space and enter the ZOOM Extents command. (Figure Step 16)

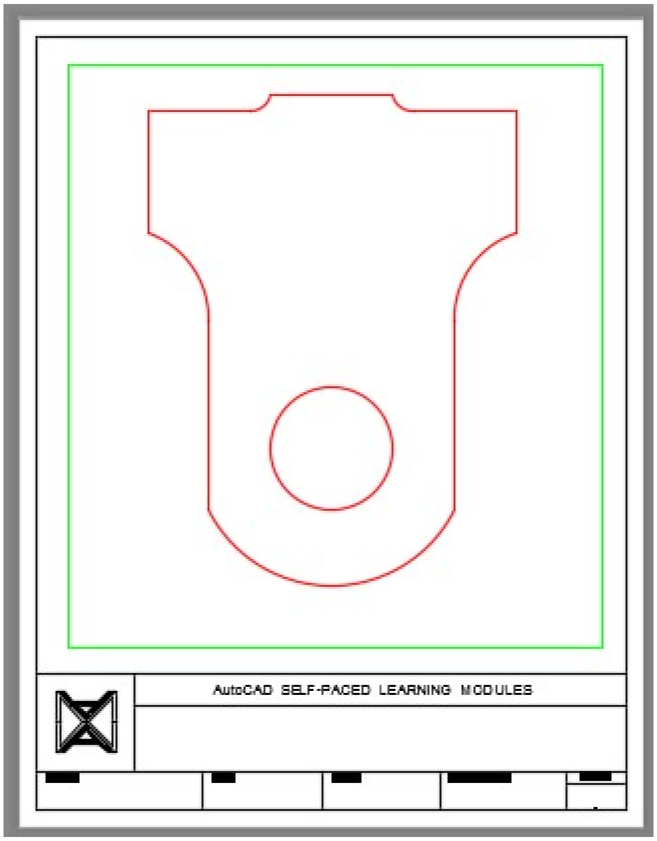
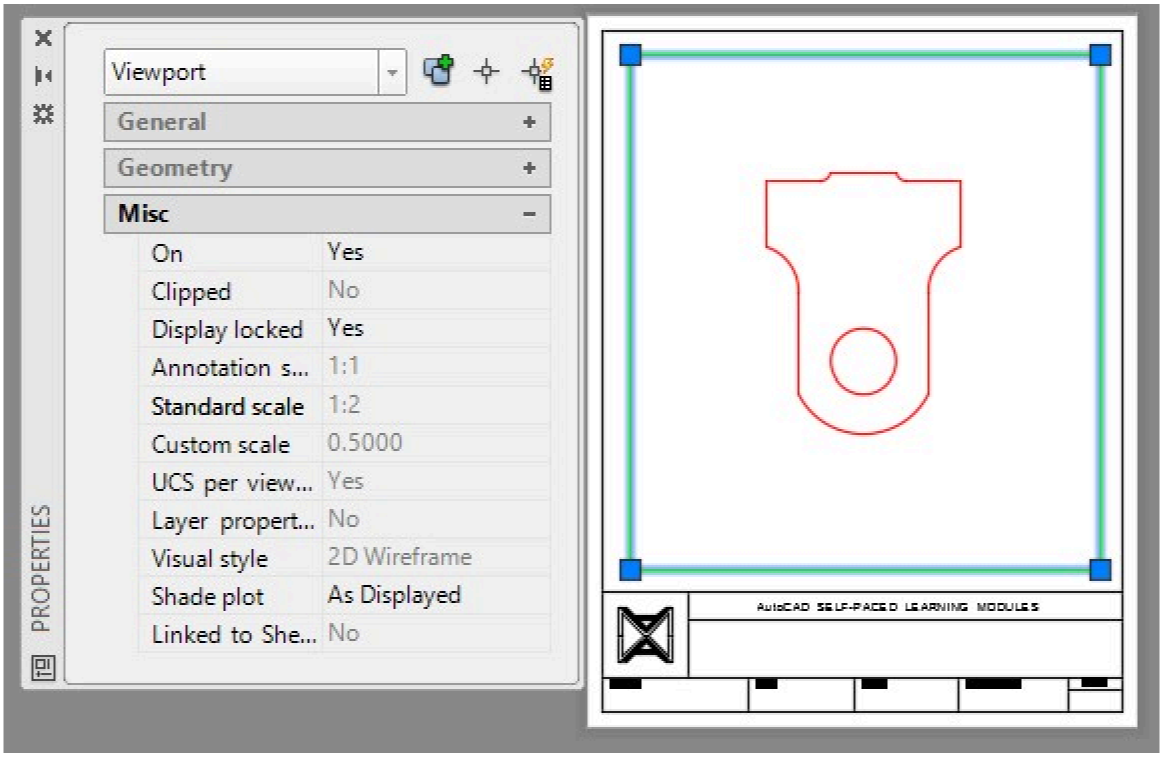

AutoCAD SELF-PACED LEARNING MODULES

Figure Step 16

Step 17

With paper space as working space, open the Properties window. Select the viewport and set the Standard scale to 1:2 and lock its display. (Figure Step 17)



Viewport

General +

Geometry +

Misc −

On	Yes
Clipped	No
Display locked	Yes
Annotation s...	1:1
Standard scale	1:2
Custom scale	0.5000
UCS per view...	Yes
Layer propert...	No
Visual style	2D Wireframe
Shade plot	As Displayed
Linked to She...	No

PROPERTIES
AutoCAD SELF-PACED LEARNING MODULES

Figure Step 17

Step 18

Turn layer: Viewport off. (Figure Step 18)

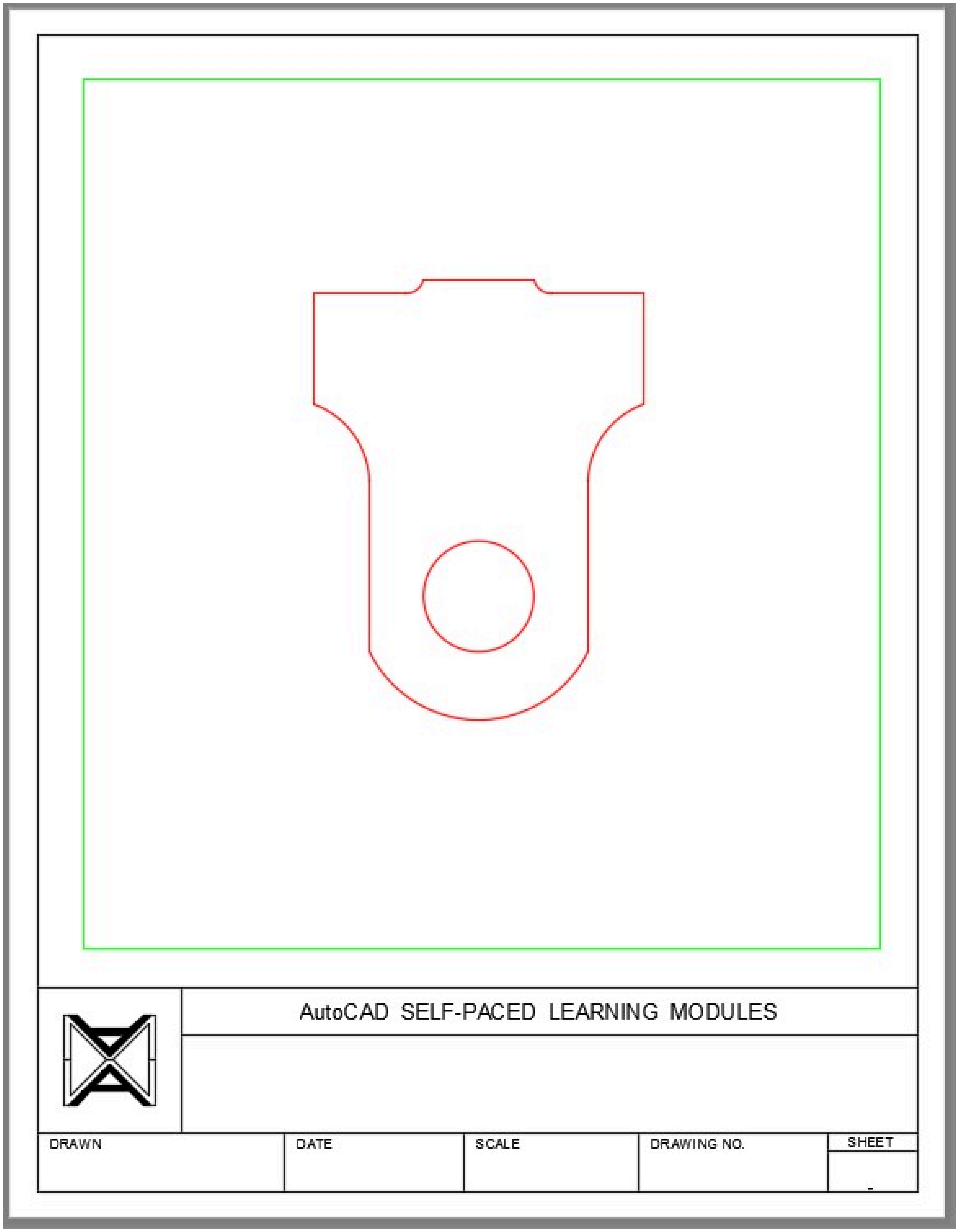

AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO.
SHEET
-

Figure Step 18

Step 19

Set layer: Titleblock Text as the current layer and set the current text style: Titleblock. Zoom in on the titleblock. (Figure Step 19)

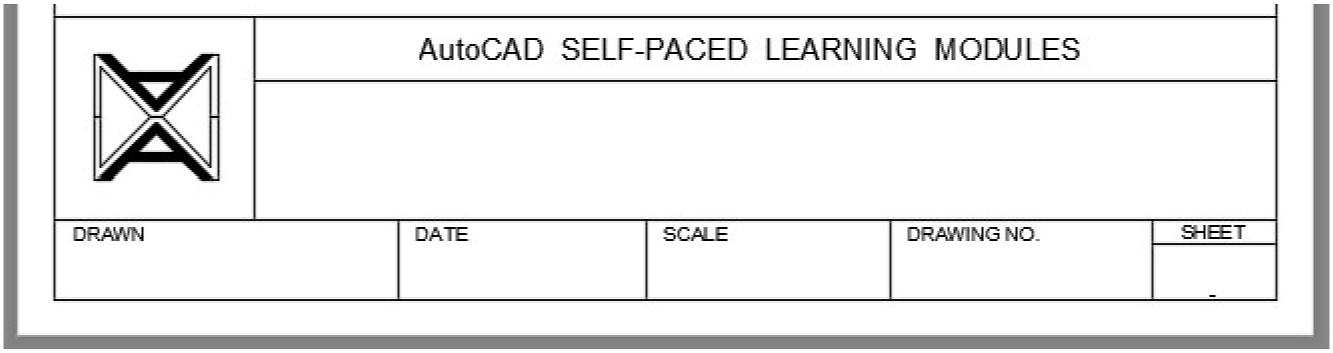

AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO.
SHEET
-

Figure Step 19

AUTHOR'S COMMENTS: There are many techniques to filling out the titleblock fast and efficiently as possible. Since it is text, it can be inserted by eye and moved around to make it look good. The following is my technique. Over time, you may find one of your own that works better for you.

Step 20

With the Paper space the current space, enter the DTEXT command and insert the drawing title text at a height of 0.25 using middle justified. Select a location by eye. It does not have to be placed exactly as you will be moving it later. (Figure Step 20)

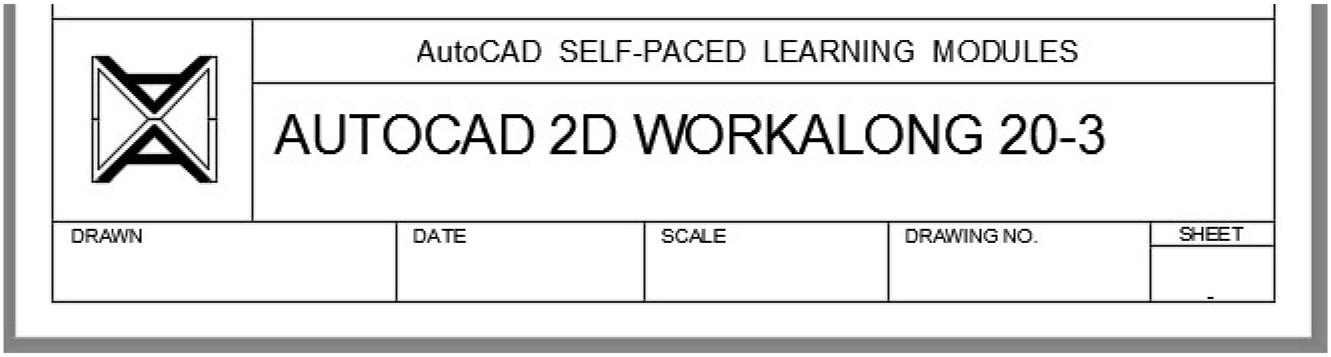

AutoCAD SELF-PACED LEARNING MODULES
AUTOCAD 2D WORKALONG 20-3
DRAWN
DATE
SCALE
DRAWING NO.
SHEET
-

Figure Step 20

Step 21

Using the MOVE command, move the text locating it by eye to center it horizontally and vertically. (Figure Step 21)

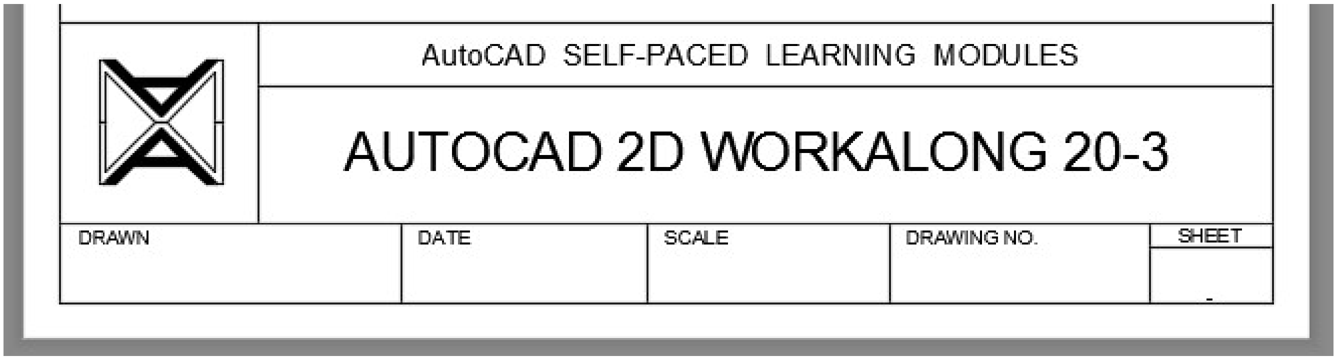

AutoCAD SELF-PACED LEARNING MODULES
AUTOCAD 2D WORKALONG 20-3
DRAWN
DATE
SCALE
DRAWING NO.
SHEET

Figure Step 21

Step 22

Enter the DTEXT command enter the text TEXT at a height of 0.14. Center it by eye using the MOVE command. (Figure Step 22)

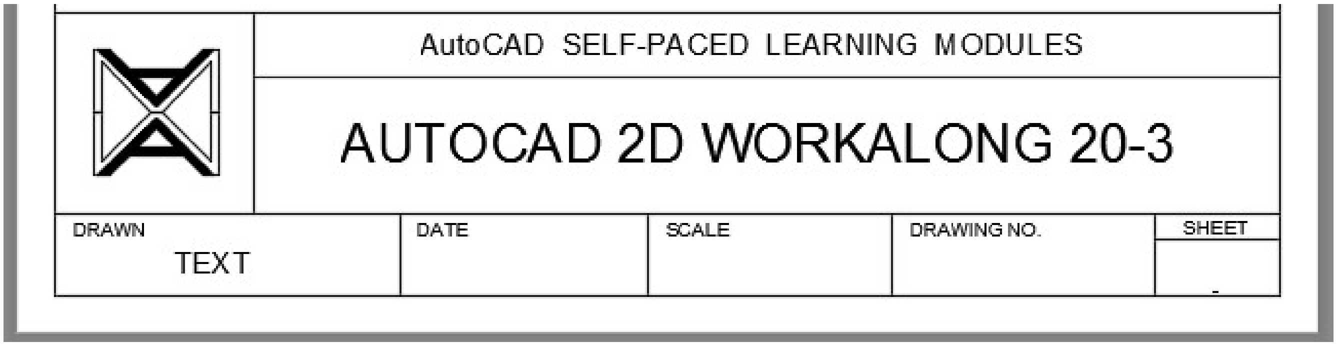

AutoCAD SELF-PACED LEARNING MODULES
AUTOCAD 2D WORKALONG 20-3
DRAWN
TEXT
DATE
SCALE
DRAWING NO.
SHEET

Figure Step 22

Step 23

Use the COPY command and Ortho mode, copy the text to the next two locations. Disable Ortho mode and copy the text into the two locations on the right and center the text by eye. (Figure Step 23)

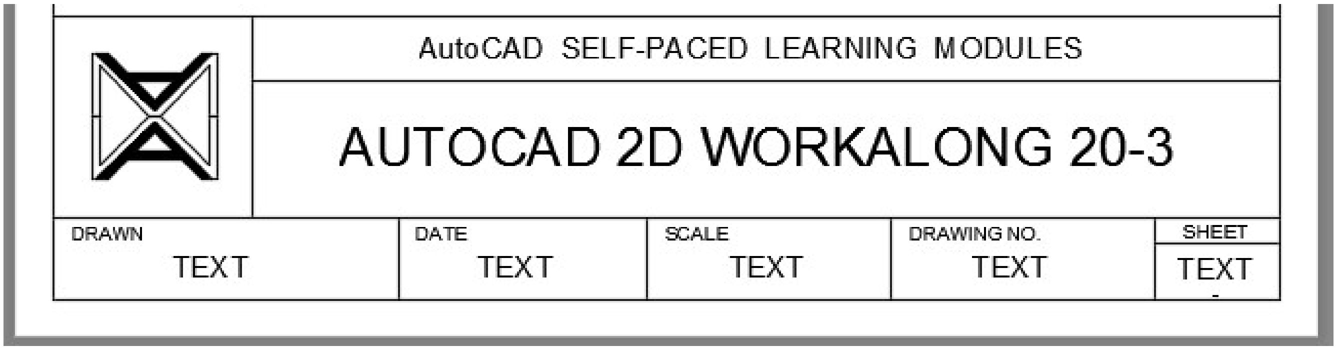
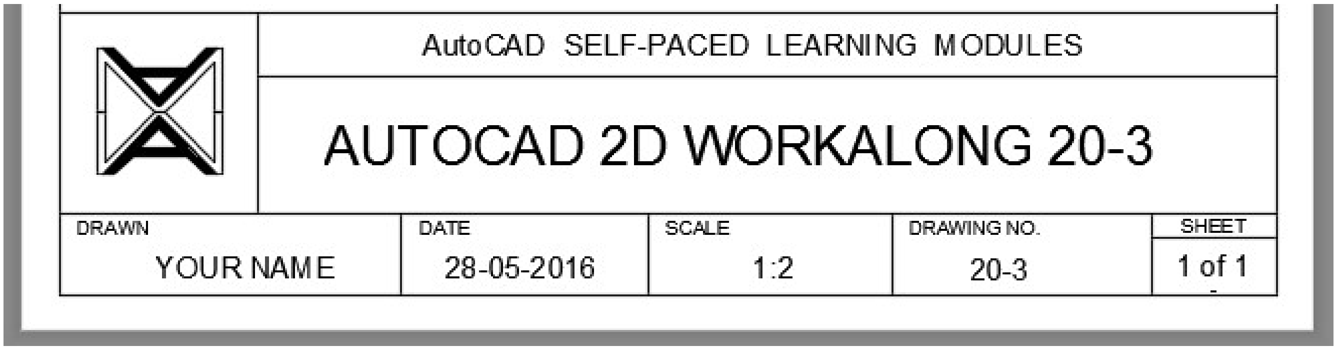

AutoCAD SELF-PACED LEARNING MODULES
AUTOCAD 2D WORKALONG 20-3
DRAWN
TEXT
DATE
TEXT
SCALE
TEXT
DRAWING NO.
TEXT
SHEET
TEXT

Figure Step 23

Step 24

Use the text editor to edit the text. (Figure Step 24)



AutoCAD SELF-PACED LEARNING MODULES

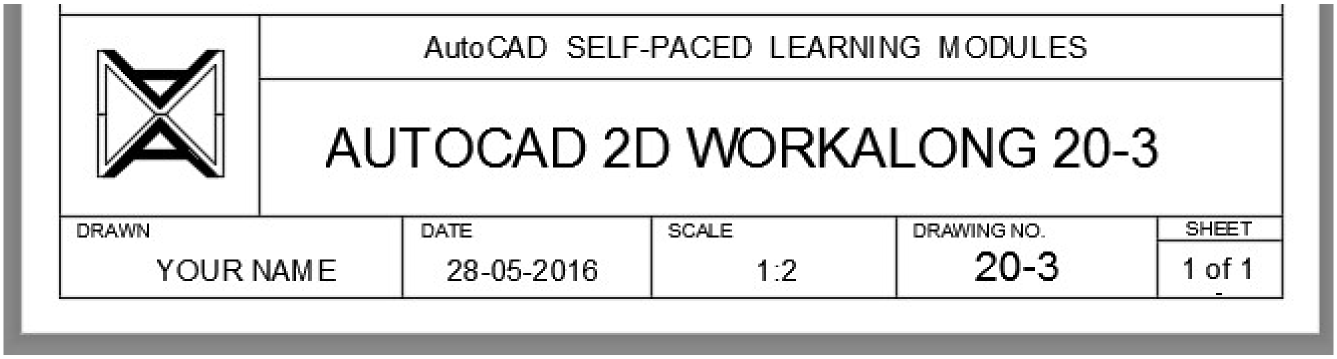
AUTOCAD 2D WORKALONG 20-3

DRAWN	DATE	SCALE	DRAWING NO.	SHEET
YOUR NAME	28-05-2016	1:2	20-3	1 of 1

Figure Step 24

Step 25

If required, change the text height to match the standards under “AutoCAD book Standards: Filling in the Titleblock”. (Figure Step 25)



AutoCAD SELF-PACED LEARNING MODULES

AUTOCAD 2D WORKALONG 20-3

DRAWN	DATE	SCALE	DRAWING NO.	SHEET
YOUR NAME	28-05-2016	1:2	20-3	1 of 1

Figure Step 25

Step 26

Your completed drawing should the figure. (Figure Step 26)

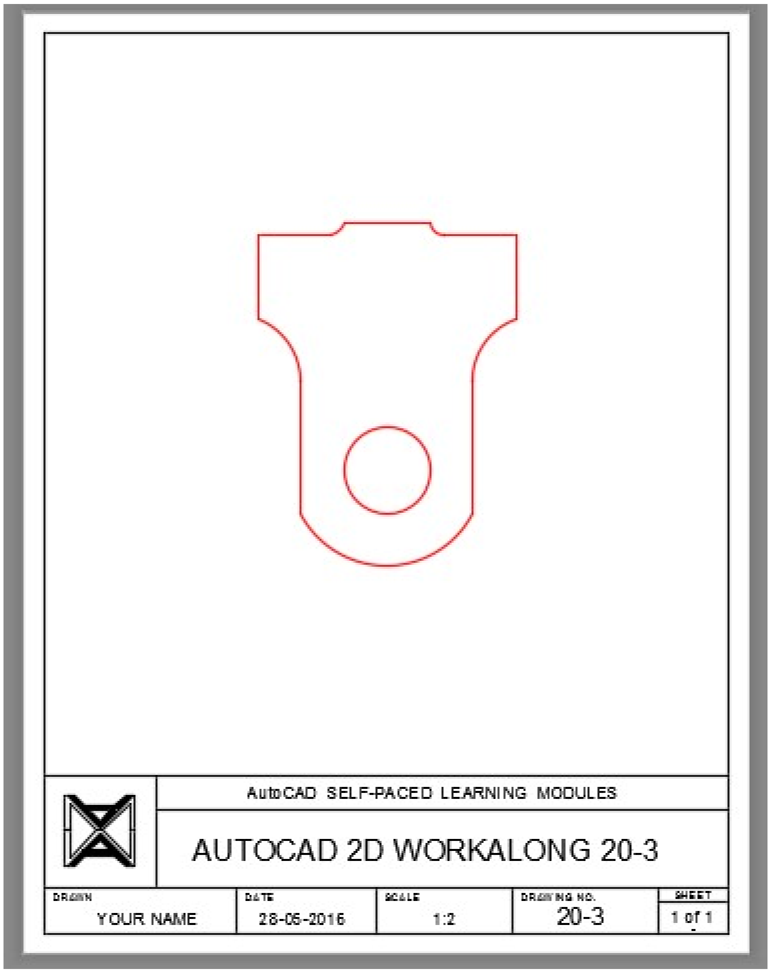
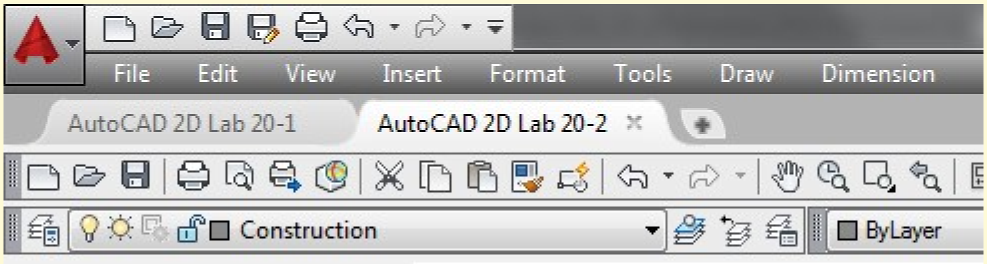

AutoCAD SELF-PACED LEARNING MODULES
AUTOCAD 2D WORKALONG 20-3
DRAWN
YOUR NAME
DATE
28-05-2016
SCALE
1:2
DRAWING NO.
20-3
SHEET
1 of 1

Figure Step 26

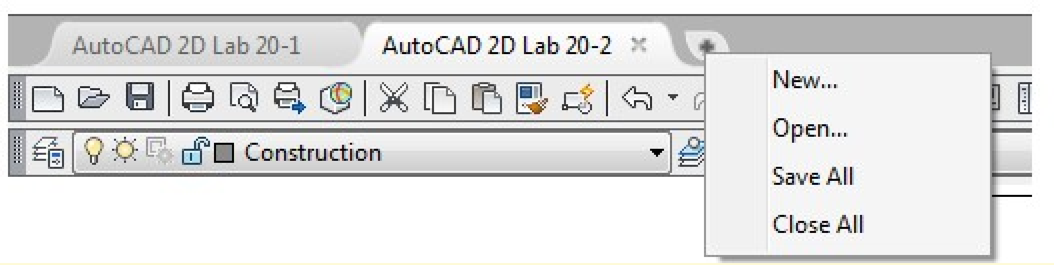
Step 27

Save and close the drawing.

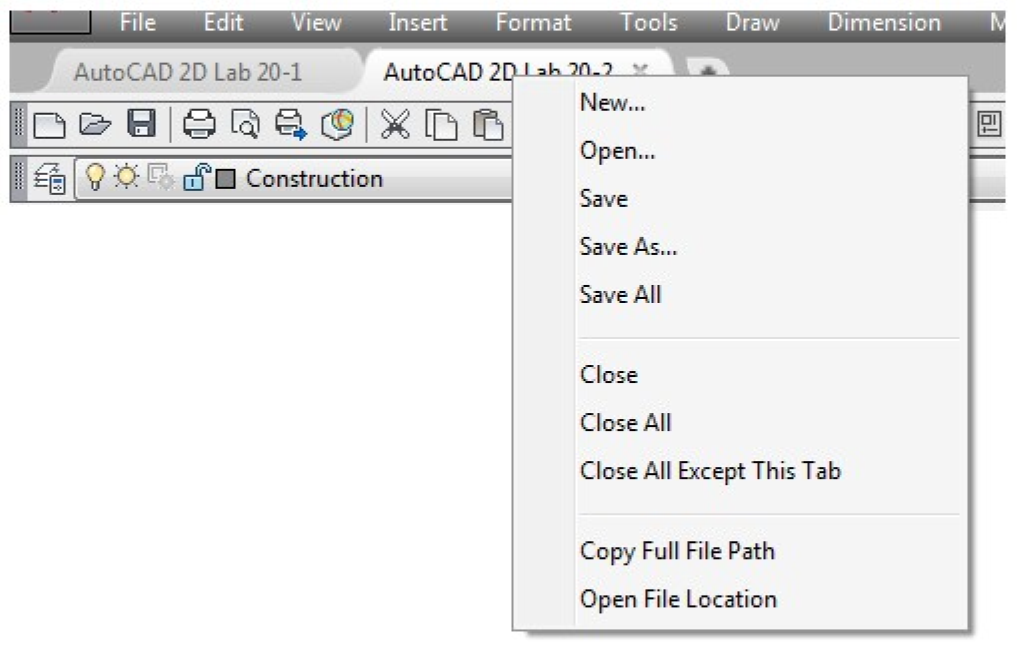
USER TIP: The FILETAB command enables the display of the drawing file tabs on top of the Graphic window and the FILETABCLOSE disables the display.


File
Edit
View
Insert
Format
Tools
Draw
Dimension
AutoCAD 2D Lab 20-1
AutoCAD 2D Lab 20-2
Construction
ByLayer

If you click the small arrow at the end of the tab, a handy menu displays as shown below.


AutoCAD 2D Lab 20-1
AutoCAD 2D Lab 20-2
Construction
New...
Open...
Save All
Close All

If you right click on a tab, a handy menu displays as shown below.


File
Edit
View
Insert
Format
Tools
Draw
Dimension
AutoCAD 2D Lab 20-1
AutoCAD 2D Lab 20-2
Construction
New...
Open...
Save
Save As...
Save All
Close
Close All
Close All Except This Tab
Copy Full File Path
Open File Location

Key Principles

Key Principles in Module 20

1. Objects only have to be drawn once in AutoCAD and then they can be used over and over again even if they are in a different drawing or at a different size or orientation.
2. When an object is copied, the copies retain all of the properties of the original object.
3. When copying objects from one drawing to another and the objects must be in the exact location as the original drawing, use the COPYBASE command with 0,0 as the base point. Use the same coordinates when pasting it into the new drawing.

Lab Exercise 20-1

Time allowed: 60 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 20-1	2D Layout English	Inches

Step 1

On layer Object, draw the object shown in the figure. (Figure Step 1A. 1B, and 1C)

Step 2

Draw all construction objects on layer: Construction.

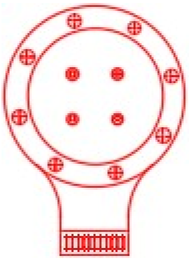
Step 3

Set the insertion units and insert the key. Check the drawing for accuracy.

Step 4

Turn layer: Key off and freeze layer: Construction.

AUTHOR'S COMMENTS: Do not delete the construction lines.



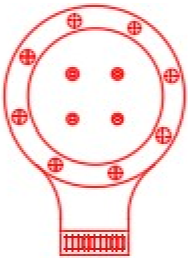


Figure Step 1A
Completed Drawing

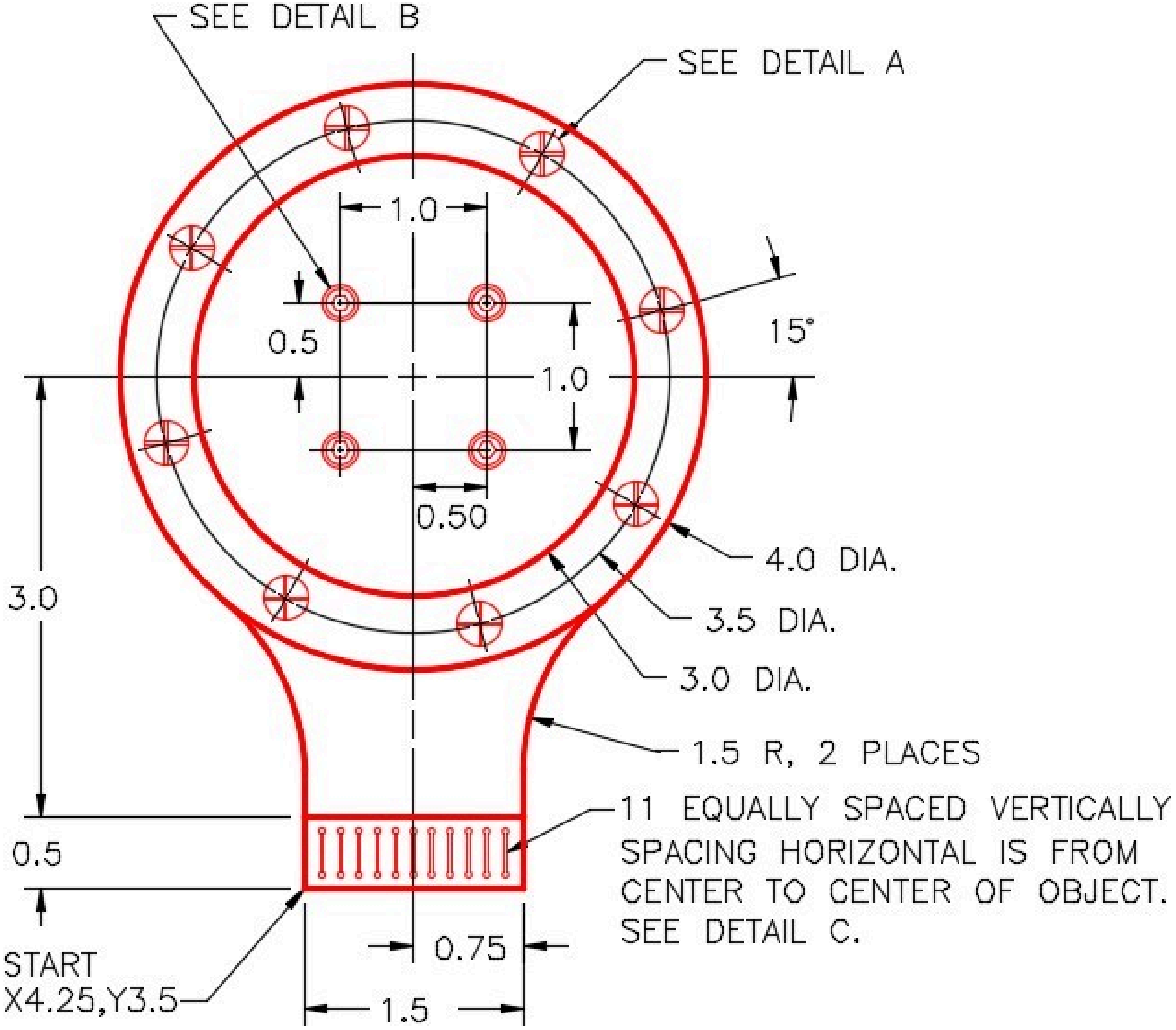

SEE DETAIL B
SEE DETAIL A
1.0
0.5
1.0
15°
0.50
3.0
4.0 DIA.
3.5 DIA.
3.0 DIA.
1.5 R, 2 PLACES
11 EQUALLY SPACED VERTICALLY SPACING HORIZONTAL IS FROM CENTER TO CENTER OF OBJECT. SEE DETAIL C.
0.5
0.75
START X4.25,Y3.5
1.5

Figure Step 1B

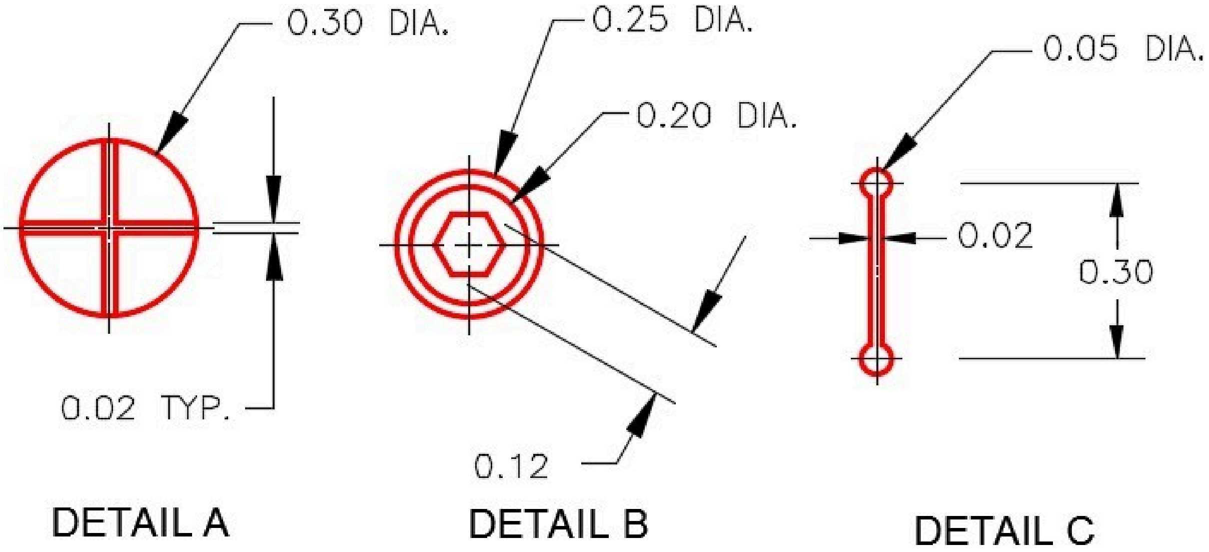

0.30 DIA.
0.02 TYP.
DETAIL A
0.25 DIA.
0.20 DIA.
0.12
DETAIL B
0.05 DIA.
0.02
0.30
DETAIL C

Figure Step 1C

Step 5

Copy all of the objects 7 units in the X direction to complete the drawing. (Figure Step 5)

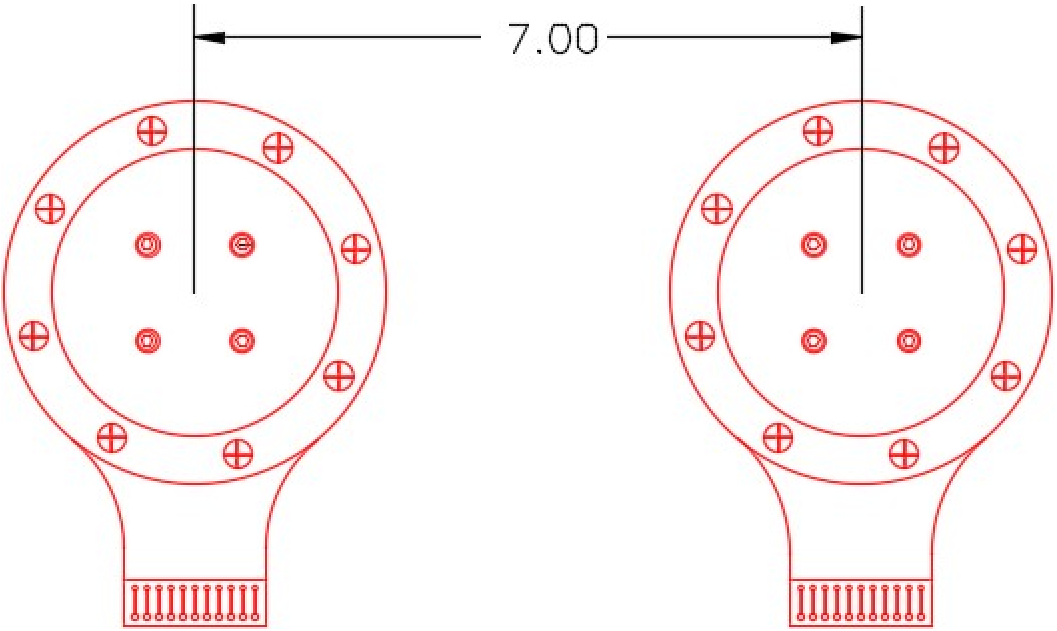


Figure Step 5

Lab Exercise 20-2

Time allowed: 45 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 20-2	2D Layout Metric	Millimeters

Step 1

On layer Object, draw the object shown in the figure. (Figure Step 1A, 1B, and 1C)

Step 2

Draw all construction objects on layer: Construction.

Step 3

Set the insertion units and insert the key. Check the drawing for accuracy.

Step 4

Turn layer: Key off and freeze layer: Construction.

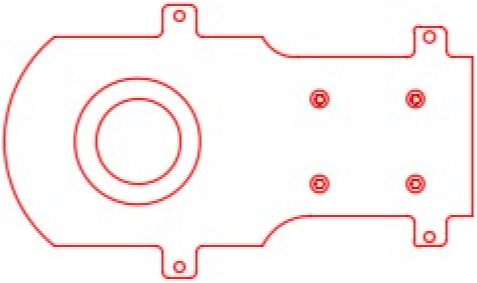


Figure Step 5A Completed Drawing

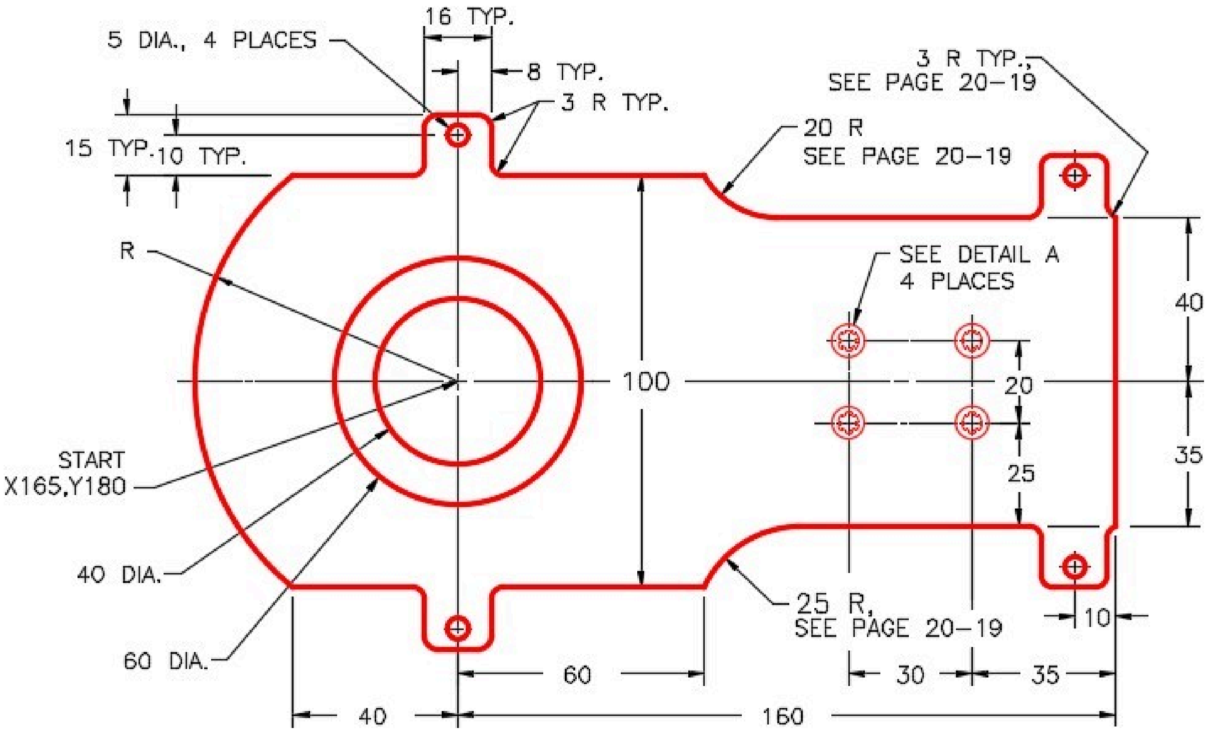
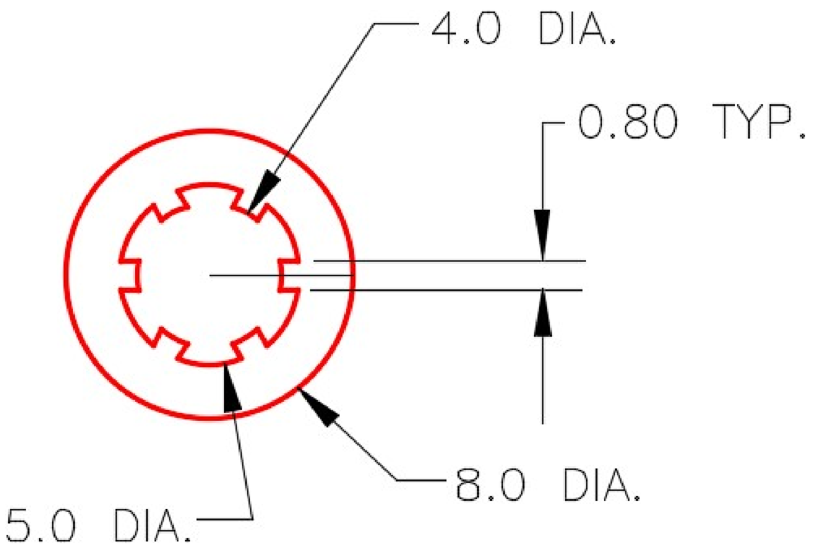

16 TYP.
5 DIA., 4 PLACES
8 TYP.
3 R TYP.
15 TYP.
10 TYP.
3 R TYP., SEE PAGE 20–19
20 R SEE PAGE 20–19
R
SEE DETAIL A 4 PLACES
40
100
20
35
25
START X165,Y180
40 DIA.
60 DIA.
25 R, SEE PAGE 20–19
10
60
30
35
40
160

Figure Step 1B

AUTHOR'S COMMENTS: Do not delete the construction lines.


4.0 DIA.
0.80 TYP.
8.0 DIA.
5.0 DIA.

Detail A
Figure Step 1C

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

Construct the object in Detail A using the following 6 steps. (Figure Hint 1)

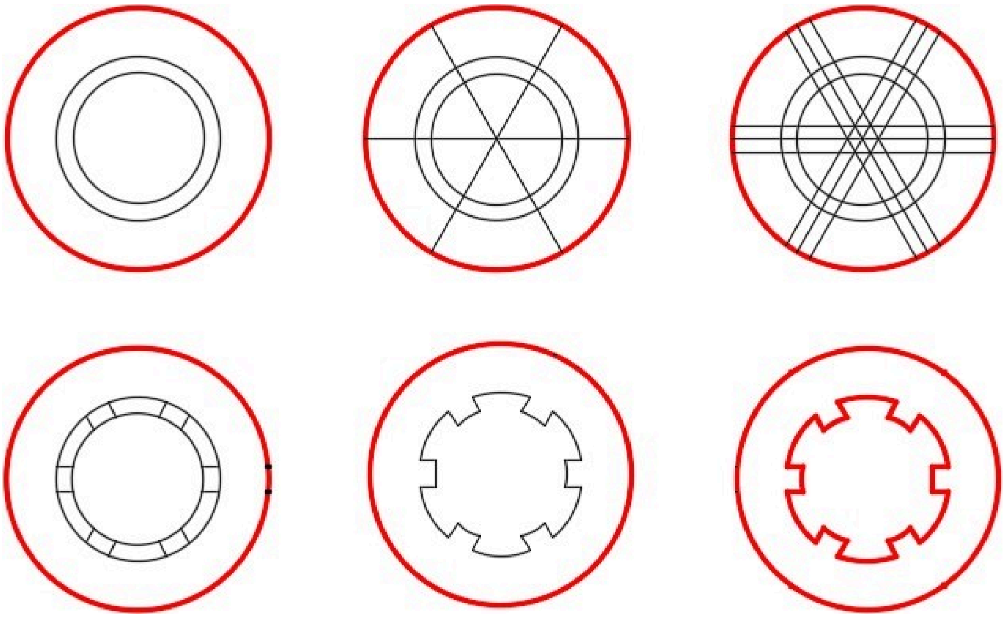


Figure Hint 1

Hint 2

There are four locations on the object that you must use the Geometry Lesson on Tangency. Figure Hint 2A and 2B shows you the three steps to draw two of them. (Figure Hint 2A and 2B)

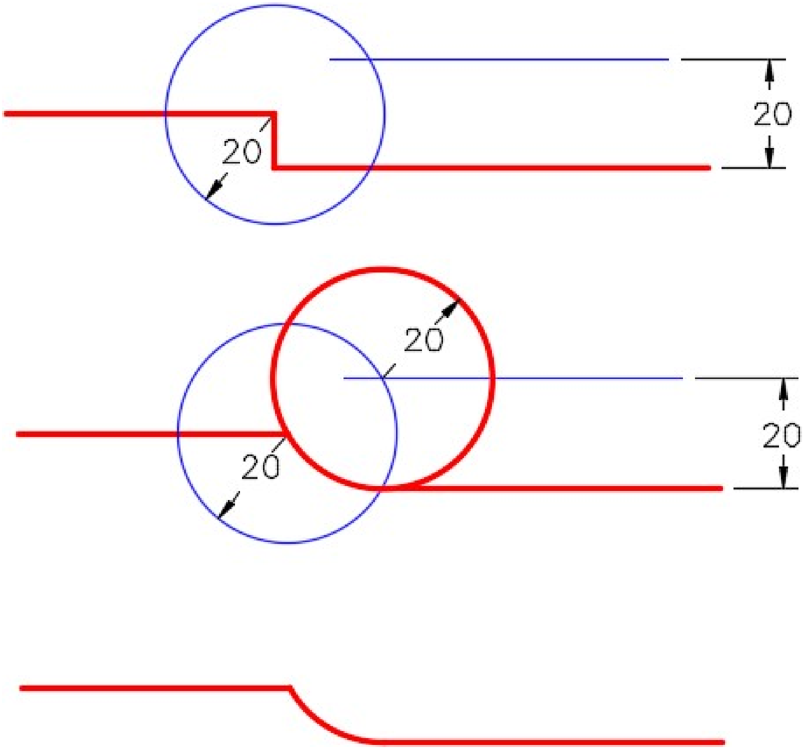


Figure Hint 2A

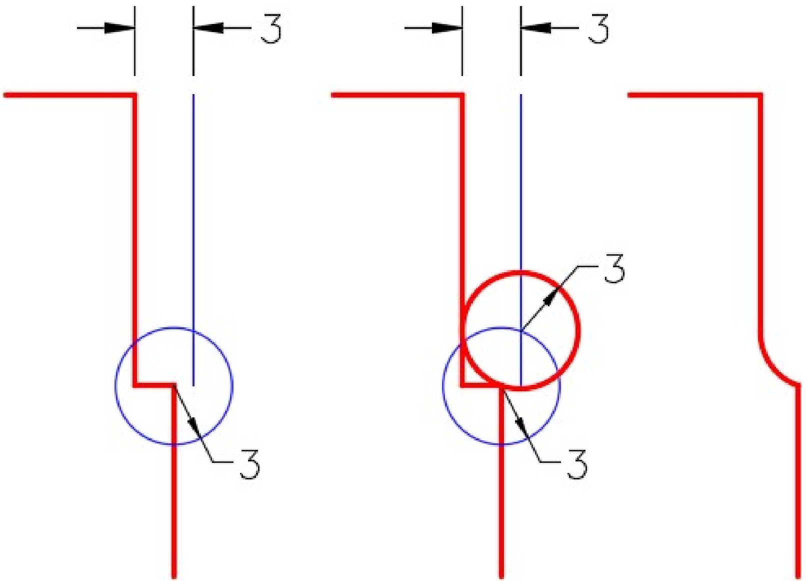


Figure Hint 2B

Lab Exercise 20-3

Time allowed: 30 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 20-3	2D Layout Metric	Millimeters

Step 1

Start a new drawing using the information above. Save and name the drawing: AutoCAD 2D Lab 20-3.

Step 2

Open drawing: AutoCAD 2D Lab 20-2. There should now be two drawings open.

Step 3

Using the COPYBASE and PASTECLIP commands, copy all of the objects from drawing Lab 20-2 into drawing Lab 20-3. Use the basepoint 0,0.

Step 4

Close drawing: AutoCAD 2D Lab 20-2.

Step 5

Edit the drawing: AutoCAD 2D Lab 20-3 to match the figure. Use the MOVE and COPY commands wherever possible. (Figure Step 5A and 5B)

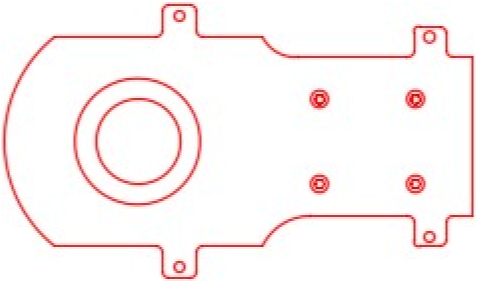


Figure Step 5A
Completed Drawing

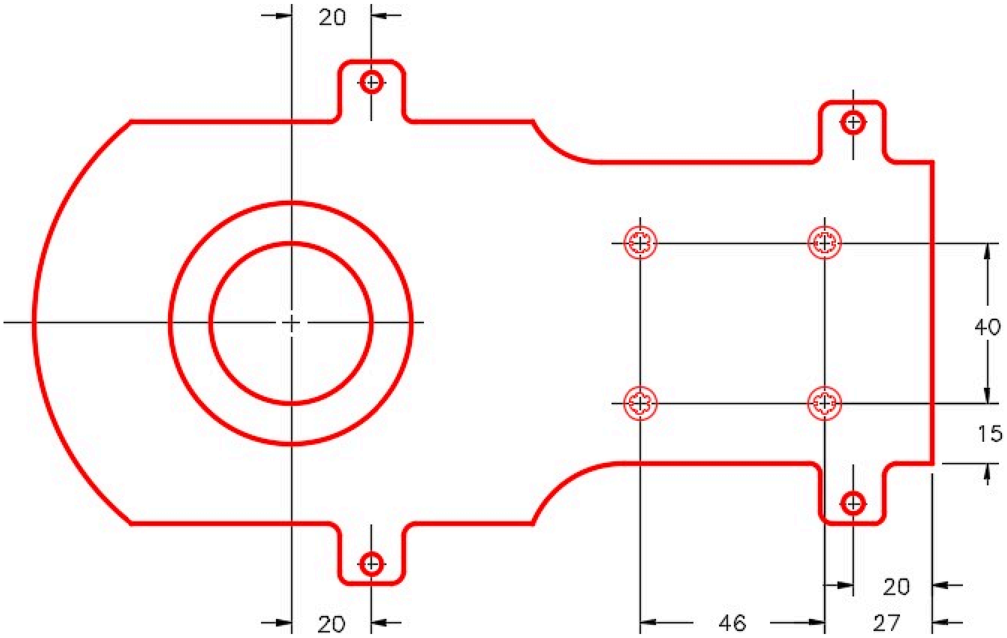


Figure Step 5B

AUTHOR'S COMMENTS: Do not delete the construction lines.

Step 6

Enable layout: Module Layout A3. On layer Viewport, use the MVIEW command and create a viewport as shown in the figure. Set the scale of the viewport to 1:1 and lock the display. (Figure Step 6)

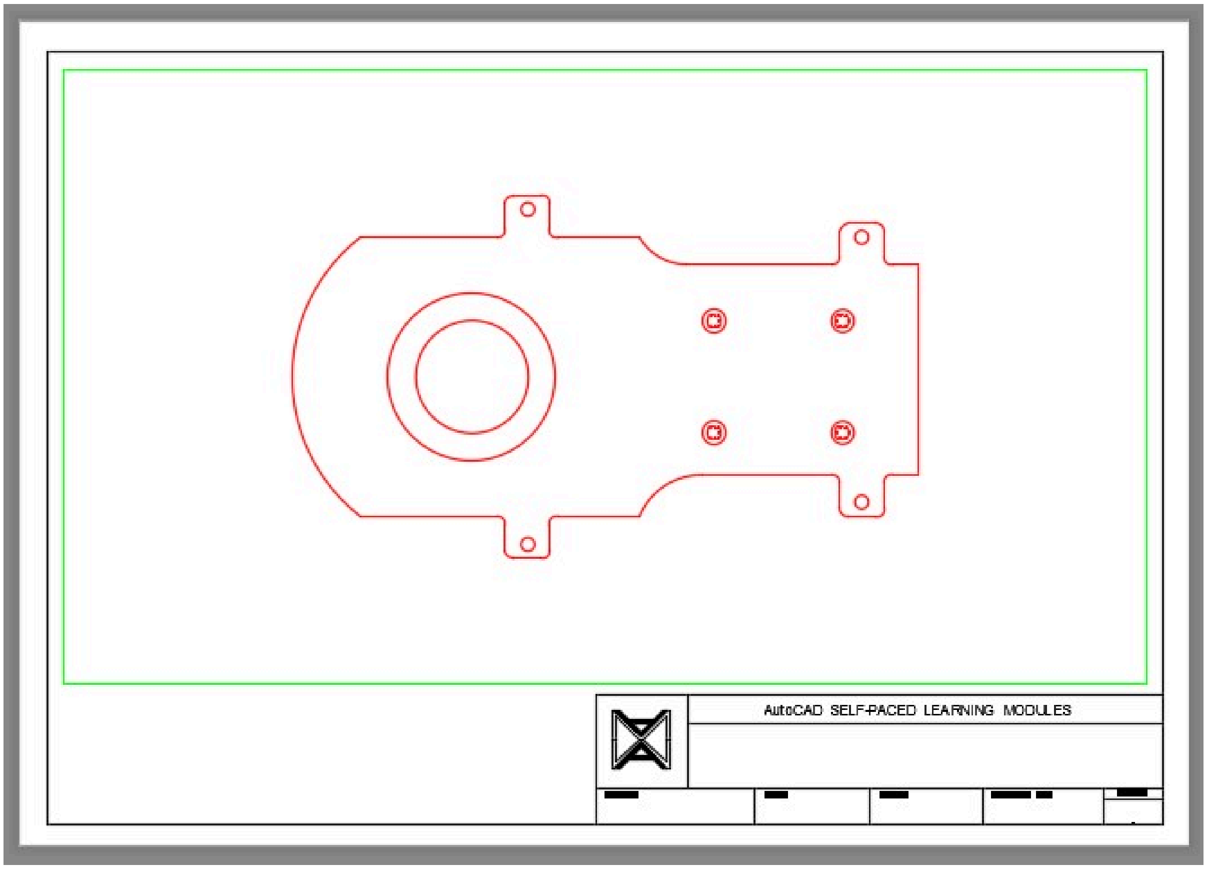
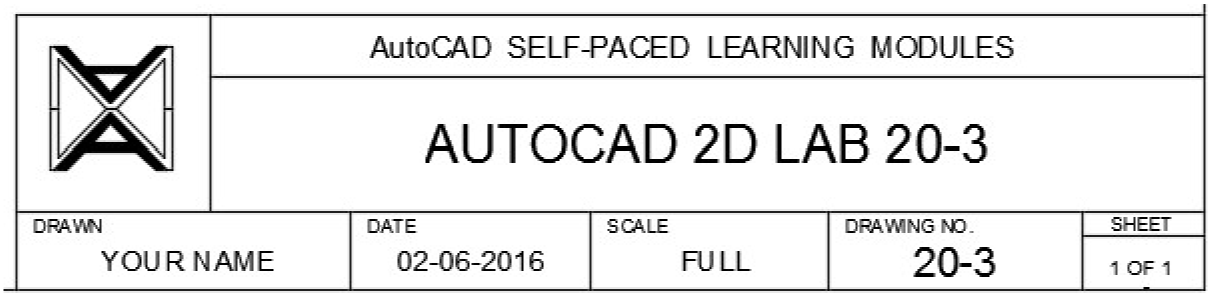


Figure Step 6

Step 7

Using the standards under "AutoCAD book Standards: Filling in the Titleblock". (Figure Step 7)



AutoCAD SELF-PACED LEARNING MODULES

AUTOCAD 2D LAB 20-3

DRAWN	DATE	SCALE	DRAWING NO.	SHEET
YOUR NAME	02-06-2016	FULL	20-3	1 OF 1

Figure Step 7

Step 8

Turn layer: Viewport off. (Figure Step 8)

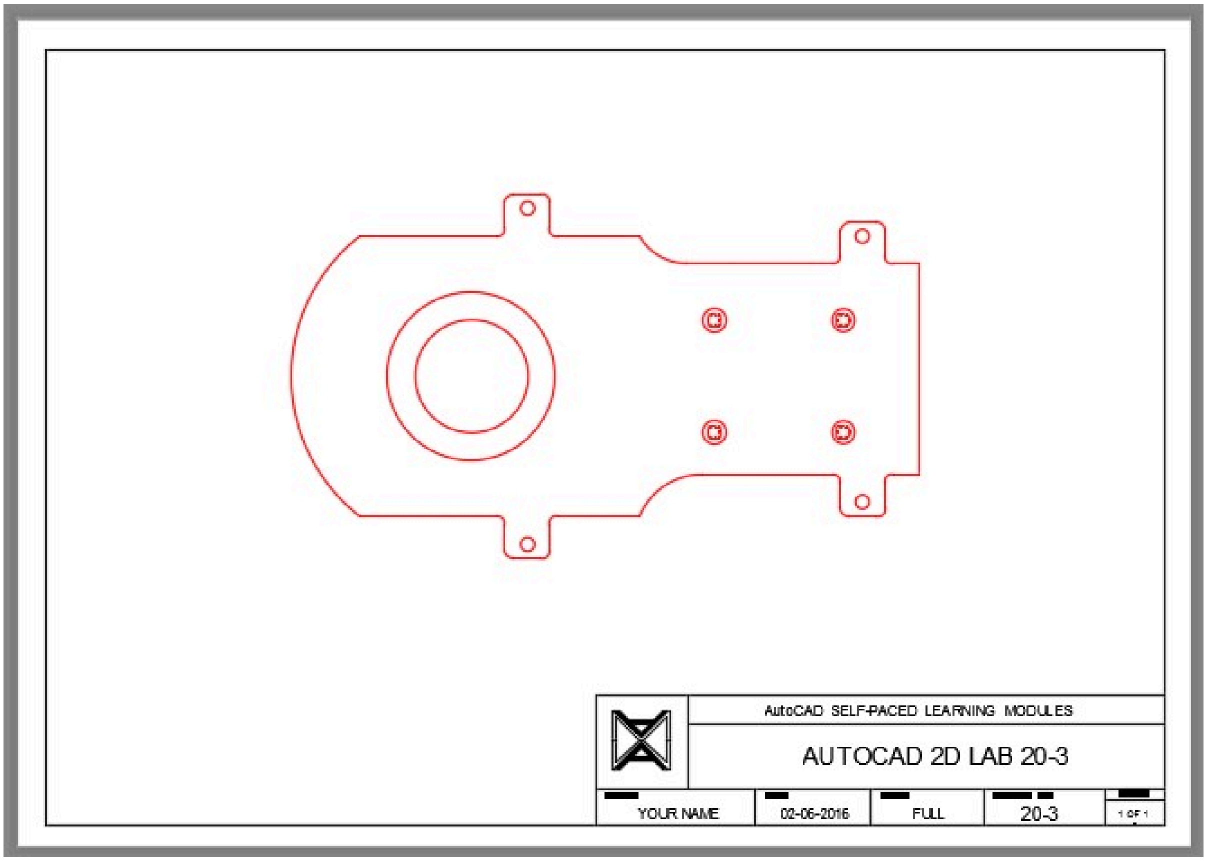

AutoCAD SELF-PACED LEARNING MODULES
AUTOCAD 2D LAB 20-3
YOUR NAME
02-06-2016
FULL
20-3
1 OF 1

Figure Step 8

Module 21: Competency Test 4 Open Book

Learning Outcomes

When you have completed this module, you will be able to:

1. Within a two hour time limit, complete a written exam and a lab exercise without the aid of a key.

The AutoCAD 2D book was written with competency based modules. What that means is that you have not completed each module until you have mastered it. The Competency Test module contains multiple choice questions and a comprehensive lab exercise to test your mastery of the set of modules that you completed. There are no answers or keys supplied in a Competency Test module since it is meant to be checked by your instructor. If there are any parts of this module that you have trouble completing, you should go back and reread the module or modules containing the information that you are having trouble with. If necessary, redo as many lab exercises required until you fully understand the material.

If you are Completing this book:

- Without the Aid of an Instructor, complete the written test and the lab exercise.
- In a Classroom with an Instructor, the instructor will give instructions on what to do after this module has been completed.

Multiple Choice Questions

Select the BEST answer.

1. What is the name of the object snap mode for a point?
    A. Snap
    B. Point
    C. On
    D. End
    E. Node
2. When enabled, what effect does Ortho mode have in a drawing?
    A. It displays grid points.
    B. It restricts the graphic cursor to the horizontal and vertical axis.

   C. It rotates the angle of the graphic cursor.
   D. It enables coordinate picks to snap to grid points.
   E. It changes the default text inserts to horizontal.

3. What should the text height be set to in the Text Style dialogue box?
   A. Zero
   B. The height you use the most.
   C. 1 unit
   D. 10 units
   E. 100 units
4. Which function key toggles Ortho mode?
   A. F5
   B. F6
   C. F7
   D. F8
   E. F9
5. What command is used to set the way units are displayed in the Properties window?
   A. UNITS
   B. GRID
   C. DISPLAY
   D. PRECISION
   E. SNAP
6. What is the main difference between the DIVIDE and MEASURE commands?
   A. DIVIDE actually divides the object into individual parts while MEASURE does not.
   B. MEASURE only works for lines while DIVIDE works for all objects.
   C. DIVIDE inserts points in an equal number of divisions while MEASURE inserts points at an equal distance entered by the user.
   D. There is no difference, they are exactly the same command.
   E. The DIVIDE command inserts points at an equal distance specified by the user starting at the end of the object clicked, while MEASURE measures out the distance of the object and breaks it in to equal parts.
7. What command is used to create a viewport in a layout?
   A. VIEW
   B. CVIEW

   C. VIEWP
   D. MVIEW
   E. PAGELAYOUT
8. Which system variable controls the rotation angle of the cursor?
   A. ANGSNAP
   B. CURANGLE
   C. ANGCUR
   D. ROTCURSOR
   E. SNAPANG
9. What command is used to change the font of an existing text object on the drawing?
   A. FONT
   B. DTEXT
   C. DDEDIT
   D. EDITTEXT
   E. STYLE
10. Which command is used to copy objects onto the clipboard with a known basepoint?
    A. BASECOPY
    B. POINTCOPY
    C. COPYBASE
    D. COPYCLIP
    E. COPY

Lab Exercise 21-1 OPEN BOOK

Time allowed: 60 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 21-1	2D Layout English	Inches

Step 1

On layer: Object, draw the object shown in the figure. Draw all construction objects on layer: Construction. (Figure Step 1A and 1B)

AUTHOR'S COMMENTS: Do not delete the construction objects. There is no key for this lab.

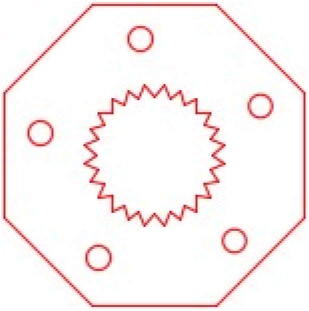


Figure Step 1A
Completed Drawing

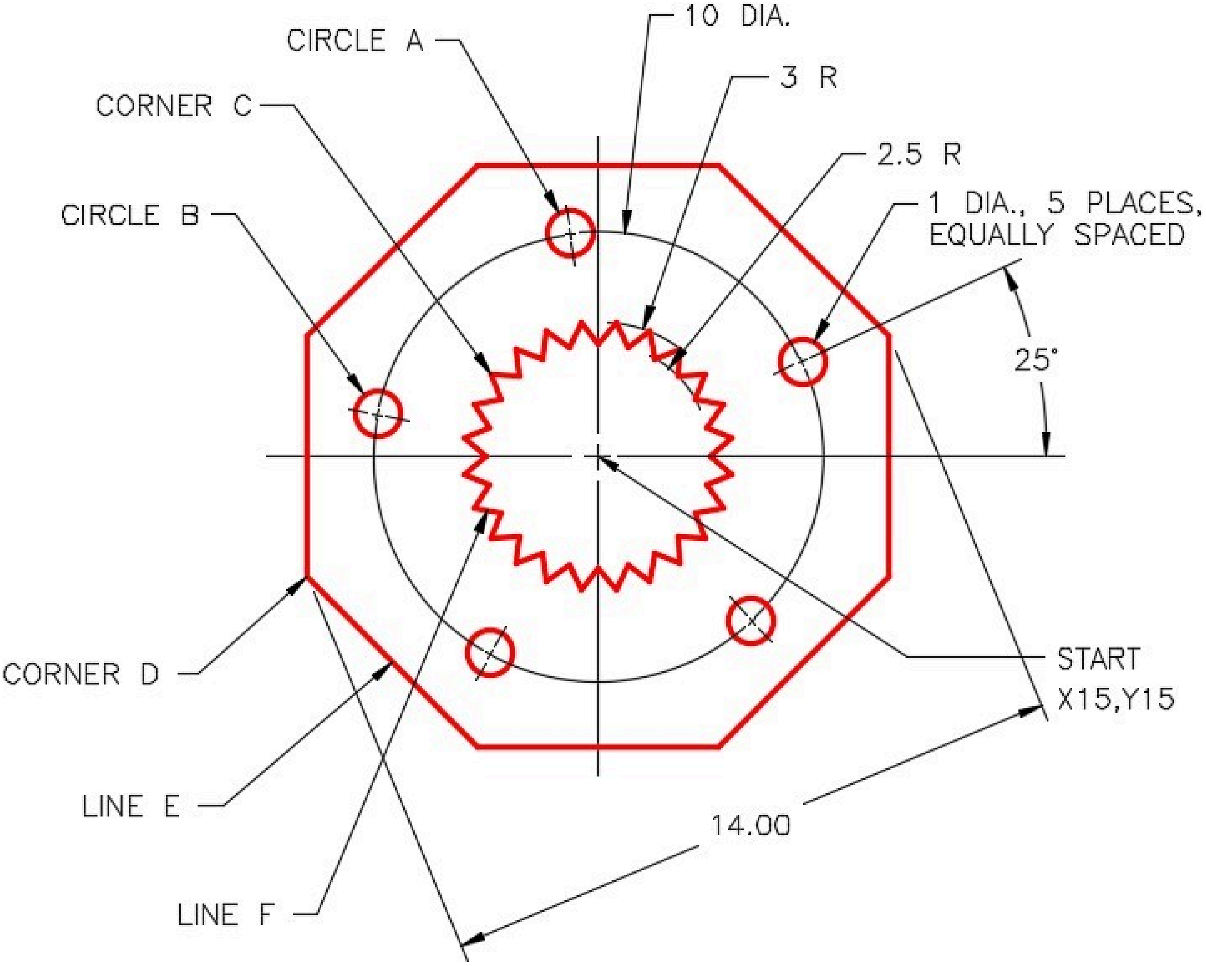

CIRCLE A
10 DIA.
3 R
CORNER C
2.5 R
1 DIA., 5 PLACES, EQUALLY SPACED
CIRCLE B
25°
START X15,Y15
CORNER D
LINE E
14.00
LINE F

Figure Step 1B

Step 2

Using inquiry commands and the Properties window, find the answers for questions A to H.

For questions A to D, find the answers to a precision of 4 decimal places.

A. What is the XY coordinate location of the center of Circle A?

X- _____________

Y- _____________

B. What is the circumference of Circle A? _____________

C. What is the distance from the center of Circle A to the center of Circle B? _____________

D. What is shortest distance between the circumference of Circle A and the circumference of Circle B? _____________

For questions E to G, find the answers to a precision of 8 decimal places.

E. What are the XY coordinates of Corner D? _____________

X- _____________

Y- _____________

F. What is the length of Line E? _____________

G. What is the shortest distance from Corner C to the circumference of Circle B? _____________

For question H, find the answer in Deg/Min/Sec to a precision of 2 decimal places (seconds).

H. What is the angle of Line F? _____________

Step 3

In layout: Module Layout B, create a viewport on layer: Viewport. Set the scale of the viewport to 1:2 and lock the display. (Figure Step 3)

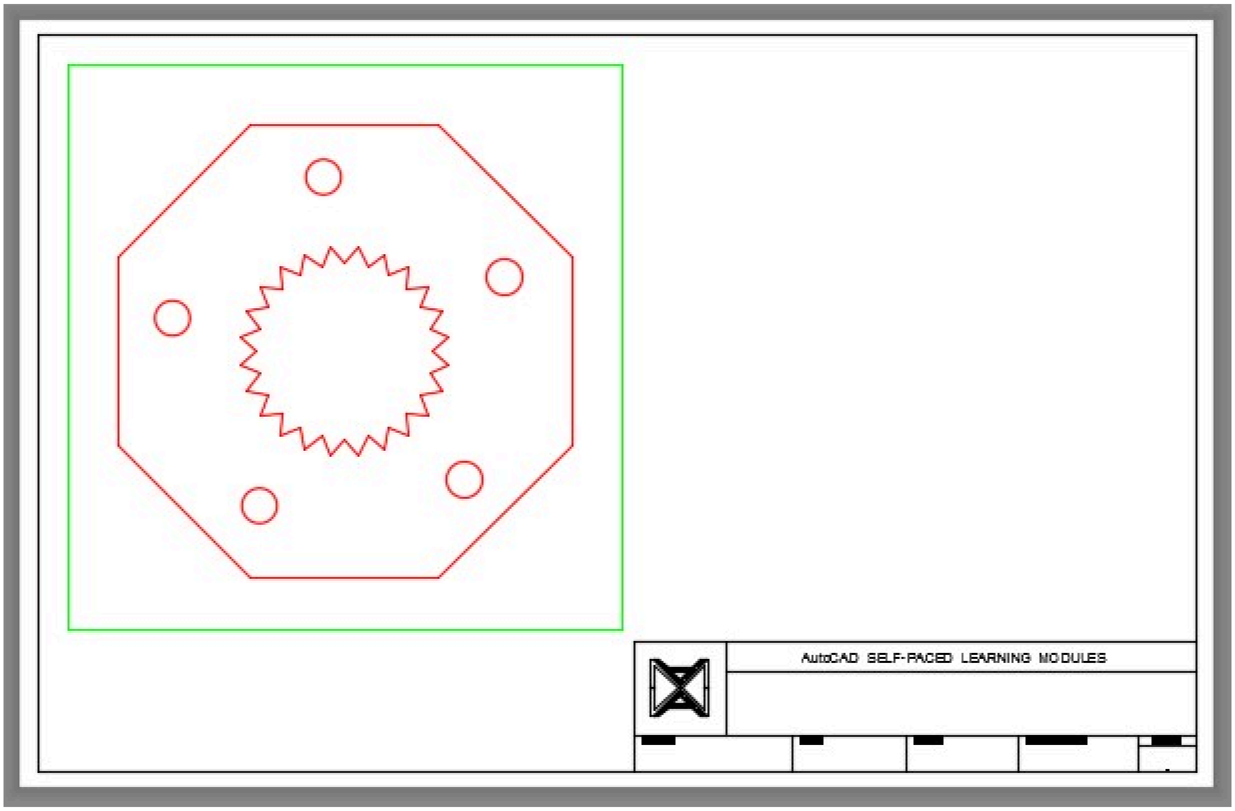
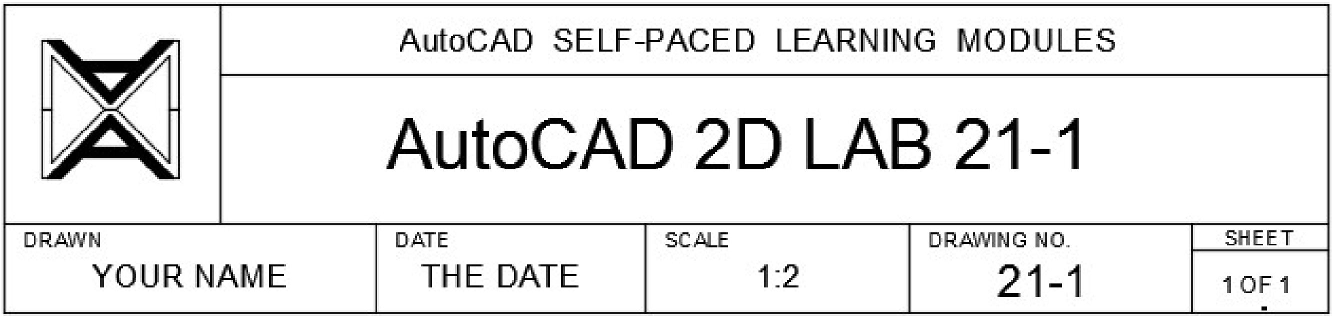


Figure Step 3

Step 4

On layer: Titleblock Text, fill in the titleblock. Use the AutoCAD Modules Titleblock Standards shown in Module 20. (Figure Step 4)



AutoCAD SELF-PACED LEARNING MODULES

AutoCAD 2D LAB 21-1

DRAWN	DATE	SCALE	DRAWING NO.	SHEET
YOUR NAME	THE DATE	1:2	21-1	1 OF 1

Figure Step 4

Step 5

Freeze layer: Viewport.

Step 6

Using the TEXT command and the answers you found for questions Step 2 (A to H), insert them on your drawing locating them by eye on the layer: Text. Insert them in Paper space. Use the text style: 2D Modules at a height of 0.20. (Figure Step 6)

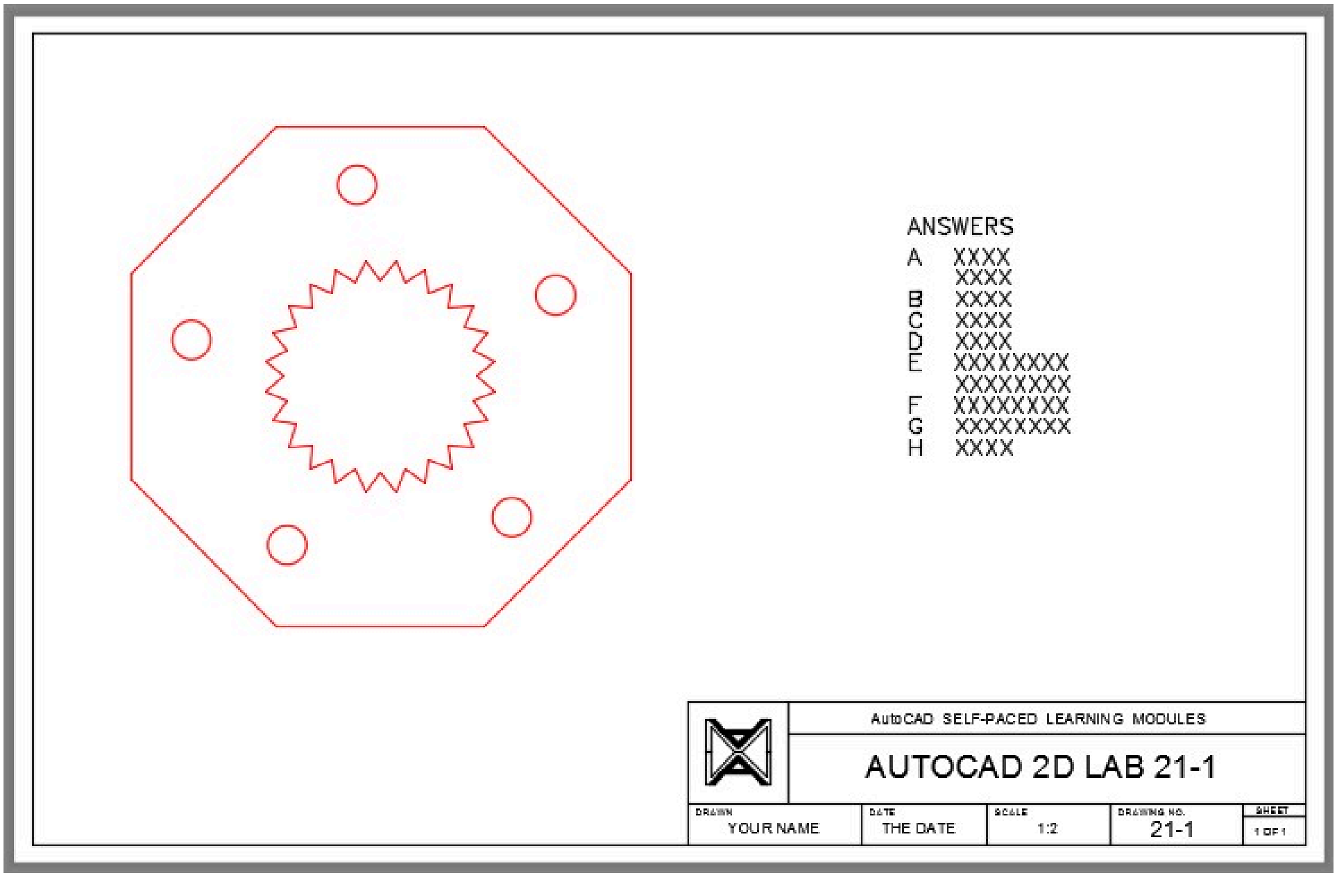

ANSWERS
A XXXX
XXXX
B XXXX
C XXXX
D XXXX
E XXXXXXXX
XXXXXXXX
F XXXXXXXX
G XXXXXXXX
H XXXX
AutoCAD SELF-PACED LEARNING MODULES
AUTOCAD 2D LAB 21-1
DRAWN
YOUR NAME
DATE
THE DATE
SCALE
1:2
DRAWING NO.
21-1
SHEET
1 OF 1

Figure Step 6

Step 7

Save and close the drawing.

Part 5

Module 22: Mirroring and Rotating

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe how drawing objects are mirrored using mirror lines.
2. Describe how drawing objects are rotated.
3. Apply the MIRROR and the ROTATE commands to mirror and rotate objects.
4. Apply the MIRRTEXT system variable.
5. Describe motion paths and explain how to draw and use them with the ROTATE command.

Mirroring

The MIRROR command is a very useful productivity tool. It is used to create a mirror image of an object saving you time that it takes to redraw the reverse image of the object. Before drawing any object, you should study it to see if it is symmetrical in any way and can be drawn faster by applying the MIRROR command.

The objects to be mirrored are flipped around an axis. This axis is called a *mirror line* or a *folding line*. The mirror line is simply defined by two XY coordinates. The distance between the two XY coordinates is not important.

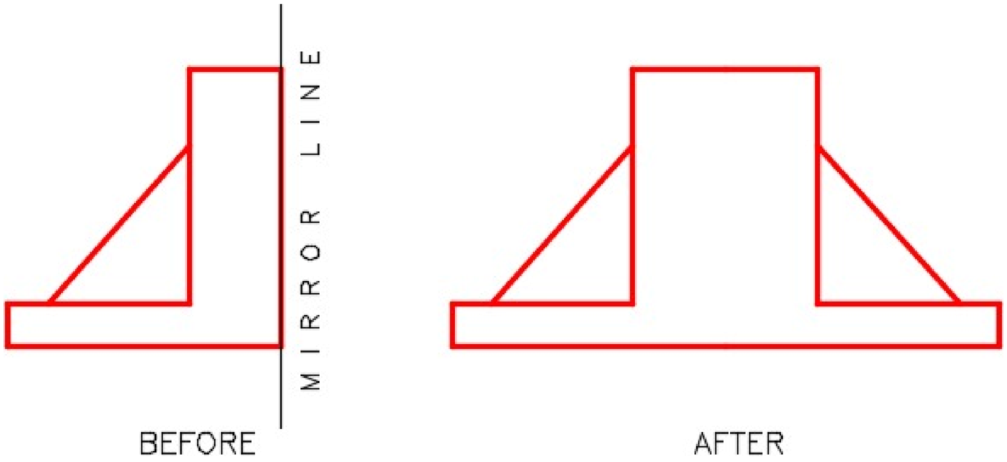


Figure 22-1
Mirroring

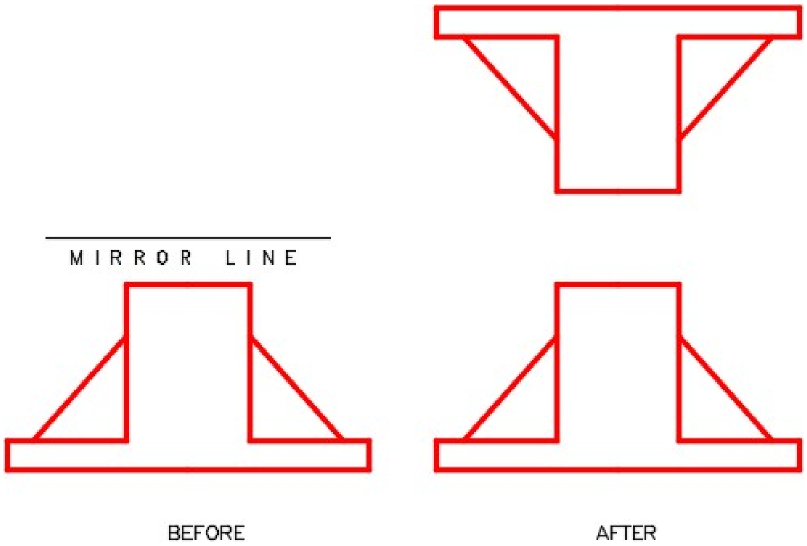
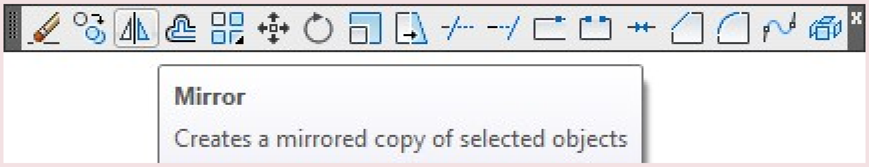

MIRROR LINE
BEFORE
AFTER

Figure 22-2
Mirroring

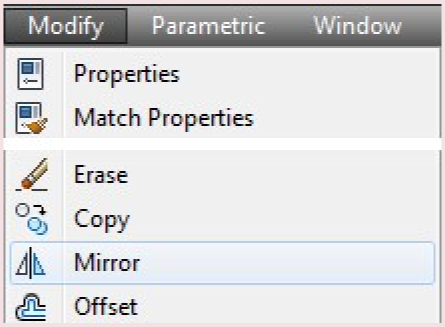
AutoCAD Command: MIRROR

The MIRROR command is used to mirror an object around a mirror line (folding line) to create a mirror image of the object.

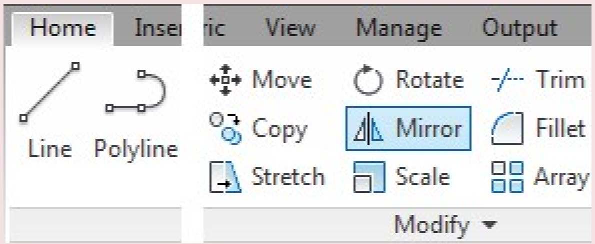
Shortcut: **MI**


Mirror
Creates a mirrored copy of selected objects

Modify Toolbar


Modify
Parametric
Window
Properties
Match Properties
Erase
Copy
Mirror
Offset

Modify Pull-down


Home
Inser
ric
View
Manage
Output
Line
Polyline
Move
Rotate
Trim
Copy
Mirror
Fillet
Stretch
Scale
Array
Modify

Home Ribbon

WORKALONG: Using the MIRROR Command – Part 1

Step 1

Start a new drawing using the template: 2D Layout English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 22-1.

Step 3

On layer: Object, draw one quarter of the object shown using the Figure Step 3A as a reference. (Figure Step 3A and 3B)

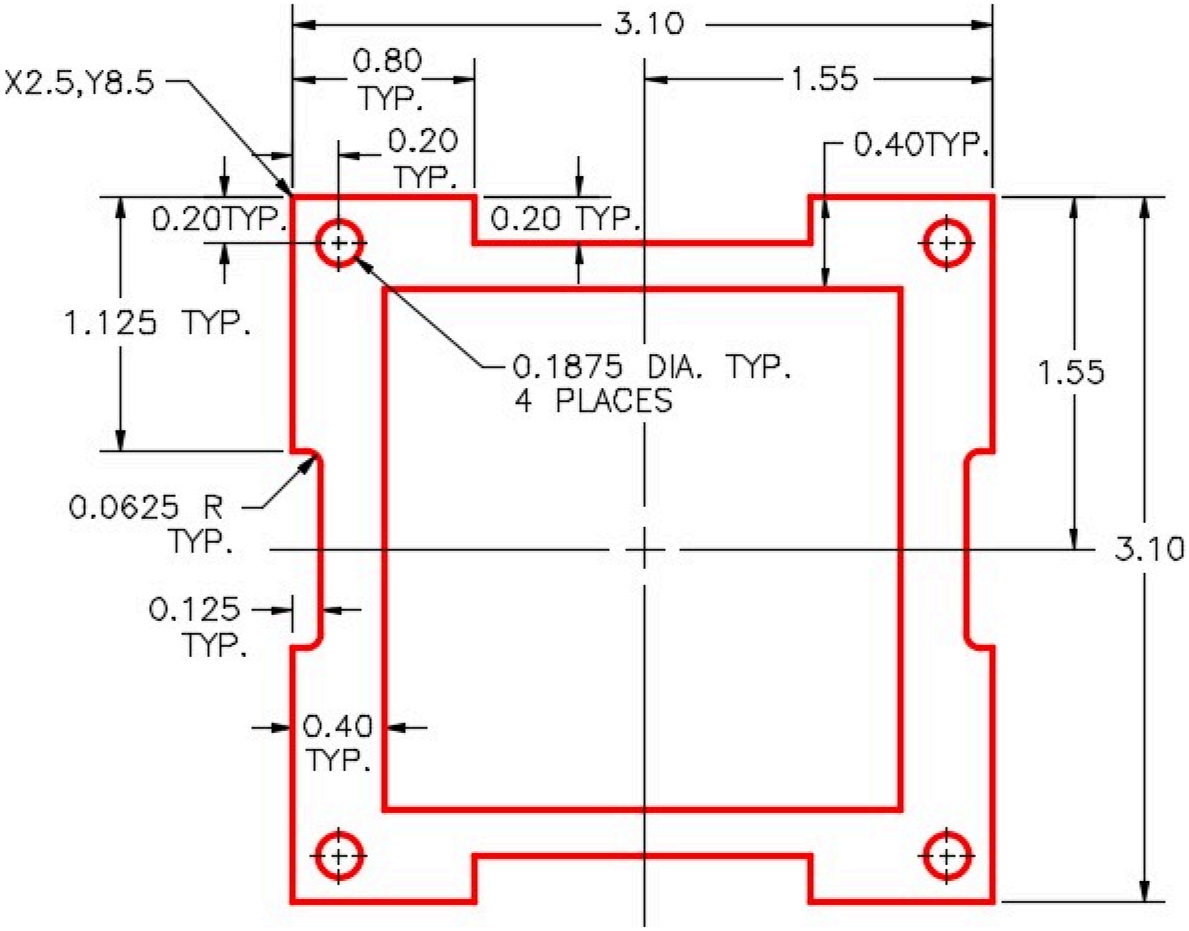

3.10
0.80
TYP.
1.55
X2.5,Y8.5
0.20
TYP.
0.40TYP.
0.20TYP.
0.20 TYP.
1.125 TYP.
0.1875 DIA. TYP.
4 PLACES
1.55
0.0625 R
TYP.
3.10
0.125
TYP.
0.40
TYP.

Figure Step 3A

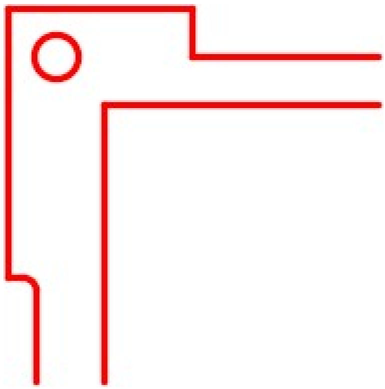


Figure Step 3B

Step 4

Enter the MIRROR command, as shown below, to mirror the object. (Figure Step 4A, 4B, 4C, and 4D)

Command: **MIRROR**

Select objects:

Specify opposite corner: 9 found

(Use a crossing window to select the objects to be mirrored.)

Select objects:

Specify first point of mirror line: *(end)*

Specify second point of mirror line: *(end)*

(Snap to the ends of the two lines to define the mirror line.)

Delete source objects? [Yes/No] <N>: **N**

(Enter No)

Command:

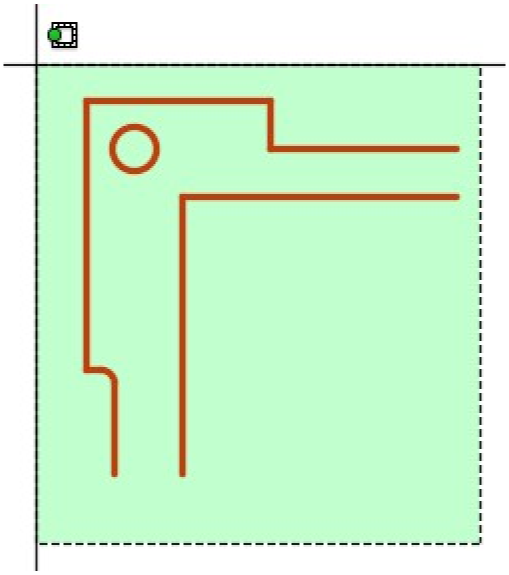


Figure Step 4A

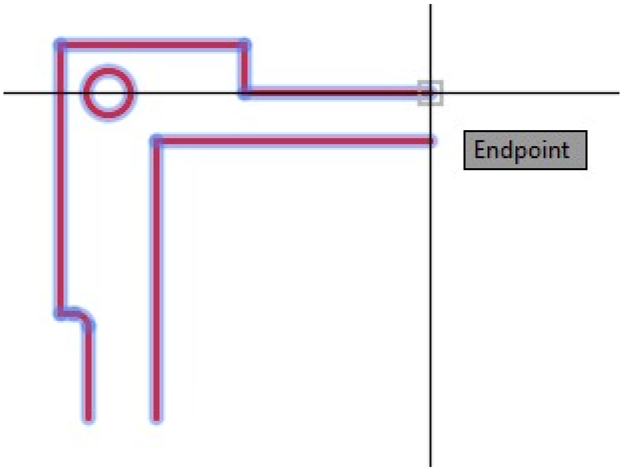


Figure Step 4B

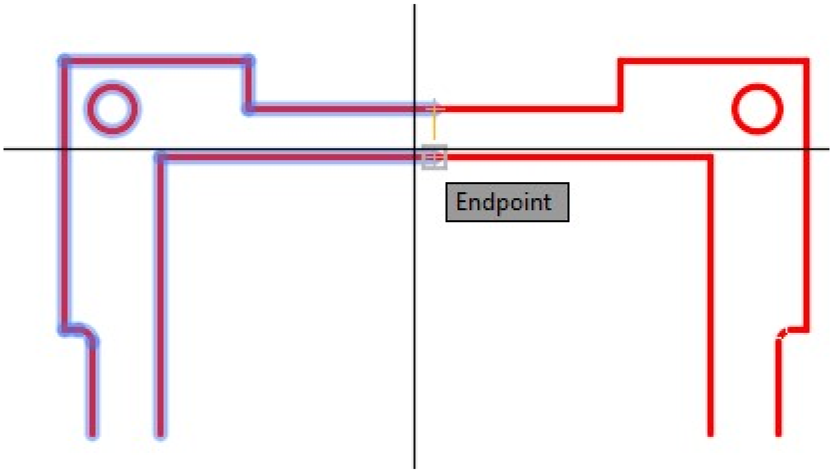


Figure Step 4C

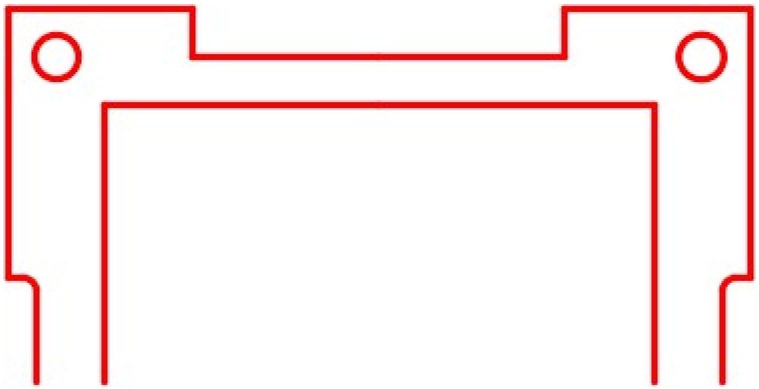


Figure Step 4D

AUTHOR'S COMMENTS: By snapping to the ends of the lines, the mirror line is vertical. See figure 22-3. The length of the mirror line is not important since AutoCAD can extend it to any length required.

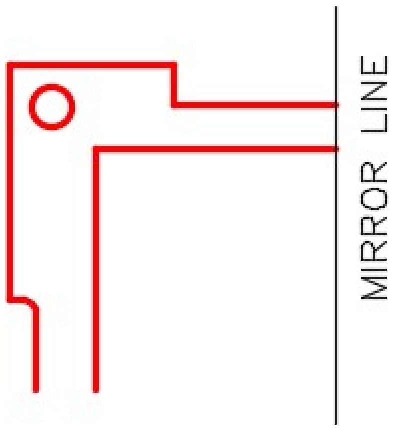


Figure 22-3
Mirroring Mirror Line Touching Object

Step 5

Enter the MIRROR command as shown below to mirror the object. (Figure Step 5A, 5B, 5C, and 5D)

Command: **MIRROR**

Select objects:

Specify opposite corner:

(Use a window to select all of the objects.)

Select objects:

Specify first point of mirror line: *(end)*

(Enable ortho mode.)

Specify second point of mirror line:

(With ortho mode enable, select any location in the -X direction.)

Delete source objects? [Yes/No] <N>: **N**

Command:

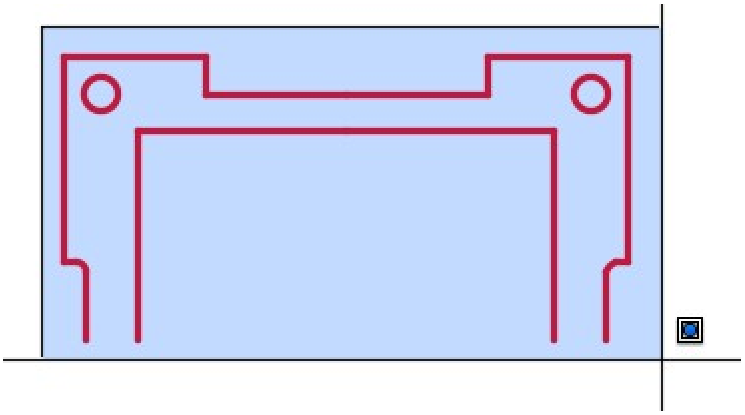


Figure Step 5A

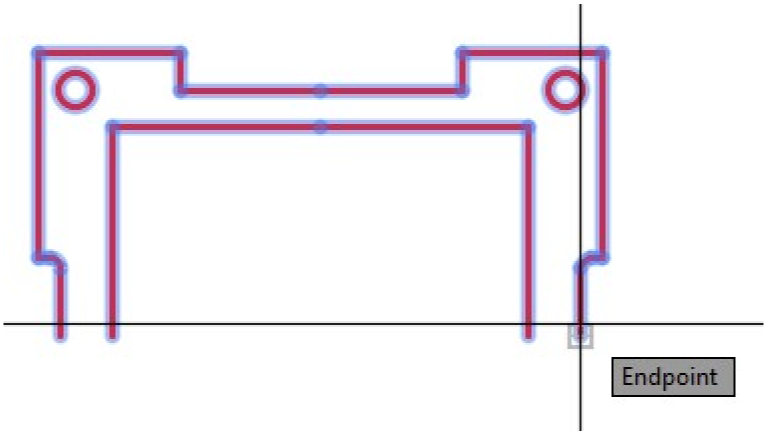


Figure Step 5B

AUTHOR'S COMMENTS: Ensure that you enable Ortho mode before you select the second point on the mirror line. By snapping to end of the object line and using Ortho mode the mirror line will be horizontal.

AUTHOR'S COMMENTS: When an object is mirrored, there are instances when lines that are mirrored will appear correct but are actually in two pieces. Even though they look correct, you cannot leave them in this condition as they may cause problems in the drawing sometime in the future. This is especially true when dimensioning, crosshatching, and working in three dimensions. It is a good practice to correct these lines as soon as the mirror is complete as this will avoid problems later. To fix them, erase one of the mirrored lines and then using the EXTEND command, extend the remaining lines.

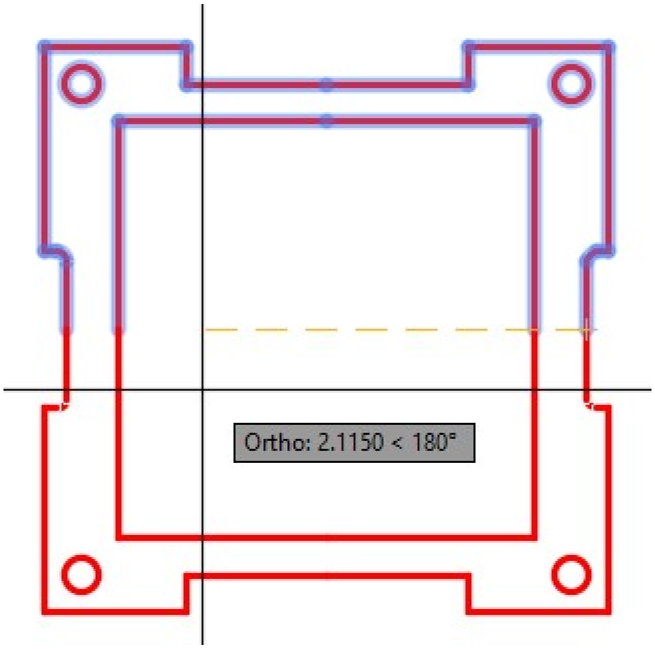

Ortho: 2.1150 < 180°

Figure Step 5C

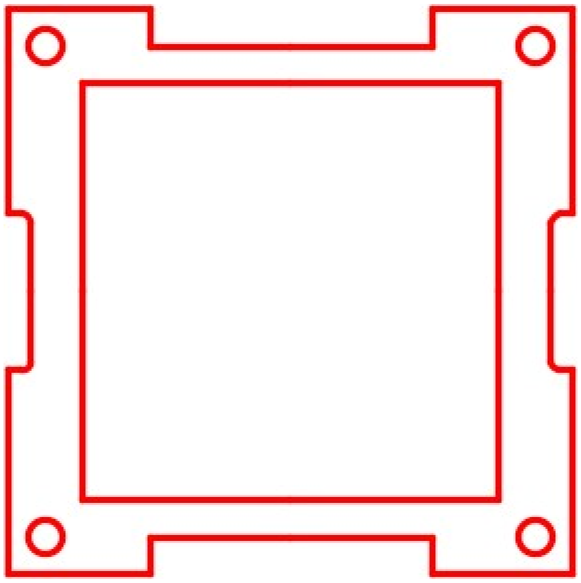


Figure Step 5D

AUTHOR'S COMMENTS: Note the location of the mirror line. See Figure 22-4.

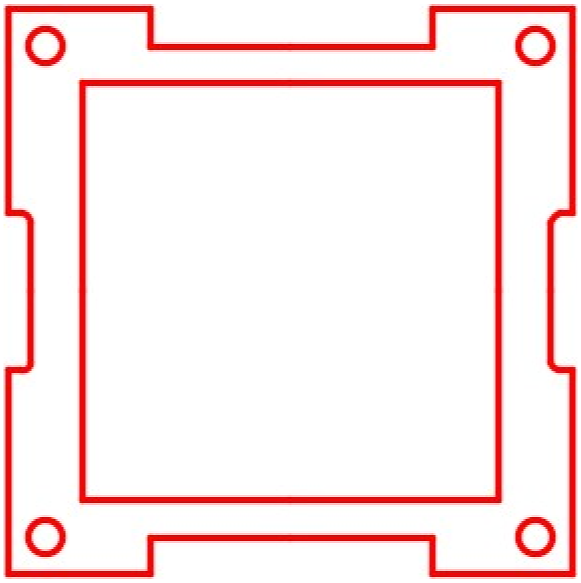


Figure 22-4
Mirror Line

Step 6

Erase half of the lines that were mirrored as shown in the figure. See the Author’s Comments under Step 5. (Figure Step 6)

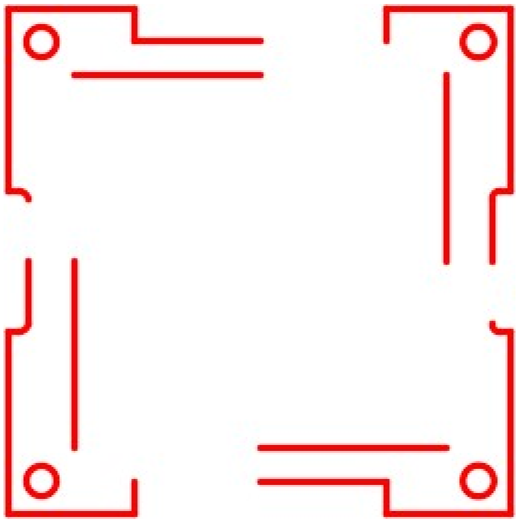


Figure Step 6

Step 7

Using the EXTEND command, extend the remaining lines to complete the object. (Figure Step 7)

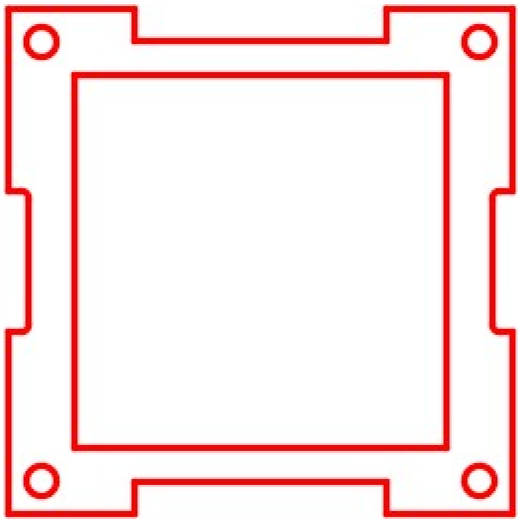


Figure Step 7

Step 8

Save and close the drawing.

MUST KNOW: When an object is mirrored, the object created in the MIRROR command will retain all the properties of the object being mirrored, including the layer it resides on. This happens regardless of the current layer when the MIRROR command is executed.

MUST KNOW: When an object is mirrored, there are instances when lines that are mirrored will appear correct but are actually in two pieces. Even though they look correct, it cannot be left in this condition as they may cause future problems in the drawing. This is especially true when dimensioning, crosshatching and working in three dimensions. It is a good practice to correct these lines as soon as the mirror is complete as this will avoid problems later. To fix them, erase one of the mirrored lines and then using the EXTEND command, extend the remaining lines.

AutoCAD Command: MIRRTEXT

The MIRRTEXT system variable is used to control how the MIRROR command reflects text.

Shortcut: **none**

WORKALONG: Using the MIRROR Command – Part 2

Step 1

Open drawing: AutoCAD 2D Lab 14-1. (Figure Step 1)

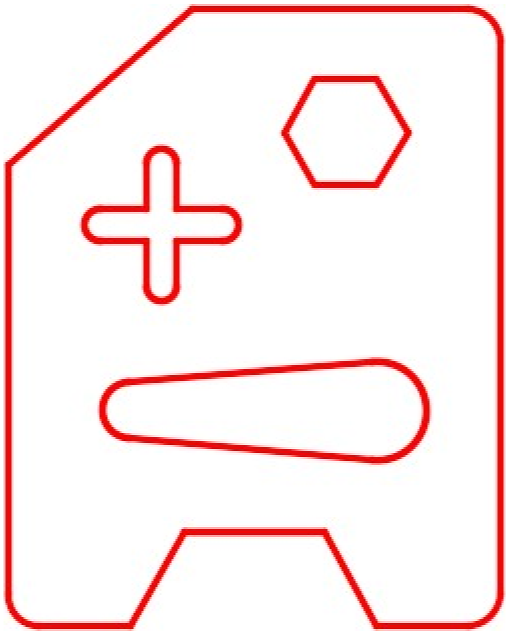


Figure Step 1

Step 2

Using the SAVEAS command, save the drawing with the name: AutoCAD 2D Workalong 22-2 Left.

Step 3

Create a text style, with the font: romans, named: Module 22. Set it as the current text style.

Step 4

Create layer: Text with the color black/white and set it as the current layer.

Step 5

Enter the TEXT command. Using the justification of Middle and the Height of 0.2, insert two lines of text as shown in the figure. Locate it by eye. (Figure Step 5)

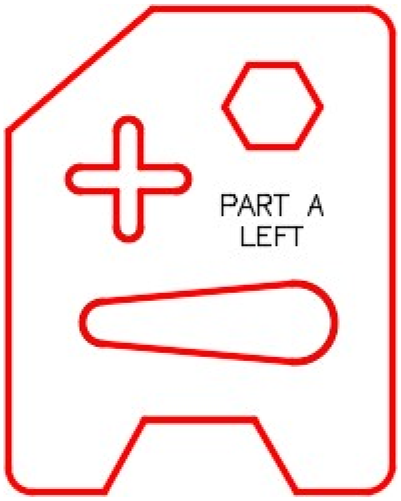


Figure Step 5

Step 6

Enter the MIRRTEXT system variable as shown below and set it to 0.

Command: **MIRRTEXT**

Enter new value for MIRRTEXT <1>: 0

Command:

AUTHOR'S COMMENTS: When the system variable MIRRTEXT is set to 0, text will NOT be mirrored in the MIRROR command.

Step 7

On layer: Construction, draw a line snapping the first end to the end of the lines. The second point can be anywhere on the drawing. (Figure Step 7).

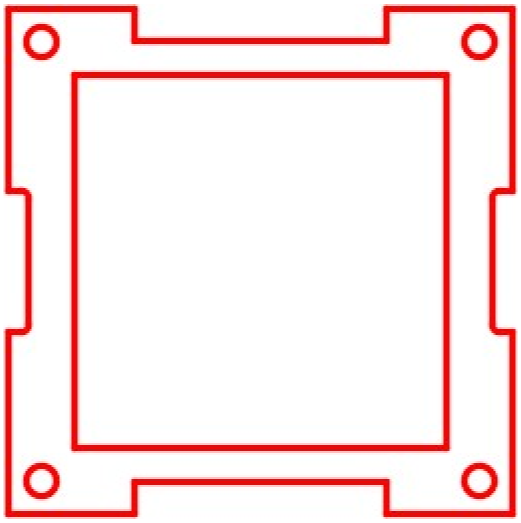


Figure Step 7

AUTHOR'S COMMENTS: Drawing the construction line is a technique I use to locate an exact location on the object. It is best to draw it inclined as shown in the figure. You will see how I use it later in the workalong.

Step 8

Enter the MIRROR command as shown below. (Figure Step 8A, 8B, and 8C)

Command: **MIRROR**

Select objects:

Specify opposite corner:

(Select the objects in a window)

Select objects:

Specify first point of mirror line:

(Select a point in space)

Specify second point of mirror line:

(With ortho enabled, select another point to draw a vertical line.)

Delete source objects? [Yes/No] <N>: **Y**

(Enter Yes)

Command:

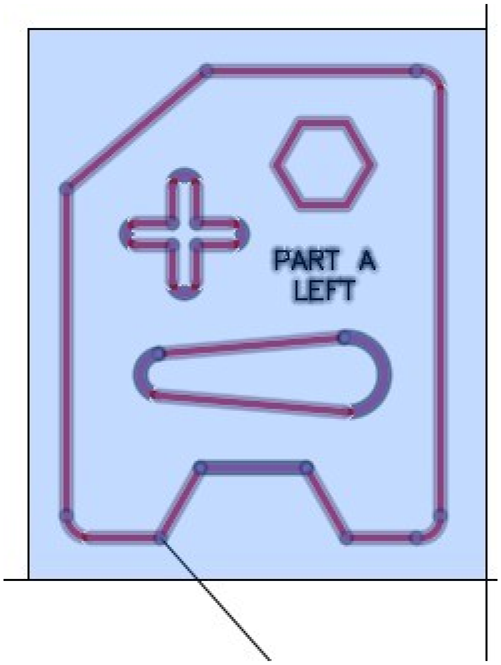


Figure Step 8A

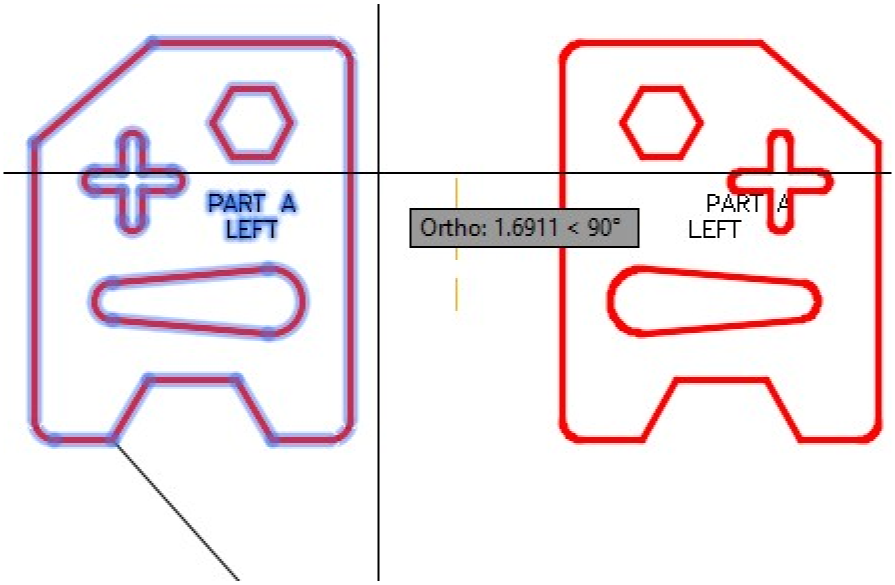

PART A
LEFT
Ortho: 1.6911 < 90°
PART A
LEFT

Figure Step 8B

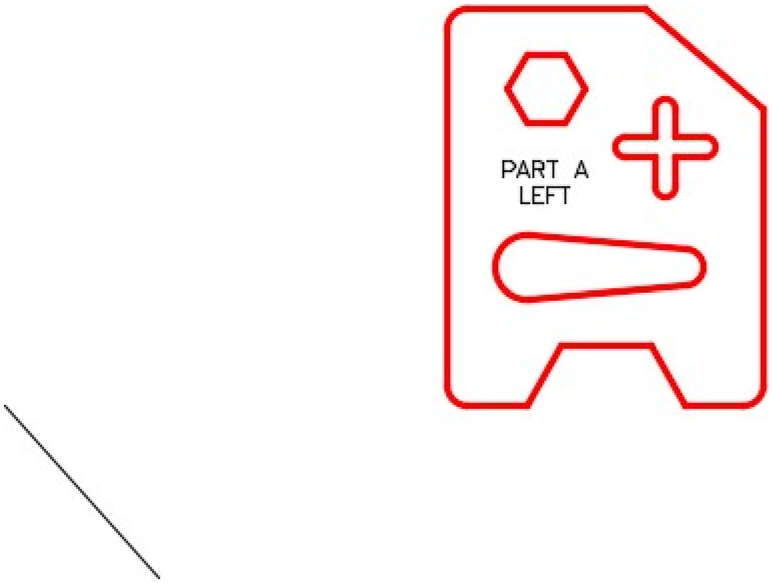


Figure Step 8C

AUTHOR'S COMMENTS: Ensure that you enable Ortho mode when you location the second point of the mirror line. Doing that will ensure that the mirror line is vertical.

Step 9

Using the MOVE command, move the object by snapping to the end of the bottom lines for the base point the end of the construction line for the second point. (Figure Step 9A and 9B)

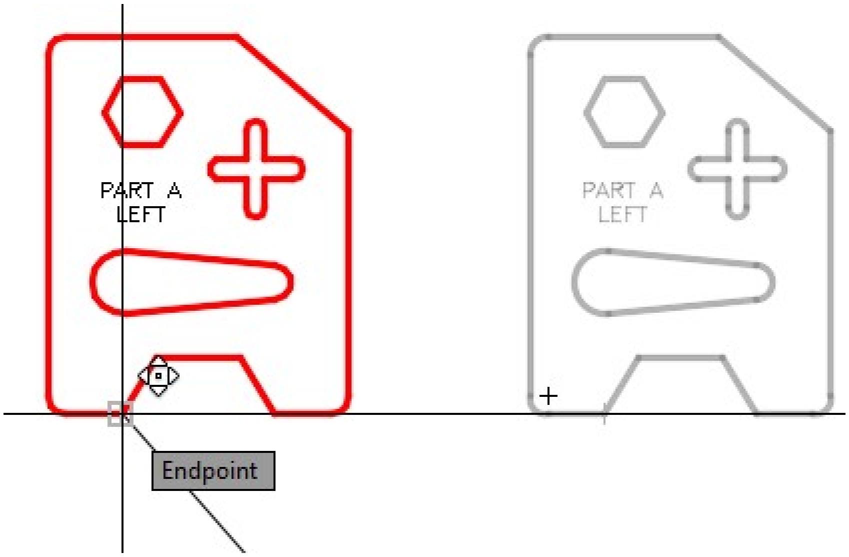


Figure Step 9A

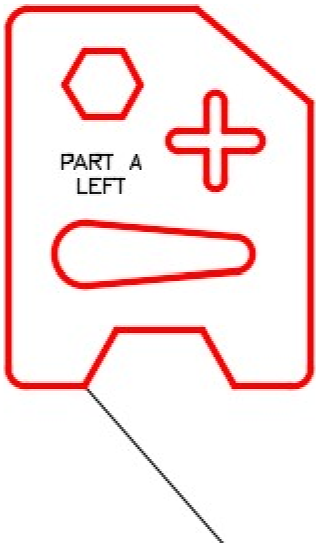


Figure Step 9B

AUTHOR'S COMMENTS: By using the construction line to locate the object, it will be in the exact location as the original object.

Step 10

Edit the text to read PART A RIGHT. Freeze layer: Construction. (Figure Step 10)

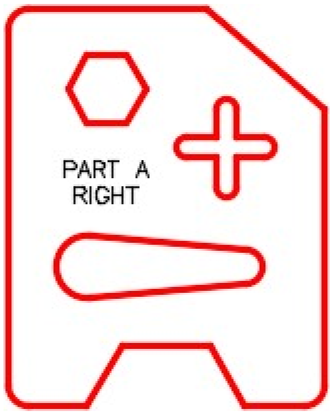


Figure Step 10

AUTHOR'S COMMENTS: It is often necessary to create a left and right part of an object. This workalong shows you a quick method of doing that without redrawing the complete object.

Step 11

Using the SAVEAS command, save the drawing with the name: AutoCAD 2D Workalong 22-2 Right.

Step 12

Save and close the drawing.

Rotating

The ROTATE command is a very useful productivity tool. It rotates selected objects around a base point the you select in the command. The base point is the only location on the rotated objects that is not changed after the rotation. The rotation can be defined by an angle or by a reference angle. By default, a positive angle rotates counterclockwise and a negative angle rotates clockwise.

Motion Paths

A *motion path* is the path along a circle an object would follow if it was rotated around a base point or moved along a line. Study Figure 22-5.

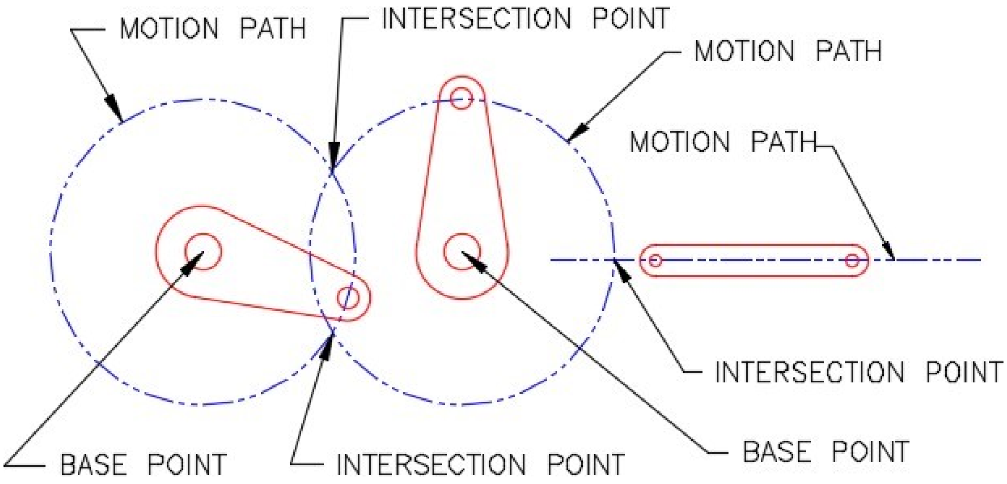
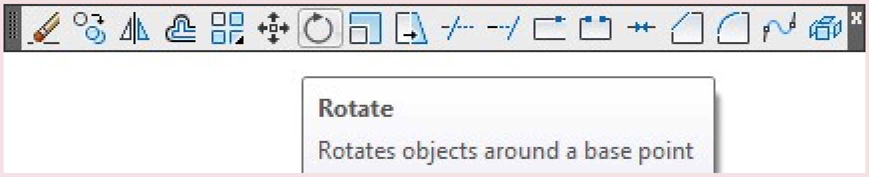

MOTION PATH
INTERSECTION POINT
MOTION PATH
MOTION PATH
INTERSECTION POINT
BASE POINT
INTERSECTION POINT
BASE POINT

Figure 22-5
Motion Paths

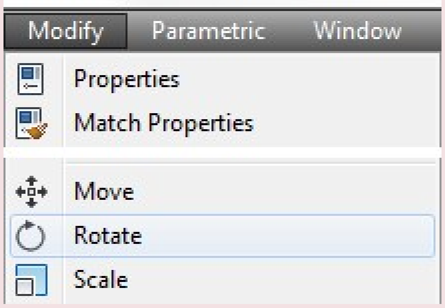
AutoCAD Command: ROTATE

The ROTATE command is used to rotate drawing objects around a base point.

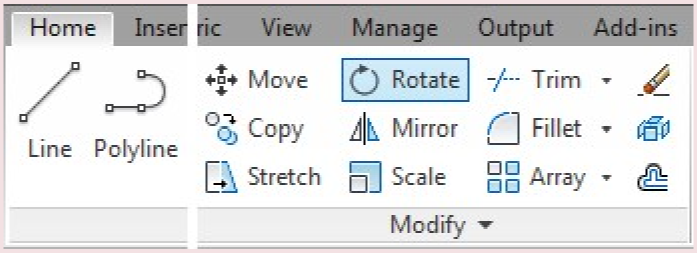
Shortcut: **RO**


Rotate
Rotates objects around a base point

Modify Toolbar


Modify
Parametric
Window
Properties
Match Properties
Move
Rotate
Scale

Modify Pull-down


Home
Inser
ric
View
Manage
Output
Add-ins
Line
Polyline
Move
Rotate
Trim
Copy
Mirror
Fillet
Stretch
Scale
Array
Modify

Home Ribbon

USER TIP: The Match Properties icon is very handy. It takes the properties of the object selected first and changes the properties of the objects selected next to match the one selected first. It is particularly handy for changing text objects. The icons have changed slightly in the different versions of AutoCAD. The icon is on the Standard toolbar.

WORKALONG: Using the ROTATE Command

Step 1

Start a new drawing using the template: 2D Layout English.

Step 2

Save the drawing with the name: AutoCAD 2D Workalong 22-3.

Step 3

On layer: Rotation 1, draw the object shown in the figure. Draw the text on layer Text using the text style: 2D Modules. Locate it by eye. (Figure Step 3A and 3B)

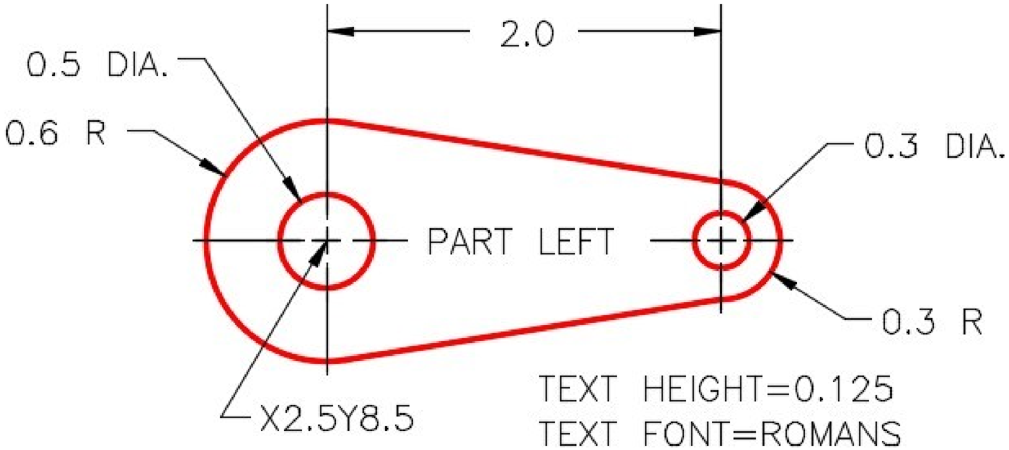

2.0
0.5 DIA.
0.6 R
0.3 DIA.
PART LEFT
0.3 R
X2.5Y8.5
TEXT HEIGHT=0.125
TEXT FONT=ROMANS

Figure Step 3A

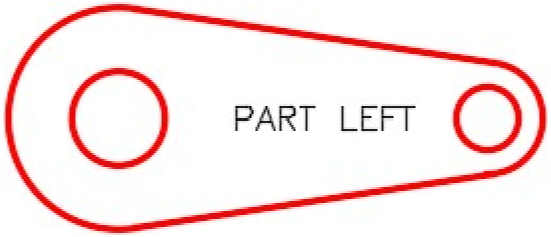


Figure Step 3B

Step 4

Enter the ROTATE command, as shown below, to rotate the object 45 degrees counterclockwise. (Figure Step 4A, 4B, and 4C)

Command: **ROTATE**

Current positive angle in UCS: ANGDIR=counterclockwise ANGBASE=0

Select objects:

Specify opposite corner: 7 found

(Select all objects using a window.)

Select objects:

Specify base point: *(cen)*

(Snap to the center of the circle on the left for the base point.)

Specify rotation angle or [Copy/Reference] <0>: **45**

(The positive rotation angle of 45 degrees is counterclockwise.)

Command:

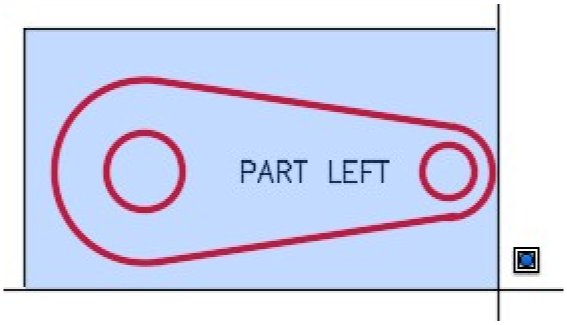


Figure Step 4A

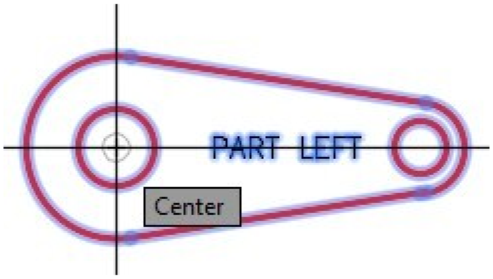


Figure Step 4B

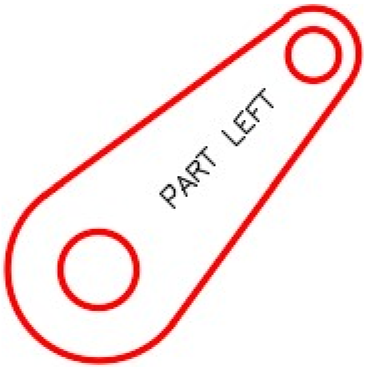


Figure Step 4C

AUTHOR'S COMMENTS: The base point in Step 4 is the center of the large circle. The base point is very important in the ROTATE command. It is the only location on the rotated object that is not relocated after the rotation.

Step 5

On layer: Construction, draw a 3.5 inch line in the positive X direction. For the first point, snap to the center of the circle. (Figure Step 5)

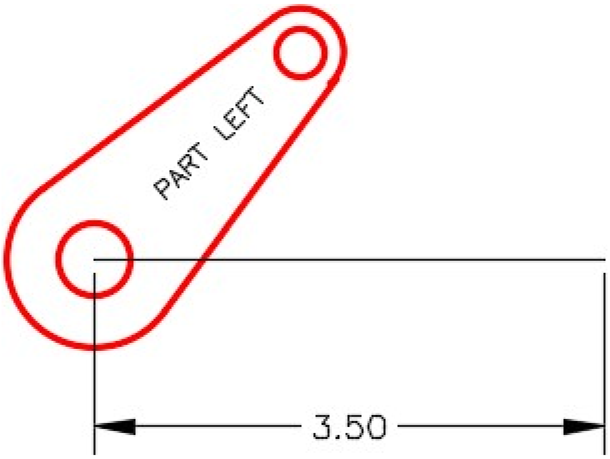


Figure Step 5

Step 6

Using what you just learned in the last workalong, mirror the object to match the figure. Snap to the midpoint of the line for first mirror line location. With Ortho mode enable, select the second point vertical. (Figure Step 6A, 6B, and 6C)

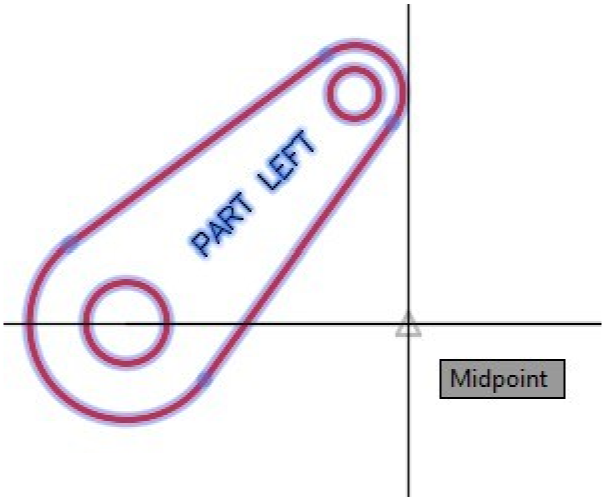


Figure Step 6A

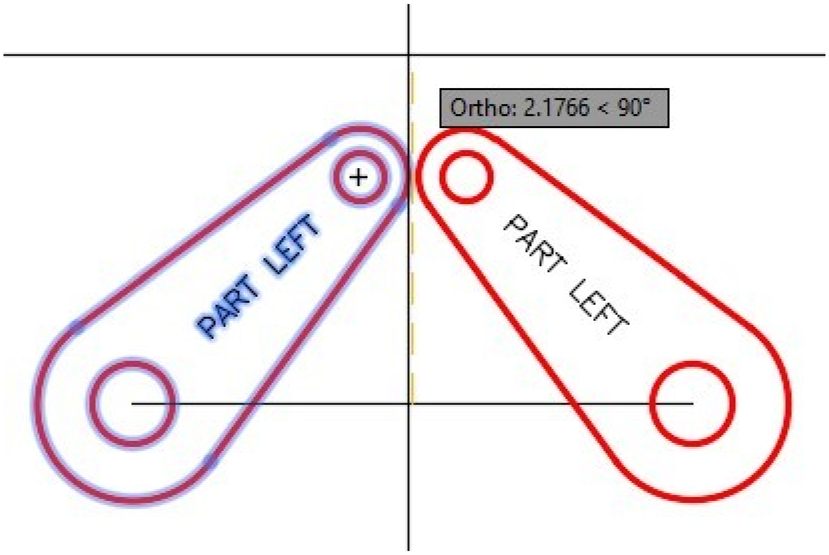

Ortho: 2.1766 < 90°
PART LEFT
PART LEFT

Figure Step 6B

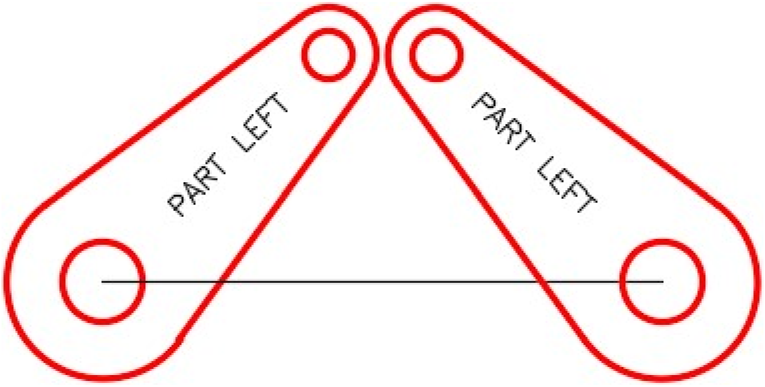

PART LEFT
PART LEFT

Figure Step 6C

Step 7

Freeze layer: Construction. Edit the text on the mirrored object to PART RIGHT. (Figure Step 7)

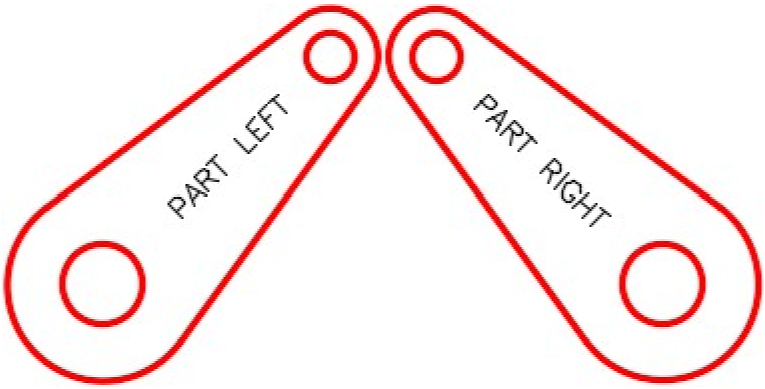

PART LEFT
PART RIGHT

Figure Step 7

Step 8

On layer: Motion Path 1, draw the motion path circle. Snap to the center of the lower left circle for the center point and the center of the upper left circle for the radius. (Figure Step 8)

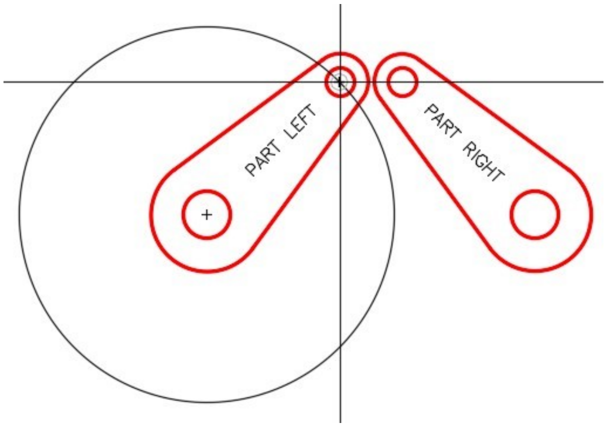


Figure Step 8

Step 9

Using what you just learned in Step 8, draw the motion path circle for the right arm. (Figure Step 9)

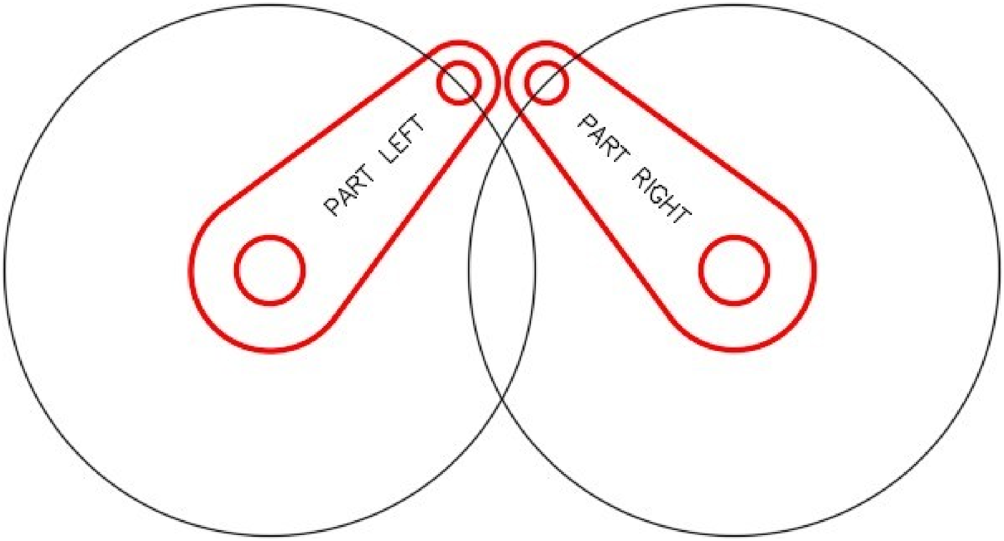


Figure Step 9

AUTHOR'S COMMENTS: The circles you just drew are motion paths for the arms assuming the base point is the large circle of the arms.

Step 10

Enter the ROTATE command, as shown below, to rotate the left arm using the Reference option. (Figure Step 10A, 10B , 10C, 10D, 10E, and 10F)

Command: **RO**

Current positive angle in UCS: ANGDIR=counterclockwise ANGBASE=0

Select objects:

Specify opposite corner: 7 found

(Select the objects in a window.)

Select objects:

Specify base point: *(cen)*

(Selecting the center of the lower circle for base point.)

Specify rotation angle or [Copy/Reference] <90>: **R**

(Use a Reference.)

Specify the reference angle <0>: *(cen)*

(Specifying the center of the large circle (the base point) as the first point of the reference.)

Specify second point: *(cen)*

(Specifying the center of the small circle as the second point of the reference.)

Specify the new angle or [Points] <0>: *(int)*

(Specifying the new angle (the intersection of the motion paths). AutoCAD will find the rotation angle using the select location.)

Command:

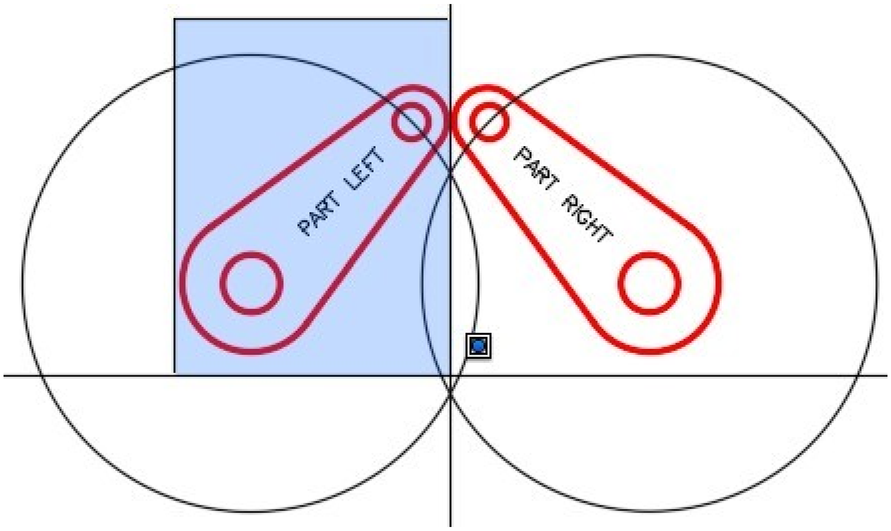


Figure Step 10A

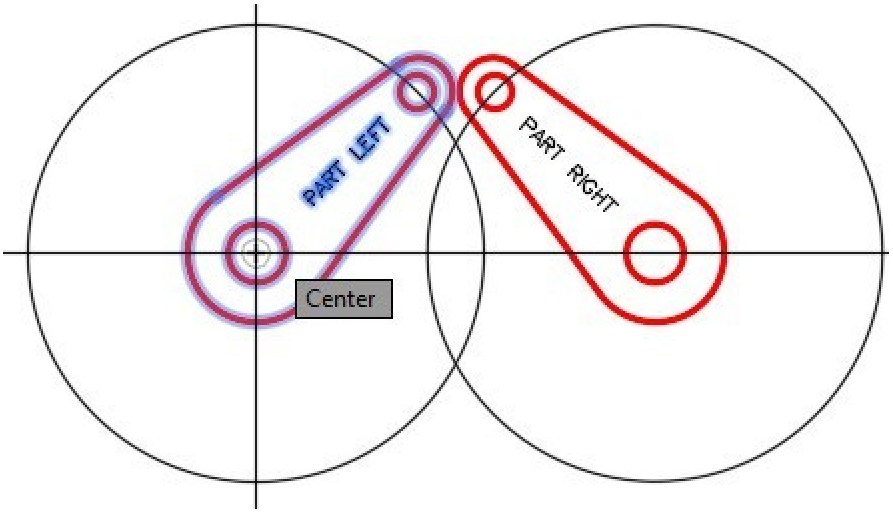

PART LEFT
PART RIGHT
Center

Figure Step 10B

AUTHOR'S COMMENTS: Take note how you must select the center of the large circle twice. The first time to locate the base point and the second time to locate the first point of the reference. The second selection is often missed by beginners.

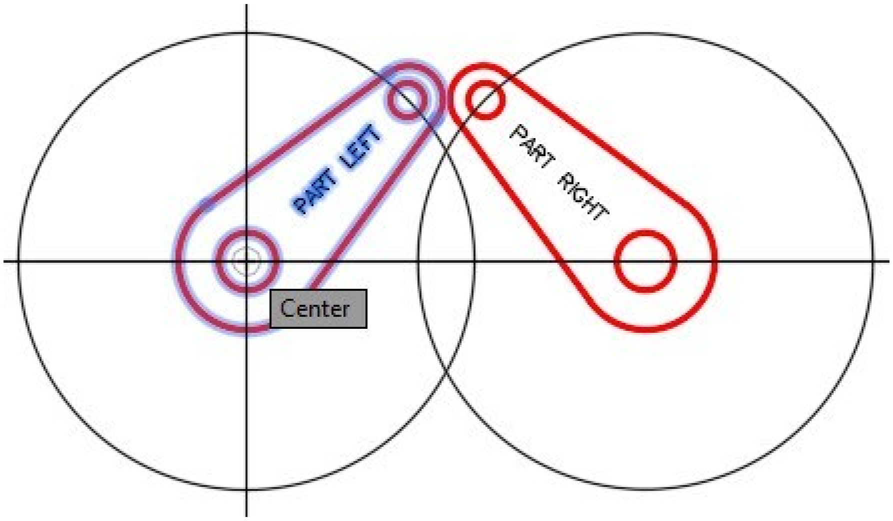

PART LEFT
PART RIGHT
Center

Figure Step 10C

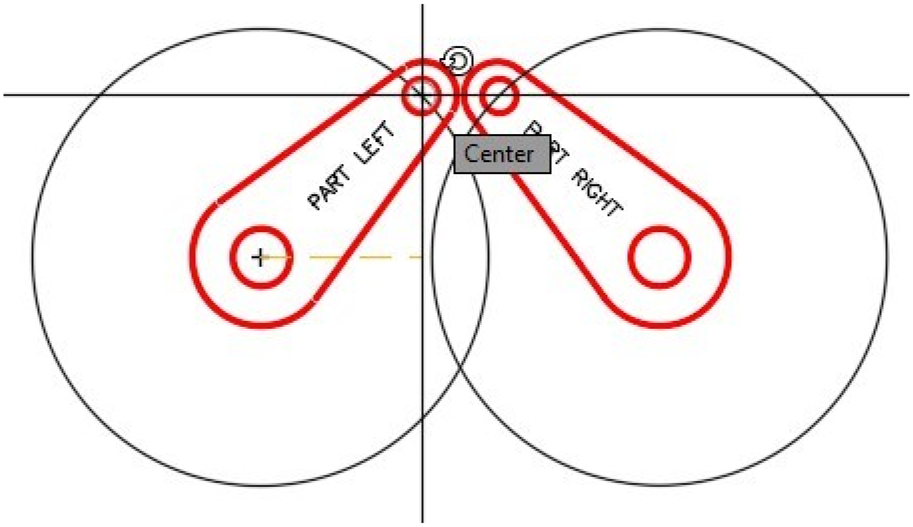

PART LEFT
PART RIGHT
Center

Figure Step 10D

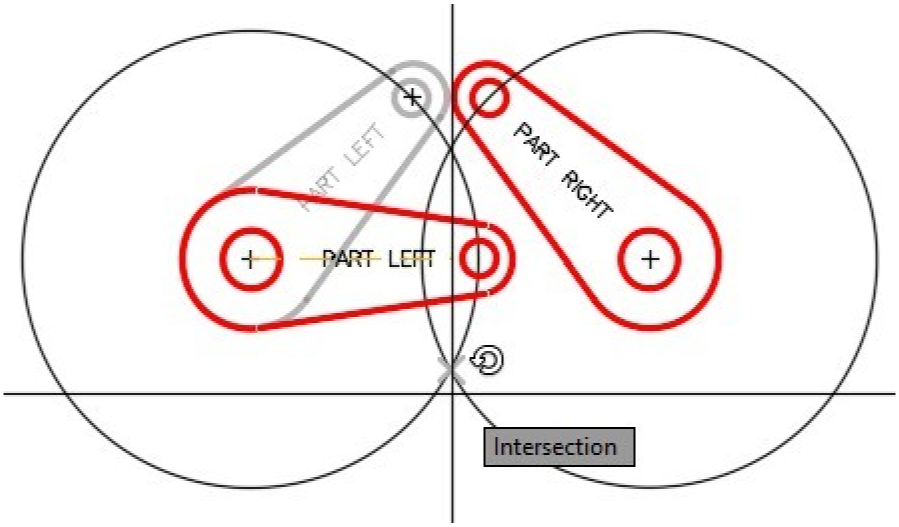

PART LEFT
PART LEFT
PART RIGHT
Intersection

Figure Step 10E

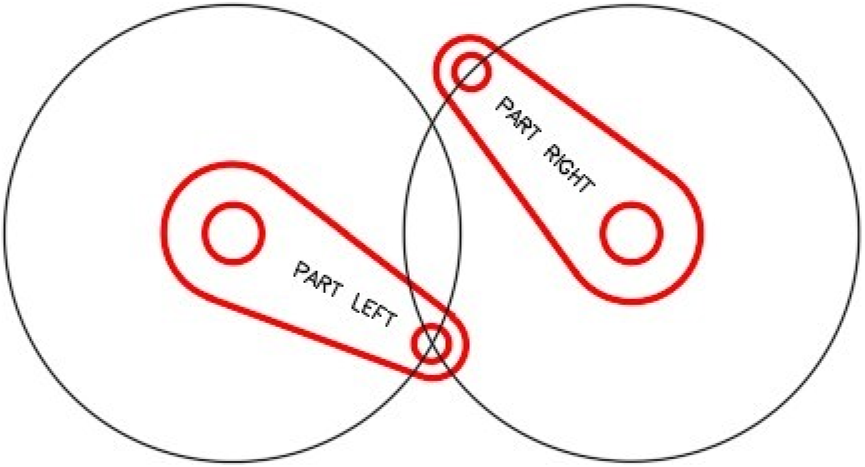

PART LEFT
PART RIGHT

Figure Step 10F

Step 11

Using what you just learned, rotate PART RIGHT to match PART LEFT. (Figure Step 11)

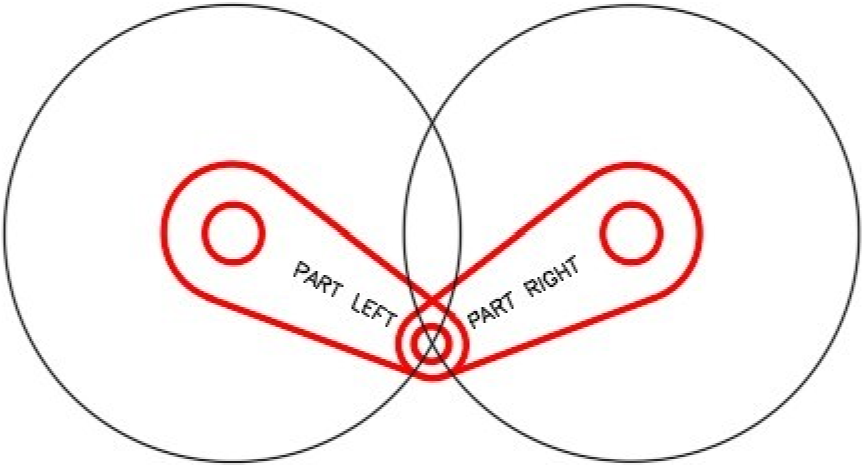

PART LEFT
PART RIGHT

Figure Step 11

Step 12

Using the COPY command, copy only the arms beside the originals. It is not important where you locate the copy. (Figure Step 12)

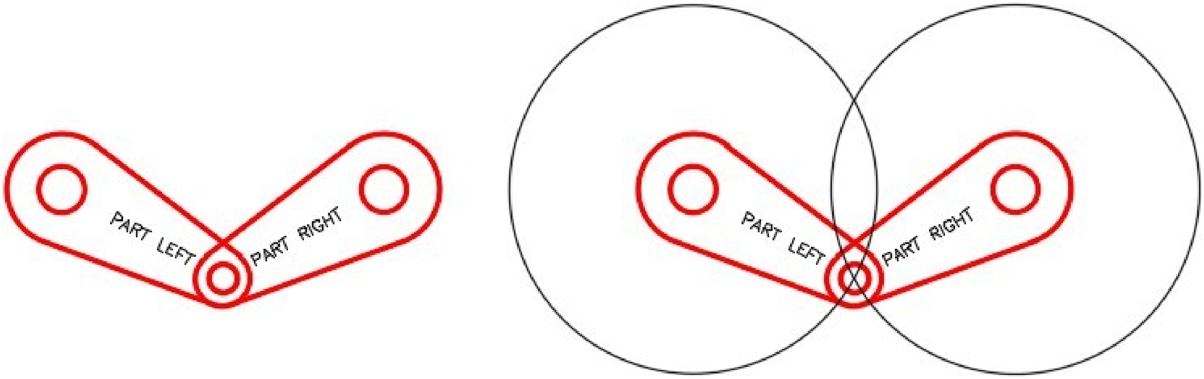

PART LEFT
PART RIGHT
PART LEFT
PART RIGHT

Figure Step 12

Step 13

Change the layer of the copied arm to layer: Rotation 2. (Figure Step 13)

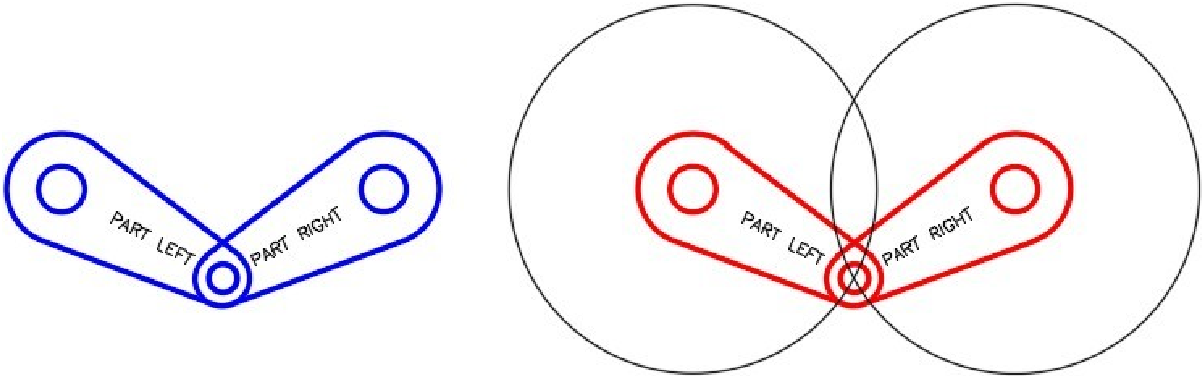

PART LEFT
PART RIGHT
PART LEFT
PART RIGHT

Figure Step 13

Step 14

Using the MOVE command, move the copied arms back to their original position on top of the original arms. Ensure that you locate them exactly on the originals by snapping the center of a circle to the center of matching circle. (Figure Step 14)

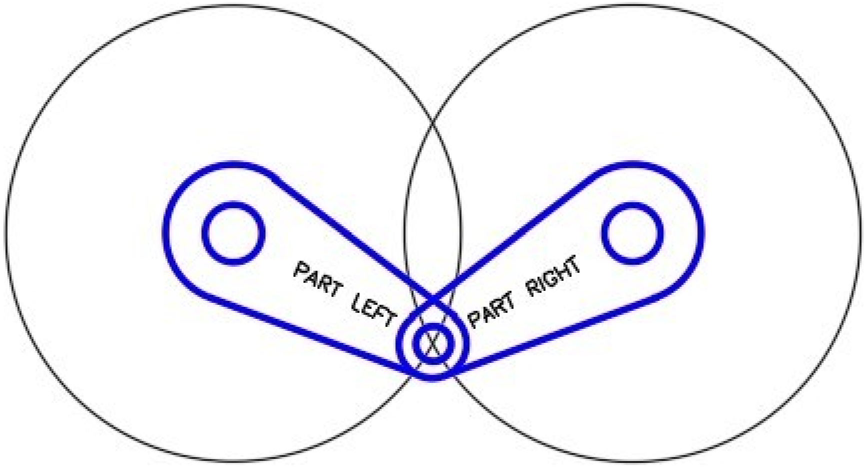

PART LEFT
PART RIGHT

Figure Step 14

AUTHOR’S COMMENTS: There are now two copies of the arms at the exact same location, but on different layers.

Step 15

Turn layer: Rotation 1 off. Using what you learned earlier in the workalong, rotated the arms on layer: Rotation 2 to match the figure. (Figure Step 15)

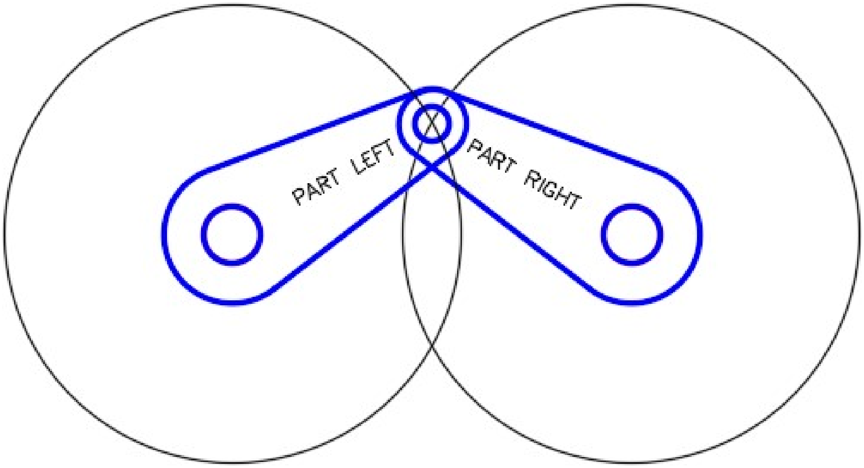

PART LEFT
PART RIGHT

Figure Step 15

Step 16

Turn layer:

Rotation 1 on. Your drawing should match the figure. (Figure Step 16)

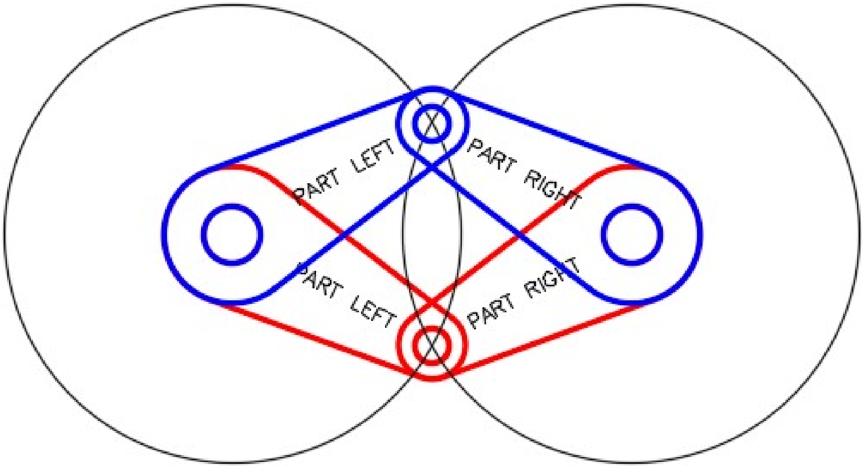

PART LEFT
PART RIGHT
PART LEFT
PART RIGHT

Figure Step 16

Step 17

Save and close the drawing.

Key Principles

Key Principles in Module 22

1. When an object is mirrored, the object that is created in the mirror will retain all the properties of the object being mirrored.
2. After mirroring, ensure that you correct any lines that appear full length but are actually two lines. You can do this by deleting one of the lines and extending the other one.
3. The ROTATE command is a very useful productivity tool. It rotates objects around a base point selected in the command. The base point is the only location on the rotated object that is not changed after the rotation.

Lab Exercise 22-1

Time allowed: 50 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 22-1	2D Layout English	Inches

Step 1

On layer: Object, draw the object shown in the figure. Start by drawing one quarter of the object, then mirror it once and then again to complete the drawing. (Figure Step 1A and 1B)

Step 2

Erase and extend all of the lines that were mirrored and are in two pieces.

Step 3

Set the insertion units and insert the key. Check the drawing for accuracy.

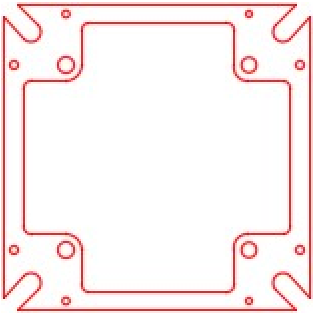


Figure Step 1A
Completed Drawing

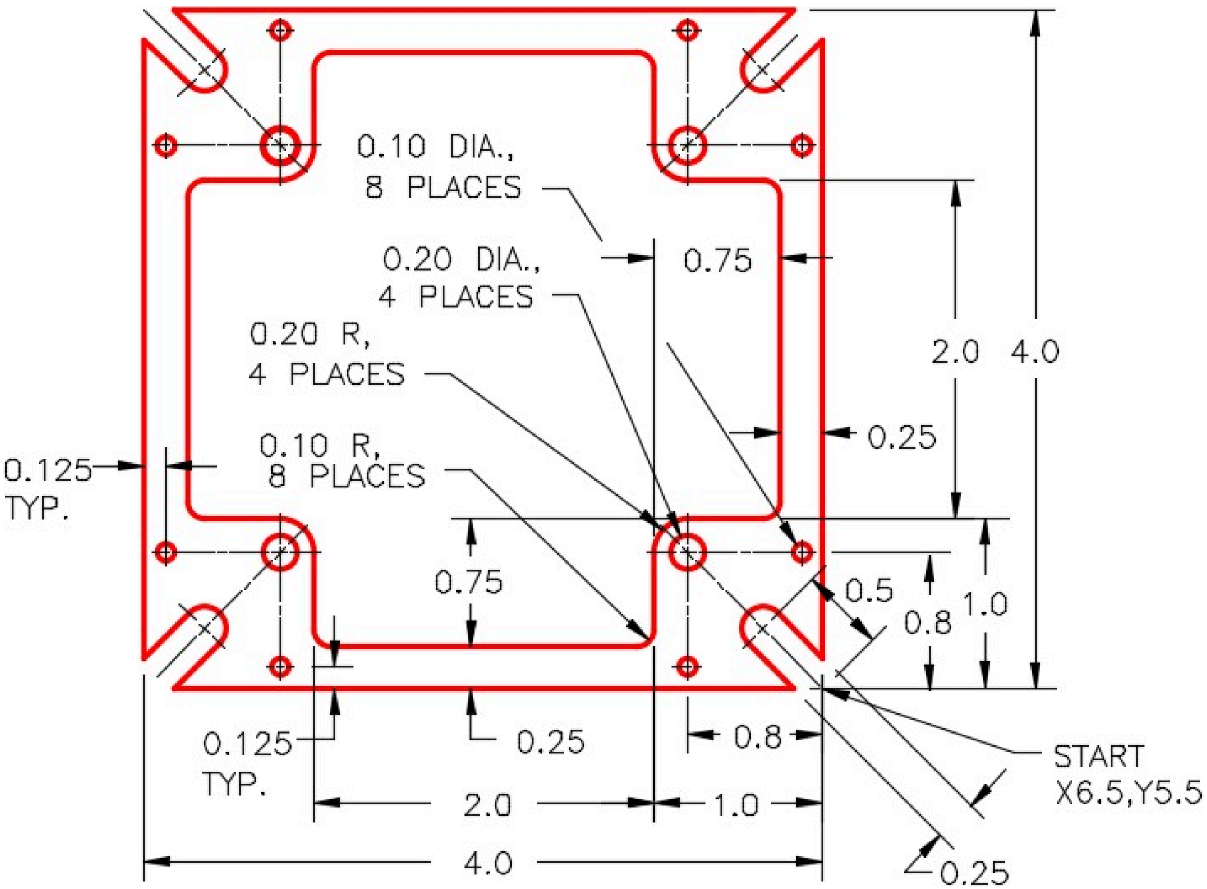
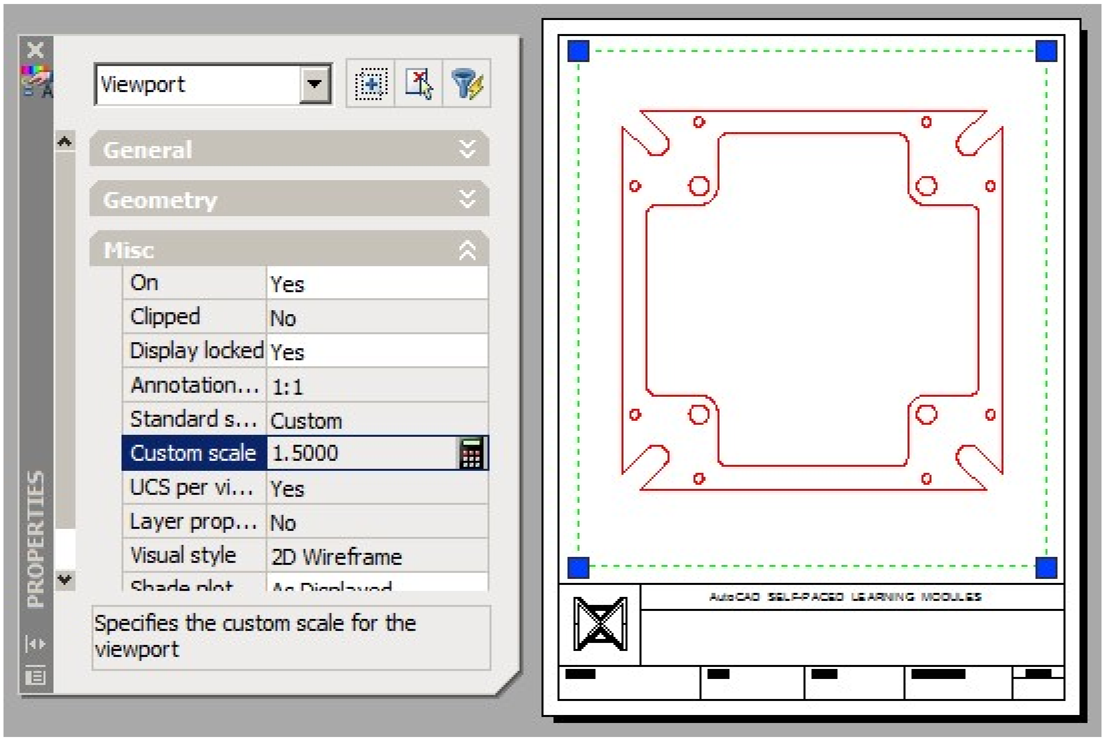

0.10 DIA.,
8 PLACES
0.20 DIA.,
4 PLACES
0.75
0.20 R,
4 PLACES
2.0
4.0
0.25
0.125
TYP.
0.10 R,
8 PLACES
0.75
0.5
0.8
1.0
0.125
TYP.
0.25
0.8
START
X6.5,Y5.5
2.0
1.0
4.0
0.25

Figure Step 1B

Step 4

On layout: Module Layout A, create a viewport on layer: Viewport. Set the scale of the viewport to 1.5 and lock the display. (Figure Step 4)



Viewport

General

Geometry

Misc

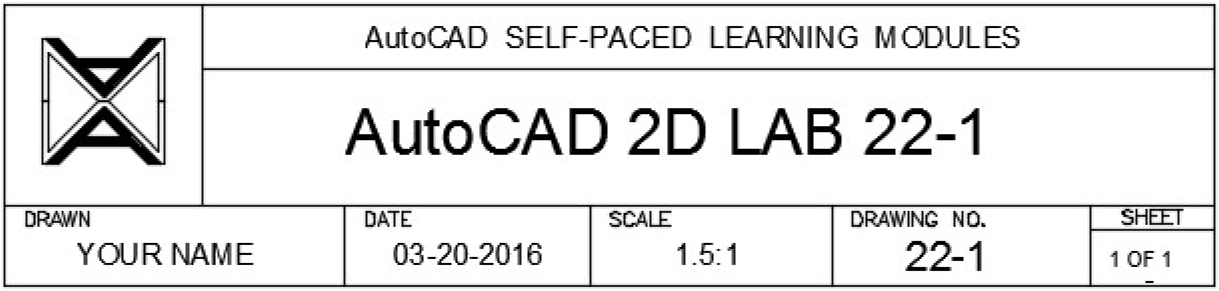
On	Yes
Clipped	No
Display locked	Yes
Annotation...	1:1
Standard s...	Custom
Custom scale	1.5000
UCS per vi...	Yes
Layer prop...	No
Visual style	2D Wireframe

Specifies the custom scale for the viewport

Figure Step 4

Step 5

Complete the titleblock using the AutoCAD book Titleblock Standards shown in Module 20. (Figure Step 5)



AutoCAD SELF-PACED LEARNING MODULES

AutoCAD 2D LAB 22-1

DRAWN	DATE	SCALE	DRAWING NO.	SHEET
YOUR NAME	03-20-2016	1.5:1	22-1	1 OF 1

Figure Step 5

Step 6

Turn layer: Viewport off. (Figure Step 6)

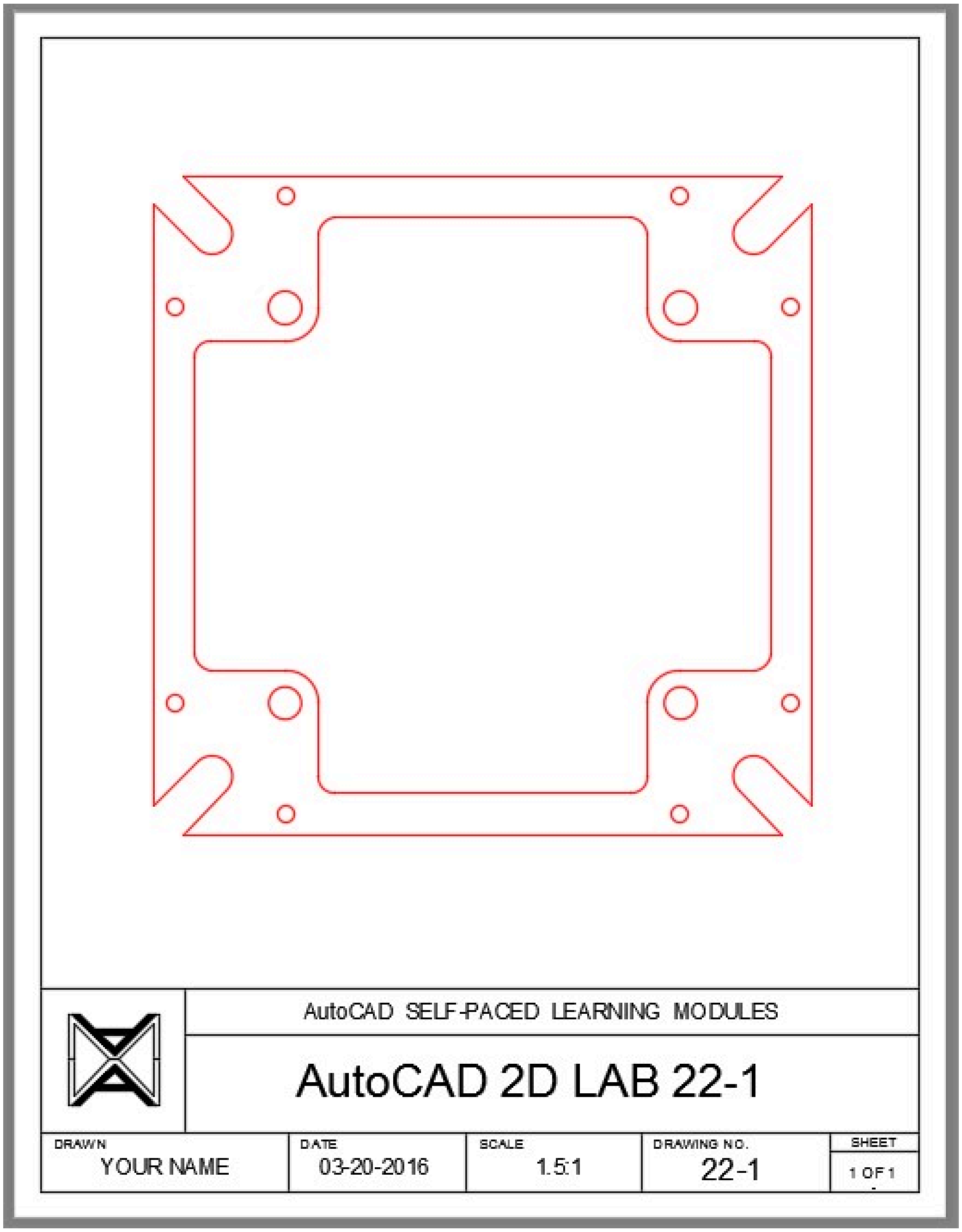

AutoCAD SELF-PACED LEARNING MODULES
AutoCAD 2D LAB 22-1
DRAWN
YOUR NAME
DATE
03-20-2016
SCALE
1.5:1
DRAWING NO.
22-1
SHEET
1 OF 1

Figure Step 6

Step 7

Save and close the drawing.

Lab Exercise 22-2

Time allowed: 60 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 22-2	2D Layout Metric	Millimeters

Step 1

On layer: Object, draw the rectangle shown in the figure. (Figure Step 1)

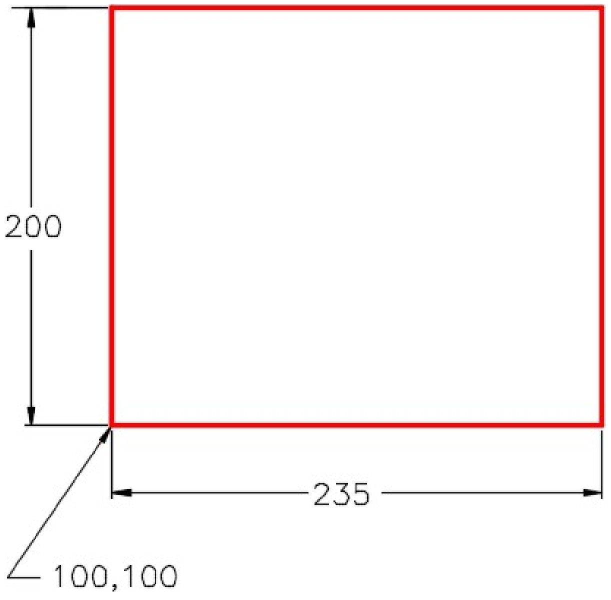


Figure Step 1

Step 2

On layer: Rotation 1, draw the object shown in Figure step 2A using the dimensioned drawing as a reference. Draw it anywhere beside the rectangle. On layer: Text, insert the text PART A using the text style: 2D Modules. Locate the text by eye and select a text height to match the figure. (Figure Step 2A and 2B)

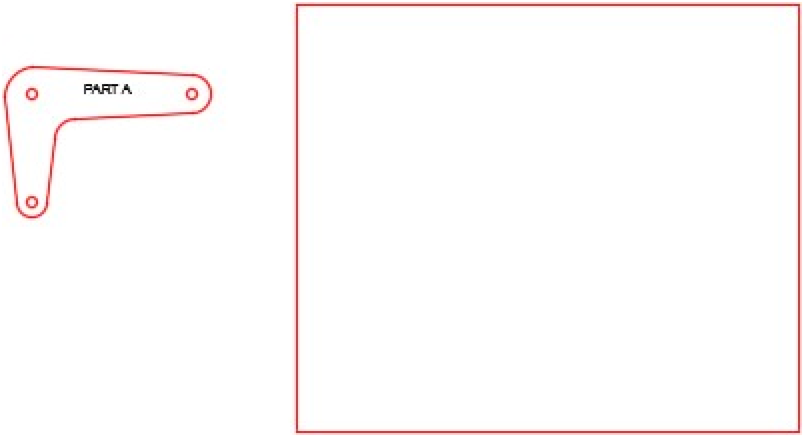


Figure Step 2A

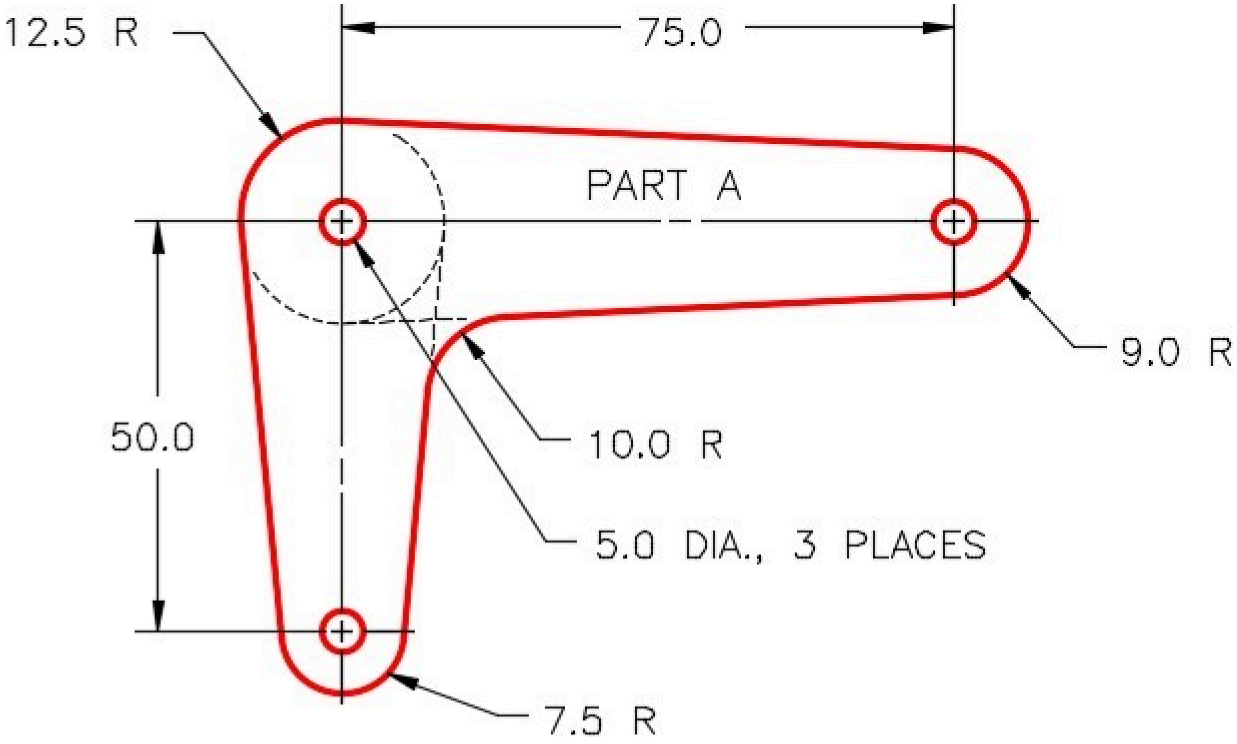

12.5 R
75.0
PART A
9.0 R
50.0
10.0 R
5.0 DIA., 3 PLACES
7.5 R

Figure Step 2B

Step 3

Mirror the object PART A. The location is not important but ensure that the mirror line is horizontal. Edit the text to PART B. (Figure Step 3)

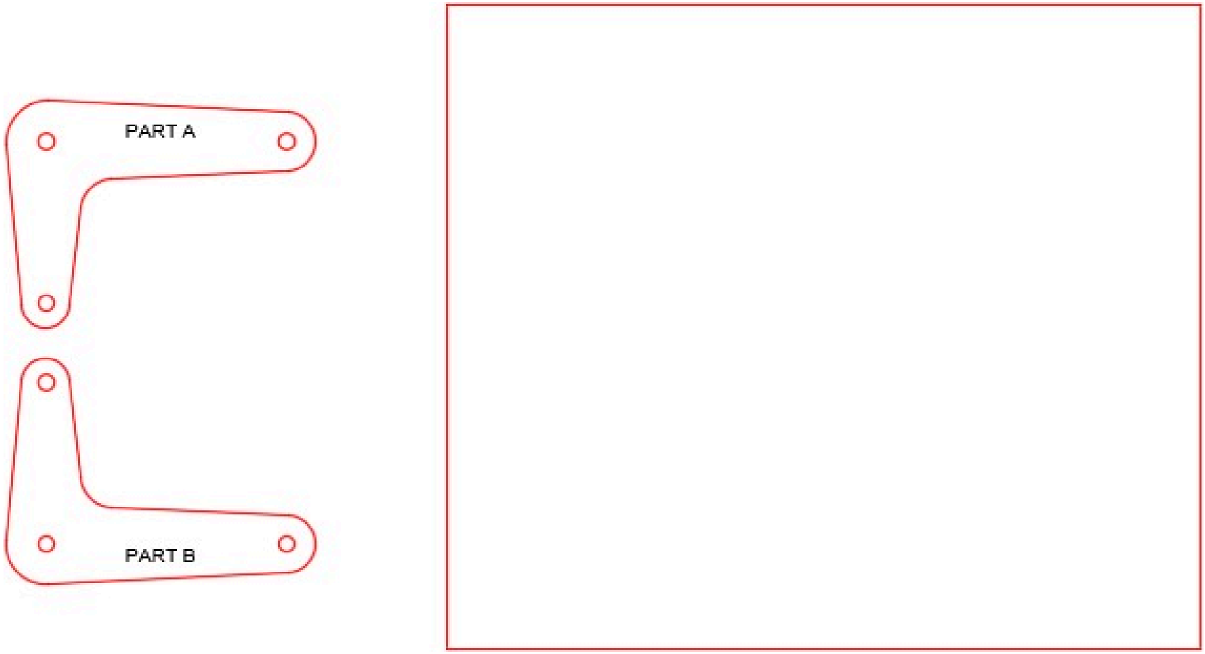

PART A
PART B

Figure Step 3

Step 4

Using the dimensioned drawing shown in the figure as a reference, move PART A and PART B to locate them as shown in the figure. (Figure Step 4)

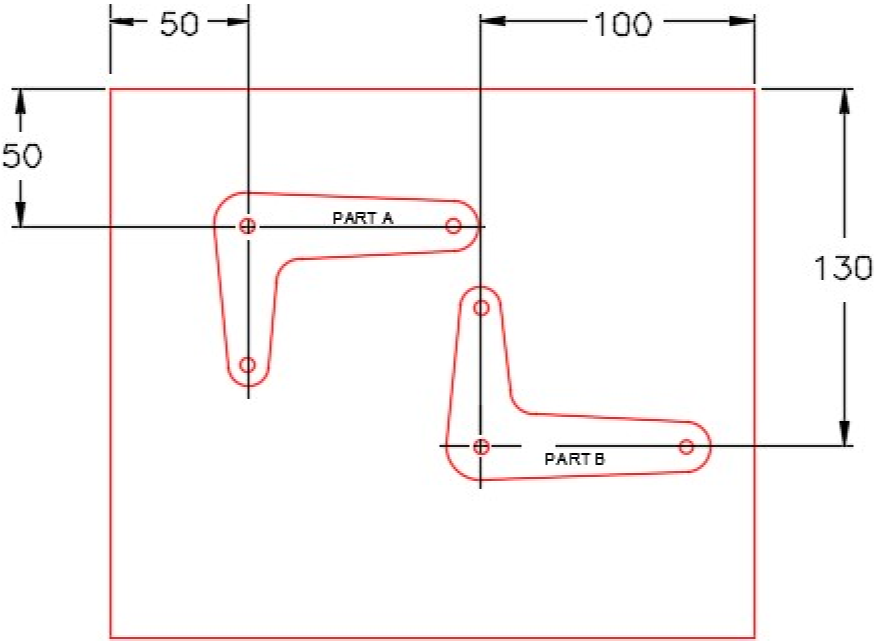

50
100
50
PART A
130
PART B

Figure Step 4

AUTHOR'S COMMENTS: Draw construction lines and snap the center of the circle to intersection of the construction lines.

Step 5

On layer: Motion Path 1, draw the circles for the motion paths. Rotate the arms to locate them as shown in the figure. (Figure Step 5A and 5B)

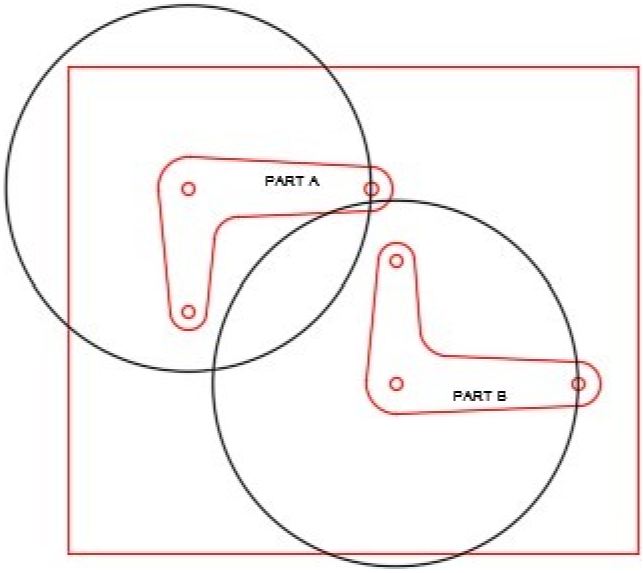


Figure Step 5A

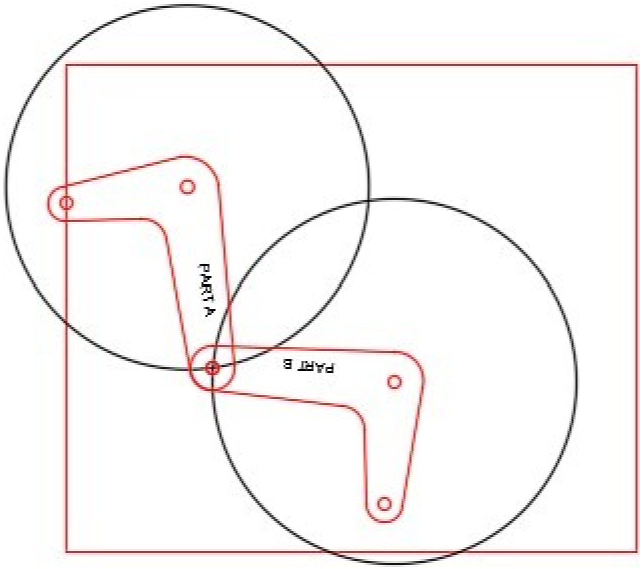


Figure Step 5B

Step 6

Turn layer: Motion Path 1 off. Copy both arms anywhere outside of the rectangle. Change the layer of the copied arms to layer: Rotation 2. (Figure Step 6)

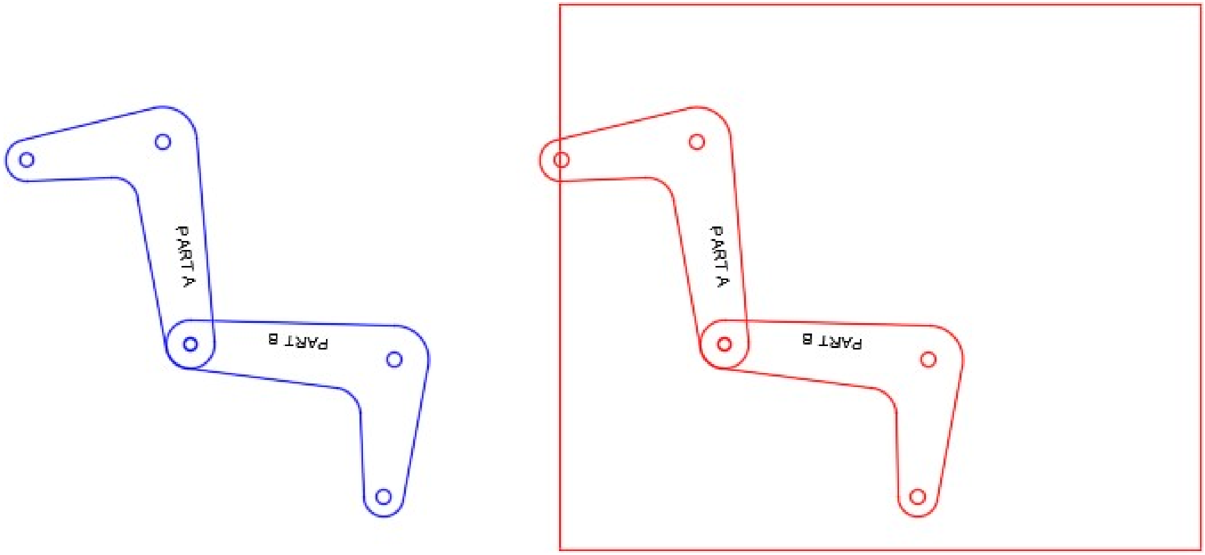

PART A
PART B
PART A
PART B

Figure Step 6

Step 7

Move the arms back to their exact original location. Turn layer: Rotation 1 off. (Figure Step 7)

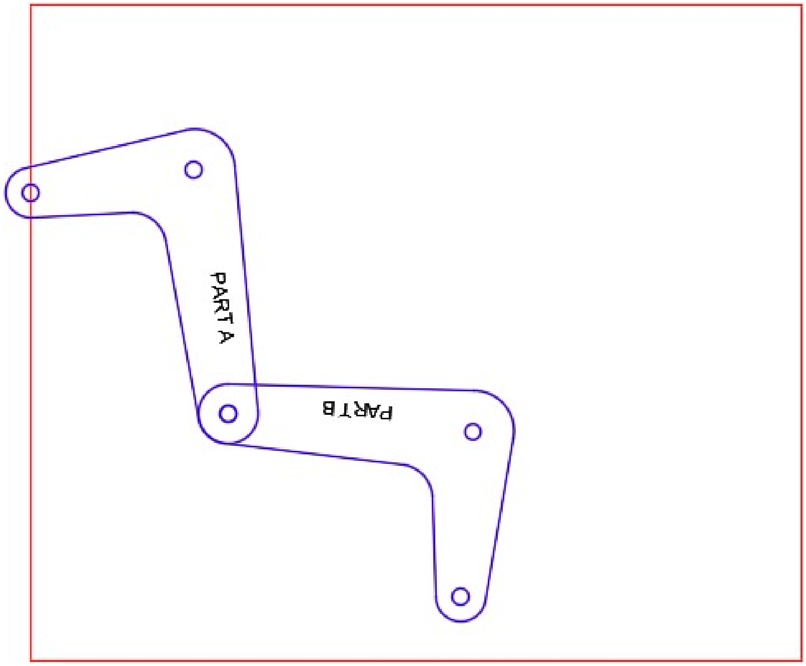

PART A
PART B

Figure Step 7

Step 8

On layer: Motion Path 2, draw the motion path circle as shown in the figure. (Figure Step 8)

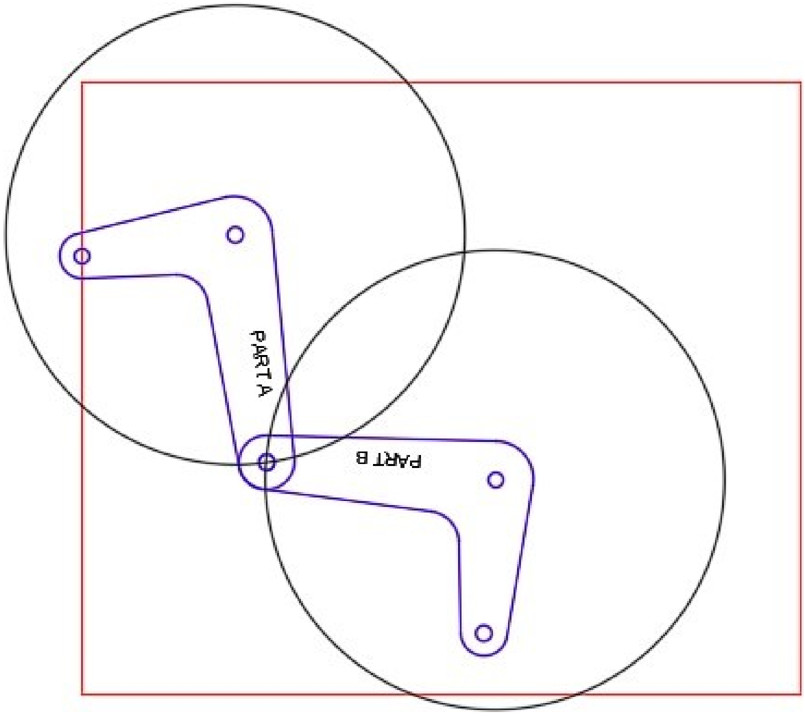


Figure Step 8

Step 9

Rotate the arms as shown in the figure. (Figure Step 9)

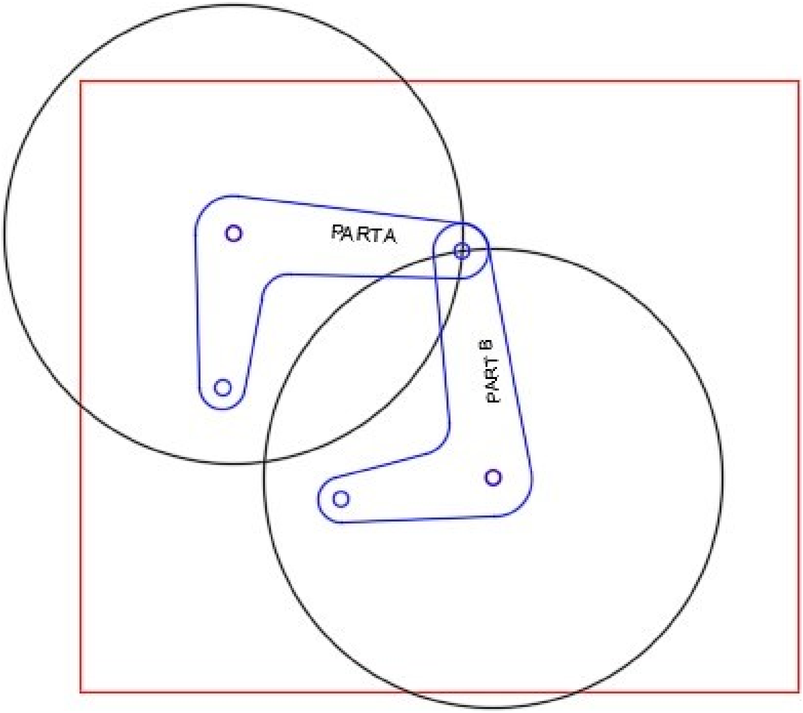

PART A
PART B

Figure Step 9

Step 10

Turn layers: Rotation 1, Rotation 2, and Text on and all other layers off. Your drawing should match the figure. (Figure Step 10)

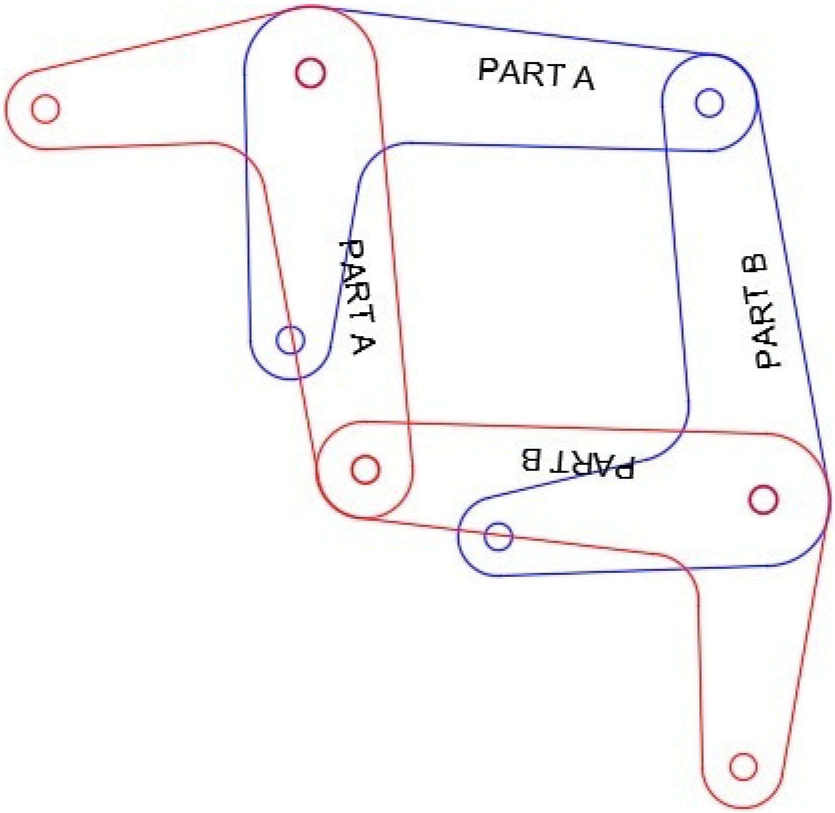


Figure Step 10

Step 11

Set the insertion units and insert the key to check your drawing for accuracy.

Step 12

Save and close the drawing.

Module 23: Arraying

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe how drawing objects are arrayed in both polar and rectangular patterns.
2. Apply the ARRAYCLASSIC command to array objects in polar or rectangular patterns.
3. Describe and apply five additional the methods of selecting objects; window polygon, cross polygon, fence, lasso window, and lasso cross window.
4. Describe how to find the center of and insert a circle inside a regular polygon.

Arraying

The ARRAY command is used to create multiple copies of objects in a *polar* or *rectangular* pattern. See Figure 23-1 and 23-2. It is another one of those powerful commands that, when used at the right times, can prove to be very productive.

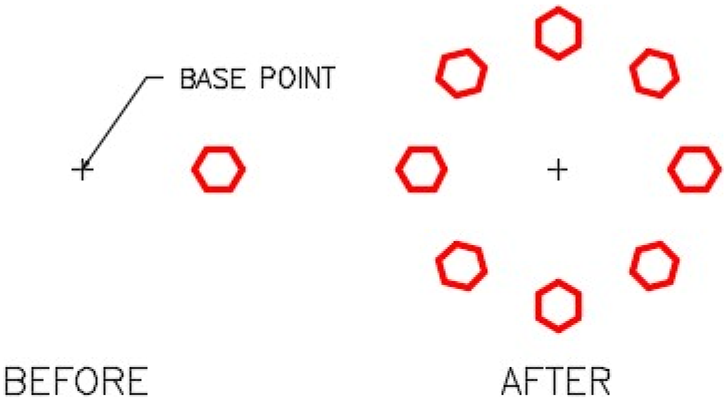


Figure 23-1 Polar Array

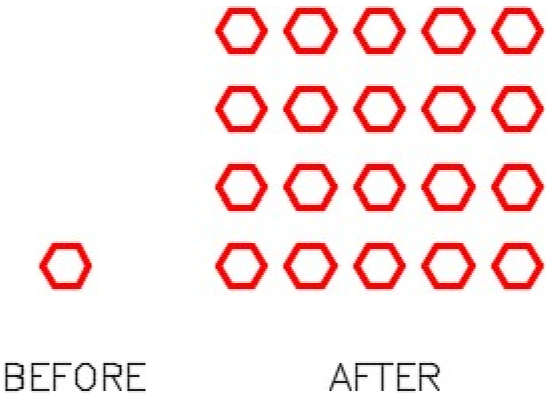


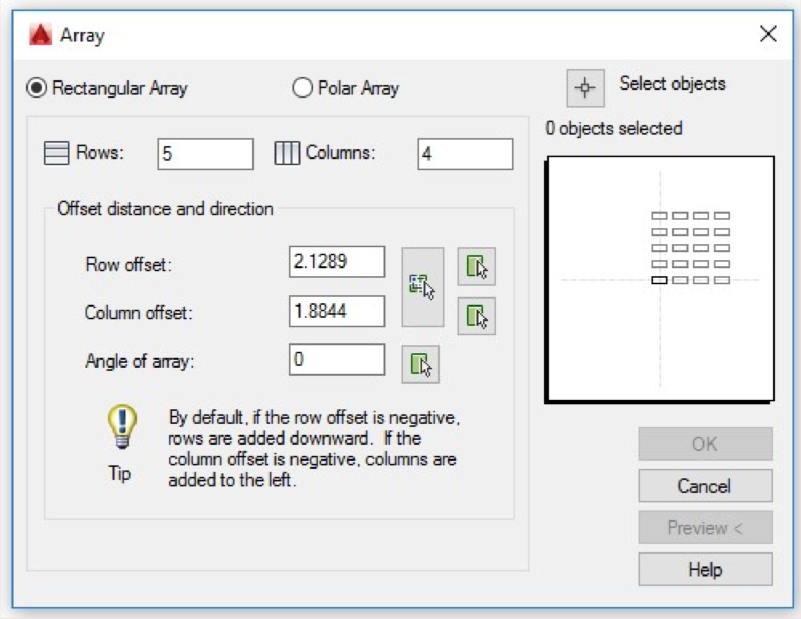
Figure 23-2 Rectangular Array

MUST KNOW: When an object is arrayed, all objects created in the array will retain the properties of the original object, even the layer the original object resides on. This happens regardless of the current layer when the ARRAY command is executed.

AutoCAD Command: ARRAYCLASSIC

The ARRAYCLASSIC command displays a dialogue box that is used to create multiple copies of objects either in a rectangular or a polar pattern.

Shortcut: **none**



● Rectangular Array ○ Polar Array

0 objects selected

Rows: 5

Columns: 4

Offset distance and direction

Row offset: 2.1289

Column offset: 1.8844

Angle of array: 0

Tip: By default, if the row offset is negative, rows are added downward. If the column offset is negative, columns are added to the left.

ARRAYCLASSIC Dialogue Box

WORKALONG: Using the ARRAY Command – Part 1

Step 1

Start a new drawing using the template: 2D Layout English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 23-1.

Step 3

On layer: Construction, draw the two circles shown in the figure. (Figure Sep 3)

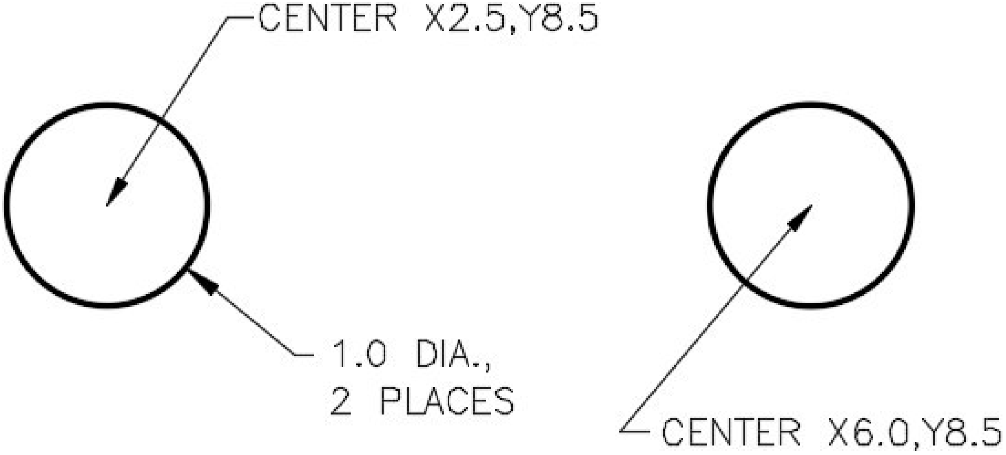


Figure Sep 3

Step 4

On layer: Text, using the text style: 2D Modules, insert the text A and B. Select the location of the text by eye and use a text height to match the figure as close as possible. (Figure Step 4)

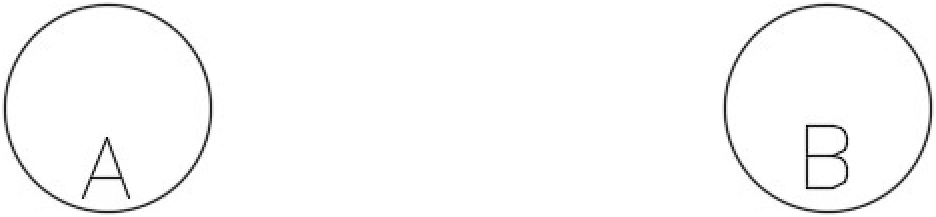


Figure Step 4

Step 5

On layer: Object, draw a 0.35 diameter circle and two lines snapping them from quad to quad as shown in the figure. Draw it anywhere on the drawing. (Figure Step 5)

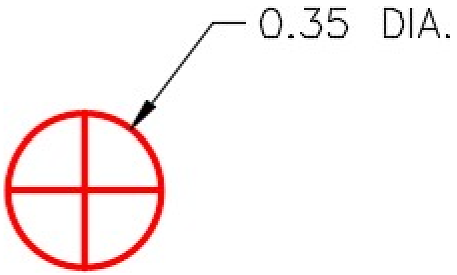


Figure Step 5

Step 6

Using the COPY command, copy the circle you drew in Step 5, four times, to the absolute coordinates shown in the figure. Add the text C and D using the same specifications as the text in Step 4. (Figure Step 6)

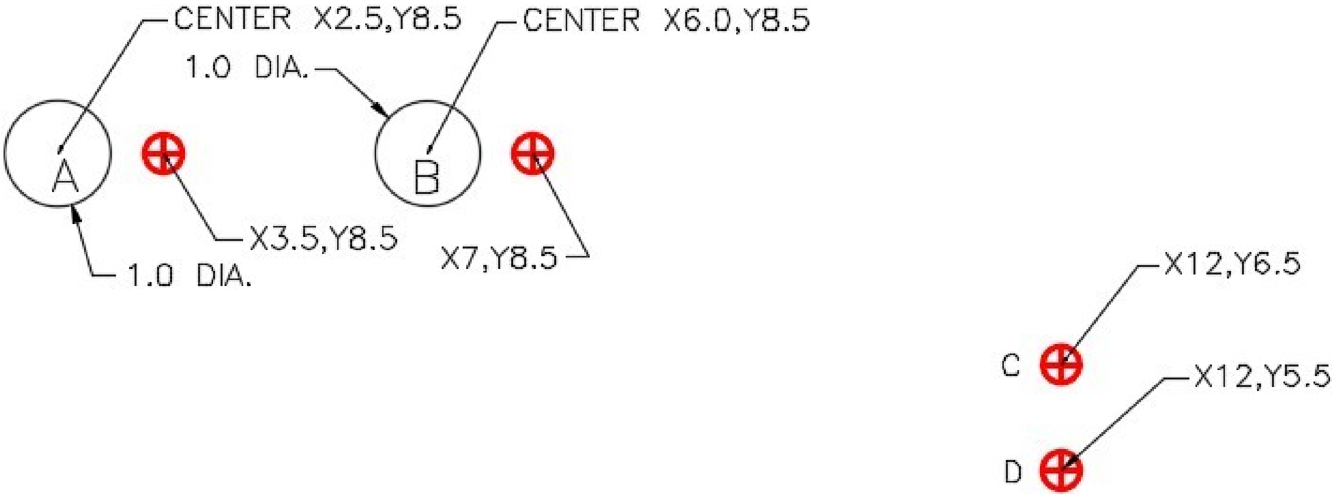
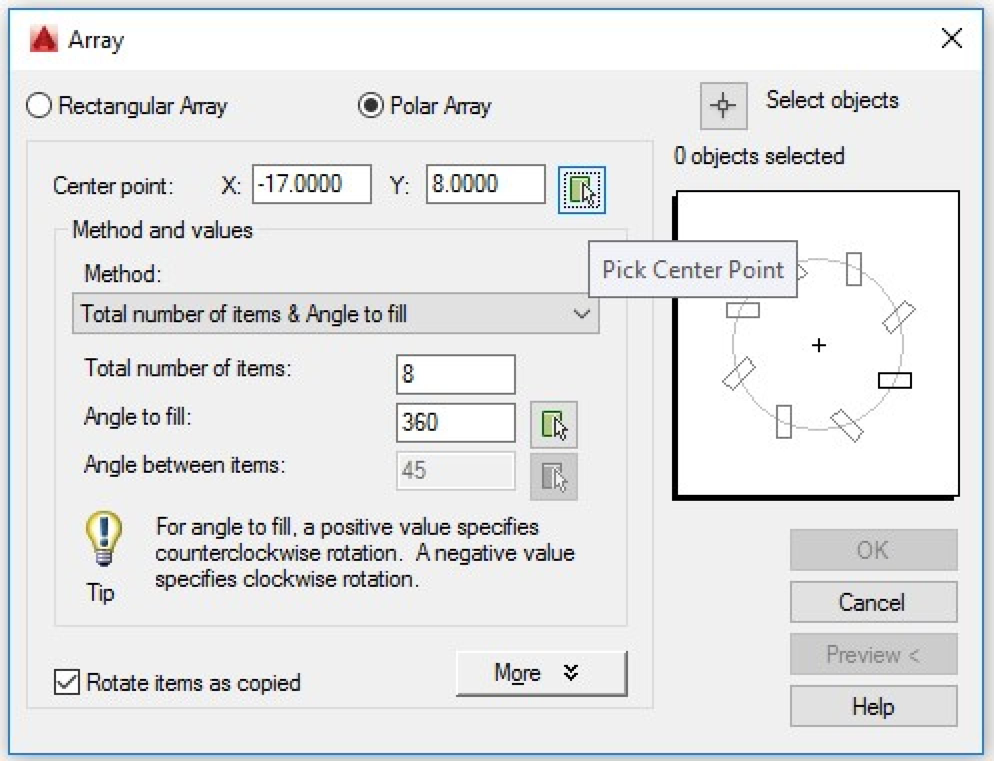

CENTER X2.5,Y8.5
CENTER X6.0,Y8.5
1.0 DIA.
A
B
X3.5,Y8.5
X7,Y8.5
1.0 DIA.
X12,Y6.5
C
X12,Y5.5
D

Figure Step 6

> **AUTHOR'S COMMENTS:** A faster method of creating the text C and D would be to copy the text A to the location of C and D. After you do that, edit the text and change the A to C and D.

Step 7

Enter the ARRAYCLASSIC command to open the Array dialogue box. In the Array dialogue box, enable Polar Array . Click the Pick Center Point show me icon. When you are prompted for the center point of the array, snap to the center of circle A. (Figure Step 7A and 7B)



Array

○ Rectangular Array ● Polar Array

Select objects

0 objects selected

Center point: X: -17.0000 Y: 8.0000

Pick Center Point

Method and values

Method: Total number of items & Angle to fill

Total number of items: 8

Angle to fill: 360

Angle between items: 45

Tip For angle to fill, a positive value specifies counterclockwise rotation. A negative value specifies clockwise rotation.

☑ Rotate items as copied

Figure Step 7A

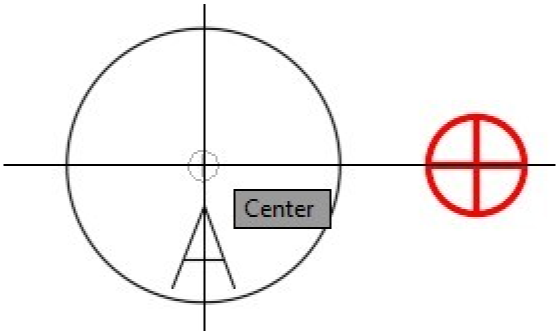
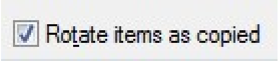


Figure Step 7B

Step 8

Enable Rotate items as copied. (Figure Step 8)



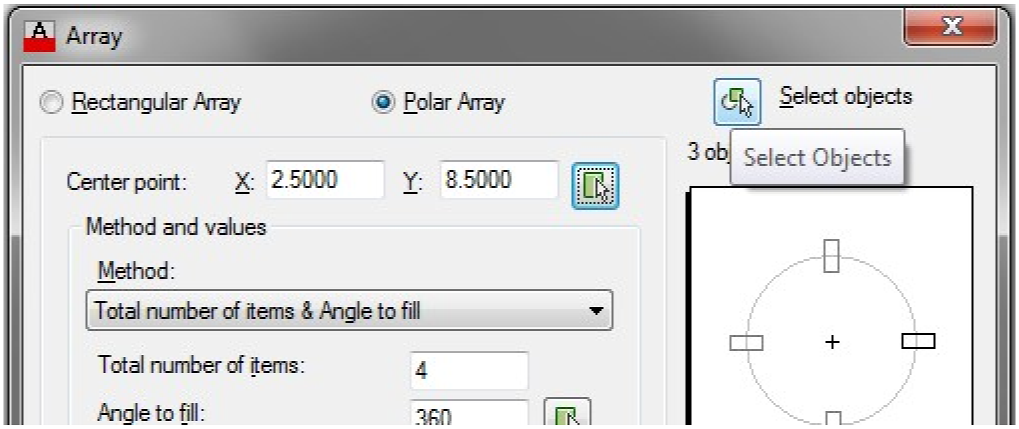
☑ Rotate items as copied

Figure Step 8

USER TIP: The Show me icon, inside a dialogue box as shown in the figure on the right, is instructing you that you can pick an object, size, or numerical value rather than entering data on the keyboard. Use these Show me icons whenever possible to improve drawing productivity.

Step 9

Click the Select objects icon. When you are prompted to Select objects, use a window to select the small circle as shown in the figure. (Figure Step 9A and 9B)



Array

○ Rectangular Array ● Polar Array

Select objects

3 ob

Select Objects

Center point: X: 2.5000 Y: 8.5000

Method and values

Method:

Total number of items & Angle to fill

Total number of items: 4

Angle to fill: 360

Figure Step 9A

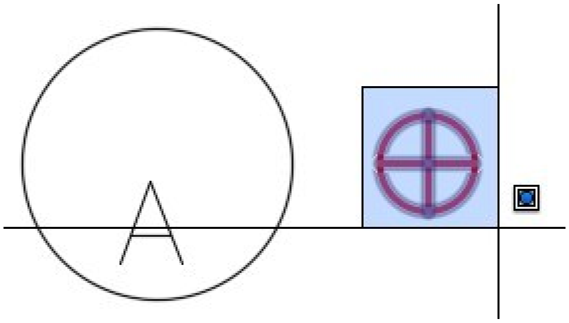
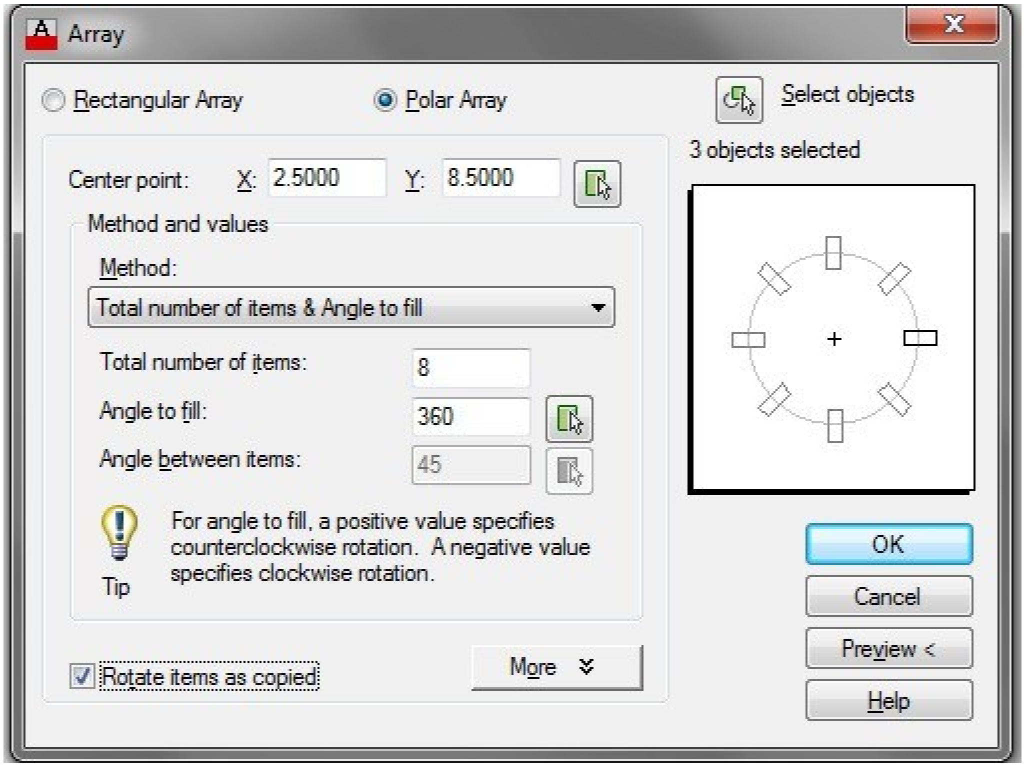


Figure Step 9B

Step 10

Set the Total number of items to 8 and Angle to fill: to 360.00. The Array dialogue box should match the figure. Click OK and your array should appear as shown in Figure Step 10B. (Figure Step 10A. 10B, and 10C)



Array

○ Rectangular Array ● Polar Array

3 objects selected

Center point: X: 2.5000 Y: 8.5000

Method and values

Method: Total number of items & Angle to fill

Total number of items: 8

Angle to fill: 360

Angle between items: 45

Tip: For angle to fill, a positive value specifies counterclockwise rotation. A negative value specifies clockwise rotation.

☑ Rotate items as copied

Figure Step 10A

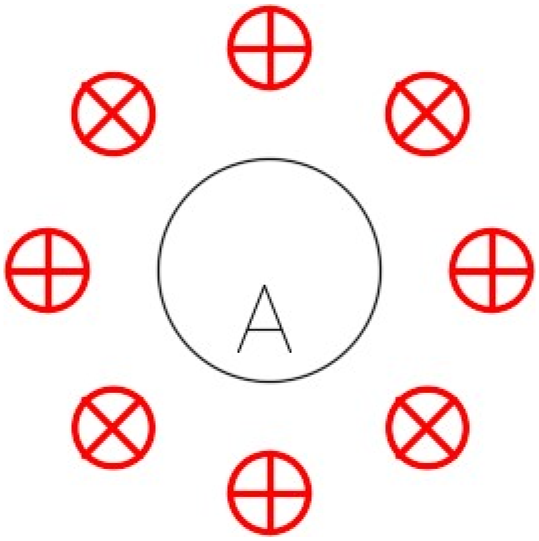


Figure Step 10B

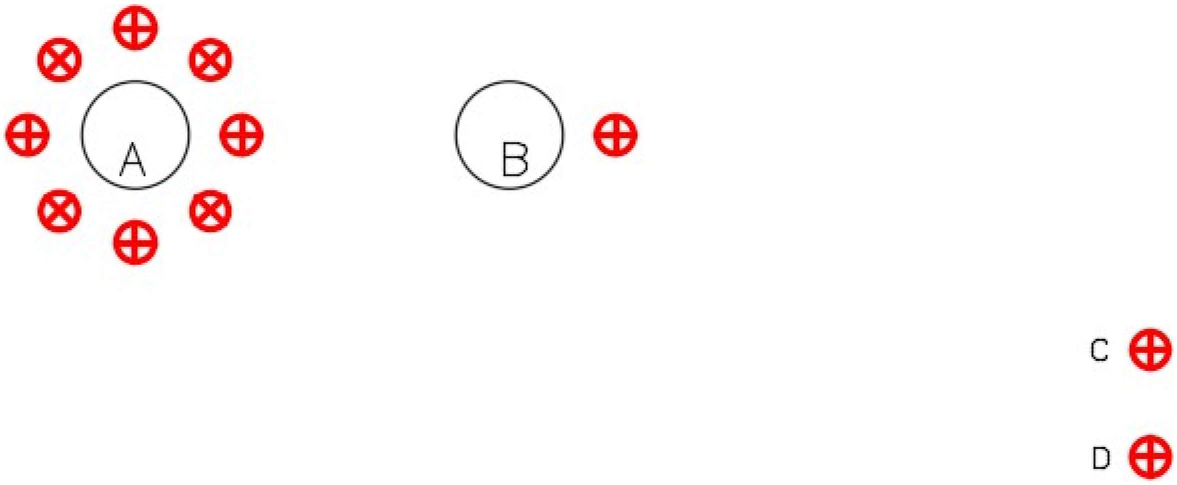


Figure Step 10C

Step 11

Enter the ARRAYCLASSIC command. Using what you learn in Step 7, enable Polar Array and click the Pick Center Point icon. When you are prompted for the center point of the array, snap to the center of circle B. (Figure Step 11)

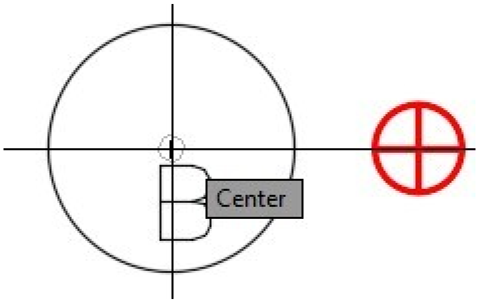
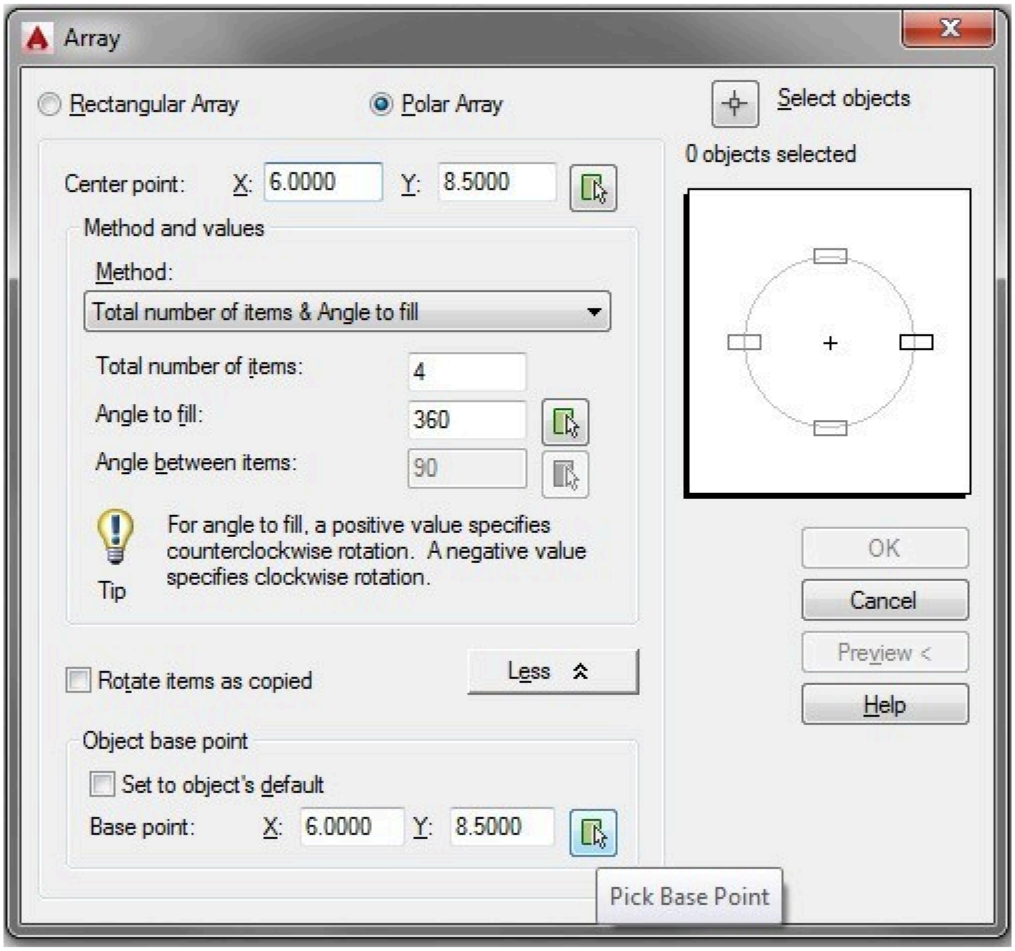


Figure Step 11

Step 12

Disable Rotate items as copied. Click the More button, to the right of the Rotate items as copied button, to expand it. Click the Pick Base Point icon. When you are prompted to select the base point, snap to the center of small circle. (Figure Step 12A and 12B)



Array

○ Rectangular Array ● Polar Array

Select objects

0 objects selected

Center point: X: 6.0000 Y: 8.5000

Method and values

Method: Total number of items & Angle to fill

Total number of items: 4

Angle to fill: 360

Angle between items: 90

Tip: For angle to fill, a positive value specifies counterclockwise rotation. A negative value specifies clockwise rotation.

☐ Rotate items as copied

Object base point

☐ Set to object's default

Base point: X: 6.0000 Y: 8.5000

Figure Step 12A

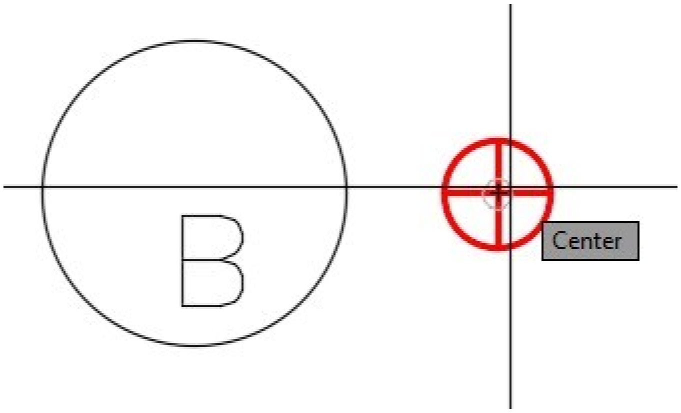
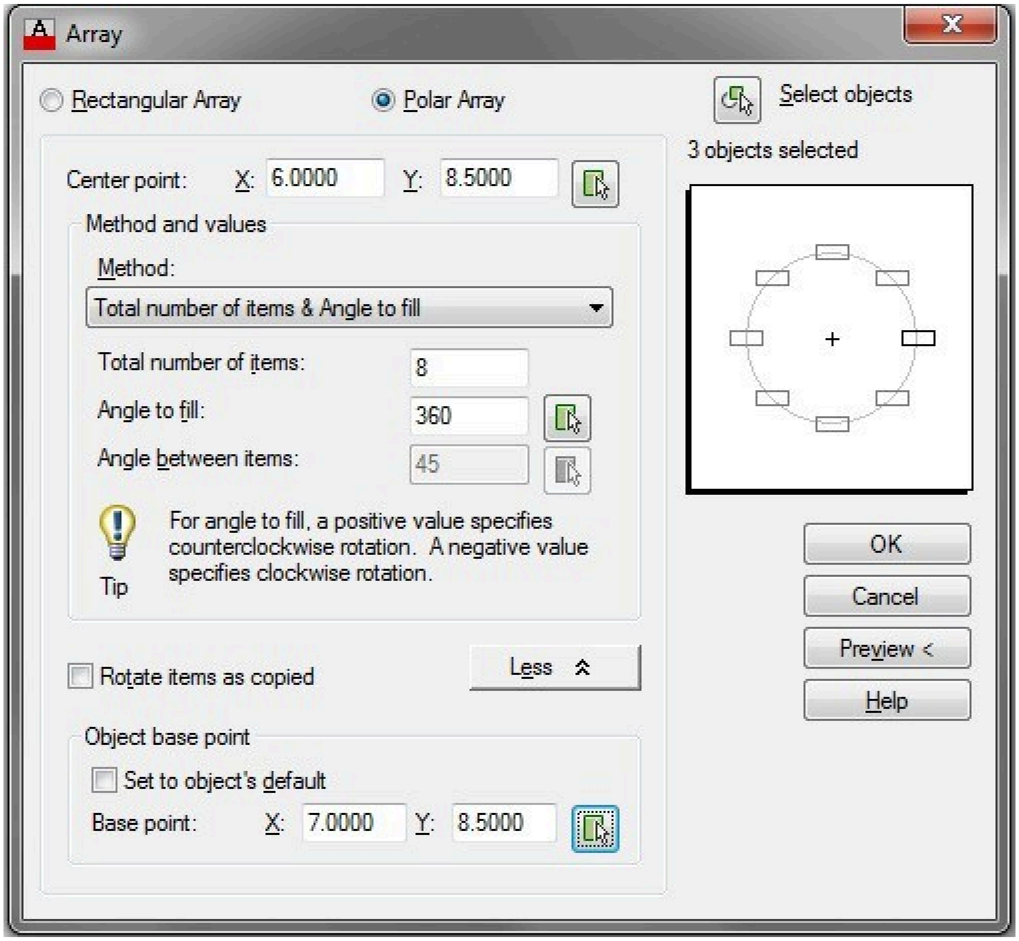


Figure Step 12B

AUTHOR'S COMMENTS: The base point on the object to be arrayed and not rotated must be its center.

Step 13

Click the Select objects button. When you are prompted to Select objects, use a window to select the small circle as shown in the figure. Set the Total number of items to 8 and Angle to fill: to 360.00. (Figure Step 13A, 13B, 13C, and 13D)



Array

○ Rectangular Array ● Polar Array

Select objects

3 objects selected

Center point: X: 6.0000 Y: 8.5000

Method and values

Method: Total number of items & Angle to fill

Total number of items: 8

Angle to fill: 360

Angle between items: 45

Tip: For angle to fill, a positive value specifies counterclockwise rotation. A negative value specifies clockwise rotation.

☐ Rotate items as copied

Object base point

☐ Set to object's default

Base point: X: 7.0000 Y: 8.5000

Figure Step 13A

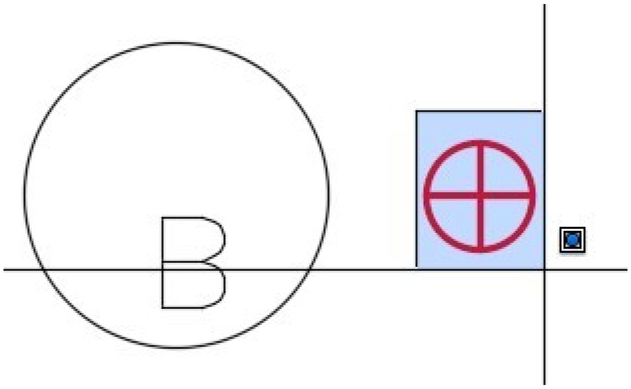


Figure Step 13B

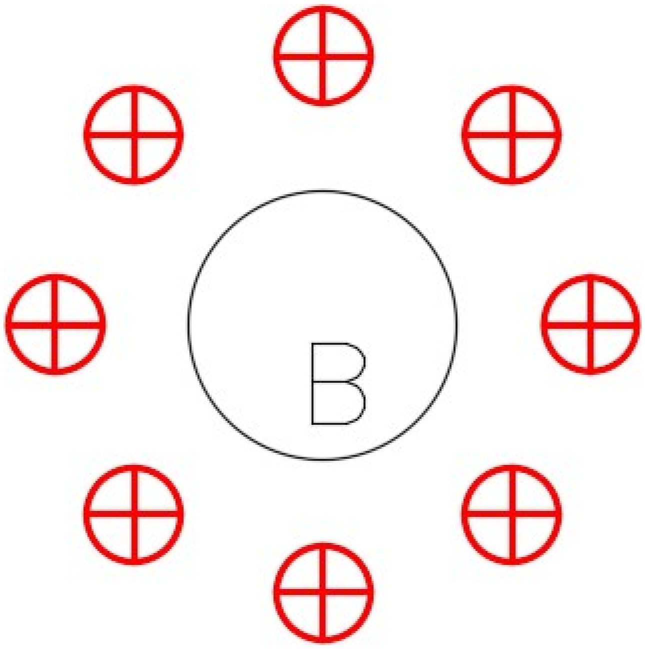


Figure Step 13C

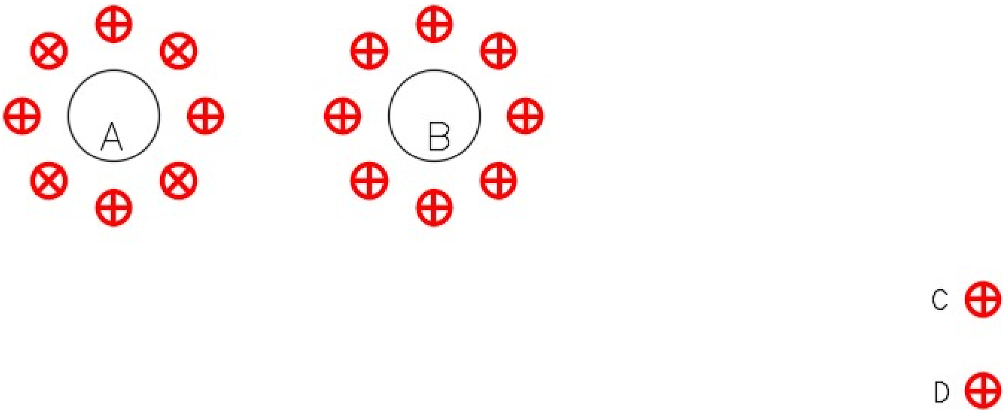


Figure Step 13D

Step 14

Save and close the drawing.

WORKALONG: Using the ARRAY Command – Part 2

Step 1

Open the drawing: AutoCAD 2D Workalong 23-1. (Figure Step 1)

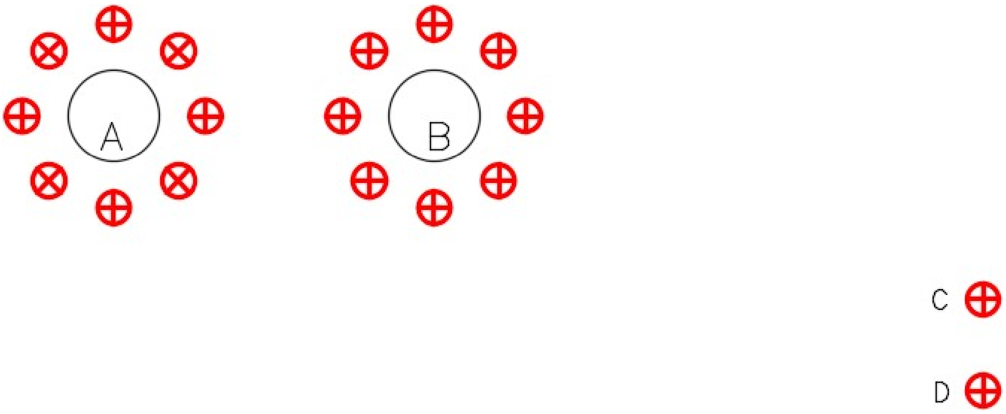


Figure Step 1

Step 2

Using the figures as a reference, draw the two circles and the line on layer: Construction. On layer: Object, draw the small inner circle. (Figure Step 2A, 2B, and 2C)

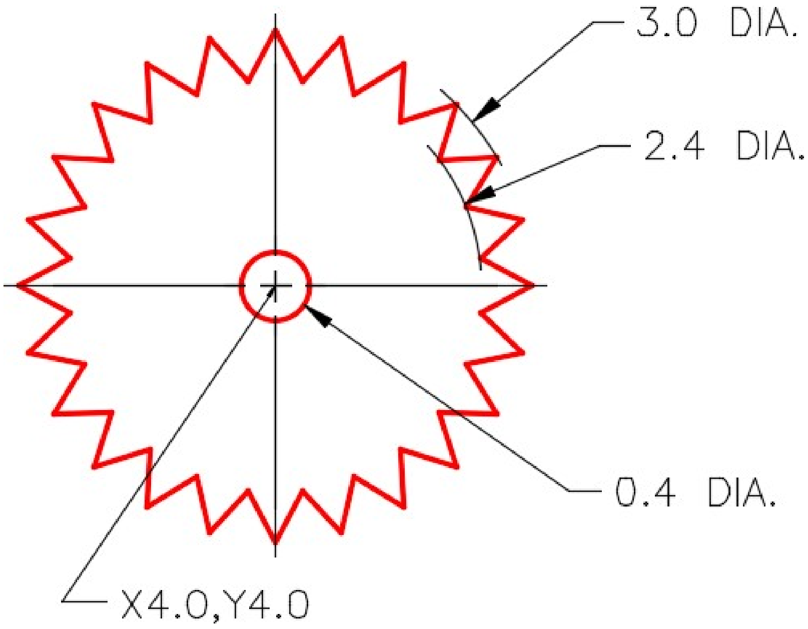


Figure Step 2A

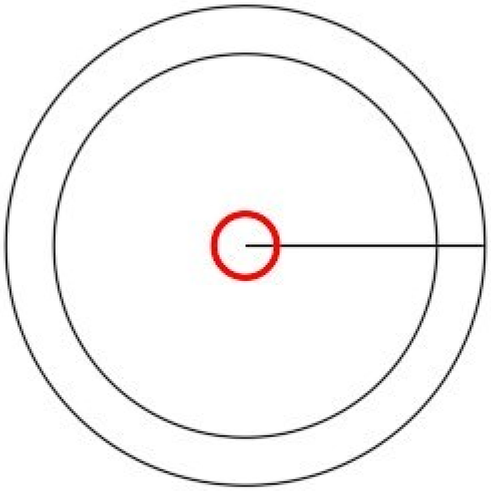


Figure Step 2B

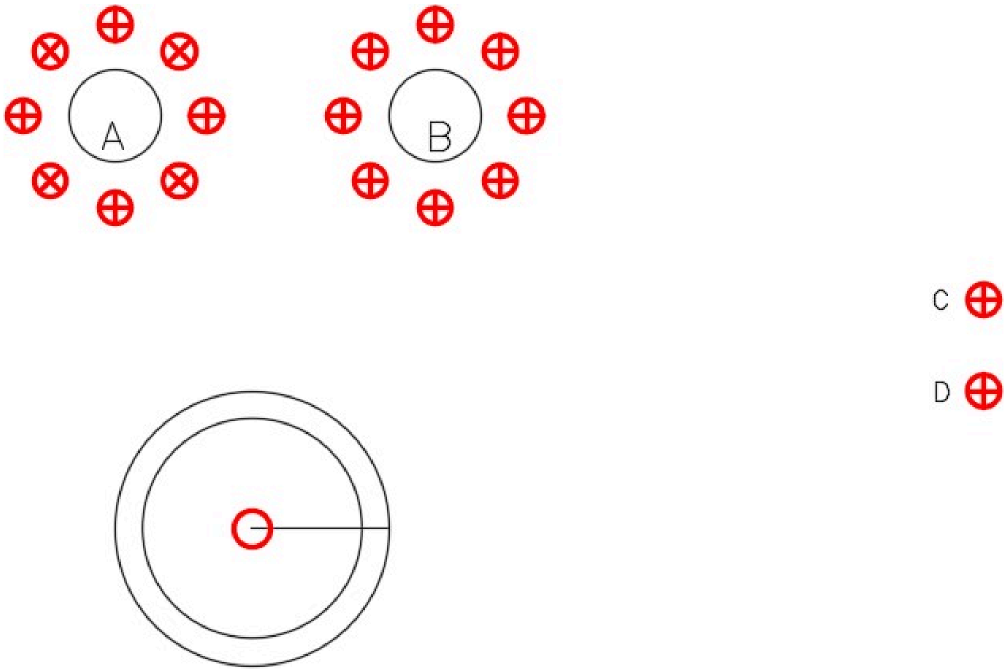
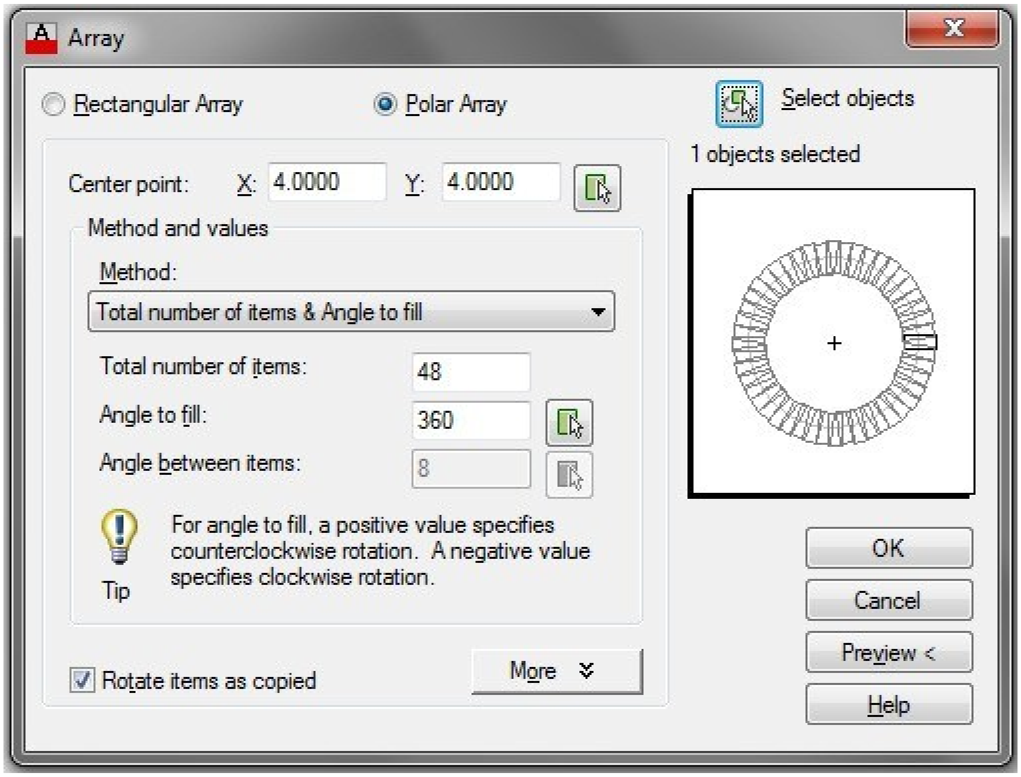


Figure Step 2C

Step 3

Enter the ARRAYCLASSIC command to open the Array dialogue box. Using what was taught earlier in this module, array the line 48 times. Rotate the line as it is arrayed. (Figure Step 3A and 3B)



Array

○ Rectangular Array ● Polar Array

Select objects

1 objects selected

Center point: X: 4.0000 Y: 4.0000

Method and values

Method:

Total number of items & Angle to fill

Total number of items: 48

Angle to fill: 360

Angle between items: 8

Tip

For angle to fill, a positive value specifies counterclockwise rotation. A negative value specifies clockwise rotation.

☑ Rotate items as copied

Figure Step 3A

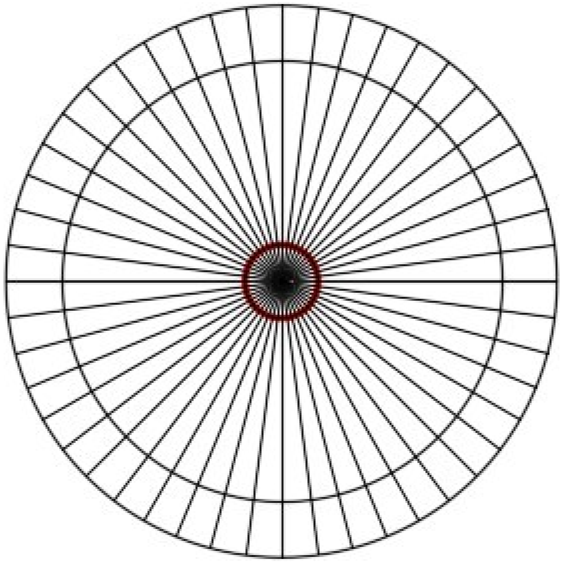


Figure Step 3B

Step 4

On layer: Object, draw one tooth by snapping to the intersection of the lines and circles as shown on the figure. (Figure Step 4)

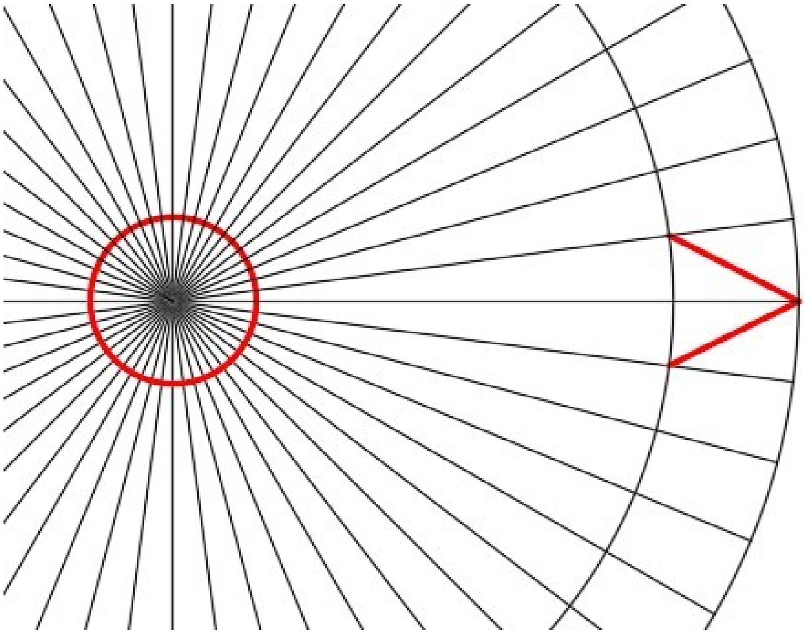


Figure Step 4

Step 5

Array the tooth 24 times. (Figure Step 5)

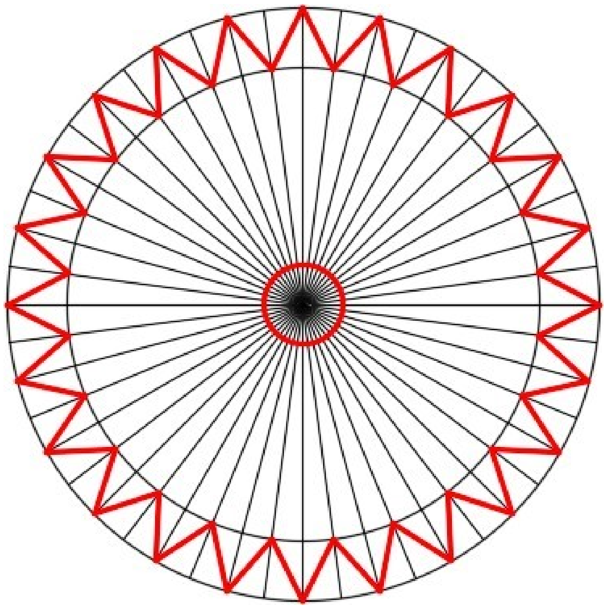


Figure Step 5

Step 6

Freeze layer: Construction. The completed drawing should match the figure. (Figure Step 6)

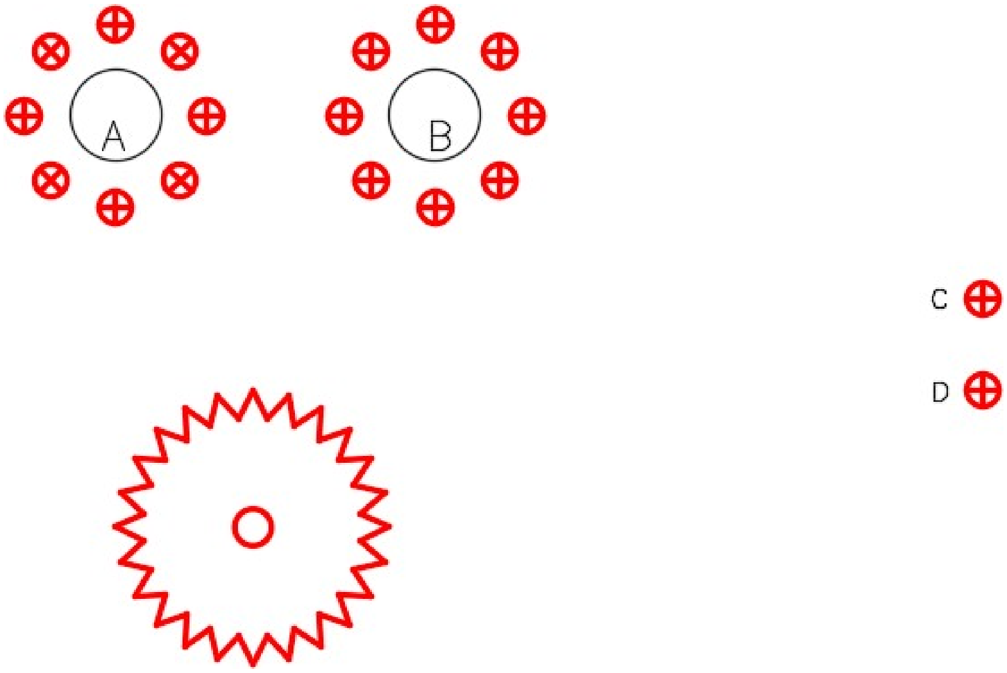
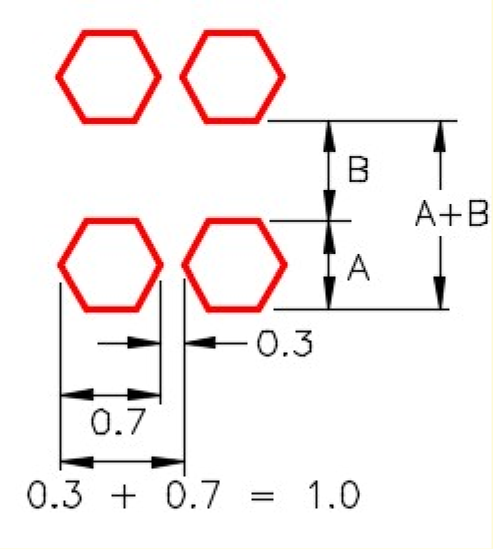


Figure Step 6

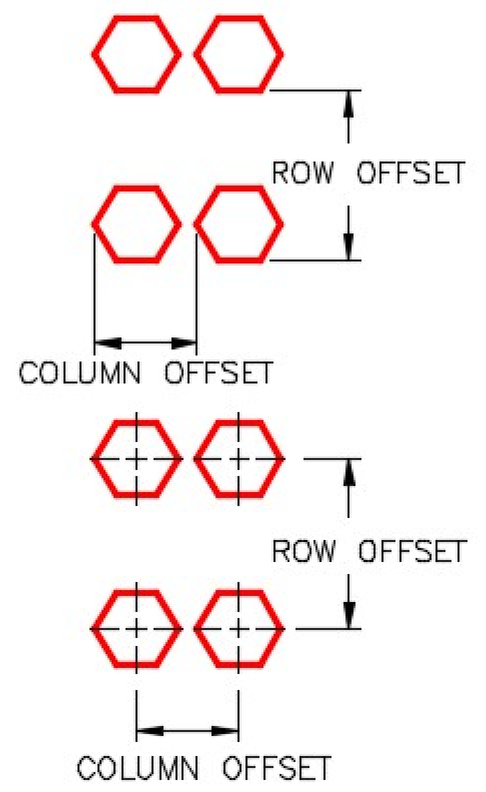
Step 7

Save and close the drawing.

USER TIP: In a rectangular array, the offset distance is measured from a point on the object to the same point on the arrayed object.



When arraying an object where only the distance between the objects is known, add the width of the object to the spacing for the column offset and height of the object to the spacing for the row offset.


ROW OFFSET
COLUMN OFFSET
ROW OFFSET
COLUMN OFFSET

If the distances from center to center of the object are known, simply use them as the offset distances.

WORKALONG: Using the ARRAY Command – Part 3

Step 1

Open the drawing: AutoCAD 2D Workalong 23-1. (Figure Step 1)

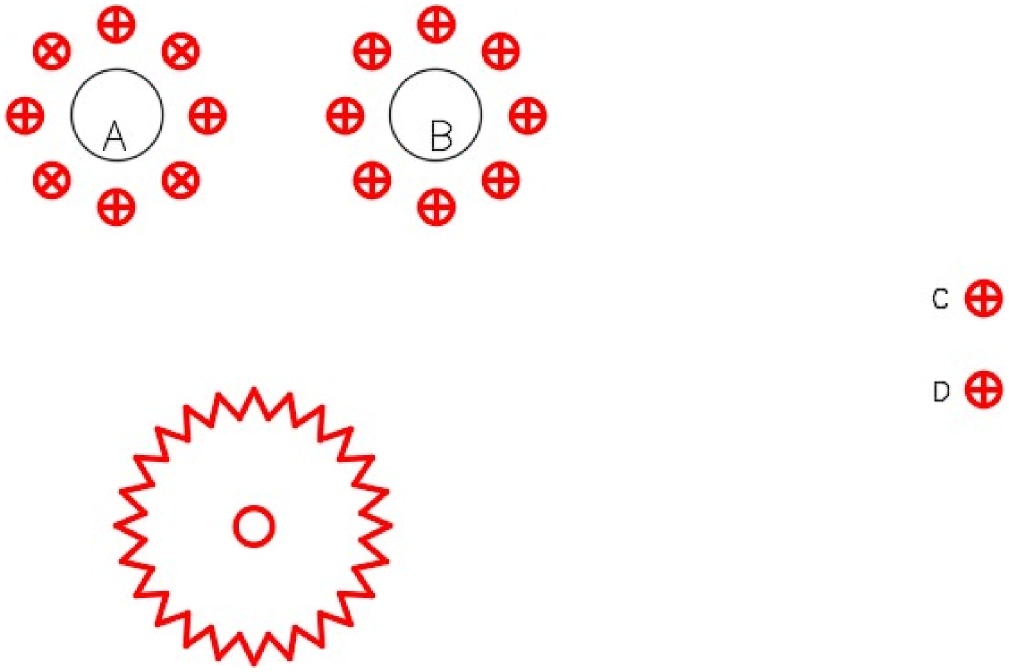


Figure Step 1

Step 2

Enter the ARRAYCLASSIC command. In the Array dialogue box, enable Rectangular Array. Set the number of rows to 3 and number of columns to 5. Set the Row offset to 1.5 and Column offset to 1.0. Click the Select object icon and when prompted to Select objects, use a window to select circle C plus the lines. (Figure Step 2A, 2B, and 2C)

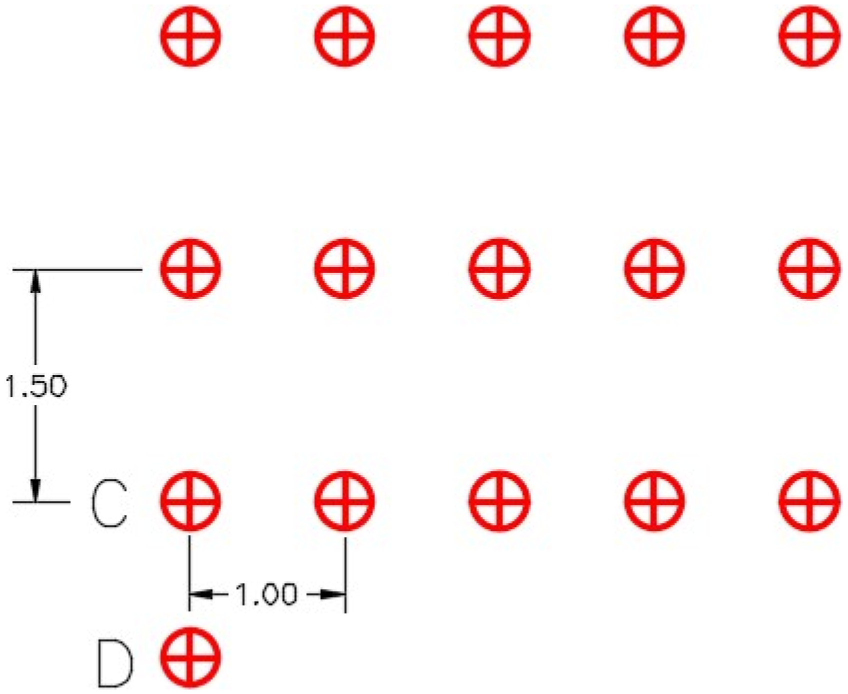
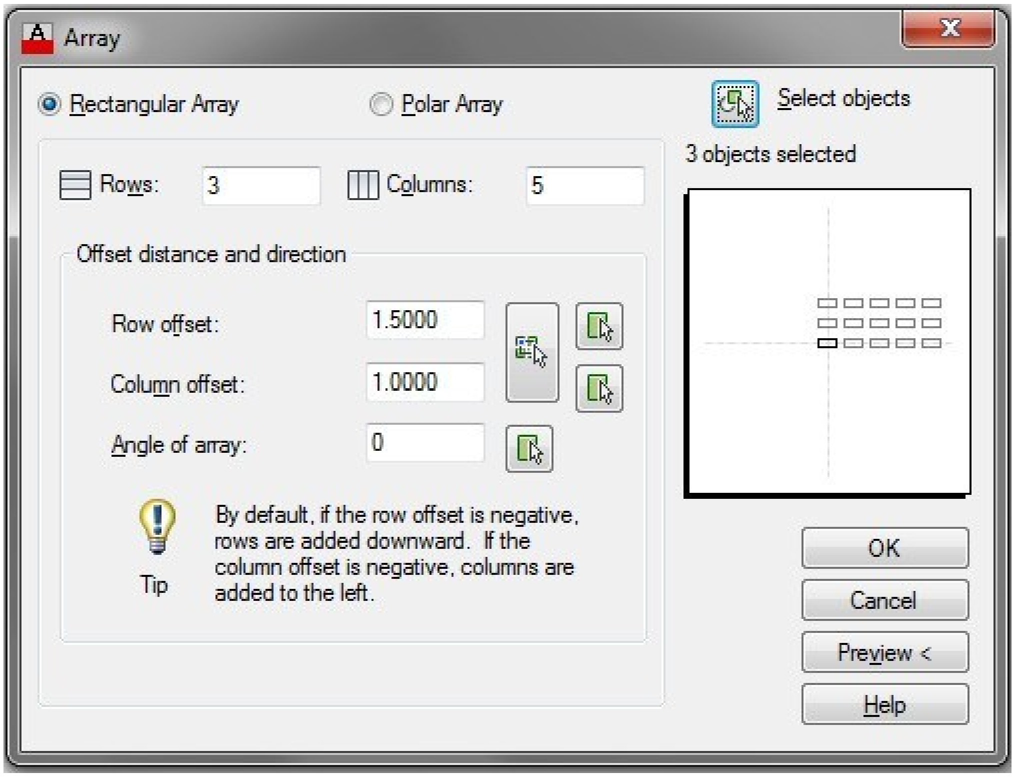


Figure Step 2A



● Rectangular Array
○ Polar Array

3 objects selected

Rows: 3
Columns: 5

Offset distance and direction

Row offset: 1.5000
Column offset: 1.0000
Angle of array: 0

By default, if the row offset is negative, rows are added downward. If the column offset is negative, columns are added to the left.

Figure Step 2B

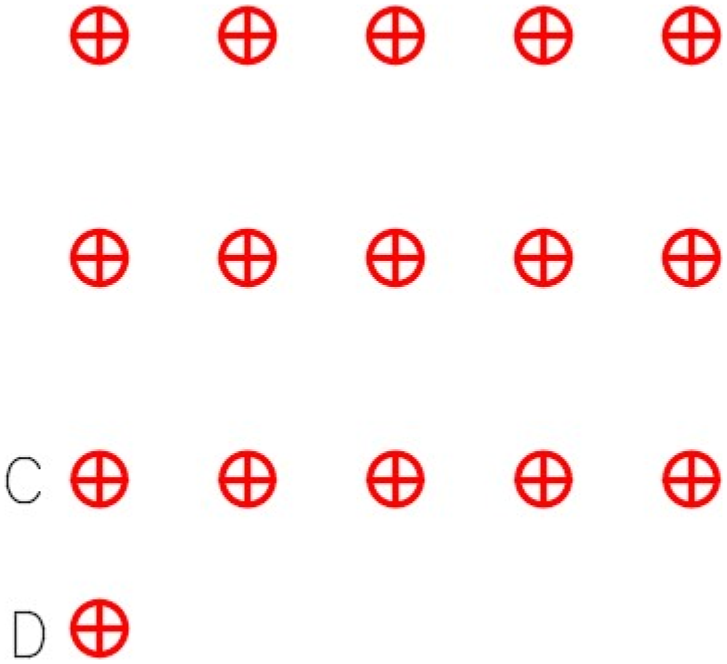


Figure Step 2C

AUTHOR'S COMMENTS: The Row offset and the Column offset are both positive in this array since you want to array in the positive X and positive Y direction.

Step 3

Enter the ARRAYCLASSIC command. In the Array dialogue box, enable Rectangular Array. Set the number of rows to 5 and number of columns to 4. Set the Row offset to – 0.5 and Column offset to -0.75. Set the Angle of array to 30. Click the Select object icon and when you are prompted to Select objects, use a window to select circle D and the two lines. (Figure Step 3A, 3B, 3C, and 3D)

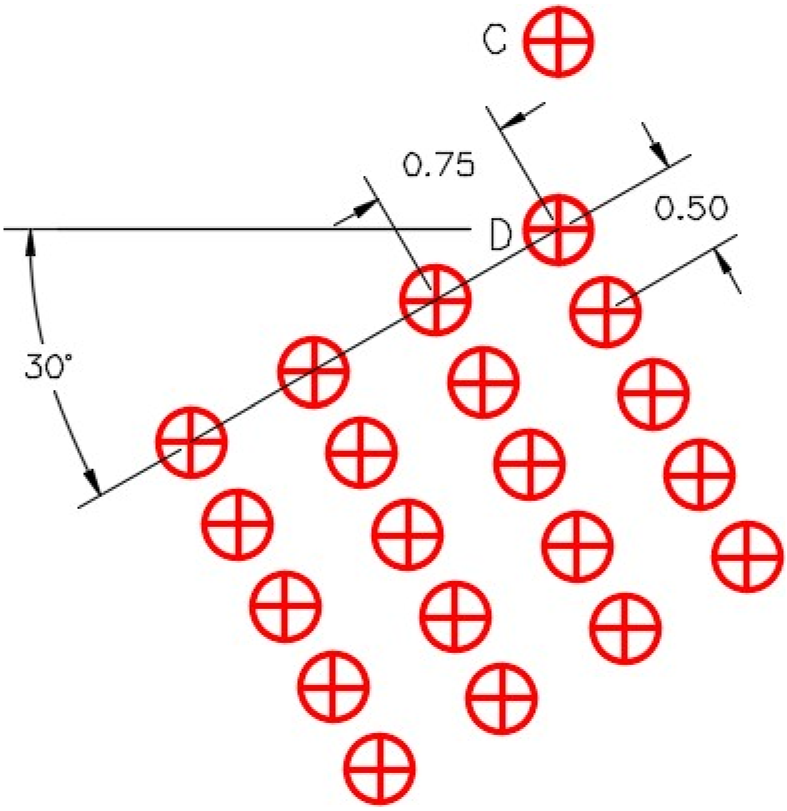
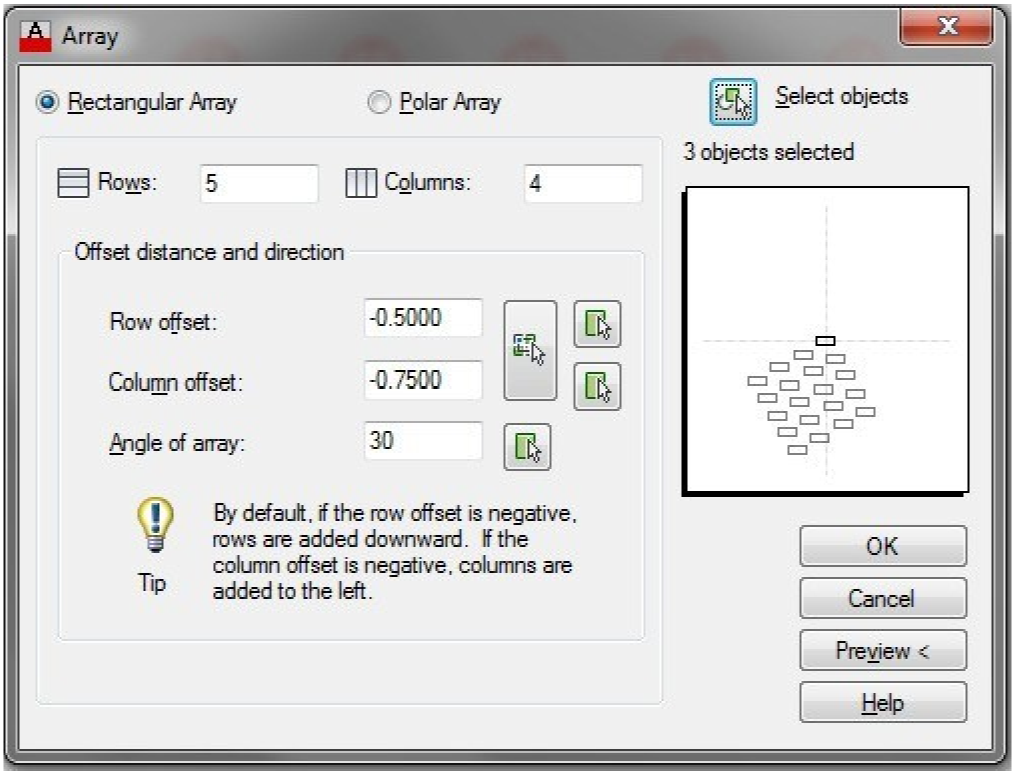


Figure Step 3A



Array

● Rectangular Array ○ Polar Array

3 objects selected

Rows: 5

Columns: 4

Offset distance and direction

Row offset: -0.5000

Column offset: -0.7500

Angle of array: 30

Tip

By default, if the row offset is negative, rows are added downward. If the column offset is negative, columns are added to the left.

Select objects
OK
Cancel
Preview <
Help

Figure Step 3B

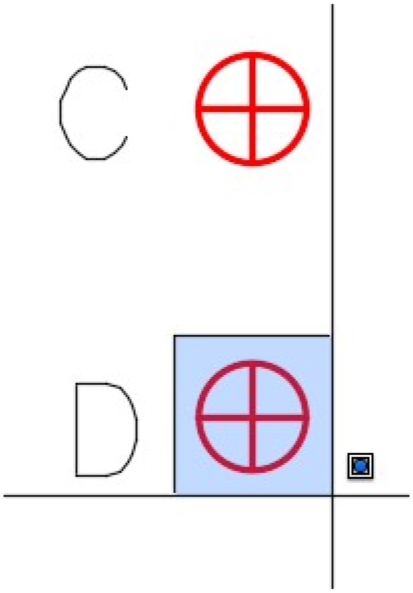


Figure Step 3C

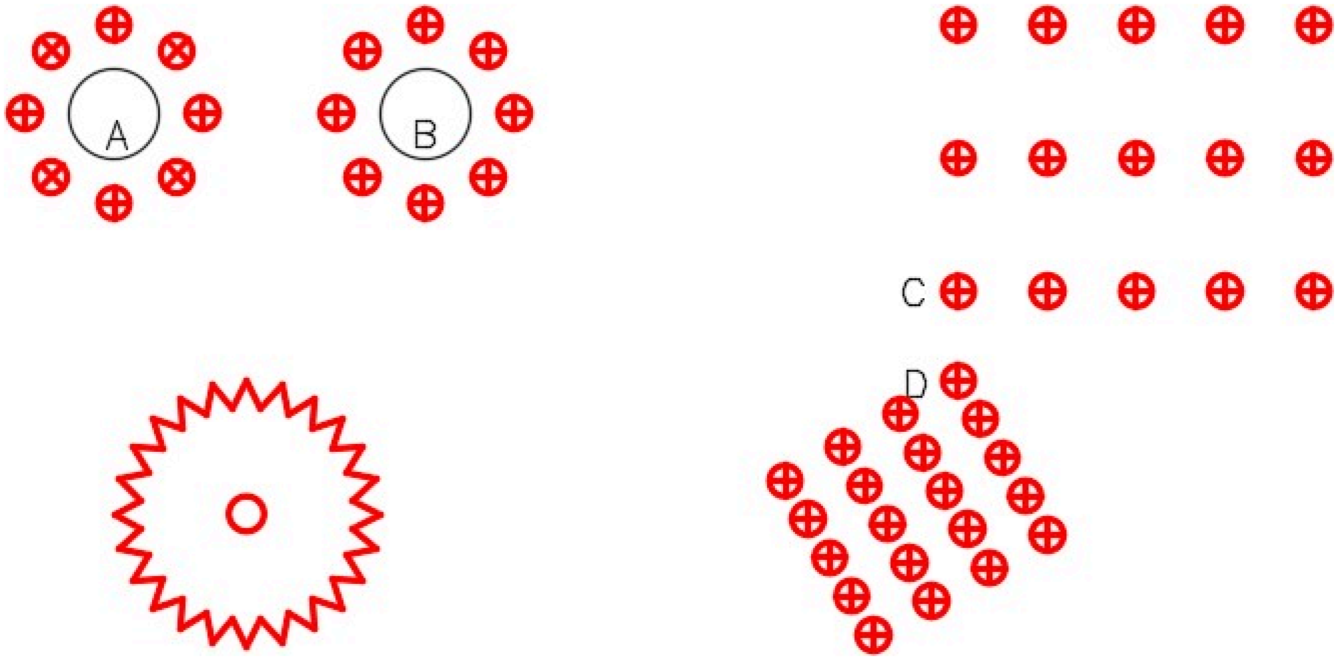
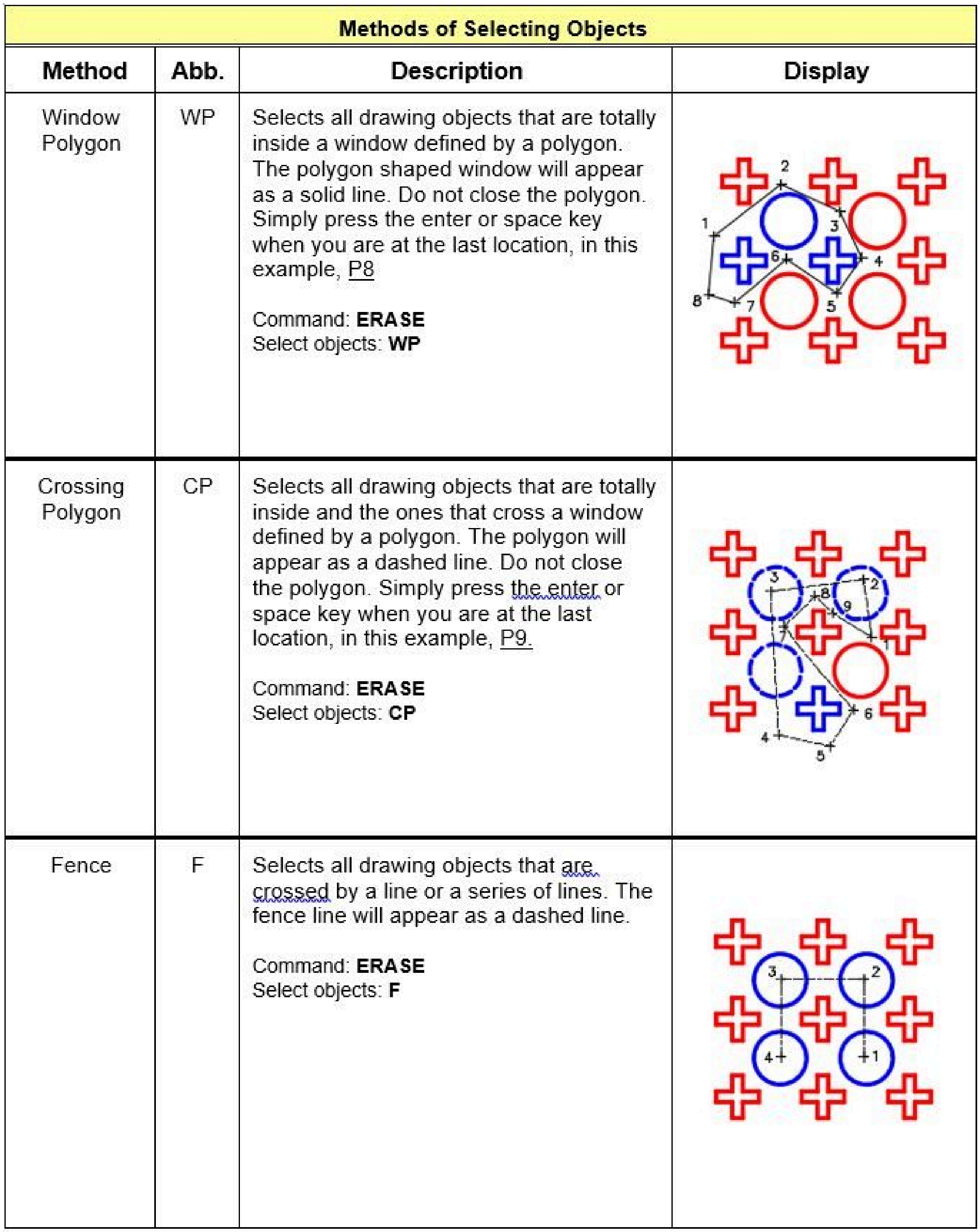


Figure Step 3D

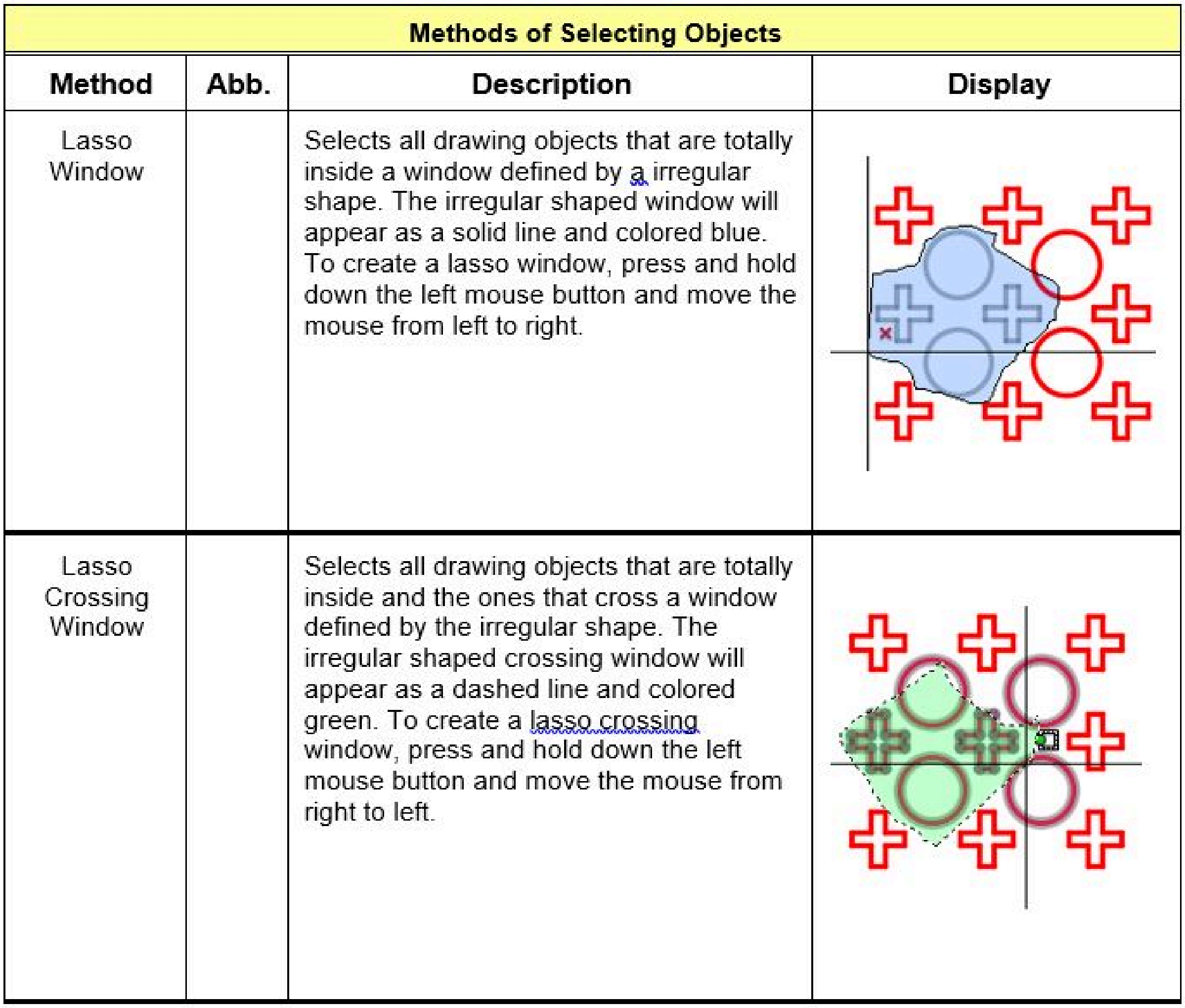
AUTHOR’S COMMENTS: The Row offset and the Column offset are both negative in this array since you want to array in the negative X and negative Y direction.

Step 4

Save and close the drawing.



Methods of Selecting Objects			
Method	**Abb.**	**Description**	**Display**
Window Polygon	WP	Selects all drawing objects that are totally inside a window defined by a polygon. The polygon shaped window will appear as a solid line. Do not close the polygon. Simply press the enter or space key when you are at the last location, in this example, P8 Command: **ERASE** Select objects: **WP**	
Crossing Polygon	CP	Selects all drawing objects that are totally inside and the ones that cross a window defined by a polygon. The polygon will appear as a dashed line. Do not close the polygon. Simply press the enter or space key when you are at the last location, in this example, P9. Command: **ERASE** Select objects: **CP**	
Fence	F	Selects all drawing objects that are crossed by a line or a series of lines. The fence line will appear as a dashed line. Command: **ERASE** Select objects: **F**	



Methods of Selecting Objects			
Method	**Abb.**	**Description**	**Display**
Lasso Window		Selects all drawing objects that are totally inside a window defined by a irregular shape. The irregular shaped window will appear as a solid line and colored blue. To create a lasso window, press and hold down the left mouse button and move the mouse from left to right.	
Lasso Crossing Window		Selects all drawing objects that are totally inside and the ones that cross a window defined by the irregular shape. The irregular shaped crossing window will appear as a dashed line and colored green. To create a lasso crossing window, press and hold down the left mouse button and move the mouse from right to left.	

Figure 23-3 Selecting Objects

WORKALONG: Using Polygon Windows and Fences to Select Objects

Step 1

Open the drawing: AutoCAD 2D Workalong 23-1.

Step 2

Using the SAVEAS command, save and name the drawing: AutoCAD 2D Workalong 23-2.

Step 3

Turn layers: Construction and Text off. (Figure Step 3)

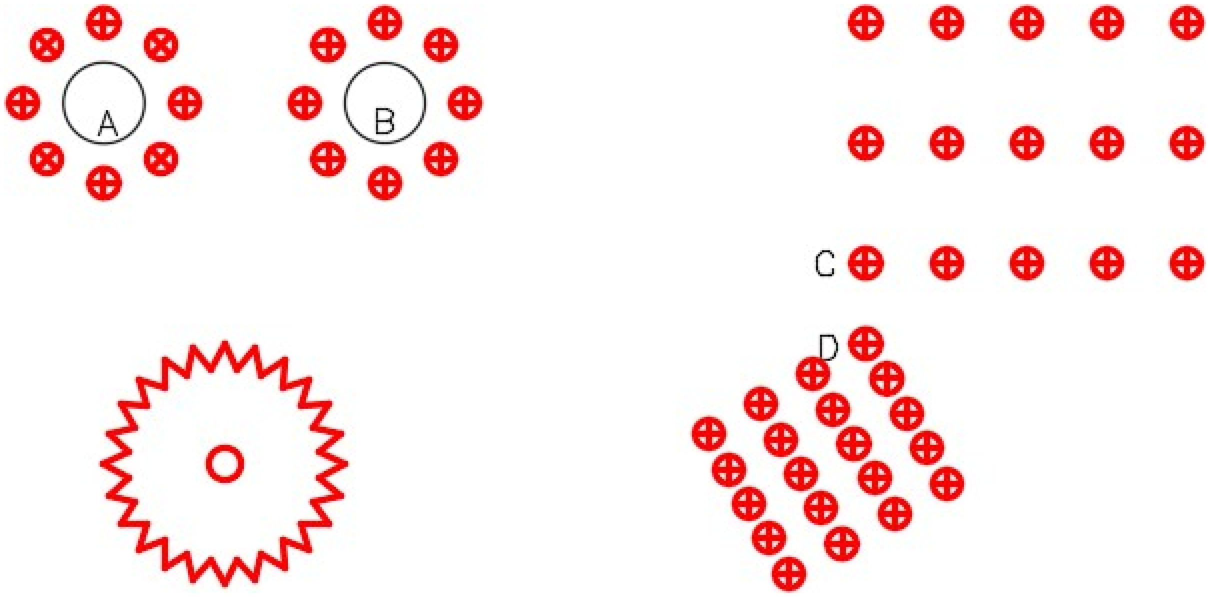


Figure Step 3

AUTHOR'S COMMENTS: Rectangular windows and crossing windows are implied, therefore you do not have to enter a W or C when you use them. To use most other windows and fences to select objects, you must enter the abbreviation after the ' Select objects ' prompt. See Figure 23-3 for the abbreviations.

Step 4

Enter the ERASE command as shown below. Use a window polygon to select the objects. (Figure Step 4A and 4B)

```
Command: ERASE Select objects: WP

First polygon point:

Specify endpoint of line or [Undo]:

Specify endpoint of line or [Undo]:

Specify endpoint of line or [Undo]:

Specify endpoint of line or [Undo]:

Specify endpoint of line or [Undo]:

Select objects:

Command:
```

AUTHOR’S COMMENTS: Do not join the last endpoint of the window to the first point. AutoCAD will do that automatically.

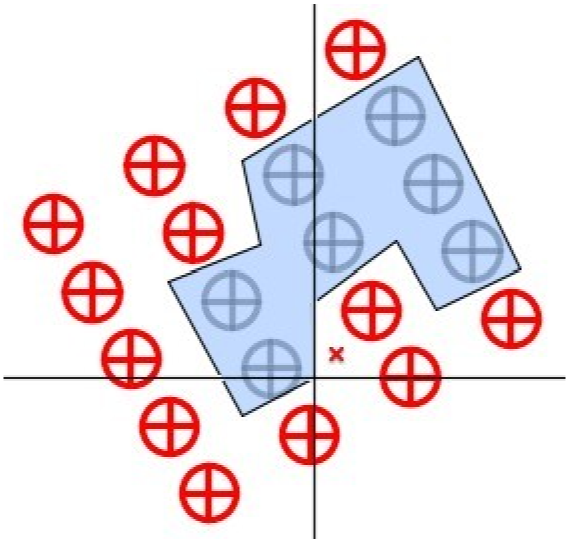


Figure Step 4A

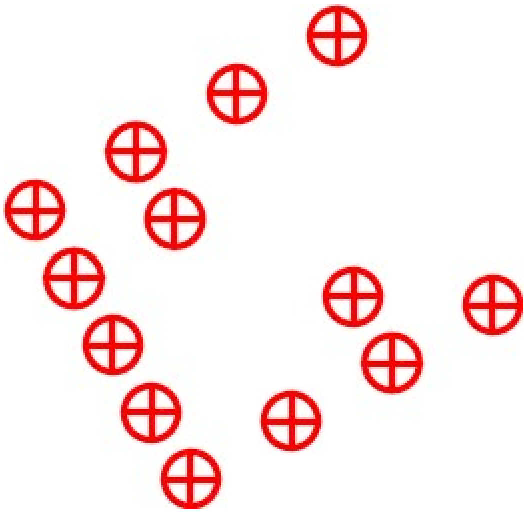


Figure Step 4B

Step 5

Enter the ERASE command as shown below. Use a cross polygon window to select the objects to be erased. (Figure Step 5A and 5B)

Command: **ERASE** Select objects: **CP**

First polygon point:

Specify endpoint of line or [Undo]:

Specify endpoint of line or [Undo]:

```
Specify endpoint of line or [Undo]:

Specify endpoint of line or [Undo]:

Specify endpoint of line or [Undo]:

Specify endpoint of line or [Undo]:

Specify endpoint of line or [Undo]:

Select objects:

Command:
```

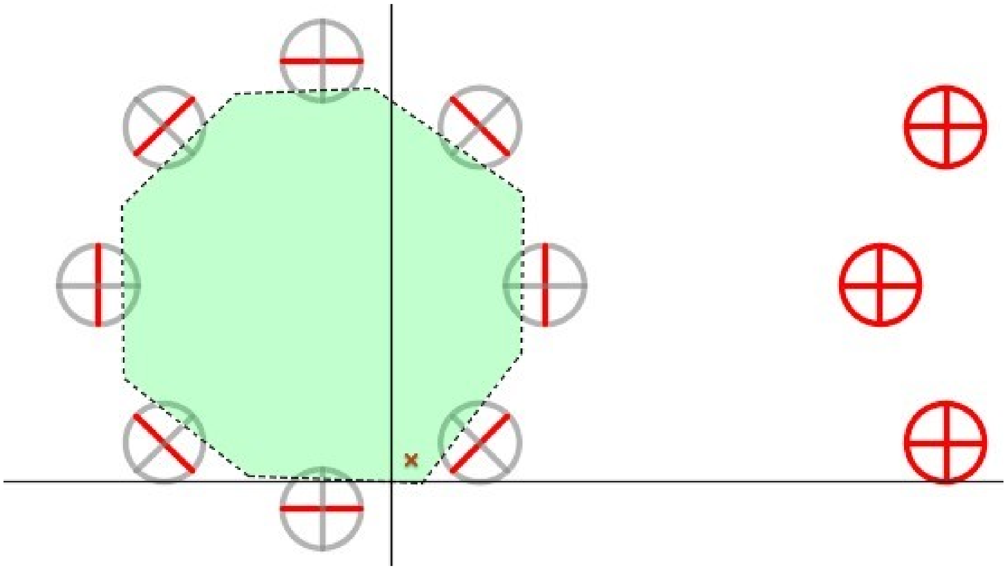


Figure Step 5A

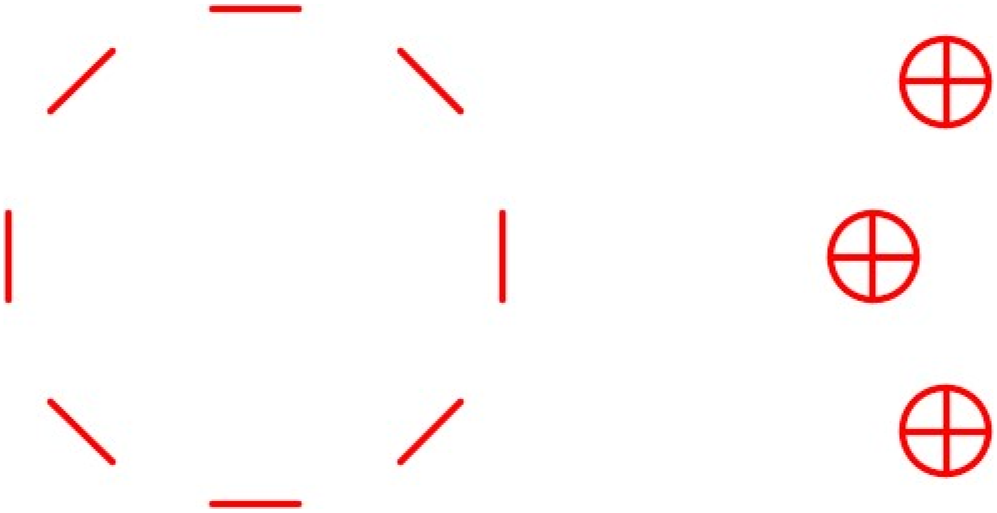


Figure Step 5B

Step 6

Enter the ERASE command as shown below. Use a fence to select the objects to be erased. (Figure Step 6A and 6B)

Command: **ERASE** Select objects: **F**

First polygon point:

Specify endpoint of line or [Undo]:

Specify endpoint of line or [Undo]:

Specify endpoint of line or [Undo]:

Specify endpoint of line or [Undo]:

Specify endpoint of line or [Undo]:

Specify endpoint of line or [Undo]:

Specify endpoint of line or[Undo]:

Select objects:

Command:

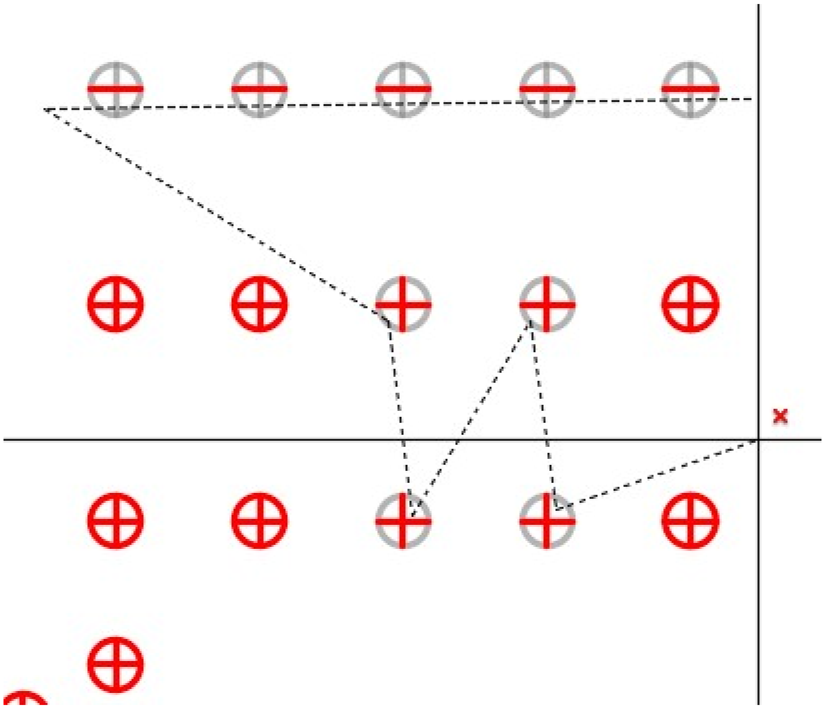


Figure Step 6A

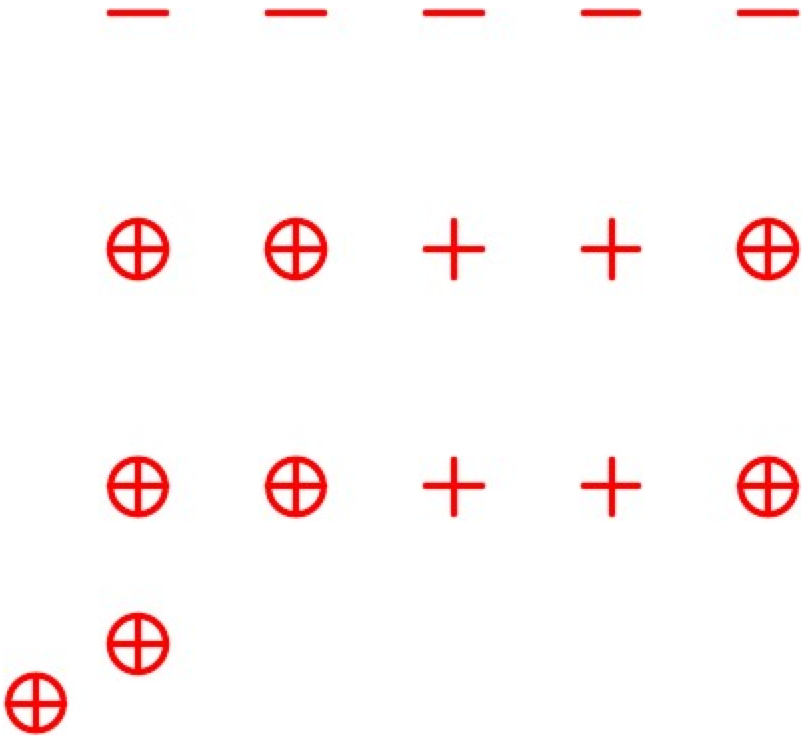


Figure Step 6B

Step 7

Using what you learned, use the COPY command and a cross polygon window to select the objects, copy the 4 circles and their lines as shown in the figure.

(Figure Step 7A, 7B, and 7C)

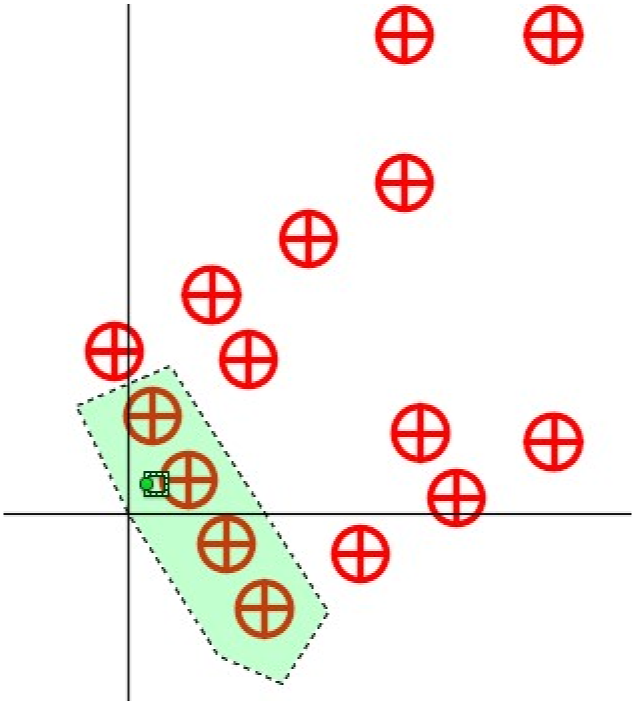


Figure Step 7A

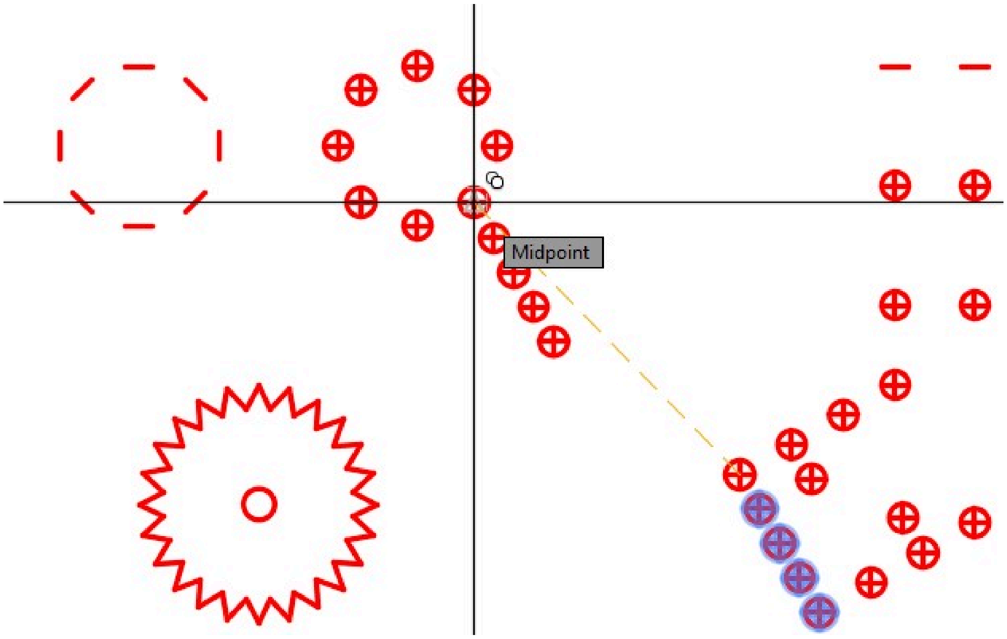


Figure Step 7B

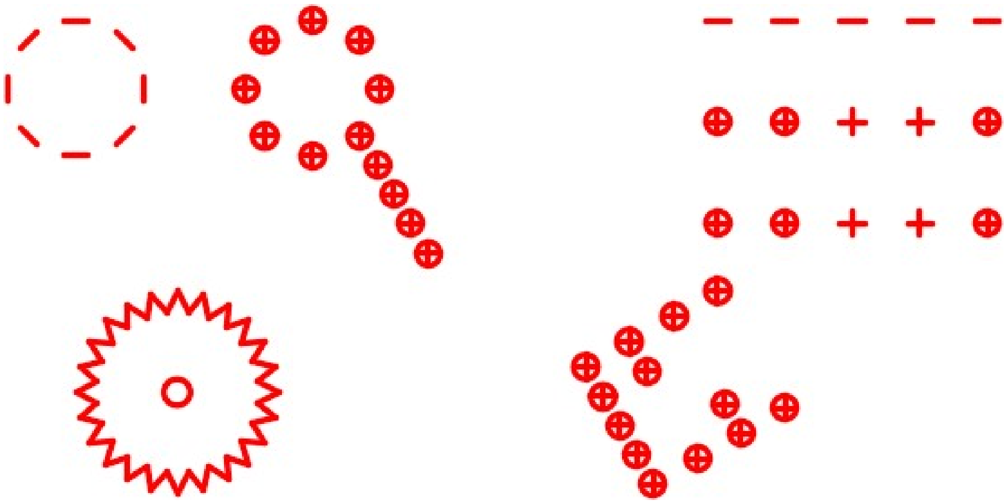


Figure Step 7C

Step 8

Use the lasso windows method to erase 5 circles. Without entering a command, press the left mouse button and while holding it down, move the mouse from left to right. The irregular shape should be colored blue. All objects totally inside the window will be selected. With the object selected, press the Delete key. (Figure Step 8A, 8B, and 8C)

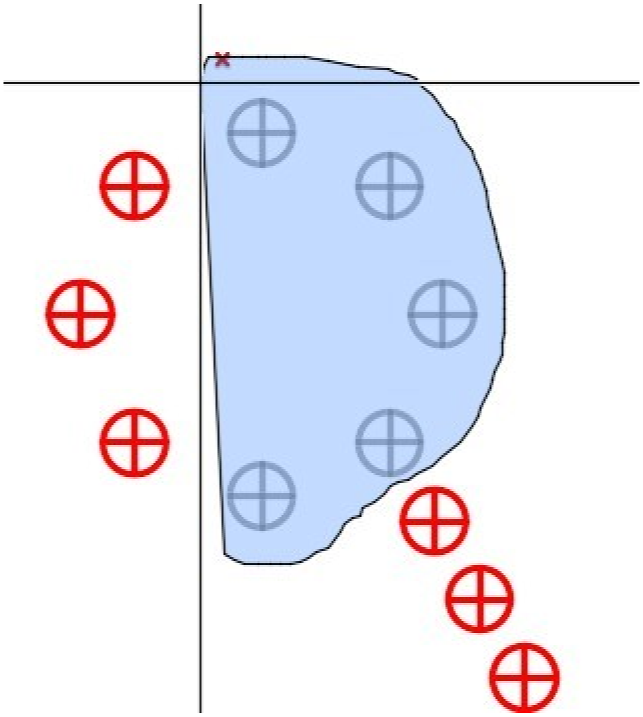


Figure Step 8A

AUTHOR'S COMMENTS: Do not join the last endpoint of the window to the first point. AutoCAD will do that.

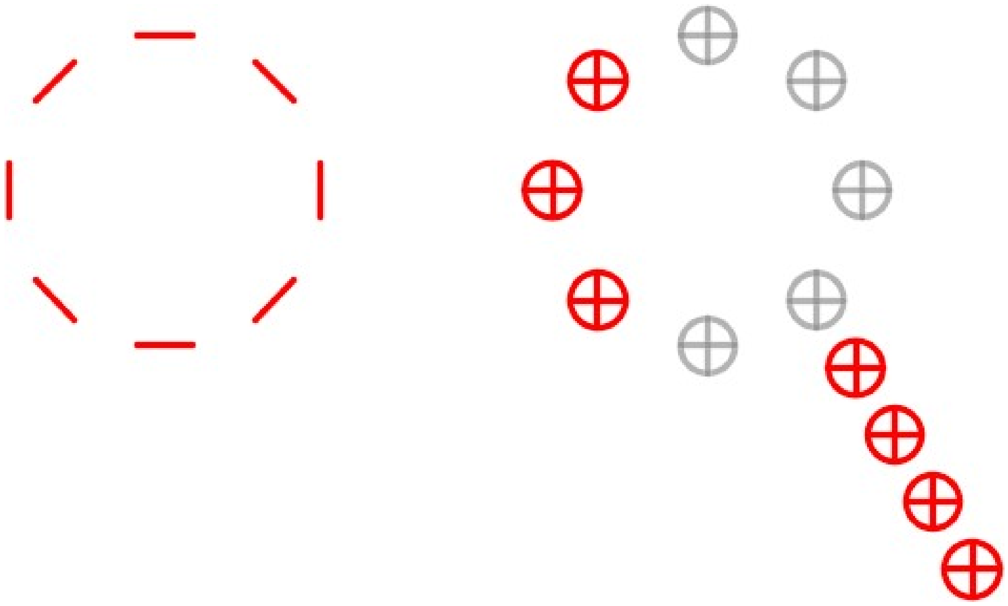


Figure Step 8B

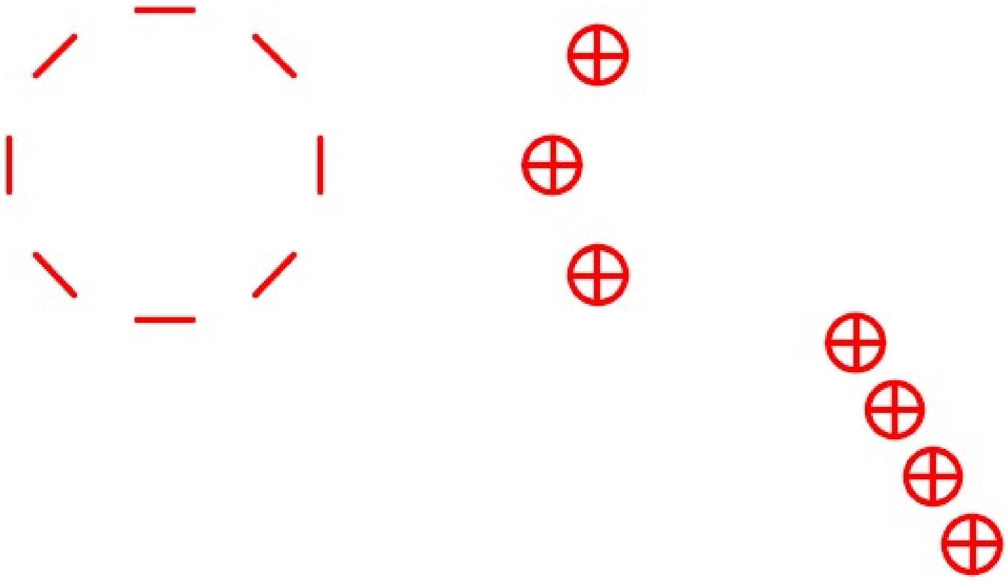


Figure Step 8C

Step 9

Use the lasso crossing windows method. Without entering a command, press the left mouse button and while holding it down, move the mouse from right to left. The irregular shape should be colored green. All objects totally inside and ones that cross the window will be selected. With the object selected, press the Delete key. (Figure Step 9A, 9B, and 9C)

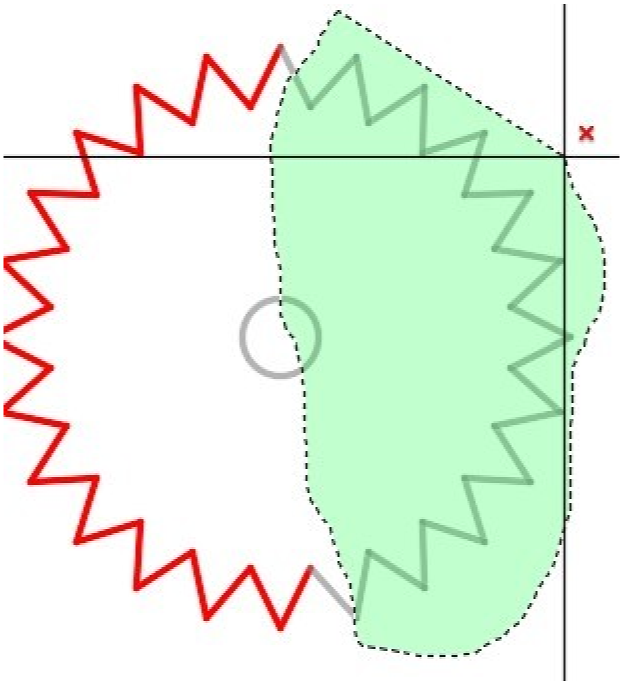


Figure Step 9A

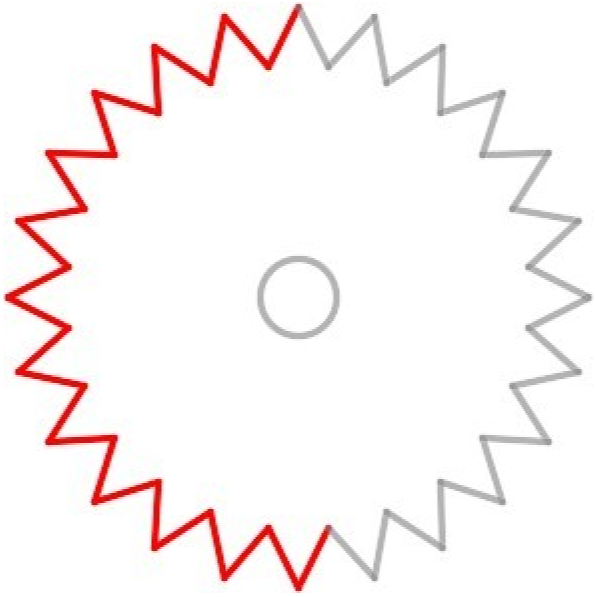


Figure Step 9B

AUTHOR'S COMMENTS: Do not join the last endpoint of the crossing window to the first point. AutoCAD will do that.

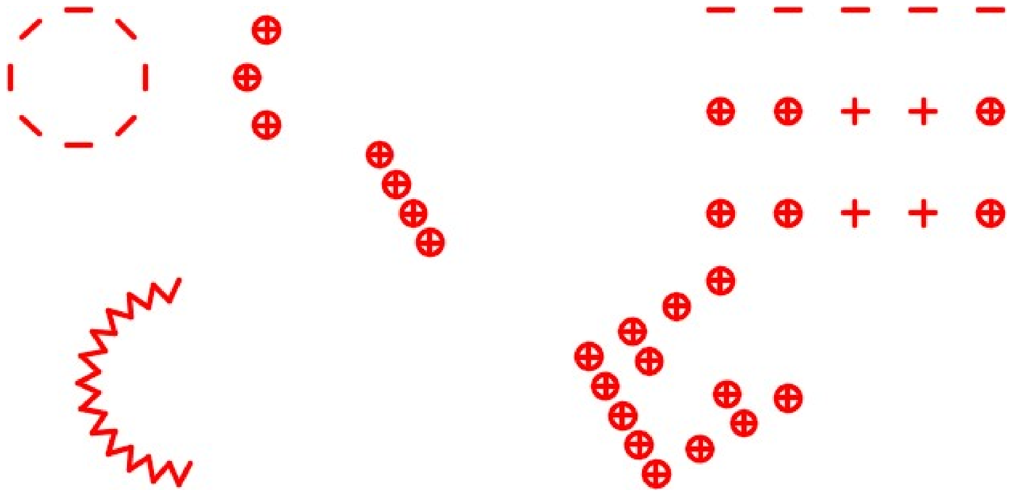


Figure Step 9C

Step 10

save and close the drawing.

Geometry Lesson: Finding Centers and Placing Circles inside Regular Polygons

Method 1

Insert a circle that is tangent to the midpoint of three sides of the regular polygon. The center of the circle is the center of the polygon. See Figure 23-4.

Insert a circle using the 3P option in the CIRCLE command. For the three points, snap to the midpoints of any three lines.

```
Command: CIRCLE

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: 3P

Specify first point on circle: (mid) P1

Specify second point on circle: (mid) P2

Specify third point on circle: (mid) P3

Command:
```

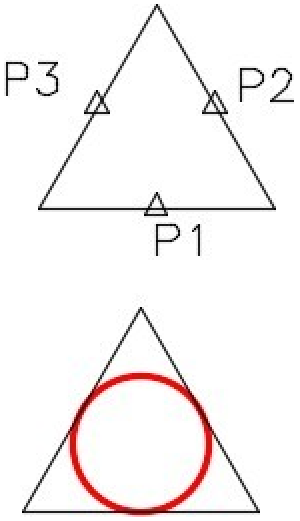


Figure 23-4

Method 2

Insert the circle using the 3P option in the CIRCLE command and snap to the tangent point of any three of sides of the regular polygon. See Figure 23-5.

```
Command: CIRCLE

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: 3P

Specify first point on circle: (tan) P4
```

Specify second point on circle: *(tan)* **P5**

Specify third point on circle: *(tan)* **P6**

Command:

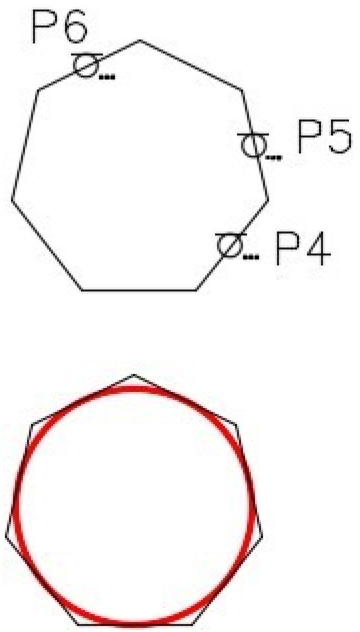


Figure 23-5

AUTHOR'S COMMENTS: Keep in mind that this only works for regular polygons. All the sides of a regular polygon are equal in length.

Key Principles

Key Principles in Module 23

1. When an object is arrayed, all objects created in the array will retain the properties of the original objects, even the layer they reside on. This happens regardless of the current layer when the ARRAYCLASSIC command is executed.
2. In a rectangular array, the offset distance is measured from the one point on the object to the same point on the arrayed object.
3. A Window Polygon selects all drawing objects that are totally inside a window defined by a polygon. The polygon shaped window will appear as a solid line. A Cross Polygon selects all drawing objects that are totally inside and the ones that cross a window defined by a polygon. The polygon will appear as a dashed line. A Fence selects all drawing objects that are crossed by a line or a series of lines.

Lab Exercise 23-1

Time allowed: 60 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 23-1	2D Layout Metric	Millimeters

Step 1

On layer: Object, draw the object shown in the figures. (figure Step 1A, 1B and 1C).

Step 2

Set the insertion units and insert the key. Check the drawing for accuracy.

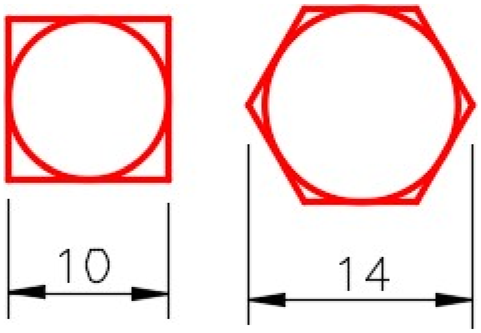


Figure Step 1A Detail

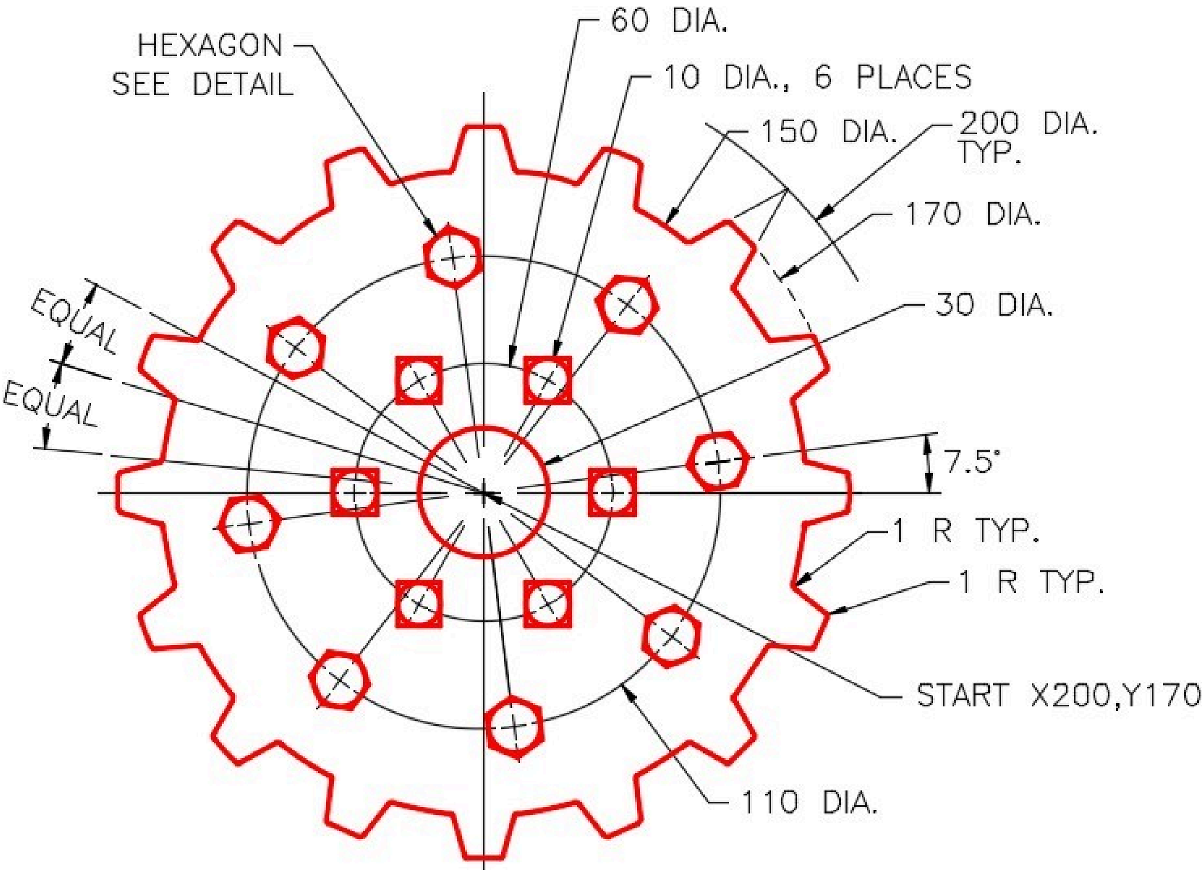

HEXAGON
SEE DETAIL
60 DIA.
10 DIA., 6 PLACES
150 DIA.
200 DIA.
TYP.
170 DIA.
30 DIA.
EQUAL
EQUAL
7.5°
1 R TYP.
1 R TYP.
START X200,Y170
110 DIA.

Figure Step 1B Dimensioned Drawing

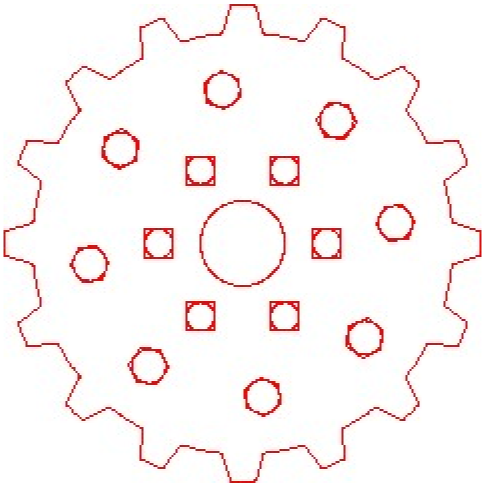


Figure Step 1C
Completed Drawing

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

Draw and then array the line 64 times and draw two teeth (Figure Hint 1)

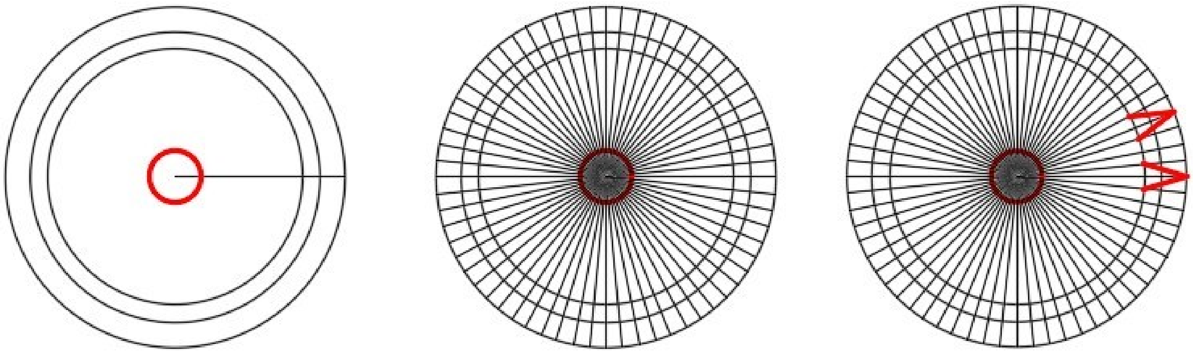


Figure Hint 1

Hint 2

Trim the circles and lines as shown in the figure. (Figure Hint 2A, 2B and 2C)

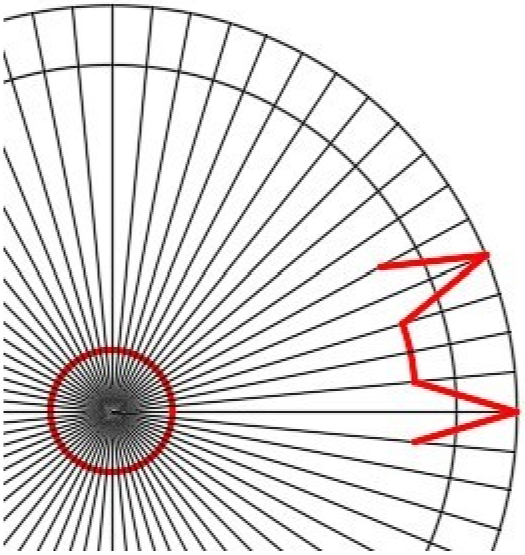


Figure Hint 2A

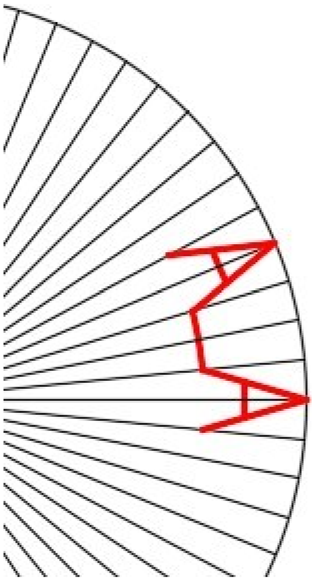


Figure Hint 2B

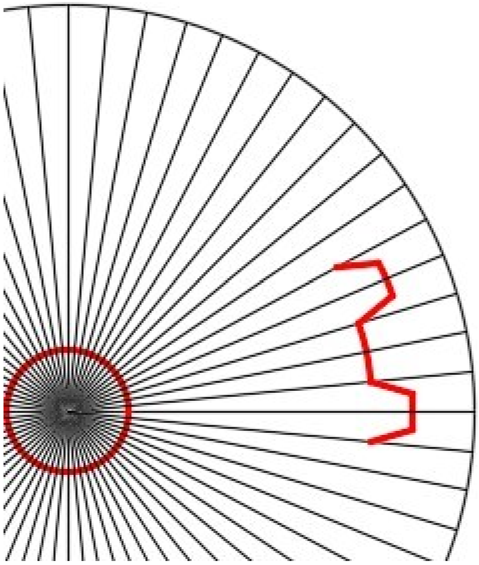


Figure Hint 2C

Hint 3

Delete the unnecessary lines and arcs. (Figure Hint 3)

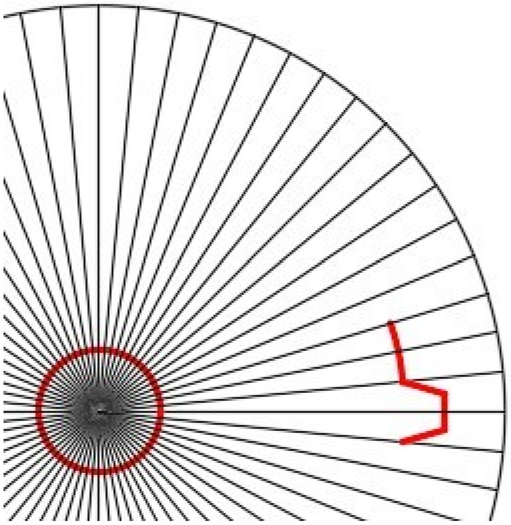


Figure Hint 3

Hint 4

Array the objects 16 times. (Figure Hint 4)

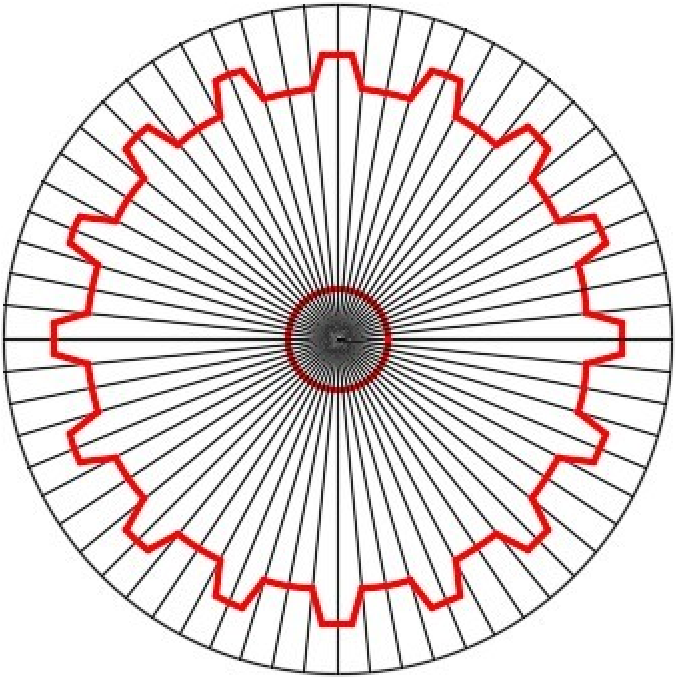


Figure Hint 4

Hint 5

Draw the hexagon first with the top and bottom lines horizontal. Locate its center using the dimensioned drawing. After it is drawn, rotate it using a Reference to the correct angle. (Figure Hint 5)

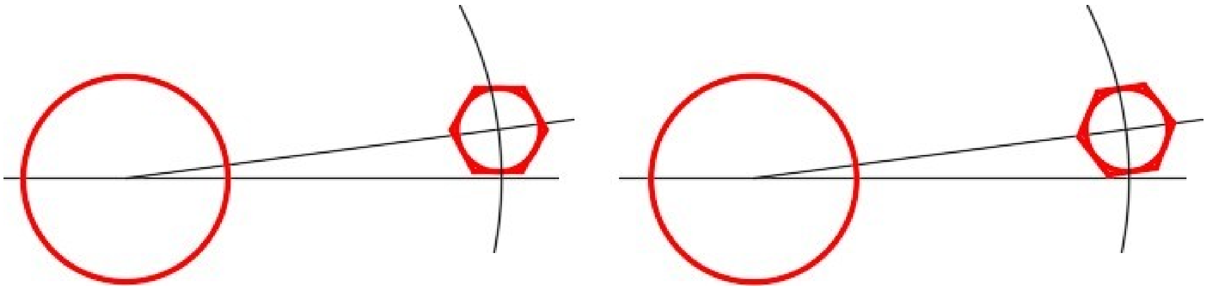


Figure Hint 5

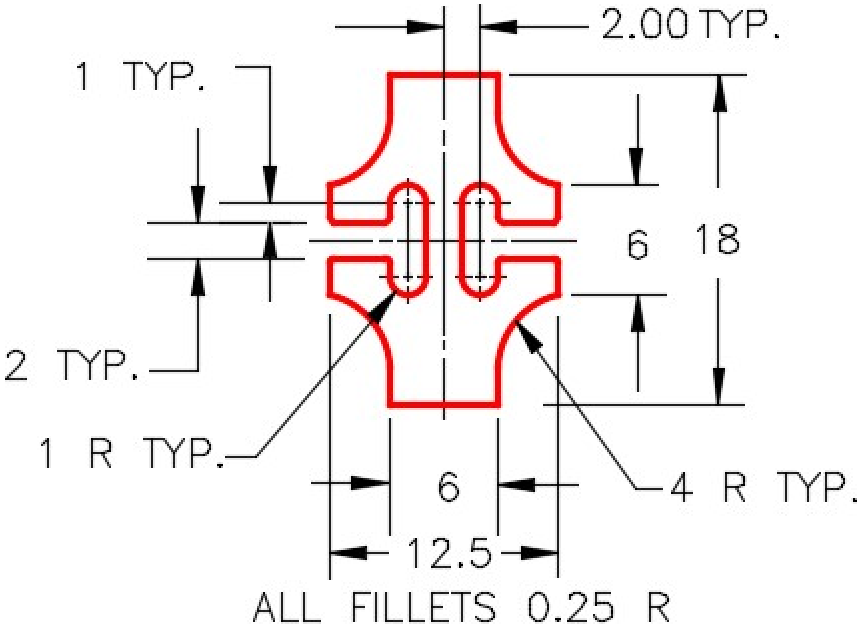
Lab Exercise 23-2

Time allowed: 60 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 23-2	2D Layout Metric	Millimeters

Step 1

On layer: Object, draw the object shown in the figure. Draw it anywhere on the drawing. (Figure Step 1)



2.00 TYP.
1 TYP.
6
18
2 TYP.
1 R TYP.
4 R TYP.
6
12.5
ALL FILLETS 0.25 R

Figure Step 1

Step 2

Copy the object to each location as shown in the figure. (Figure Step 2)

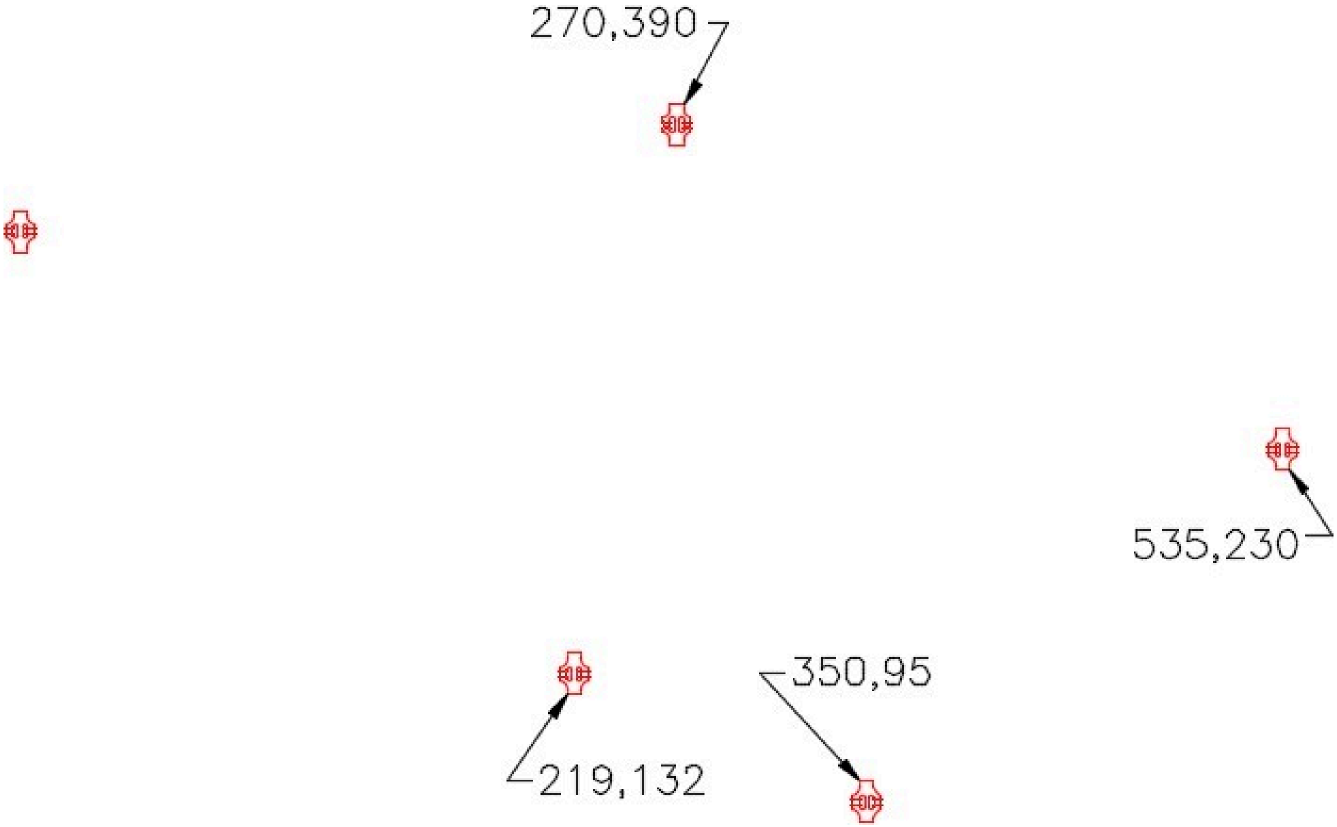

270,390
535,230
350,95
219,132

Figure Step 2

Step 3

Array the object as shown below. (Figure Step 3)

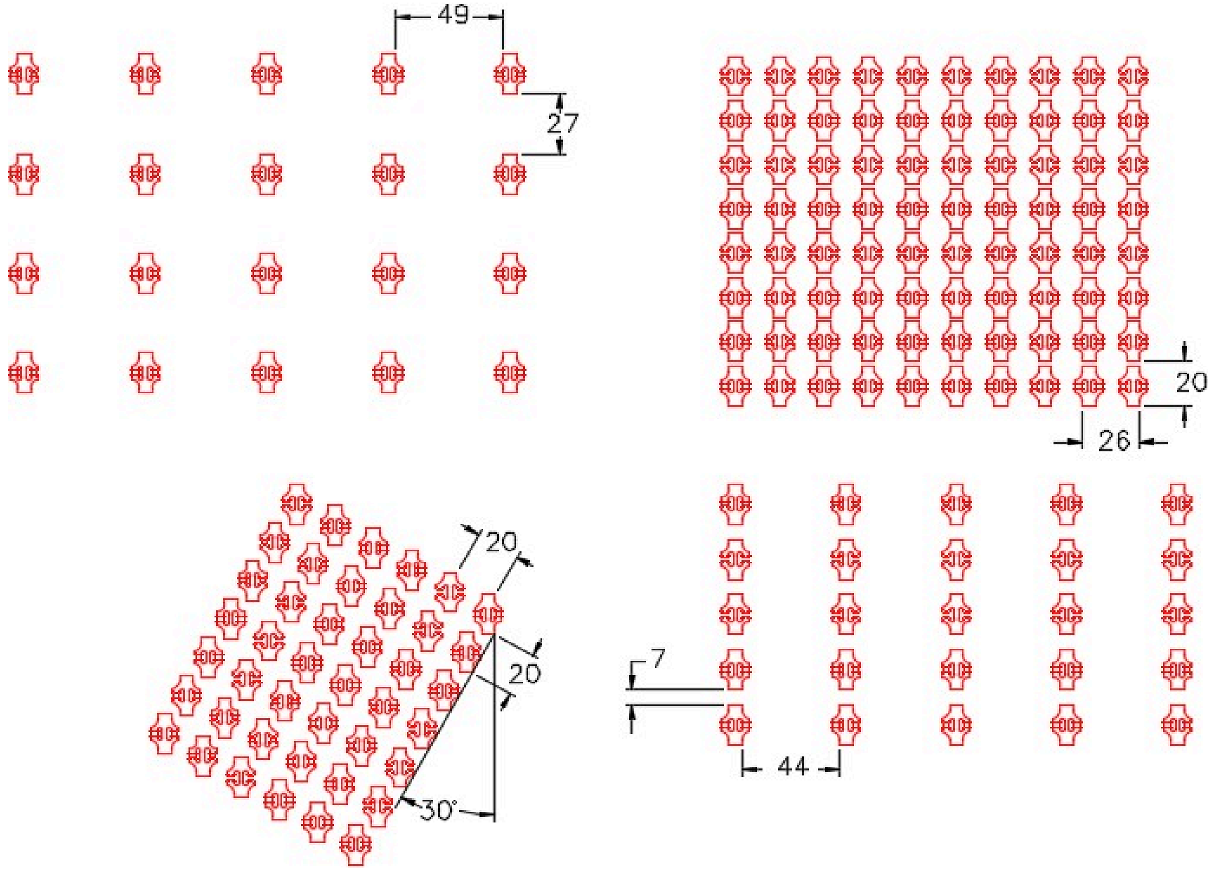


Figure Step 3

AUTHOR'S COMMENTS: Be careful of the spacing as the dimensions are tricky. See the User Tip at the end of "WORKALONG: Using the ARRAY Command – Part 2" to help you figure out the Row offset and the Column offset. In a rectangular array, the offset distance is measured from the one point on the object to the same point on the arrayed object.

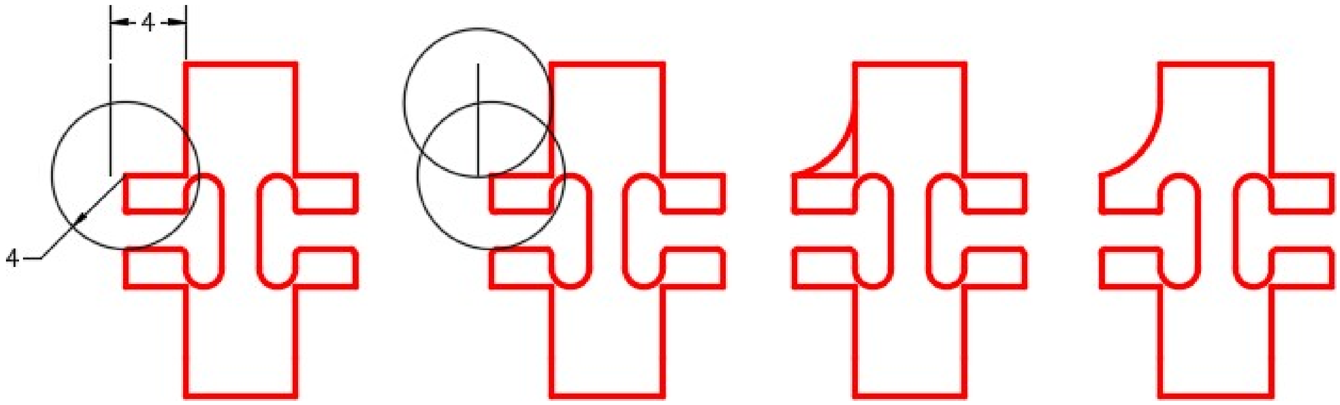
Step 4

Save and close the drawing.

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

Draw one arc as shown. The four step are shown in the figure. If you need help understanding you can see the Geometry Lesson in Module 20. (Figure Hint 1)

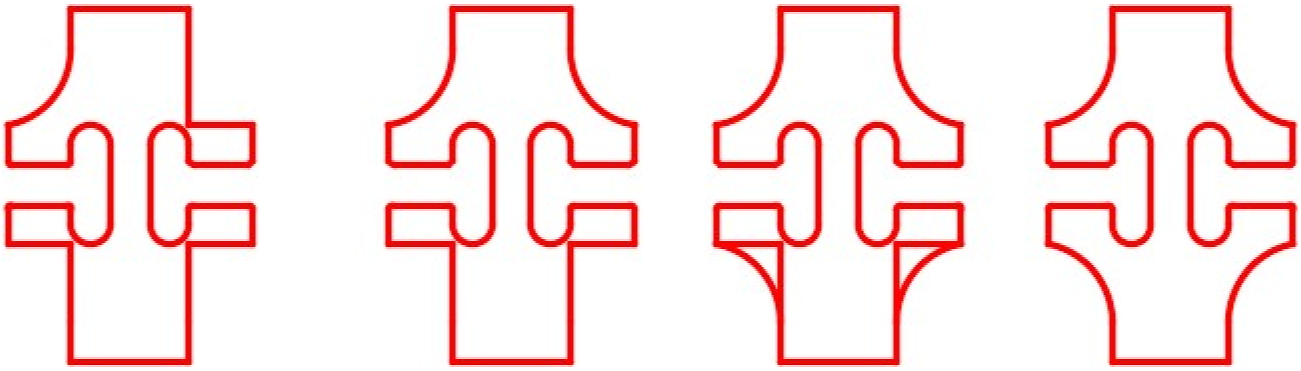


Step 1 *Step 2* *Step 3* *Step 4*

Figure Hint 1

Hint 2

After one arc is drawn, mirror it to the right side. Then mirror the two top arc to the bottom. Trim and erase the lines to complete. (Figure Hint 2)



Step 1 *Step 2* *Step 3* *Step 4*

Figure Hint 2

Module 24: Scaling and Stretching

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe how drawing objects are stretched and scaled to change their original shape and size.
2. Apply the STRETCH and SCALE commands to change the size and shape of existing objects by stretching or scaling them.

Stretching Objects

The STRETCH command has many uses. Using this command is a good way to edit the shape of objects without having to erase and redraw portions of them as shown in Figure Step 24-1. Existing objects can be stretched larger or smaller.

The objects to be stretched MUST be selected using a crossing or cross polygon window. The objects that are totally inside the crossing or cross polygon window are moved and the objects that cross the window are stretched.

Scaling Objects

The SCALE command can be also be used in many ways. An object can be scaled either larger or smaller by using a scale factor or by using a reference and letting AutoCAD calculate the scale factor. See Figure 24-2. The object to be scaled is scaled around a base point.

The base point is the only location that does not change as the object is scaled around it.

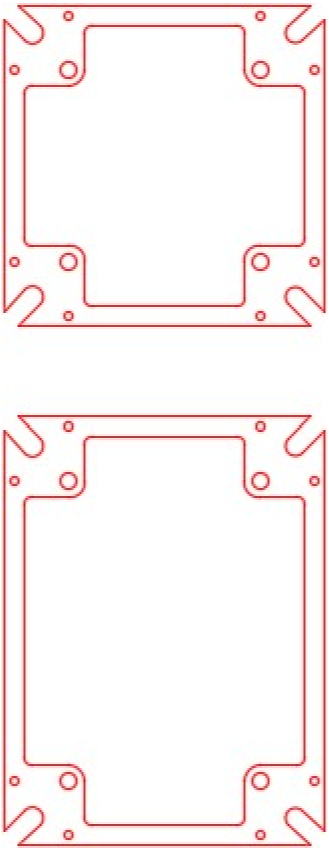


Figure 24-1 Stretching

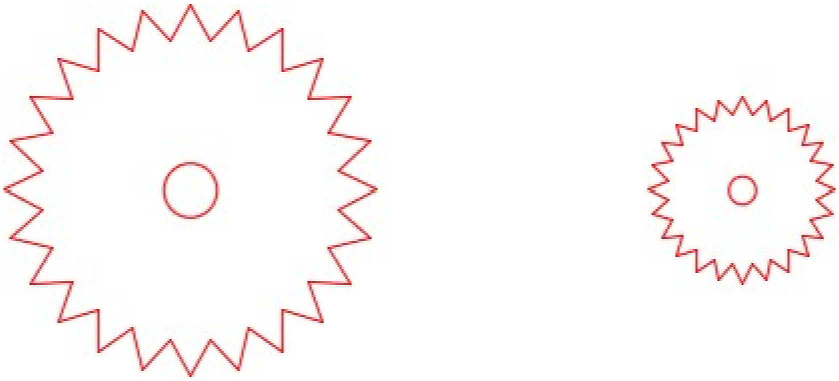


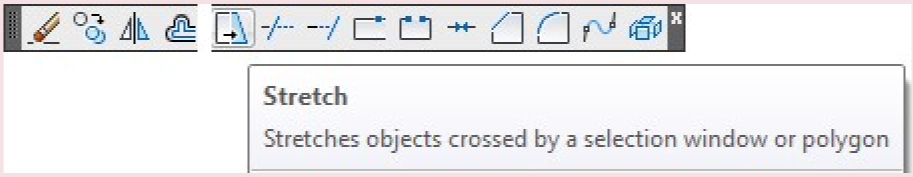
Figure 24-2
Scaling

AUTHOR'S COMMENTS: As soon as you able to identify when and where to use the STRETCH and SCALE commands, your drawing speed and productivity will increase.

AutoCAD Command: STRETCH

The STRETCH command is used to stretch drawing objects either larger or smaller.

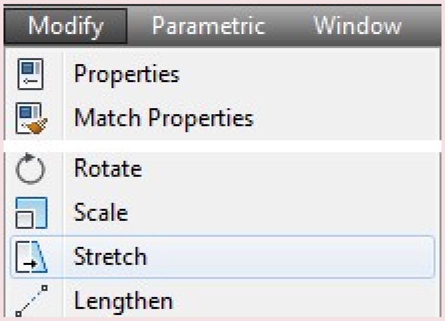
Shortcut: **S**



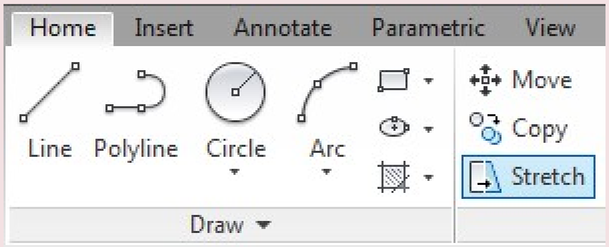
Stretch

Stretches objects crossed by a selection window or polygon

Modify Toolbar


Modify
Parametric
Window
Properties
Match Properties
Rotate
Scale
Stretch
Lengthen

Modify Pull-down

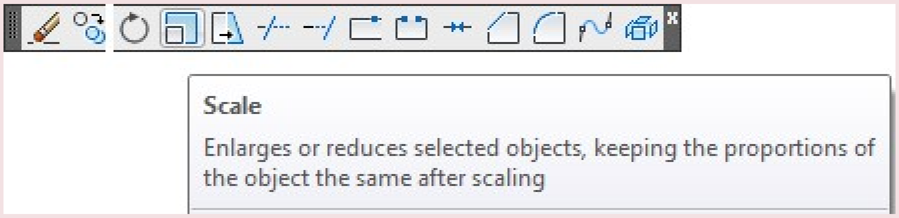

Home
Insert
Annotate
Parametric
View
Line
Polyline
Circle
Arc
Move
Copy
Stretch
Draw

Home Ribbon

AutoCAD Command: SCALE

The SCALE command is used to scale drawing objects, around a base point, either larger or smaller.

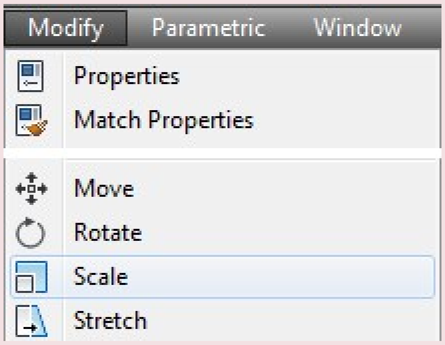
Shortcut: **SC**



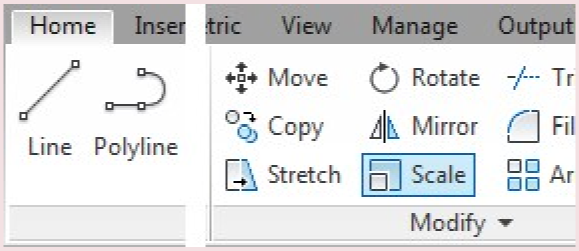
Scale

Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling

Modify Toolbar


Modify
Parametric
Window
Properties
Match Properties
Move
Rotate
Scale
Stretch

Modify Pull-down


Home
Inser
tric
View
Manage
Output
Line
Polyline
Move
Rotate
Tr
Copy
Mirror
Fil
Stretch
Scale
Ar
Modify

Home Ribbon

MUST KNOW: The STRETCH command requires the user to use a crossing or cross polygon window to select the objects. It moves the objects that are totally inside the window and stretches the objects that cross the window. A crossing or cross polygon window MUST be used at all times with the STRETCH command.

WORKALONG: Stretching and Scaling Objects

Step 1

Start a new drawing using the template: 2D Layout English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 24-1.

Step 3

On layer: Object, draw the object shown in the figure. (Figure Step 3)

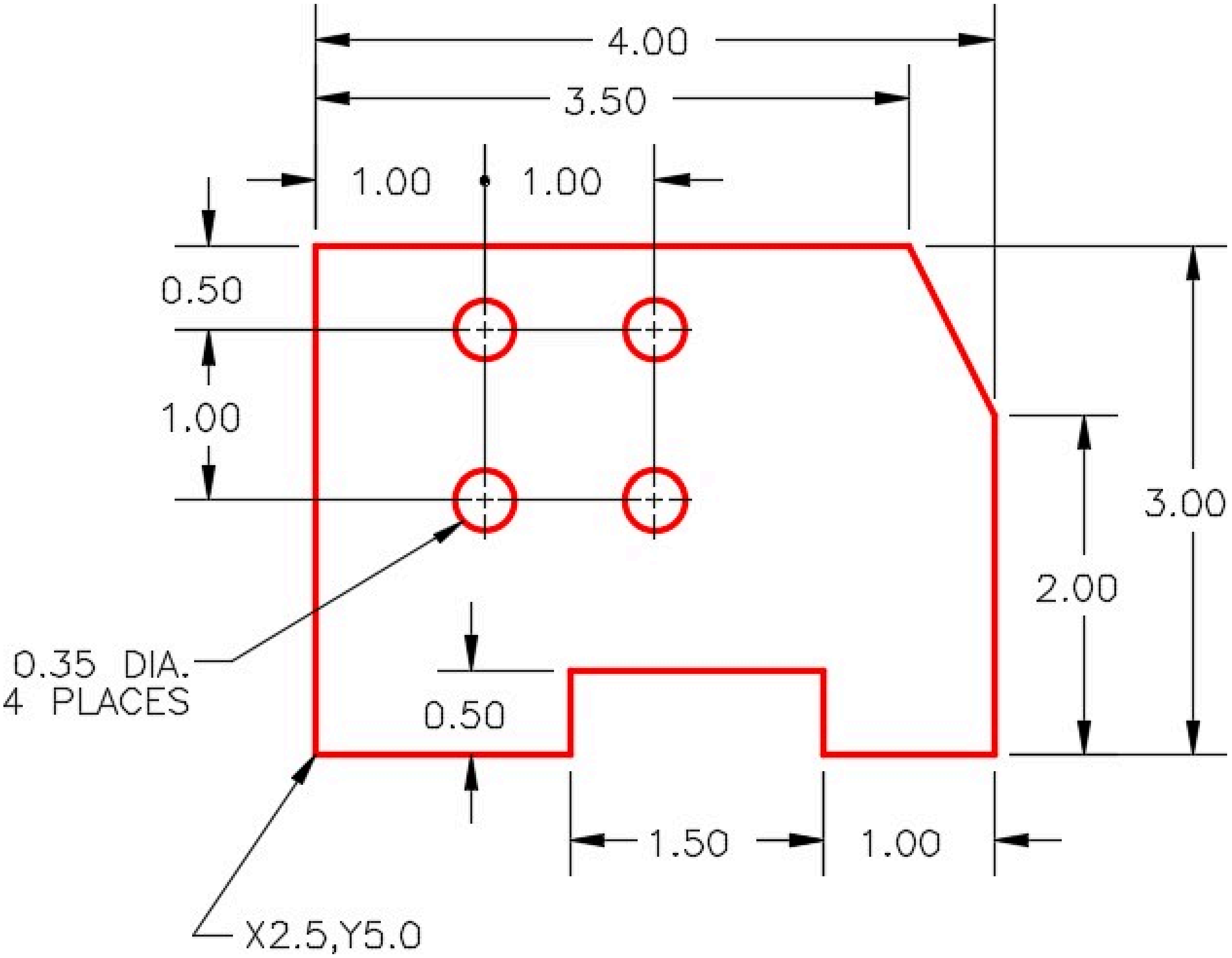

4.00
3.50
1.00
1.00
0.50
1.00
3.00
2.00
0.35 DIA.
4 PLACES
0.50
1.50
1.00
X2.5,Y5.0

Figure Step 3

Step 4

Enter the STRETCH command, as shown below, to stretch the object longer. (Figure Step 4A and 4B)

Command: **STRETCH**

Select objects to stretch by crossing-window or crossing-polygon…

Select objects:

Specify opposite corner: 4 found

(Use a crossing window and locate it as shown in the figure. You MUST use a crossing window in the STRETCH command.)

Select objects:

Specify base point or [Displacement] <Displacement>: **0,0**

Specify second point or <use first point as displacement>: **@1,0**

(The lines that the crossing window cross are stretched 1 unit longer in the positive X direction. The lines totally inside the crossing window are moved 1 unit in the positive X direction.)

Command:

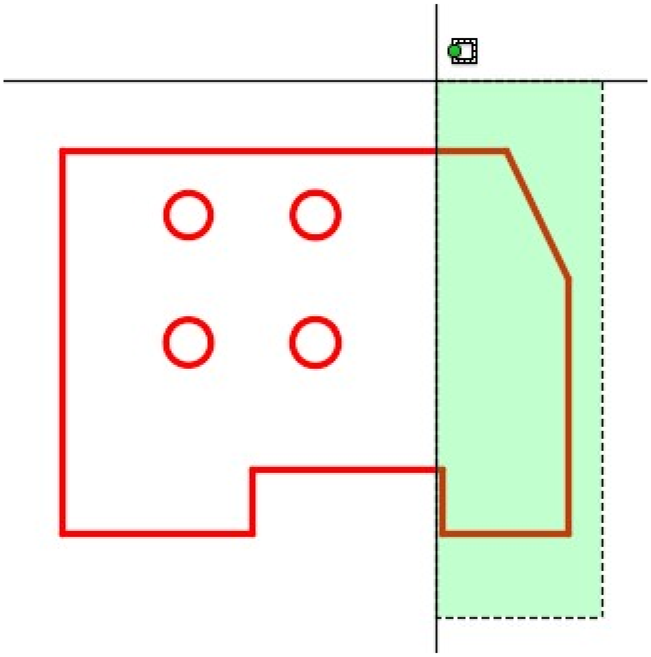


Figure Step 4A

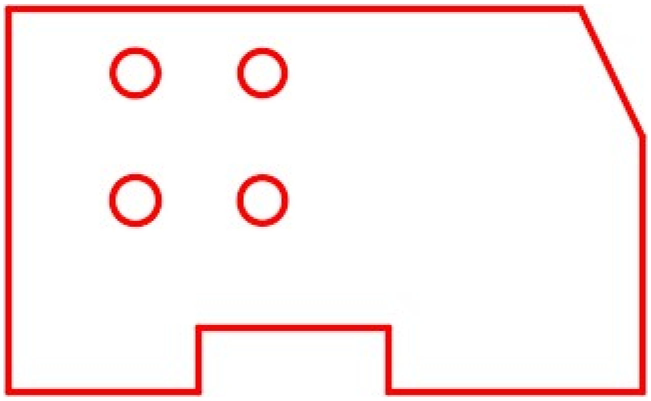


Figure Step 4B

AUTHOR'S COMMENTS: Objects that cross the crossing window are stretched 1 inch longer and the objects that are totally inside the crossing window are moved 1 inch to the right.

Step 5

Enter the STRETCH command, as shown below, to stretch the object higher. (Figure Step 5A and 5B)

Command: **STRETCH**

Select objects to stretch by crossing-window or crossing-polygon…

Select objects:

Specify opposite corner: 4 found

(Use a crossing window and locate it as shown in the figure)

Select objects:

Specify base point or [Displacement] <Displacement>: **0,0**

Specify second point or <use first point as displacement>: **@0,1.5**

(The lines that the crossing window cross are stretched 1.5 units in the positive Y direction and the lines and circles that are totally inside the crossing window are moved 1.5 units in the positive Y direction.)

Command:

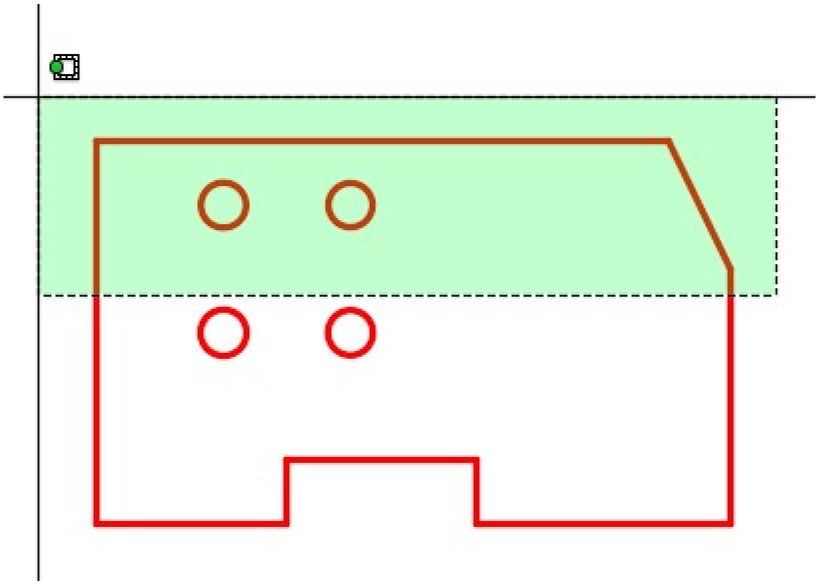


Figure Step 5A

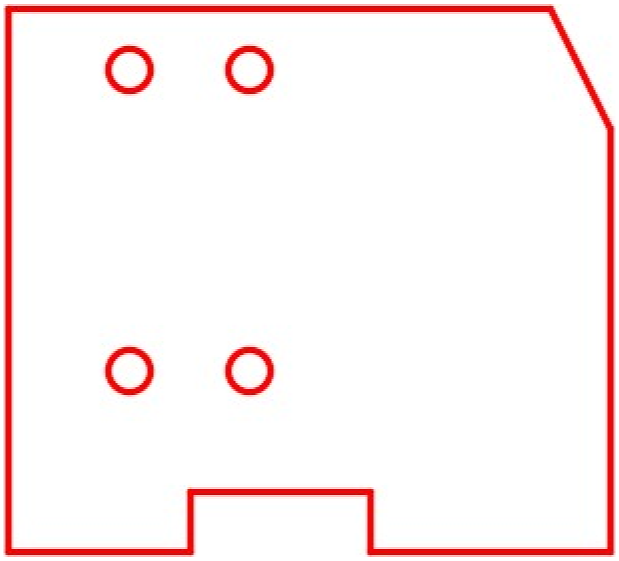


Figure Step 5B

Step 6

Enter the STRETCH command, as shown below, to stretch the object. (Figure Step 6A, 6B, and 6C)

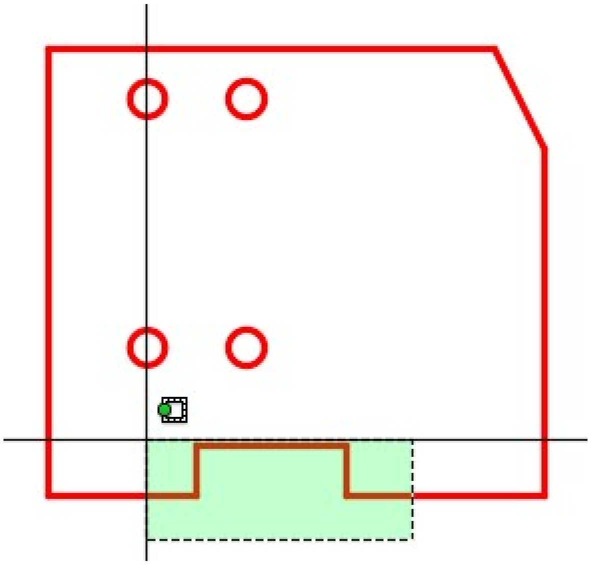


Figure Step 6A

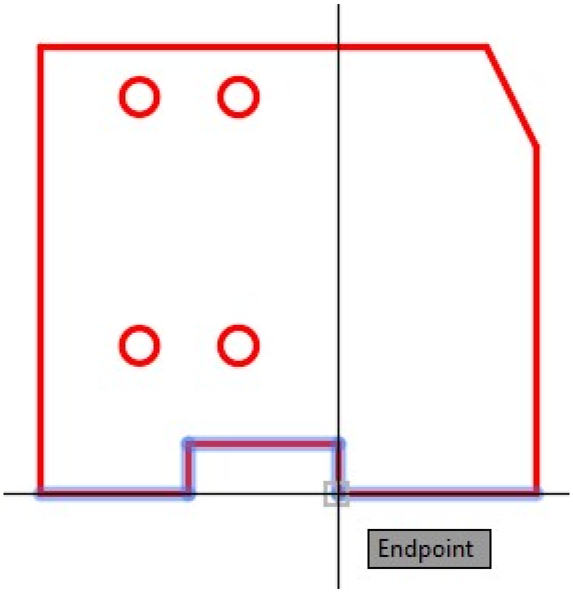


Figure Step 6B

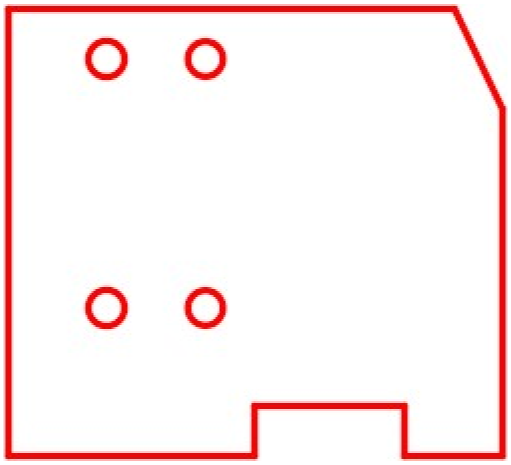


Figure Step 6C

Command: **STRETCH**

Select objects to stretch by crossing-window or crossing-polygon…

Select objects:

Specify opposite corner: 3 found

(Use a crossing window and locate it as shown in the figure.)

Select objects:

Specify base point or [Displacement] <Displacement>: *(end)*

(Snap to the end of the line.)

Specify second point or <use first point as displacement>: **@1,0**

(The lines that the crossing window crosses are stretched 1 unit, one longer and one shorter. The three lines inside the crossing window are moved 1 unit in the positive X direction.)

Command:

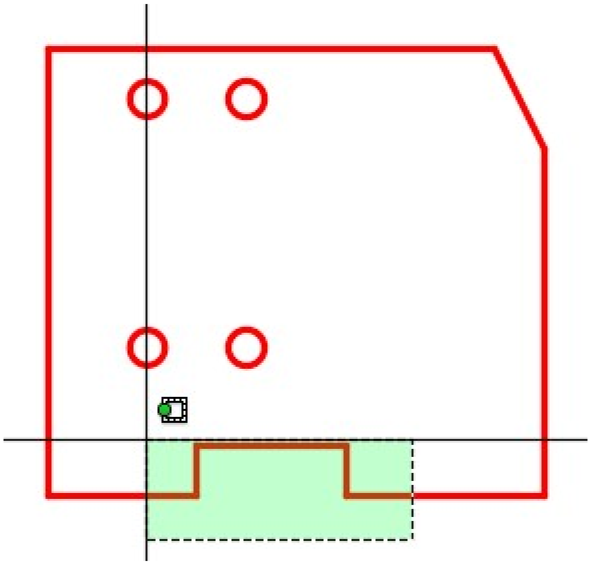


Figure Step 6A

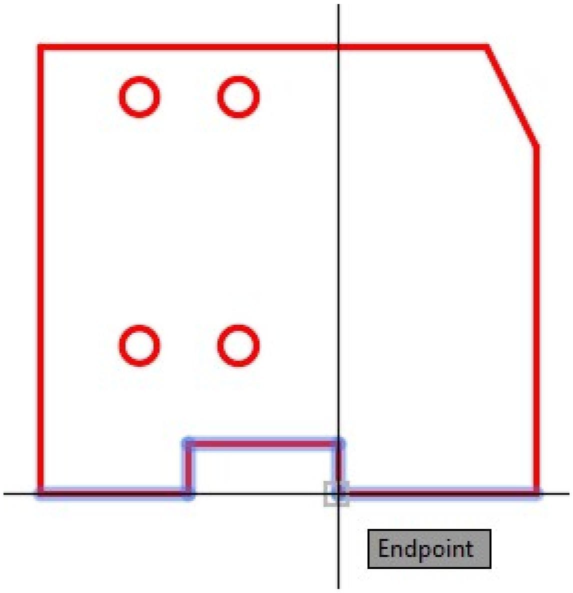


Figure Step 6B

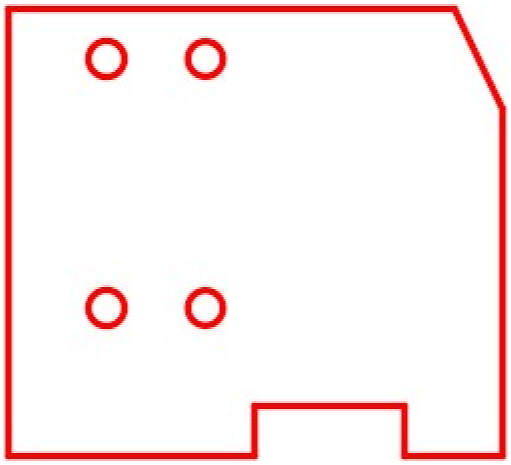


Figure Step 6C

AUTHOR’S COMMENTS: As you can see, lines can stretched longer or shorter.

Step 7

Using what you already learned, use the STRETCH command to stretch the object to match Figure Step 7A. See Figure Step 7B for the location of the cross polygon window that you have to use. (Figure Step 7A, 7B, and 7C)

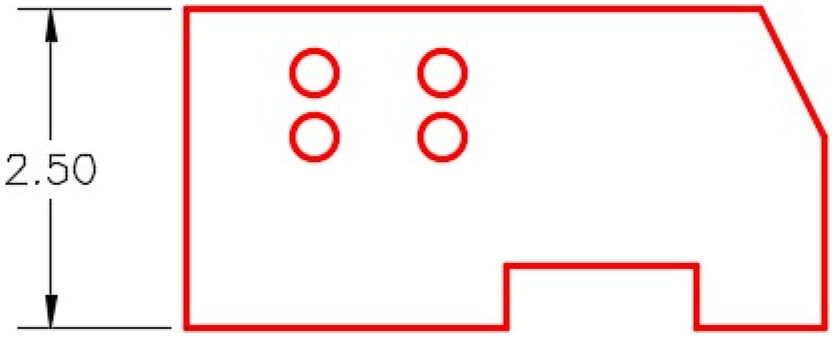


Figure Step 7A

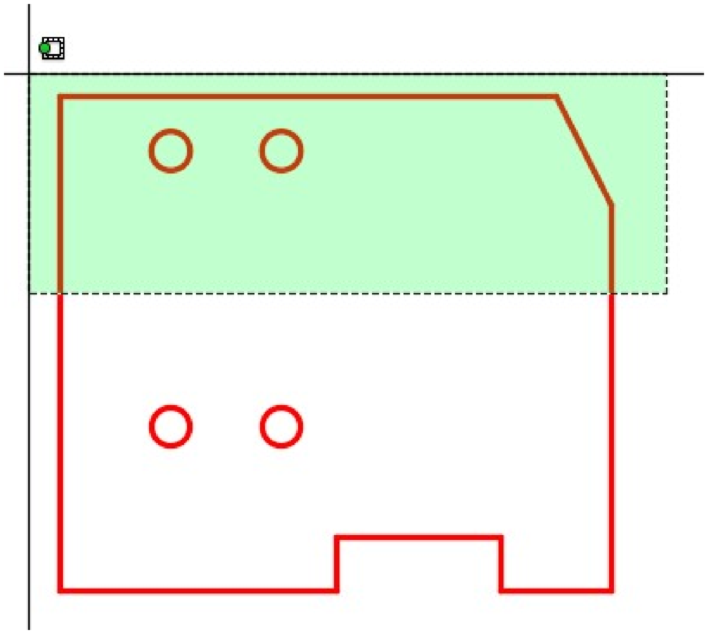


Figure Step 7B

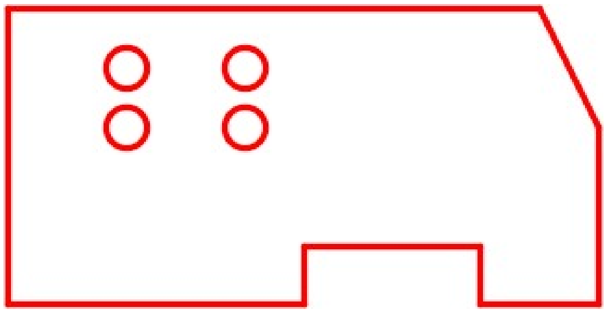


Figure Step 7C

Step 8

Using what you already learned, use the STRETCH command to stretch the object to match Figure Step 8A. See Figure Step 8B for the location of the cross polygon window that you have to use. (Figure Step 8A, 8B, and 8C)

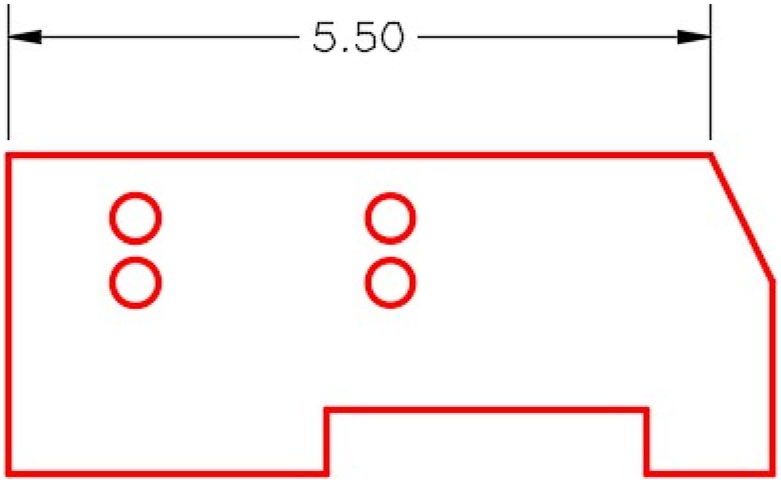


Figure Step 8A

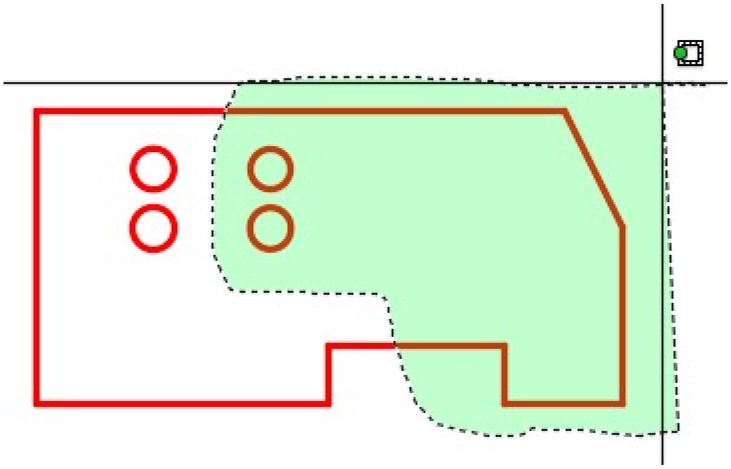


Figure Step 8B

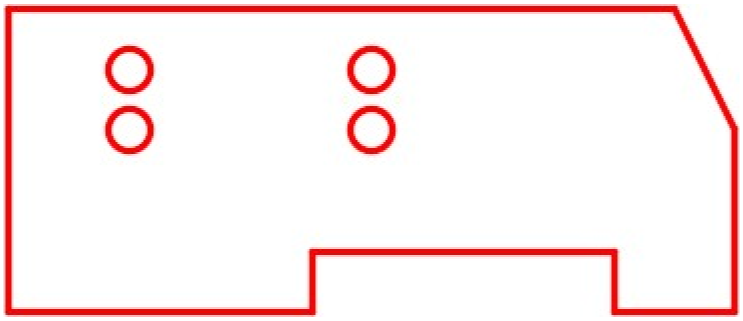


Figure Step 8C

Step 9

Using what you already learned, use the MIRROR command to mirror the object to match Figure Step 9A. (Figure Step 9A and 9B)

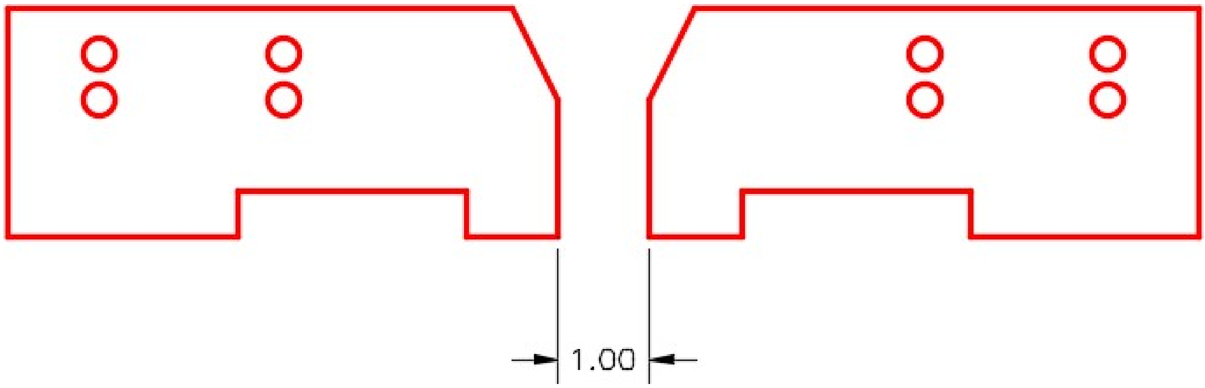


Figure Step 9A

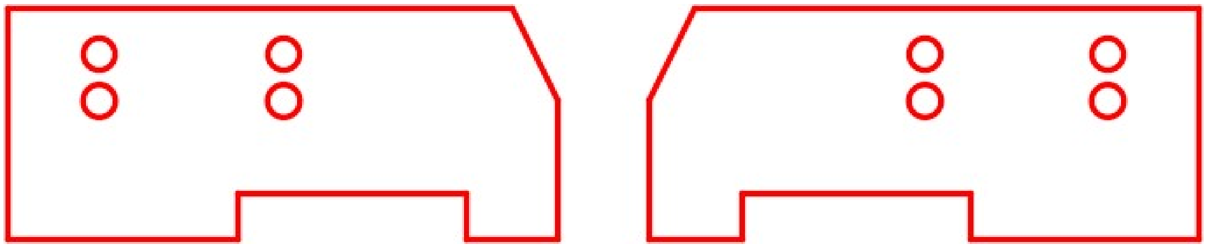


Figure Step 9B

Step 10

Enter the SCALE command, as shown below, to scale the object on the right by a scale factor of 0.5. (Figure Step 10A, 10B, and 10C)

Command: **SCALE**

Select objects:

Specify opposite corner: 13 found

(Select the objects to be scaled in a window.)

Select objects:

Specify base point: *(end)*

(Snap to the end of bottom left line to establish the base point. This point will remain at the same location and the remainder of the object will be scaled.)

Specify scale factor or [Copy/Reference] <1.0000>: 0**.5**

(Scale the object by a scale factor of one-half.)

Command:

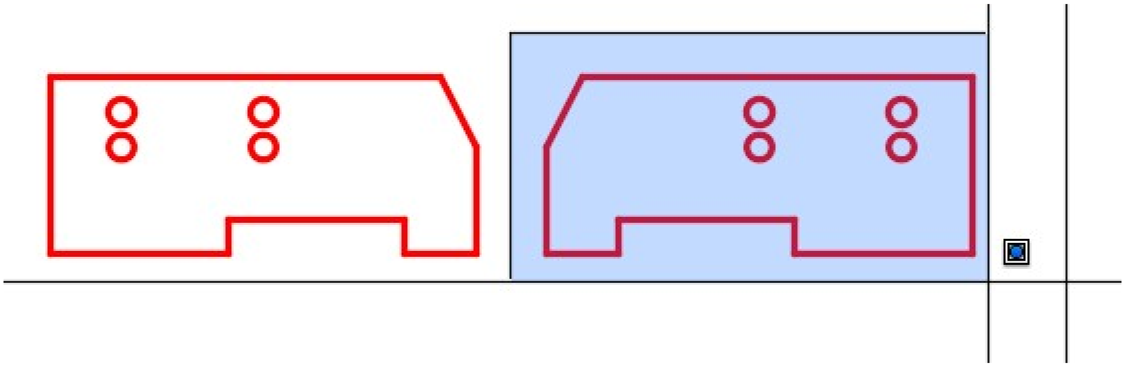


Figure Step 10A

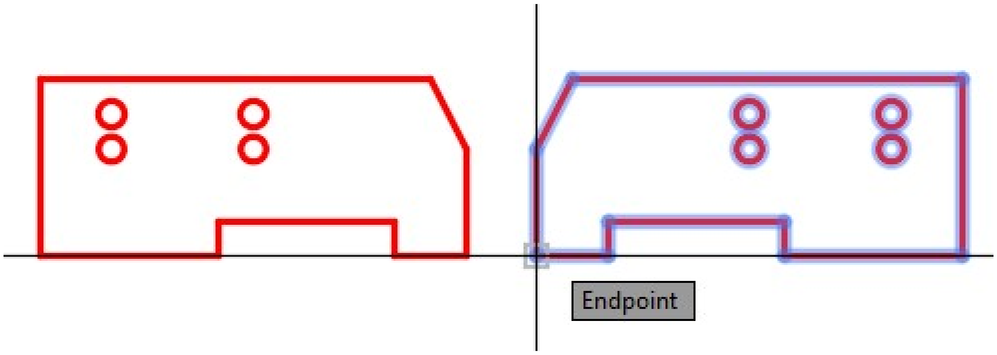


Figure Step 10B

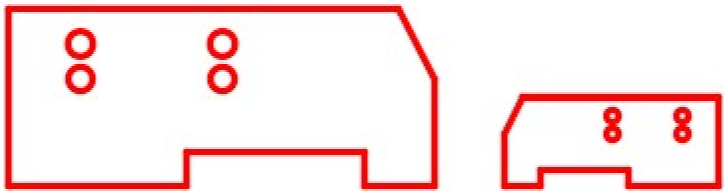


Figure Step 10C

AUTHOR'S COMMENTS: You must be careful when using a scale factor. When you apply a scale factor of 0.5, you are scaling the object by one-half the height and one-half the width. Therefore, the object will be one-quarter of the size it was before the scale. The opposite happens when you scale the object larger. For example, when you apply the scale factor of 2, it scales the object 2 times the width and 2 times the height making the object 4 times its size before the scale.

Step 11

On layer: Construction, enable Ortho mode and draw two lines. Snap to the endpoints of existing lines, as shown in the figure, drawing one vertical line and one horizontal line. The length of the lines is not important but they must intersect. (Figure Step 11)

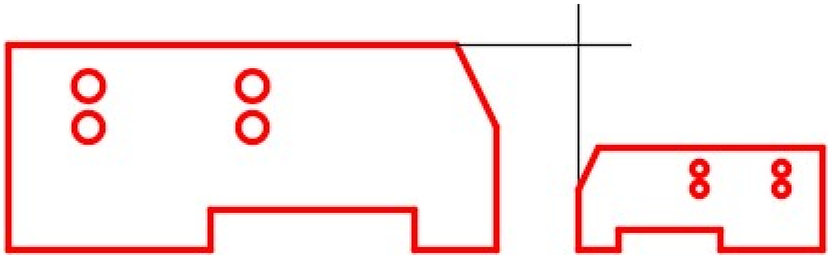


Figure Step 11

Step 12

Enter the SCALE command, as shown below, to scale using the Reference option. (Figure Step 12A, 12B, 12C, 12D, and 12E)

Command: **SCALE**

Select objects:

Specify opposite corner: 13 found

(Select the object on the right using a window as shown Figure Step 12A.)

Select objects:

Specify base point: *(end)*

(Snap to the end of the line for the base point.)

Specify scale factor or [Copy/Reference] <1.5000>: **R**

(Use a Reference.)

Specify reference length <1.0000>: *(end)*

Specify second point: *(end)*

(Specifying a reference length.)

Specify new length or [Points] <1.0000>: *(end)*

(Specifying the new length of the reference length.)

Command:

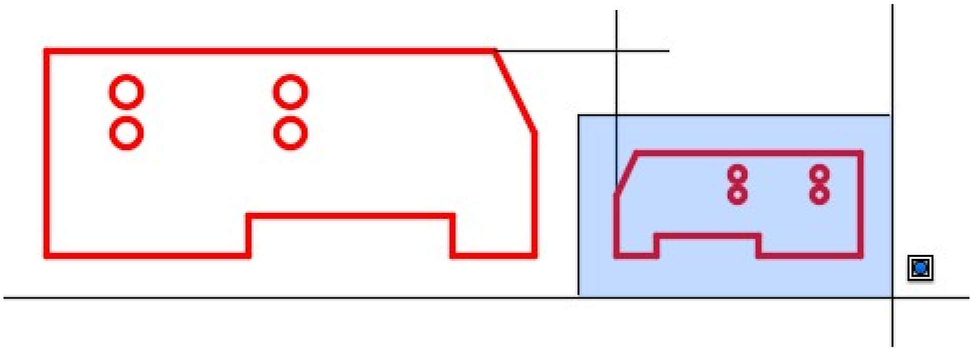


Figure Step 12A

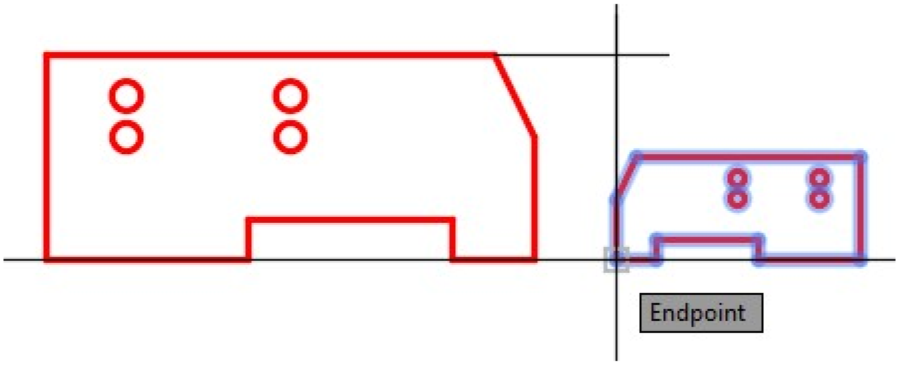


Figure Step 12B

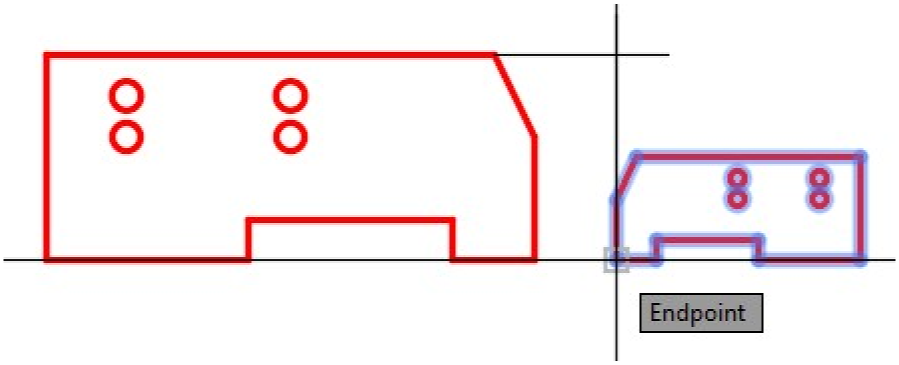


Figure Step 12C

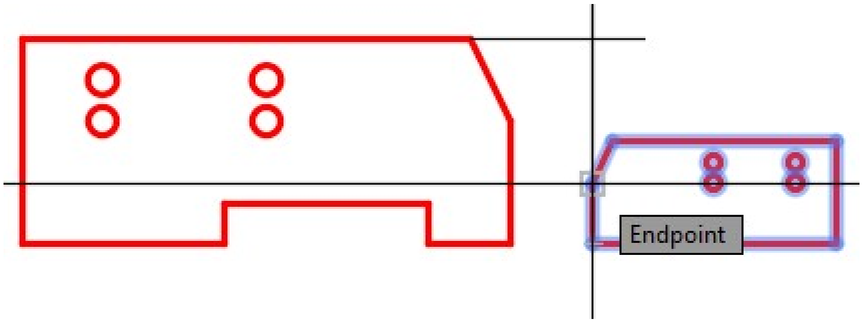


Figure Step 12D

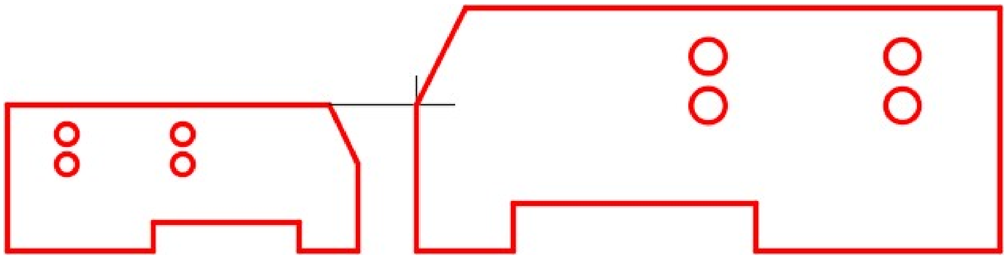


Figure Step 12E

AUTHOR'S COMMENTS: This example instructs AutoCAD to measure the distance between the two endpoints. It then calculates the scale factor using the distance between the locations selected.

Step 13

Save and close the drawing.

WORKALONG: Rotating and Scaling Objects

Step 1

Start a new drawing using the template: 2D Layout English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 24-2.

Step 3

On layer: Object, draw the object shown in the figure. (Figure Step 3)

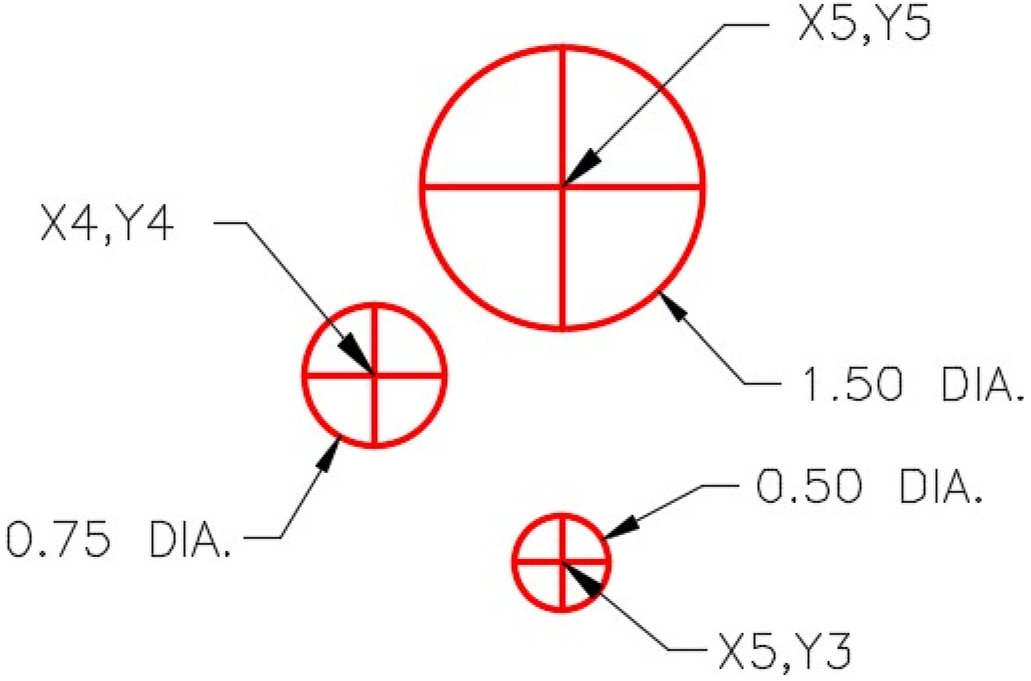

X5,Y5
X4,Y4
1.50 DIA.
0.50 DIA.
0.75 DIA.
X5,Y3

Figure Step 3

Step 4

On layer: Construction, draw a line from center of the large circle to the center of the medium circle as shown in the figure. (Figure Step 4).

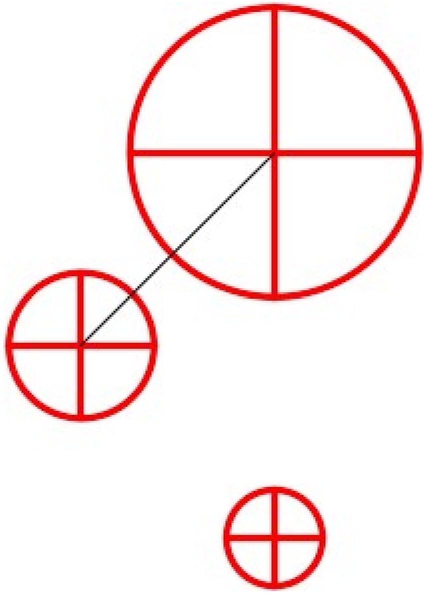


Figure Step 4

Step 5

Using the ROTATE command, rotate the large circle, using the Reference option, to line up the quad lines with the construction line. (Figure Step 5).

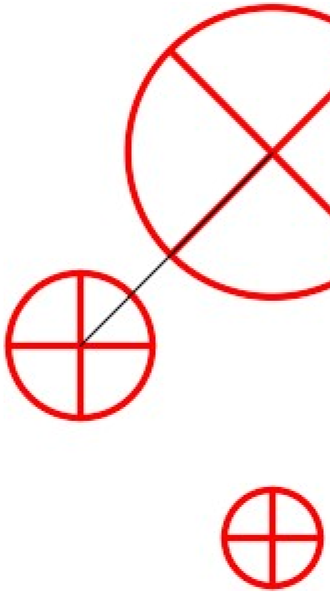


Figure Step 5

Step 6

Using the ROTATE command, rotate the medium size circle, using the Reference option, to line up the quad lines with the construction line. (Figure Step 6).

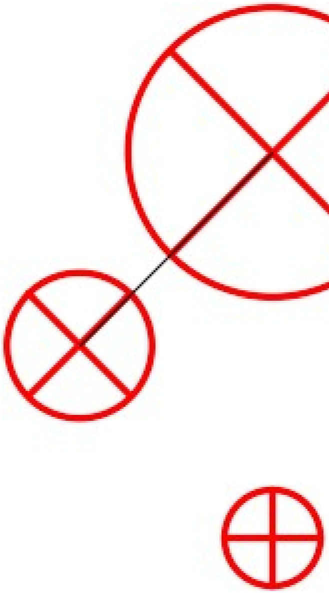


Figure Step 6

Step 7

Using the MOVE command, move the small circle snapping the bottom quad to the intersection of the medium circle and the construction line. (Figure Step 7)

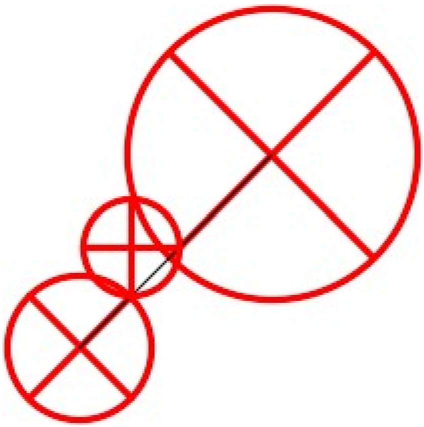


Figure Step 7

Step 8

Using the ROTATE command, rotate the small circle, using the Reference option, to line up the quad lines with the construction line. (Figure Step 8)

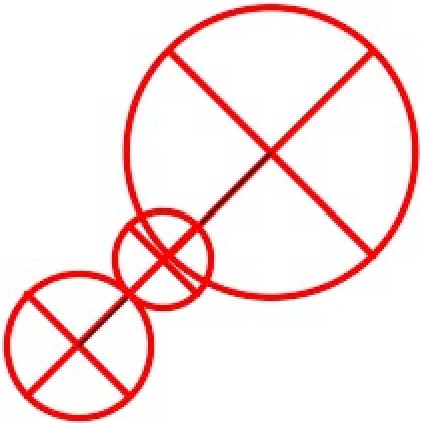


Figure Step 8

Step 9

Using the SCALE command, scale the small circle, using the Reference option, to fit exactly between the other two circles. (Figure Step 9)

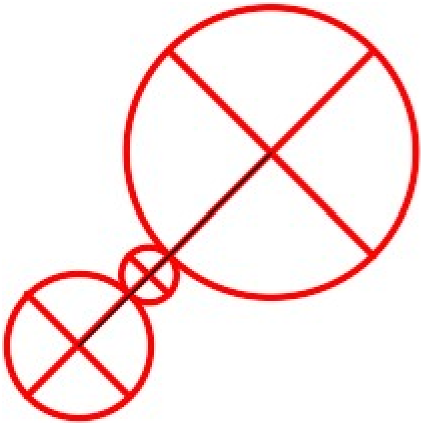


Figure Step 9

Step 10

Freeze layer: Construction. Your completed drawing should appear as shown in the figure. (Figure Step 10)

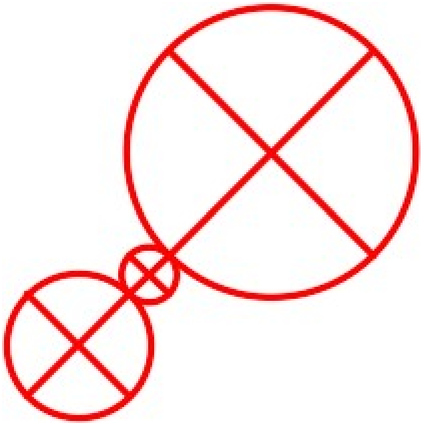


Figure Step 10

Step 11

Save and close the drawing.

Key Principles

Key Principles in Module 24

1. The STRETCH command requires that a crossing or cross polygon window be used to select the objects. It moves the objects that are totally inside the window and stretches the objects that cross the window. You MUST use a crossing or cross polygon window at all times with the STRETCH command.
2. The SCALE command can scale an object either larger or smaller by using a scale factor or a reference and let AutoCAD calculate the scale factor. The objects to be scaled is scaled around a base point. The base point is the only location that does not change as the objects are scaled around it.
3. A scale factor of 0.5 scales the object by one-half its height and one-half its width and therefore scales the object one-quarter of its original size.

Lab Exercise 24-1

Time allowed: 45 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 24-1	2D Layout English	Inches

Step 1

On layer: Object, draw the object shown in the figure. (Figure Step 1A and 1B)

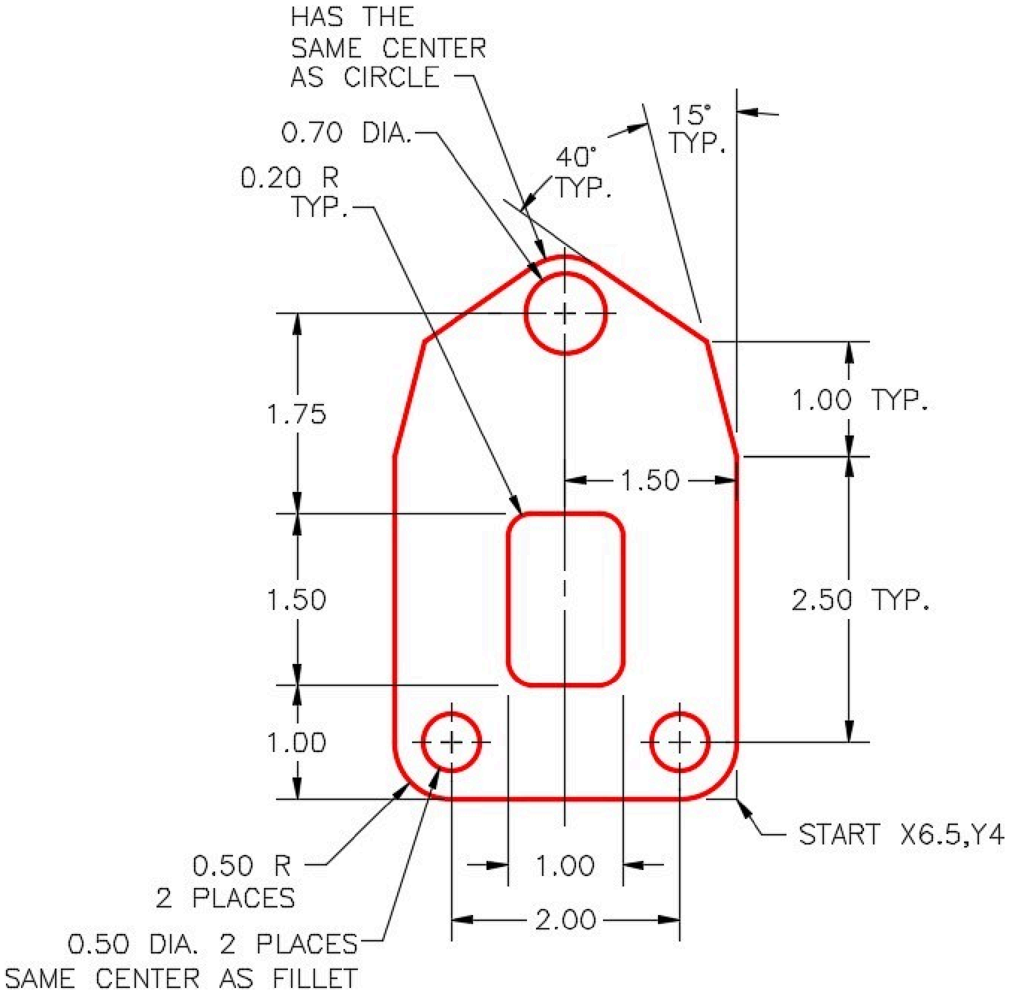

HAS THE
SAME CENTER
AS CIRCLE
0.70 DIA.
0.20 R
TYP.
40°
TYP.
15°
TYP.
1.75
1.50
1.00
1.00 TYP.
1.50
2.50 TYP.
START X6.5,Y4
0.50 R
2 PLACES
0.50 DIA. 2 PLACES
SAME CENTER AS FILLET
1.00
2.00

Figure Step 1A

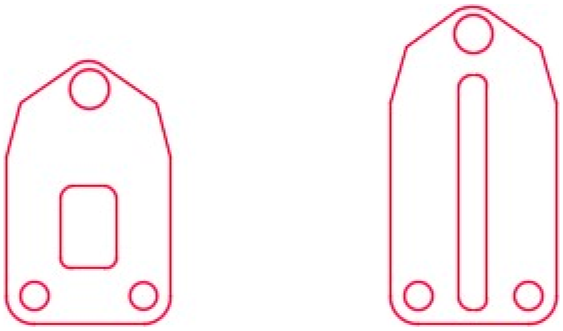


Figure Step 1B
Completed Drawing

Step 2

Set the insertion units and insert the key. Check your drawing for accuracy.

Step 3

Copy the completed object 7 inches in the positive X direction. (Figure Step 3)

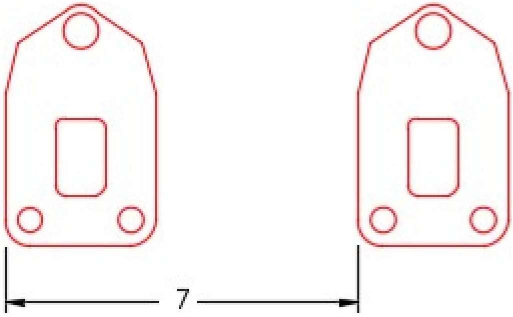


Figure Step 3

Step 4

Using the STRETCH command only, edit the object on the right to match the figure. (Figure Step 4A and 4B)

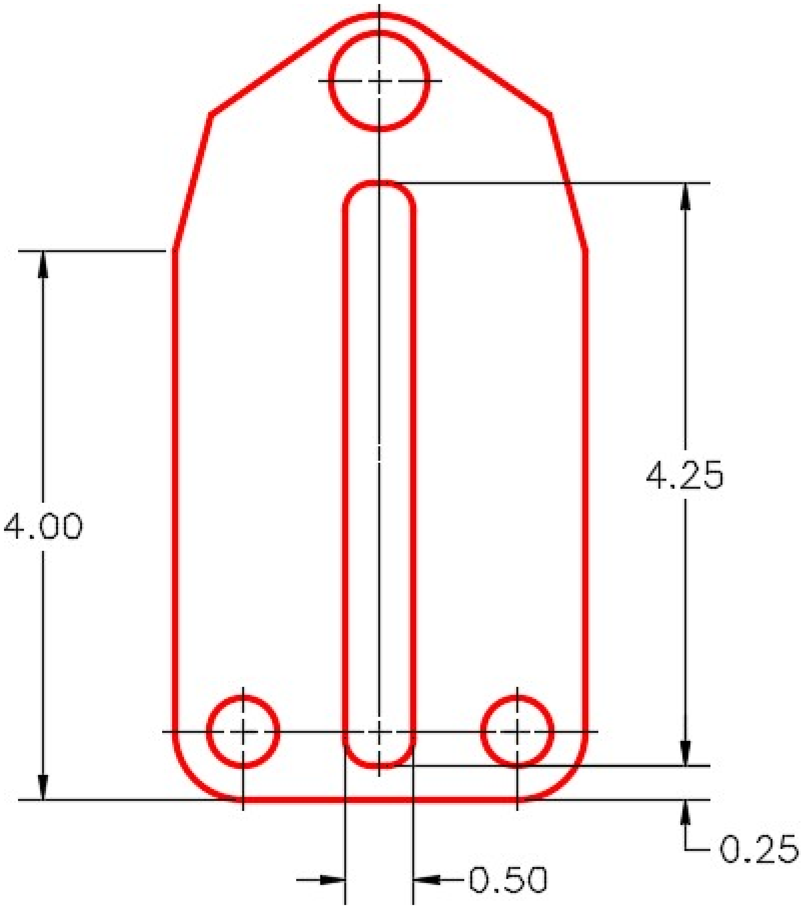


Figure Step 4A

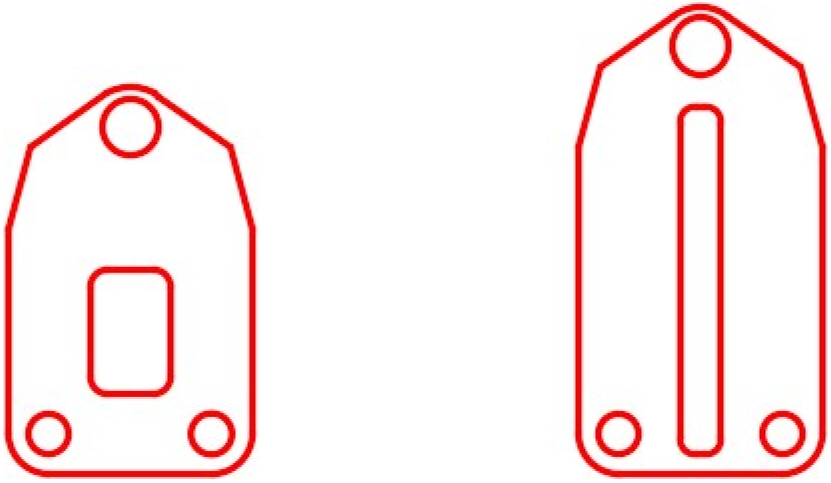


Figure Step 4B

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

To draw the arc at the top, draw a construction line by snapping the first point to the center of the circle and the second point snapping it perpendicular to the inclined line. Draw a circle, on layer: Object, snapping to the center of the circle for the center point and the end of the construction line for the radius. Trim the circle. (Figure Hint 1)

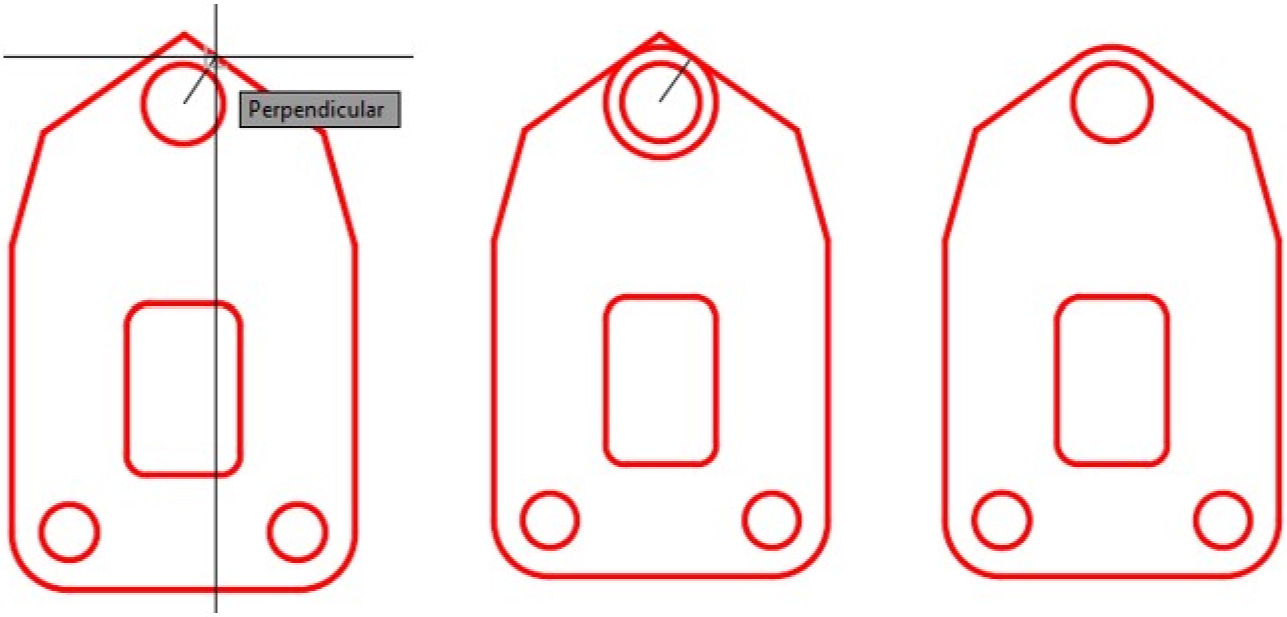


Figure Hint 1

AUTHOR'S COMMENTS: A line drawn from the center of the circle perpendicular to the inclined line locates the tangent point and the radius for the arc.

Hint 2

See the following steps to edit the object using the STRETCH command.

Step 1

Stretch the object 1 inch in the positive Y direction. (Figure Hint 2 – Step 1)

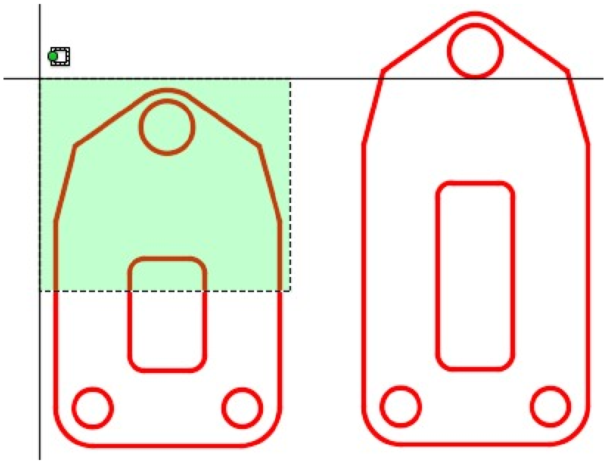


Figure Hint 2 – Step 1

Step 2

Stretch the right side of the slot 0.25 inches towards the center. (Figure Hint 2 – Step 2)

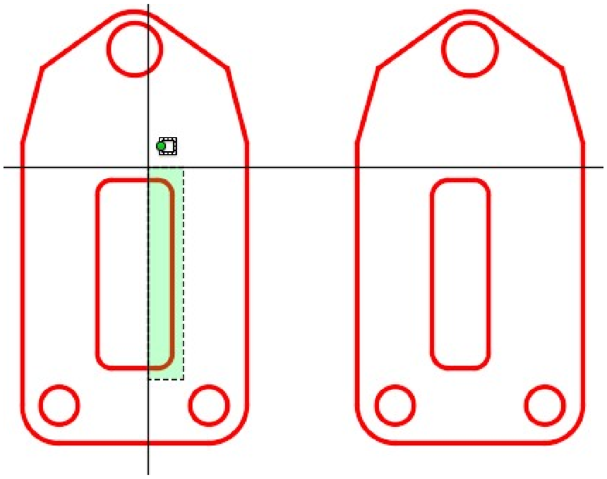


Figure Hint 2 – Step 2

Step 3

Stretch the left side of the slot 0.25 inches towards the center. (Figure Hint 2 – Step 3)

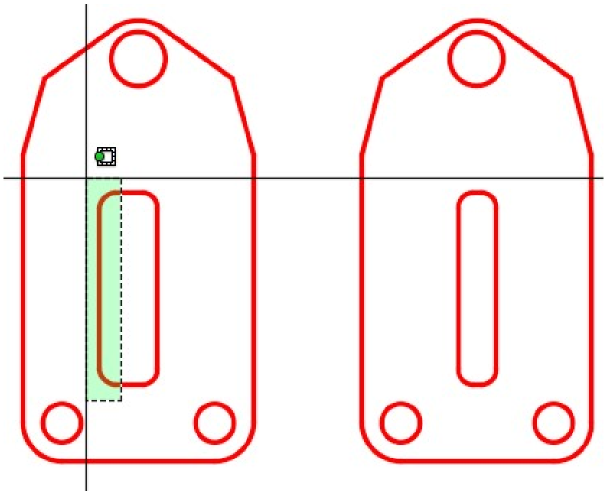


Figure Hint 2 – Step 3

Step 4

Draw construction lines using the dimensioned drawing in Step 1 as a reference. (Figure Hint 2 – Step 4)

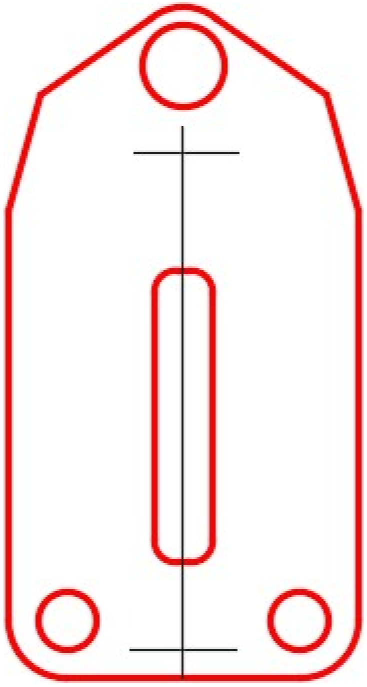


Figure Hint 2 – Step 4

Step 5

Stretch the bottom of the slot using the construction lines to snap to. Do the same for the top of the slot. (Figure Hint 2 – Step 5)

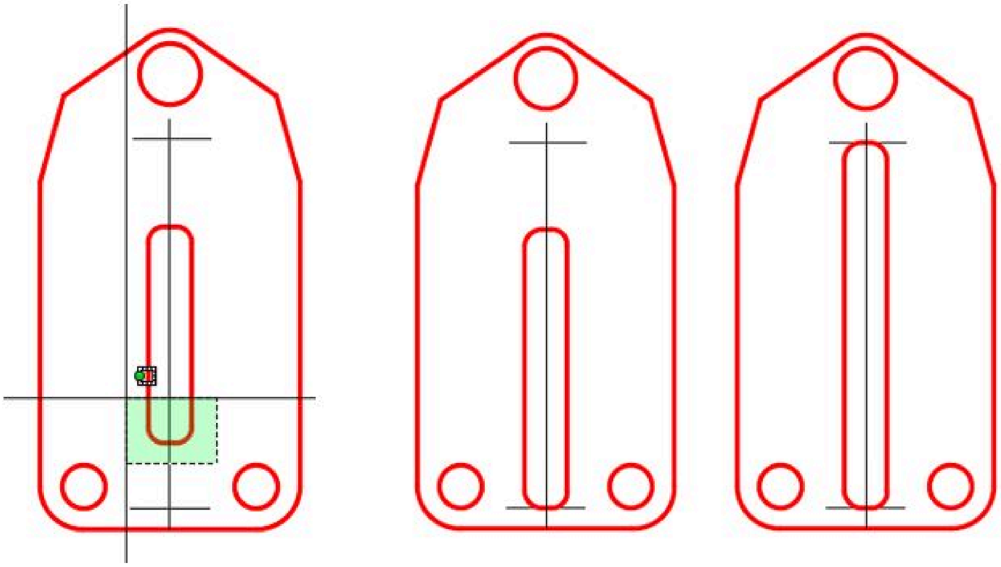


Figure Hint 2 – Step 5

Lab Exercise 24-2

Time allowed: 60 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 24-2	2D Layout English	Inches

Step 1

On layer: Object, draw the object shown in the figures. This object will be referred to as the original object in this lab exercise. (Figure Step 1A and 1B)

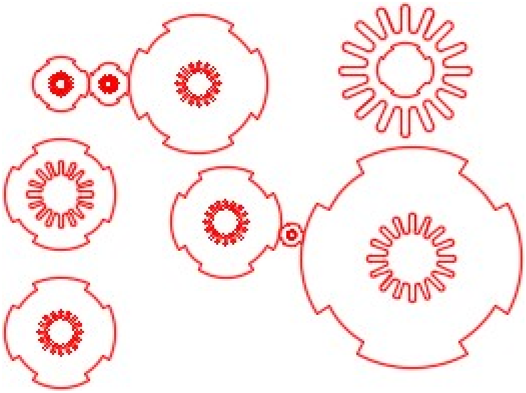


Figure Step 1A
Completed Drawing

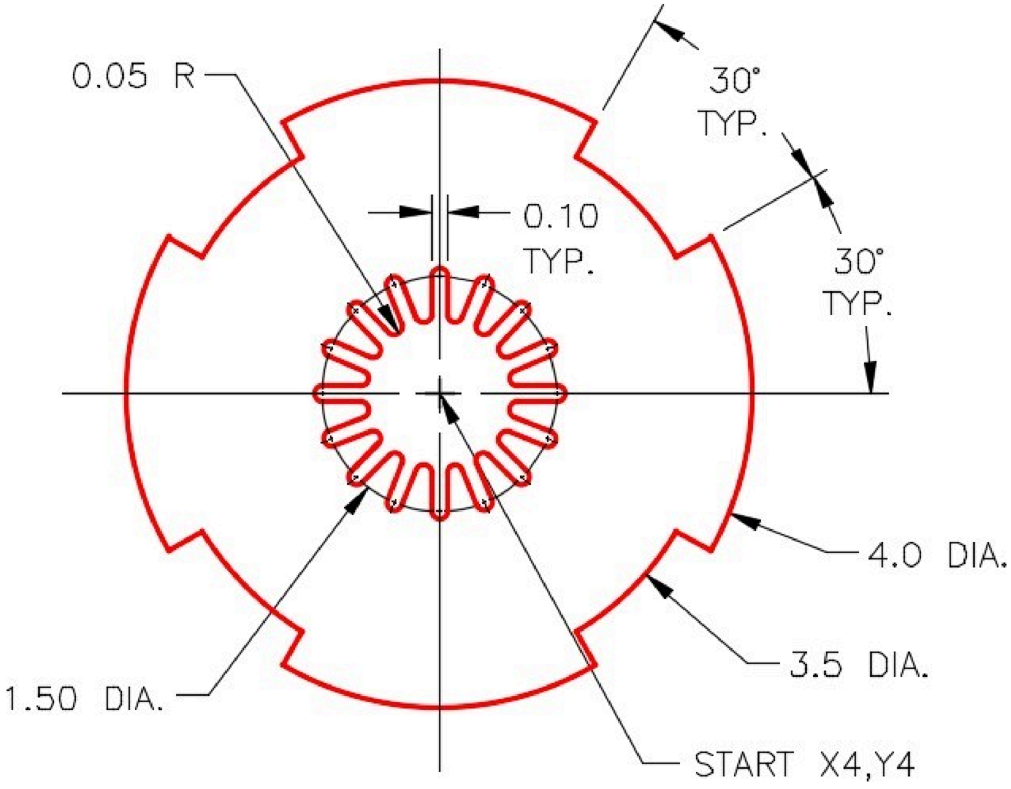

0.05 R
30° TYP.
0.10 TYP.
30° TYP.
4.0 DIA.
3.5 DIA.
1.50 DIA.
START X4,Y4

Figure Step 1B Dimensioned Drawing

Step 2

Copy and scale the original object in steps A to H: (Figure Step 2)

A Copy the center to X4,Y9. Scale the inside array only. Use the scale factor of 1.5. Use the center as base point.

B Copy the center to X4,Y13. Scale the whole object at a scale factor of 0.5. Use the center as base point.

C Copy the center to X9,Y13. Scale the outside array only. Use a scale factor of 1.25 Use the center as base point.

D Copy the center to X10,Y8.

E Copy the center to X16.75,Y6.7. Scale the whole object at a scale factor of 2. Use the center as base point.

F Copy the center to X16.5,Y13.5. Scale the inside array only at a scale factor of 3. Scale the outside array at a scale factor of 0.5. Use the center as the base point.

G Copy the object and scale it to fit exactly between the existing objects B and C. Scale it to fit exactly between the midpoint on the arcs. Use the Reference option do not use math.

AUTHOR'S COMMENTS: Draw a construction line between the centers of B and C.

H Copy the object and scale it to fit exactly between the closest distance between the two existing objects D and E. Rotate the object so the midpoints of the arcs are touching the existing objects. Use the Reference option. Do not use math.

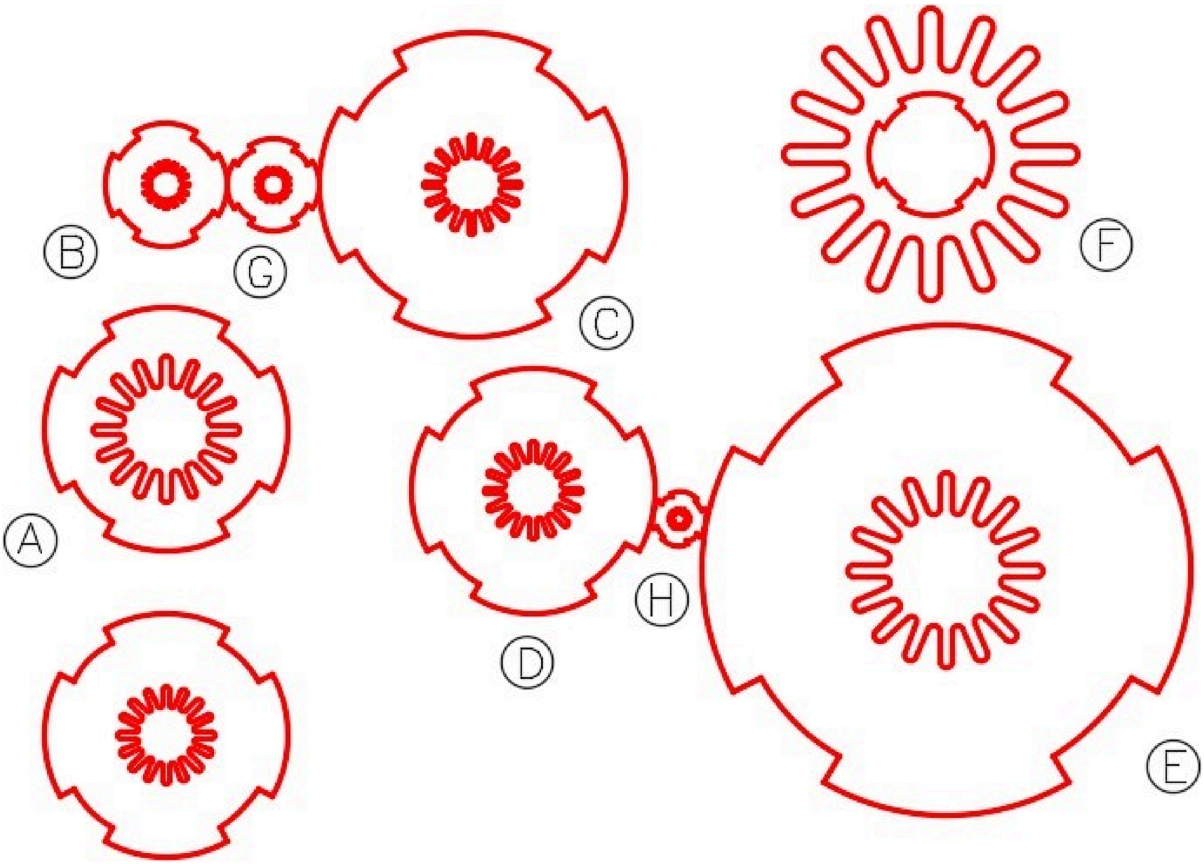


Figure Step 2

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

Using the dimensioned drawing, complete the following steps:

Step 1

Draw 2 lines and an the arc.

Step 2

Array them 16 times.

Step 3

Deleted unwanted objects and insert two 0.05 radius fillets.

Step 4

Deleted the unnecessary objects.

Step 5

Array it 16 times.

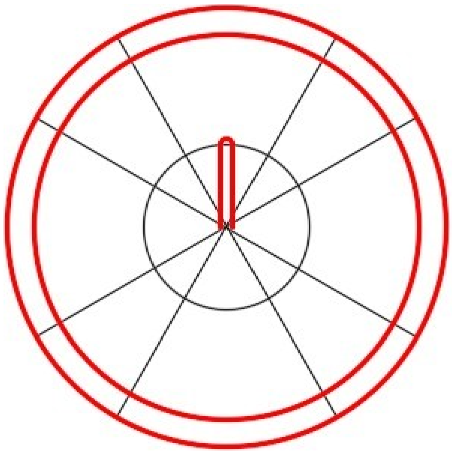


Figure Hint 1 – Step 1

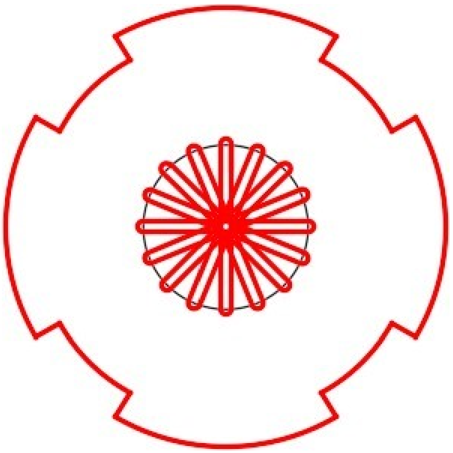


Figure Hint 1 – Step 2

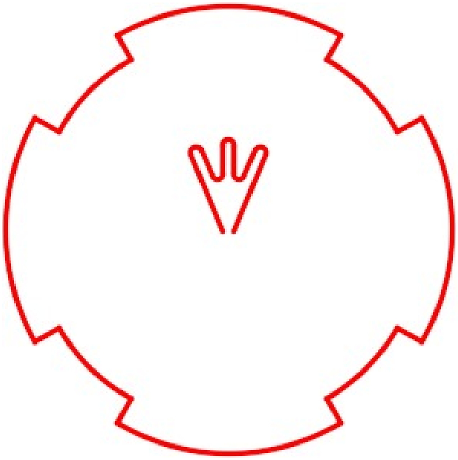


Figure Hint 1 – Step 3

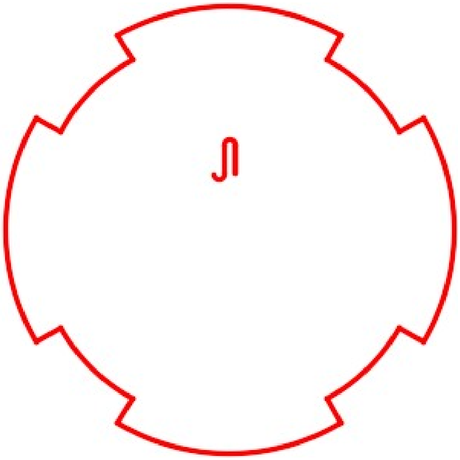


Figure Hint 1 – Step 4

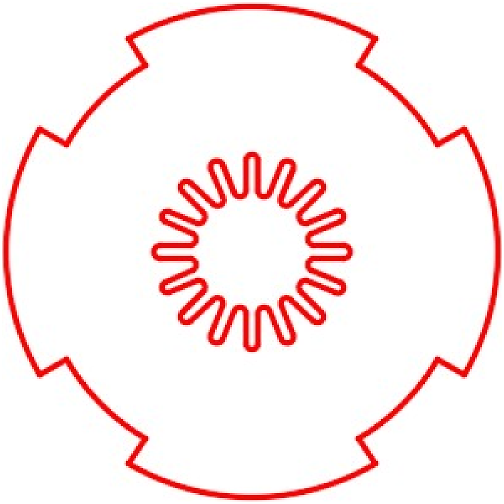


Figure Hint 1- Step 5

Hint 2

Draw construction lines from center to center of the objects to help you complete the drawing. (Figure Hint 2)

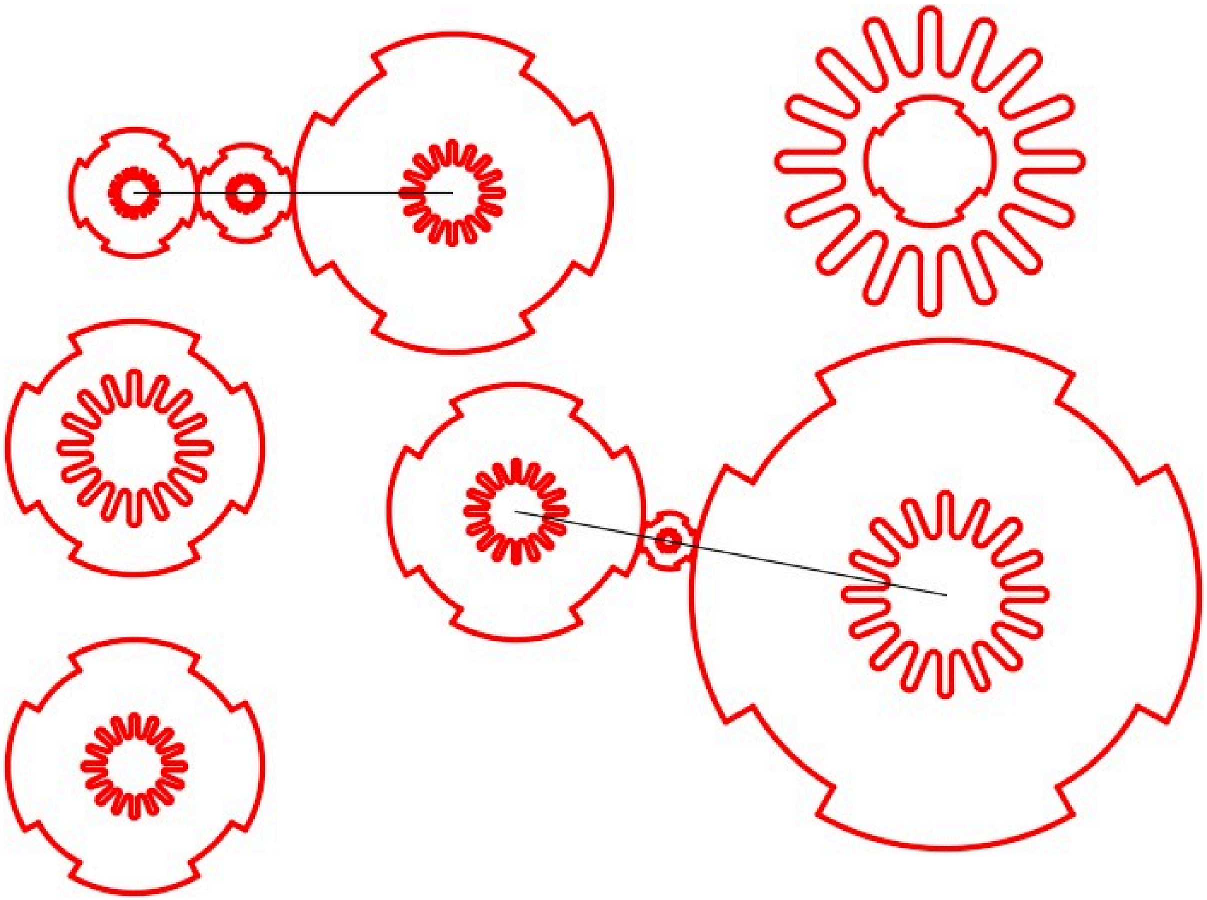


Figure Hint 2

Module 25: Text – Part 2

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe a mtext object and explain how to create, edit and explode mtext.
2. Apply the MTEXT, QTEXT, and SPELL commands.
3. Describe and compare shape and true type text fonts and their characteristics.
4. Describe special text characters and the use of control codes.

Multiline Text

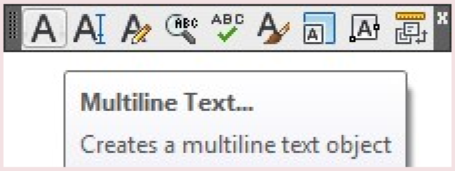
Unlike single line text, that was taught Module 19, *multiline text*, also called *mtext*, is a text object that can contain one or more lines of text. AutoCAD contains an editor that inserts and edits mtext similar to a word processor. Mtext uses the current text style for its properties, just like single line text, but it allows the font color and appearance of the text, with the same style, to be changed. This cannot be done with single line text.

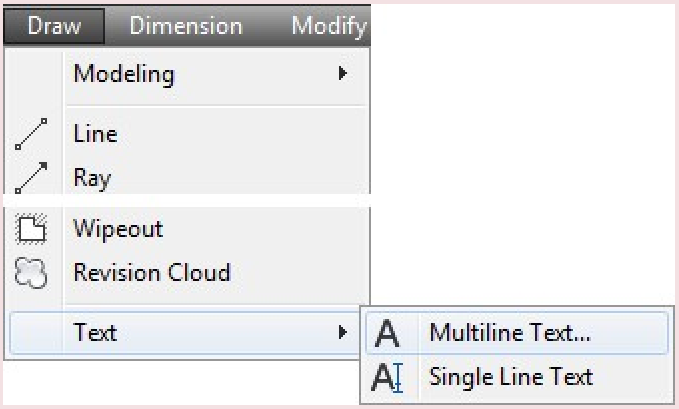
Mtext can be exploded using the EXPLODE command. When exploded, it is converted into single line text objects and loses all of its mtext properties except the color, height, and text style.

AutoCAD Command: MTEXT

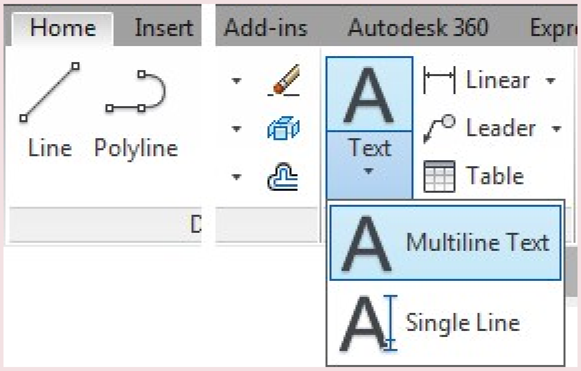
The MTEXT command is used to insert one or more lines of multiline text objects. Each mtext is a single drawing object.

Shortcut: **MT**

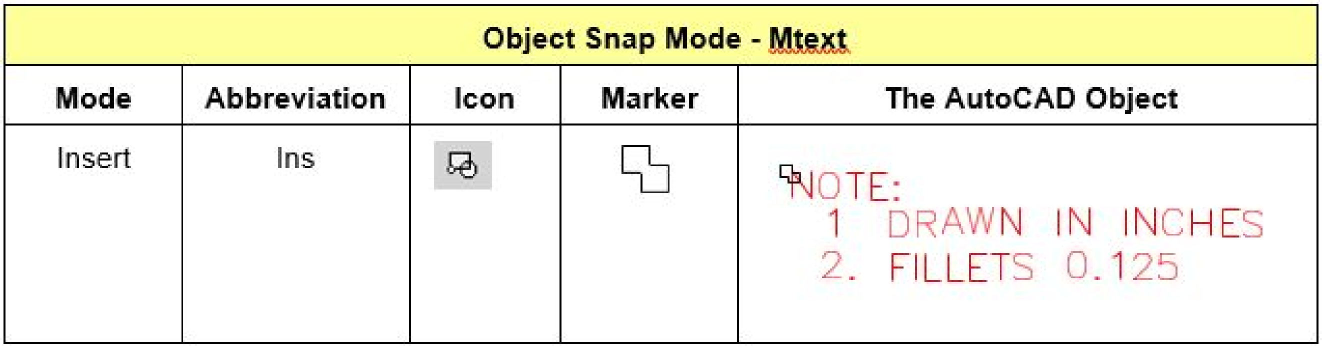



Draw
Dimension
Modify
Modeling
Line
Ray
Wipeout
Revision Cloud
Text
Multiline Text...
Single Line Text

Draw Pull-down


Home
Insert
Add-ins
Autodesk 360
Expr
Line
Polyline
Text
Linear
Leader
Table
Multiline Text
Single Line

Home Ribbon



Object Snap Mode - Mtext				
Mode	**Abbreviation**	**Icon**	**Marker**	**The AutoCAD Object**
Insert	Ins			NOTE: 1 DRAWN IN INCHES 2. FILLETS 0.125

Figure 25-1 Object Snap Mode for Mtext and Single Line Text

WORKALONG: Using the MTEXT Command

Step 1

Start a new drawing using the template: 2D Layout English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 25-1.

Step 3

Create a new text style naming it: Module 25 and assign the font: romans.shx. Set Standard as the current text style.

Step 4

Enable layout: Module Layout A.

Step 5

Set layer: Text as the current layer and while in paper space, enter the MTEXT command as shown below. (Figure Step 5)

Command: **MTEXT**

Current text style: “Standard” Text height: 0.2000

Specify first corner:

Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]:

(Select two corners to form a window as shown in the figure. The size and location are not that important but try to match the figure the best you can.)

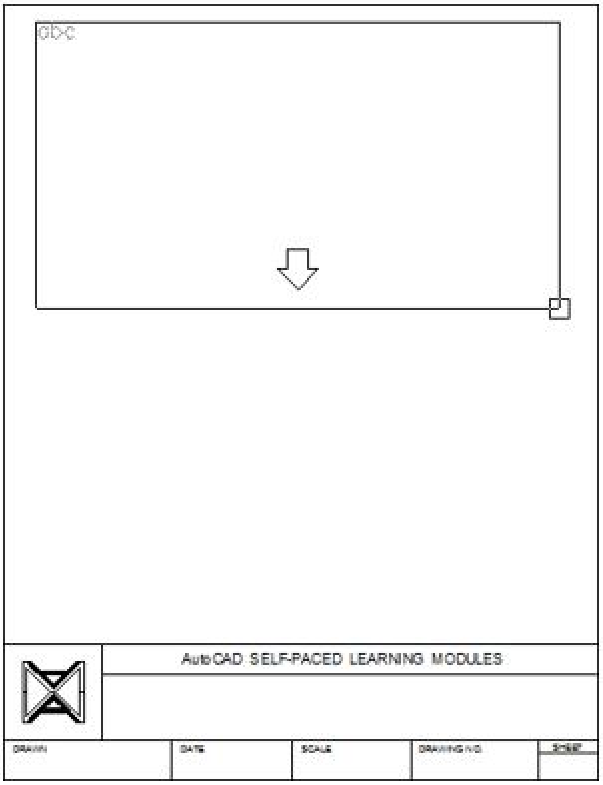

abc
AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO.
SHEET

Figure Step 5

AUTHOR’S COMMENTS: The window you created in the MTEXT command is called the *bounding box*.

Step 6

The MTEXT command will open the Text Formatting dialogue box. It works similar to a simple word processor. (Figure Step 6)

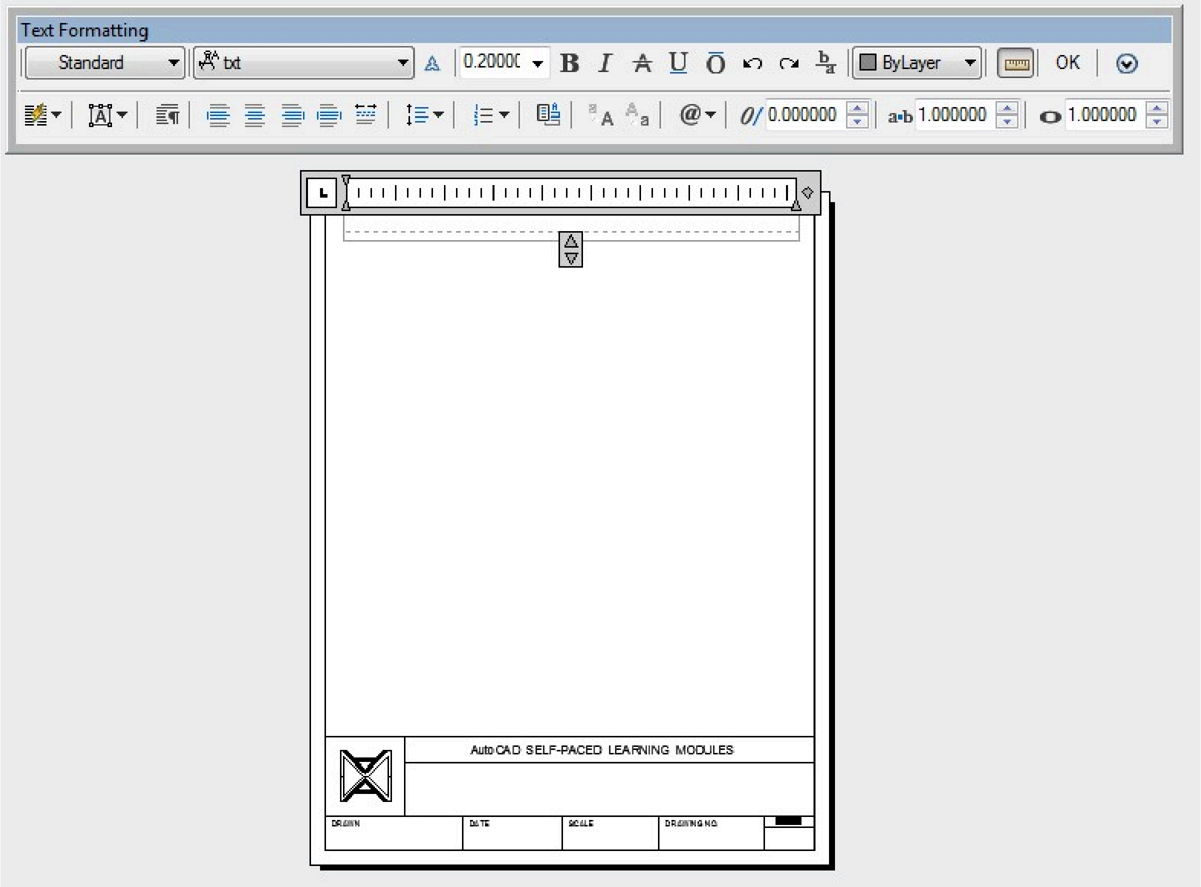

Text Formatting
Standard
txt
0.2000C
ByLayer
OK
0.000000
1.000000
1.000000
AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO.

Figure Step 6

Step 7

Set layer: Text as the current layer. Set the text style to: Module 25, the text height to 0.4 and the color to Red as shown in the figure. (Figure Step 7)

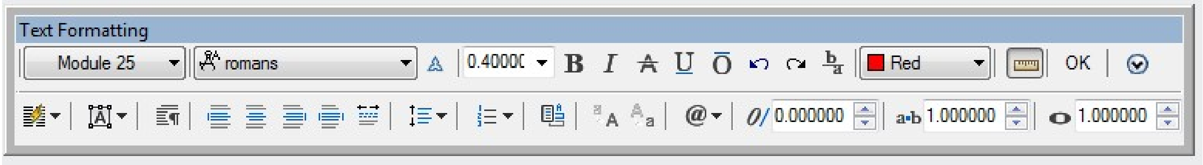
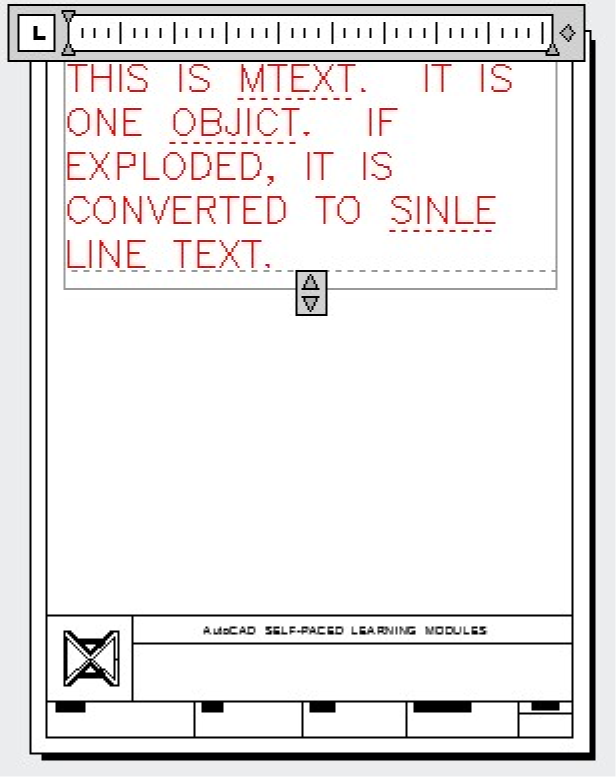

Text Formatting
Module 25
romans
0.4000C
Red
OK
0.000000
1.000000
1.000000

Figure Step 7

Step 8

In the bounding box, enter the text shown below. Use uppercase letters. Do not press the Enter key at the end of the line, just keep typing. Do not correct the spelling errors in the text, type it as shown. (Figure Step 8)

THIS IS MTEXT. IT IS ONE OBJICT. IF EXPLODED, IT IS CONVERTED TO SINLE LINE TEXT.



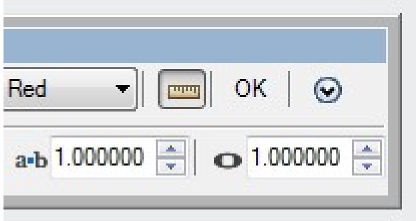
THIS IS MTEXT. IT IS ONE OBJICT. IF EXPLODED, IT IS CONVERTED TO SINLE LINE TEXT.

AutoCAD SELF-PACED LEARNING MODULES

Figure Step 8

Step 9

Click OK after you have entered all the text. (Figure step 9)



Red
OK
a•b 1.000000
1.000000

Figure Step 9

Step 10

Enter the MTEXT command as shown below. (Figure Step 10)

Command: **MTEXT**

Current text style: “Module 25” Text height: 0.4000

Specify first corner:

Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]:

(Select two corners to form a window. The size and location are not that important but try to match the figure the best you can.)

Command:

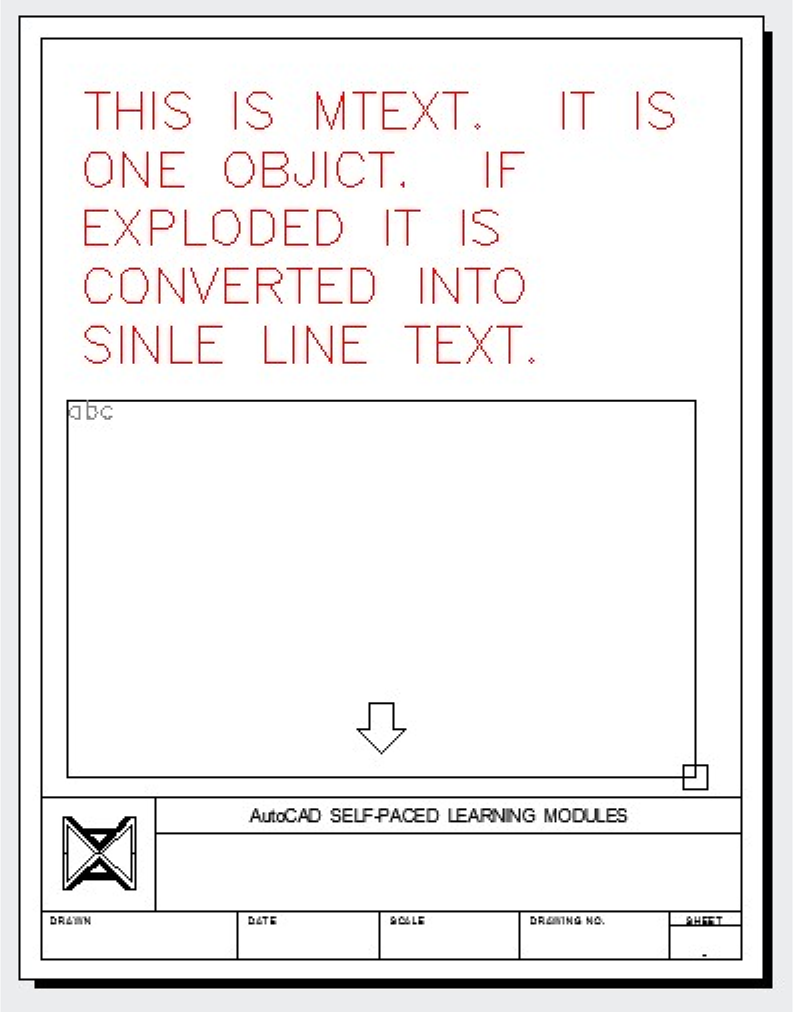
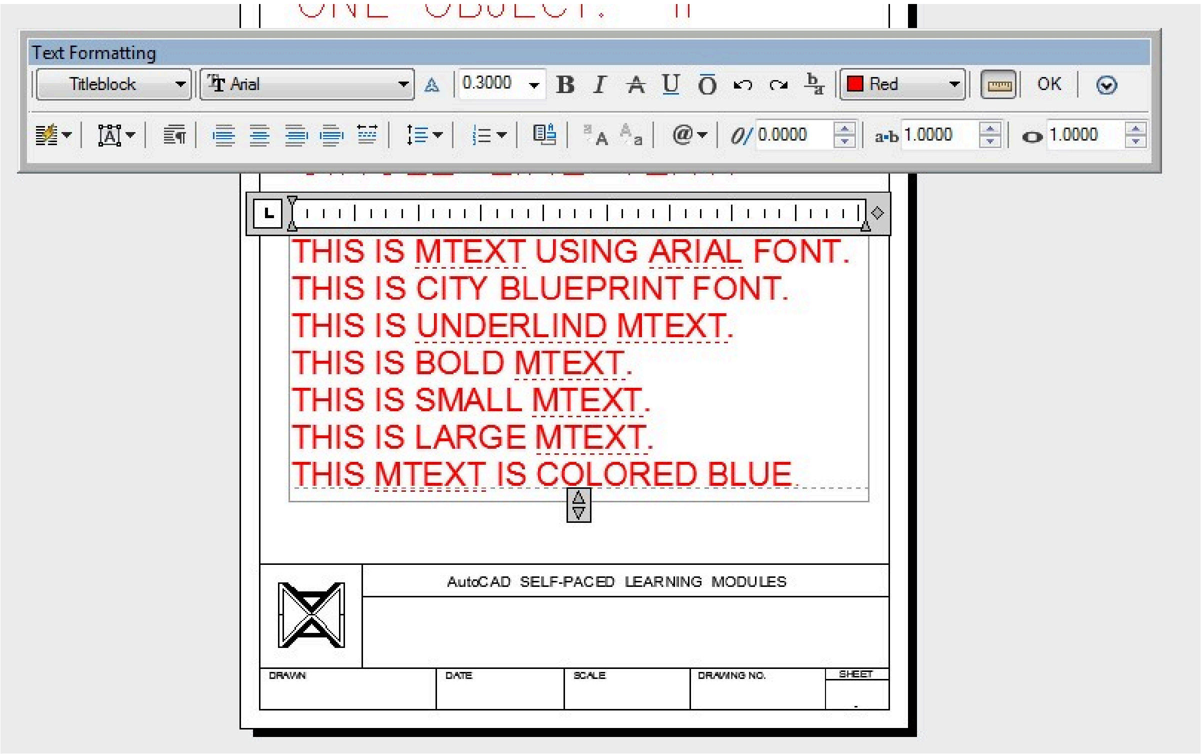

THIS IS MTEXT. IT IS ONE OBJICT. IF EXPLODED IT IS CONVERTED INTO SINLE LINE TEXT.
abc
AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO.
SHEET

Figure Step 10

Step 11

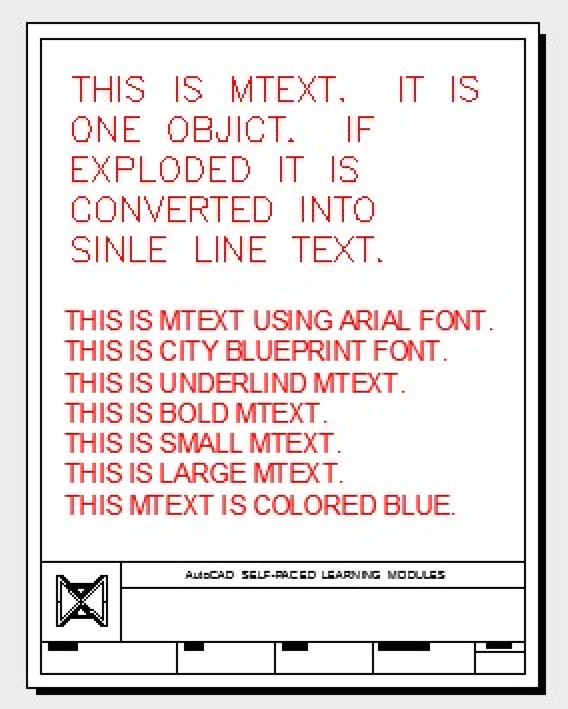
Set the text style to: Titleblock, font Arial and height to 0.3 as shown in the figure. In the bounding box, enter the text shown below. Enter it in upper case characters. At the end of each line press the Enter key. Do not correct the spelling errors in the text, type it as shown. (Figure Step 11A and 11B)



Text Formatting
Titleblock
Arial
0.3000
Red
OK
0.0000
1.0000
1.0000
AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO.
SHEET

THIS IS MTEXT USING ARIAL FONT.
THIS IS CITY BLUEPRINT FONT.
THIS IS UNDERLIND MTEXT.
THIS IS BOLD MTEXT.
THIS IS SMALL MTEXT.
THIS IS LARGE MTEXT.
THIS MTEXT IS COLORED BLUE.

Figure Step 11A



THIS IS MTEXT. IT IS ONE OBJICT. IF EXPLODED IT IS CONVERTED INTO SINLE LINE TEXT.

THIS IS MTEXT USING ARIAL FONT.
THIS IS CITY BLUEPRINT FONT.
THIS IS UNDERLIND MTEXT.
THIS IS BOLD MTEXT.
THIS IS SMALL MTEXT.
THIS IS LARGE MTEXT.
THIS MTEXT IS COLORED BLUE.

Figure Step 11B

THIS IS MTEXT USING ARIAL FONT.

THIS IS CITY BLUEPRINT FONT.

THIS IS UNDERLIND MTEXT.

THIS IS BOLD MTEXT. THIS IS SMALL MTEXT.

THIS IS LARGE MTEXT.

THIS MTEXT IS COLORED BLUE.

Step 12

Block the second line of the text. While it is blocked, select the font: City Blueprint. (Figure Step 12)

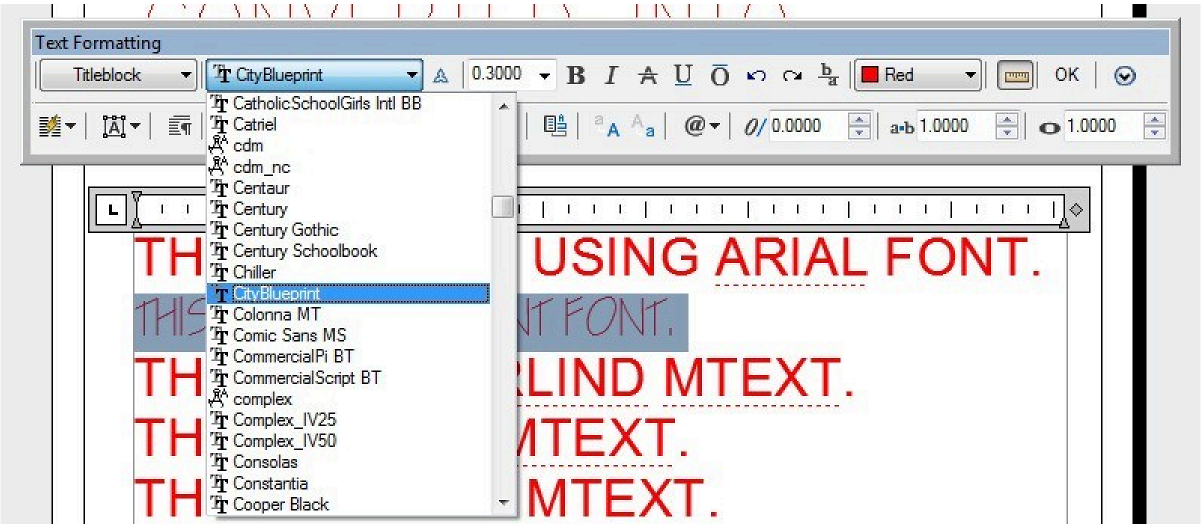
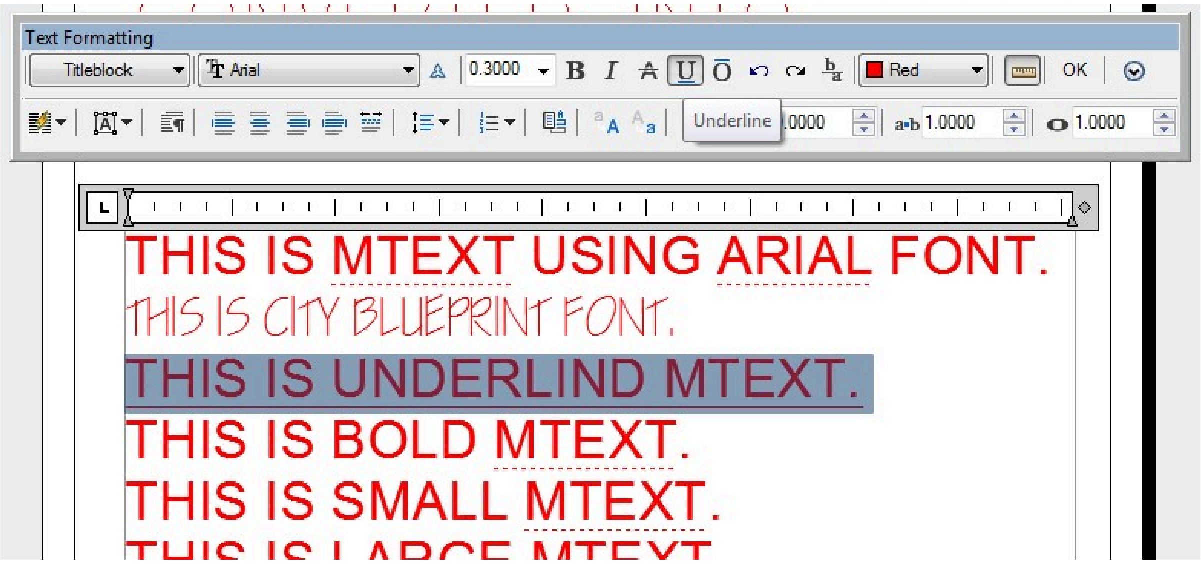

Text Formatting
Titleblock
CityBlueprint
0.3000
B
I
U
Red
OK
0.0000
1.0000
1.0000
CatholicSchoolGirls Intl BB
Catriel
cdm
cdm_nc
Centaur
Century
Century Gothic
Century Schoolbook
Chiller
CityBlueprint
Colonna MT
Comic Sans MS
CommercialPi BT
CommercialScript BT
complex
Complex_IV25
Complex_IV50
Consolas
Constantia
Cooper Black
TH USING ARIAL FONT.
THIS T FONT.
TH LIND MTEXT.
TH MTEXT.
TH MTEXT.

Figure Step 12

AUTHOR’S COMMENTS: The easiest way to block text is to hold down the left mouse button as you pass the cursor over the text you want to block.

Step 13

Block the third line. While it is blocked, click the Underline icon. (Figure Step 13)

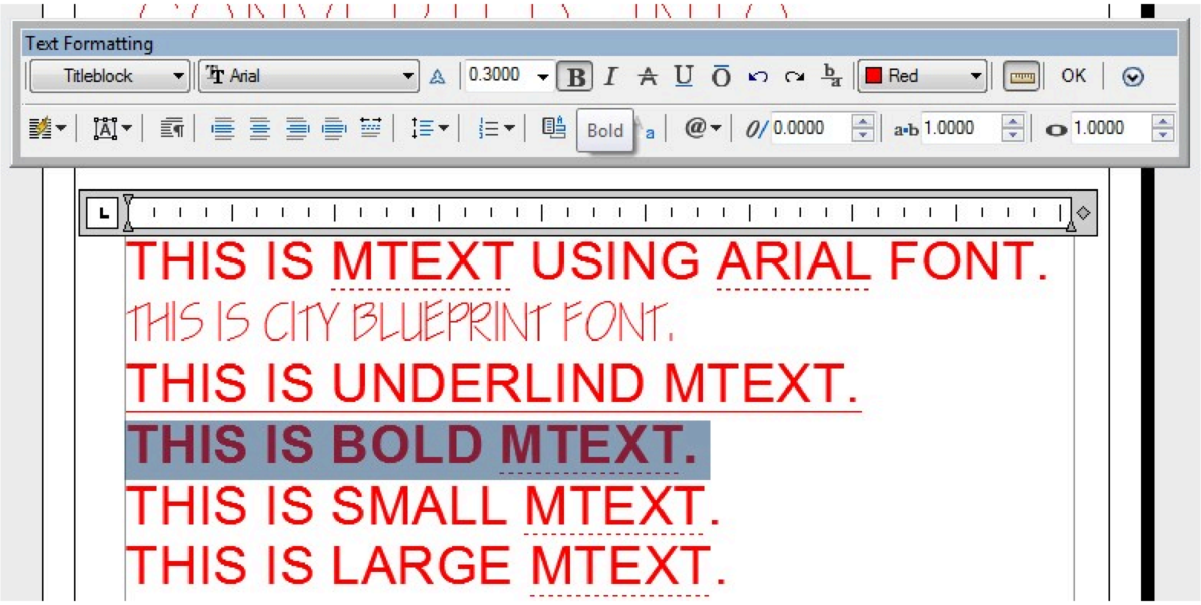

Text Formatting
Titleblock
Arial
0.3000
Red
OK
Underline
1.0000
1.0000

THIS IS MTEXT USING ARIAL FONT.
THIS IS CITY BLUEPRINT FONT.
THIS IS UNDERLIND MTEXT.
THIS IS BOLD MTEXT.
THIS IS SMALL MTEXT.

Figure Step 13

Step 14

Block the forth line. While it is blocked, click the Bold icon. (Figure Step 14)

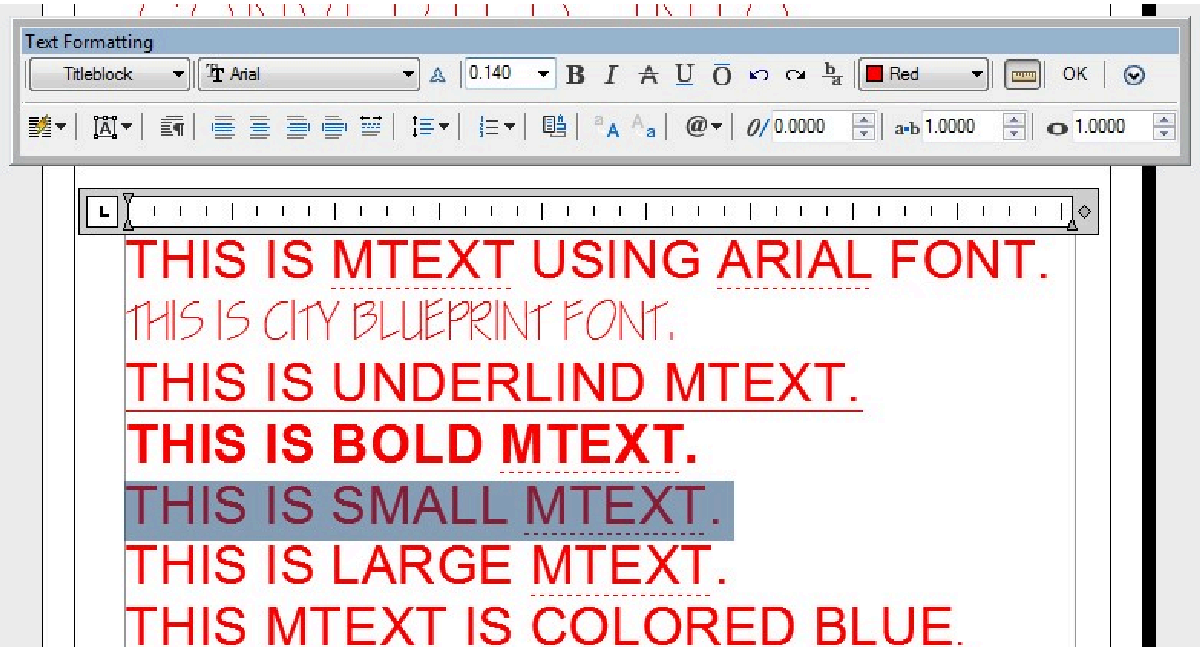

Text Formatting
Titleblock
Arial
0.3000
Red
OK
Bold
0.0000
1.0000
1.0000

THIS IS MTEXT USING ARIAL FONT.
THIS IS CITY BLUEPRINT FONT.
THIS IS UNDERLIND MTEXT.
THIS IS BOLD MTEXT.
THIS IS SMALL MTEXT.
THIS IS LARGE MTEXT.

Figure Step 14

Step 15

Block the fifth line. While it is blocked, set the text height to 0.14. (Figure Step 15)

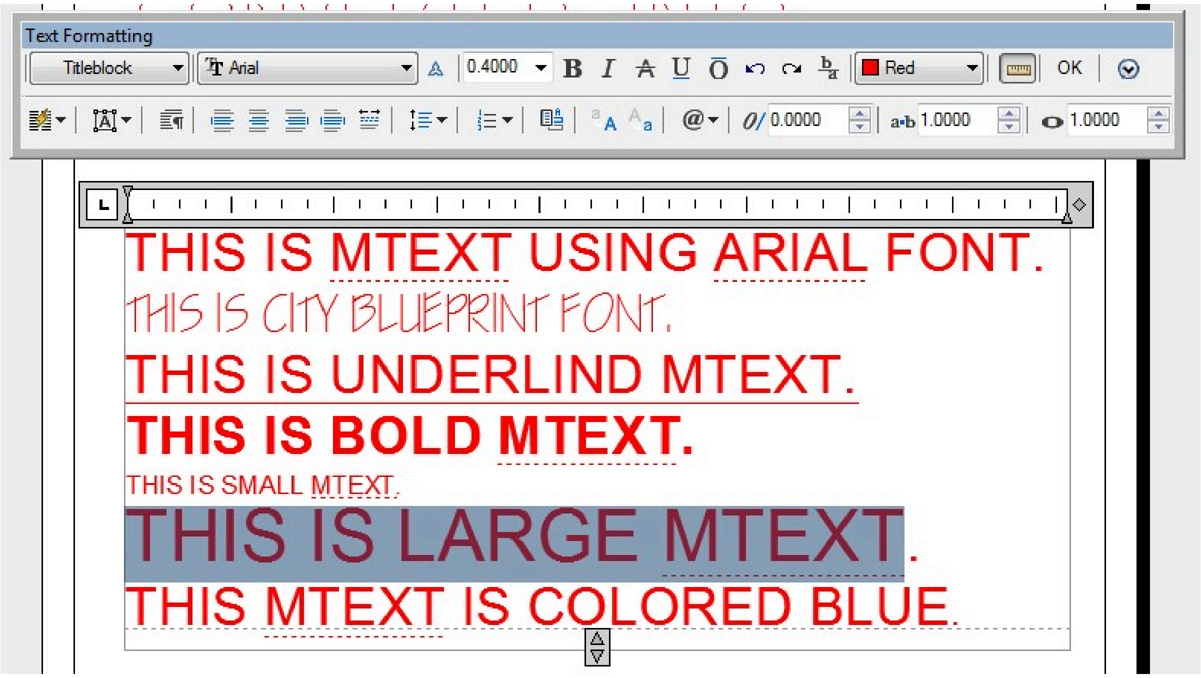

Text Formatting
Titleblock
Arial
0.140
Red
OK
0.0000
1.0000
1.0000

THIS IS MTEXT USING ARIAL FONT.
THIS IS CITY BLUEPRINT FONT.
THIS IS UNDERLIND MTEXT.
THIS IS BOLD MTEXT.
THIS IS SMALL MTEXT.
THIS IS LARGE MTEXT.
THIS MTEXT IS COLORED BLUE.

Figure Step 15

Step 16

Block the sixth line. While it is blocked, set the text height to 0.40. (Figure Step 16)

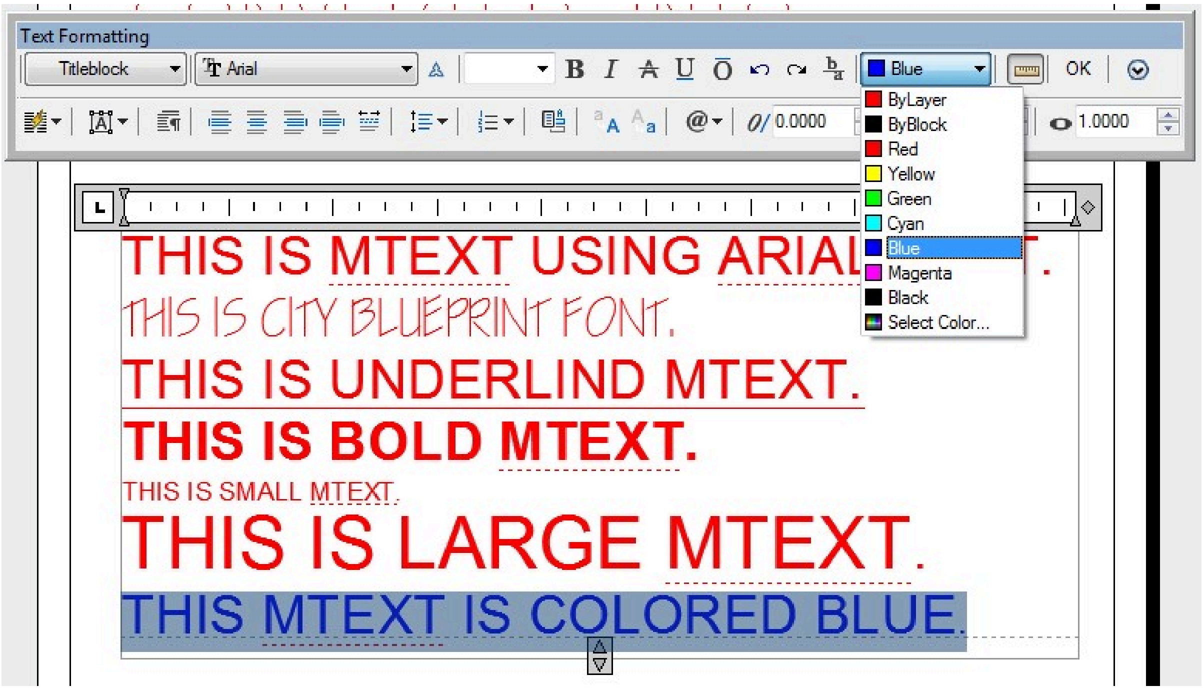

Text Formatting
Titleblock
Arial
0.4000
Red
OK
0.0000
1.0000
1.0000

THIS IS MTEXT USING ARIAL FONT.
THIS IS CITY BLUEPRINT FONT.
THIS IS UNDERLIND MTEXT.
THIS IS BOLD MTEXT.
THIS IS SMALL MTEXT.
THIS IS LARGE MTEXT.
THIS MTEXT IS COLORED BLUE.

Figure Step 16

Step 17

Block the seventh line. While it is blocked, set the color to Blue. (Figure Step 17)


Text Formatting
Titleblock
Arial
Blue
OK
ByLayer
ByBlock
Red
Yellow
Green
Cyan
Blue
Magenta
Black
Select Color...
0.0000
1.0000

THIS IS MTEXT USING ARIAL

THIS IS CITY BLUEPRINT FONT.

THIS IS UNDERLIND MTEXT.

THIS IS BOLD MTEXT.

THIS IS SMALL MTEXT.

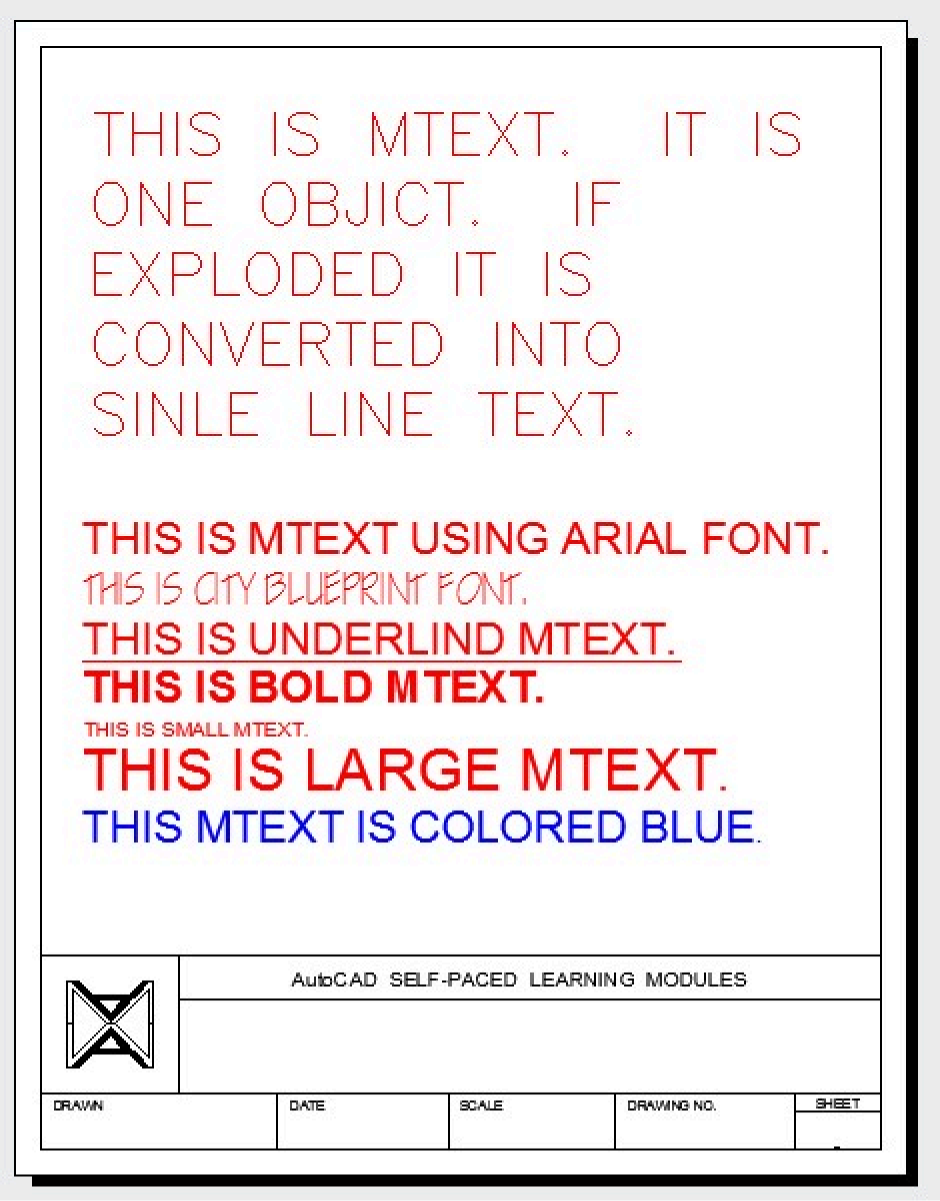
THIS IS LARGE MTEXT.

THIS MTEXT IS COLORED BLUE.

Figure Step 17

Step 18

The completed drawing should appear as shown in the figure. (Figure Step 18)



THIS IS MTEXT. IT IS ONE OBJICT. IF EXPLODED IT IS CONVERTED INTO SINLE LINE TEXT.

THIS IS MTEXT USING ARIAL FONT.

THIS IS CITY BLUEPRINT FONT.

THIS IS UNDERLIND MTEXT.

THIS IS BOLD MTEXT.

THIS IS SMALL MTEXT.

THIS IS LARGE MTEXT.

THIS MTEXT IS COLORED BLUE.

Figure Step 18

Step 19

Save and close the drawing.

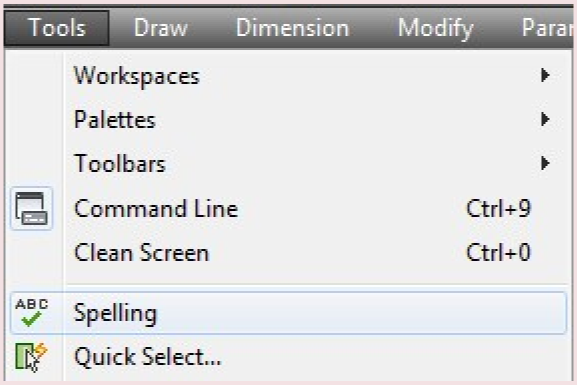
Editing Mtext

The DDEDIT command, taught in Module 19, is used to edit mtext. You can enter the DDEDIT command on the command line or enter it from one of the menus and then select the mtext to edit. A much faster method to edit mtext is to double click the mtext. When mtext is double- clicked, it will open the Text Formatting dialogue box and display the selected mtext.

AutoCAD Command: SPELL

The SPELL command is used to spell check selected text or all text in the current drawing.

Shortcut: **SP**


Tools
Draw
Dimension
Modify
Parar
Workspaces
Palettes
Toolbars
Command Line Ctrl+9
Clean Screen Ctrl+0
Spelling
Quick Select...

Tool Pull-down

AutoCAD Command: QTEXT

The QTEXT command is used to change the display of all text objects on the drawing to rectangles that represent the location of the text.

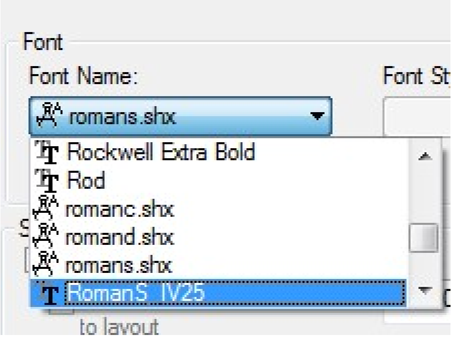
Shortcut: none

Font Types

There are two basic font types available in AutoCAD. They are shape fonts and true type fonts.

Shape Font

A *shape* font is a compiled font created using vector lines and arcs. The text characters in a shape font are made from lines and arcs, the same as drawings. They are created in a text file, saved as a shape file, and compiled before they can be used in a drawing. A shape file is saved with the extension .shp and after it is compiled, is given a .shx extension. See Figure 25-2.



Font Name: romans.shx

Figure 25-2
True Type and Shape Fonts

True Type Fonts

A *true type* font is a raster font. The text characters in a raster font are created from dots or pixels. A true type font does not have a file extension assigned to the name.

A Shape Font Verses a True Type Font

The fonts available in the Text Style dialogue box are either shape fonts or true type fonts. Shape fonts have the extension .shx while true type fonts do not have an extension. That is how you can tell which font type is being used. See Figure 25-2.

The main differences between shape fonts and true type fonts are as follows:

1. Shape fonts take much less space in an AutoCAD drawing's data base then true type
2. Drawings regenerate faster with shape fonts then they do with true type
3. True type fonts can be assigned appearances like bolding and italicized while shape fonts cannot.

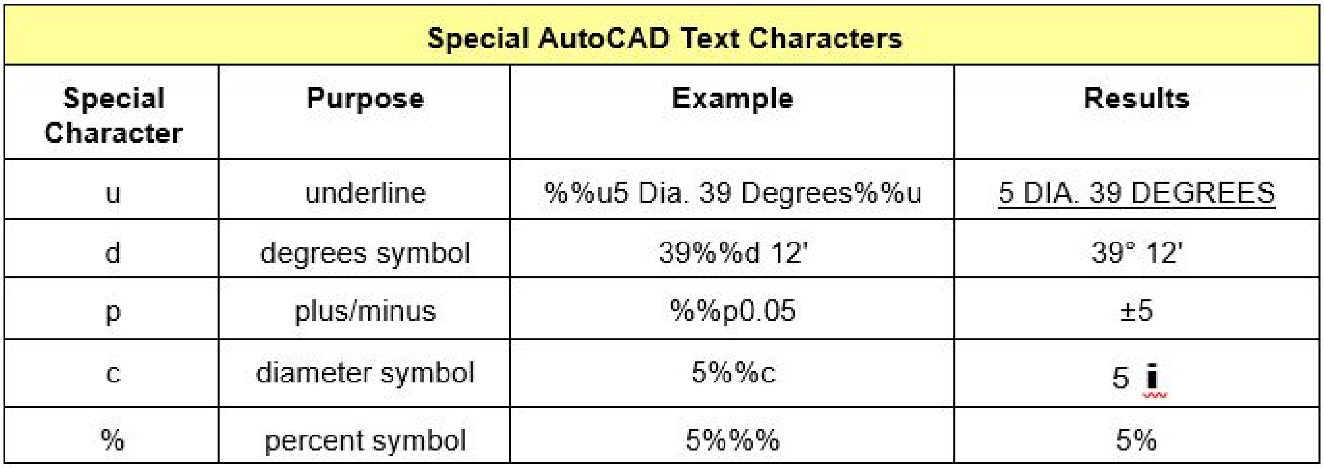
AUTHOR'S COMMENTS: It is best to use a shape font, whenever possible, in an AutoCAD drawing.

Special Text Characters

Not all characters or symbols that are required on drawings are available on a keyboard. These characters are called *special characters*, see Figure 25-3. To enter them as text, control codes must be used.

Control Codes

The *control code* used by AutoCAD is a pair of percent signs %%. Added before a special character, AutoCAD's text commands will interpret it differently than normal text characters. Shape and true type fonts as well as single line and multiline text work differently with special characters. Therefore, you will have to experiment with the text type and font being used.



Special AutoCAD Text Characters			
Special Character	**Purpose**	**Example**	**Results**
u	underline	%%u5 Dia. 39 Degrees%%u	5 DIA. 39 DEGREES
d	degrees symbol	39%%d 12'	39° 12'
p	plus/minus	%%p0.05	±5
c	diameter symbol	5%%c	5 i
%	percent symbol	5%%%	5%

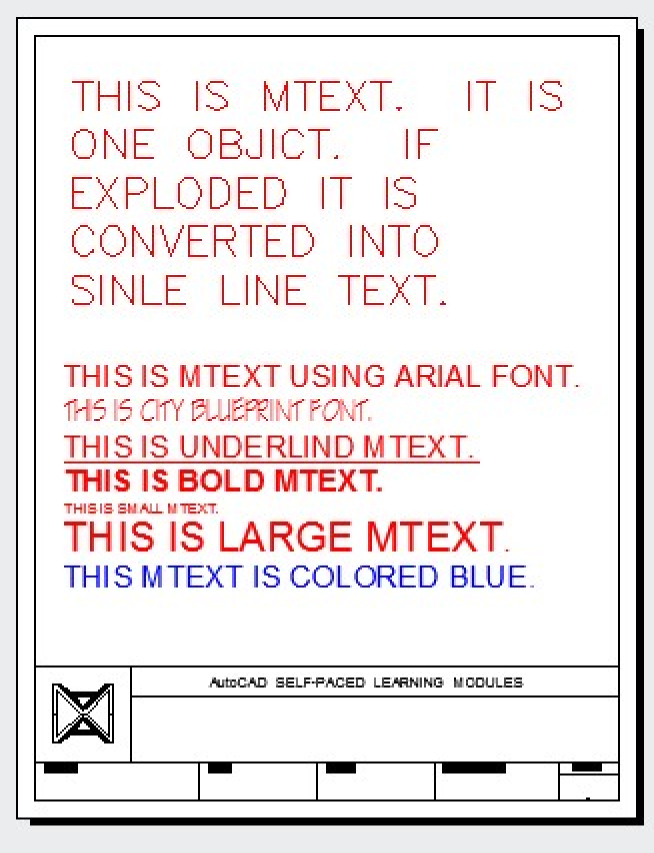
Figure 25-3 Special AutoCAD Text Characters

WORKALONG: Using the SPELL and QTEXT Commands plus Special Characters

Step 1

Open the drawing:

AutoCAD 2D Workalong 25-1. Enable layout: Module Layout A. (Figure Step 1)



THIS IS MTEXT. IT IS ONE OBJICT. IF EXPLODED IT IS CONVERTED INTO SINLE LINE TEXT.

THIS IS MTEXT USING ARIAL FONT.

THIS IS CITY BLUEPRINT FONT.

THIS IS UNDERLIND MTEXT.

THIS IS BOLD MTEXT.

THIS IS SMALL MTEXT.

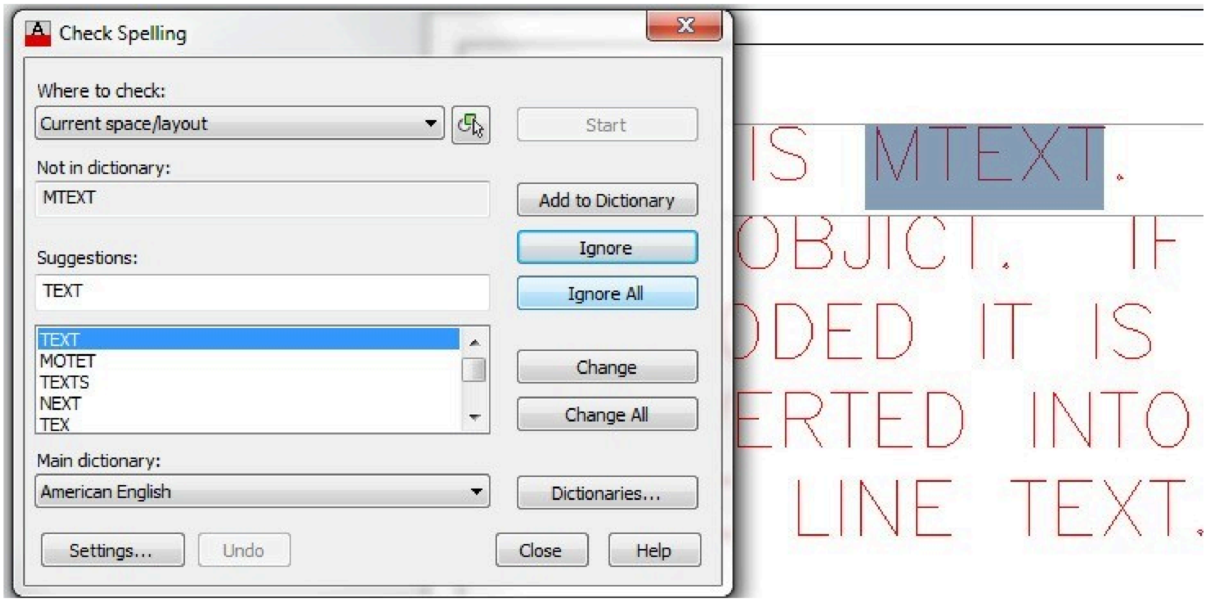
THIS IS LARGE MTEXT.

THIS MTEXT IS COLORED BLUE.

Figure Step 1

Step 2

Enter the SPELL command. It will open the Check Spelling dialogue box. Pull down the Where to check: list and select Current space/layout. Press the Start button. (Figure Step 2)


Check Spelling
Start
Add to Dictionary
Ignore
Ignore All
Change
Change All
Dictionaries...
Settings...
Undo
Close
Help
IS MTEXT.
OBJICT. IF
DDED IT IS
ERTED INTO
LINE TEXT.

Where to check: Current space/layout

Not in dictionary: MTEXT

Suggestions: TEXT

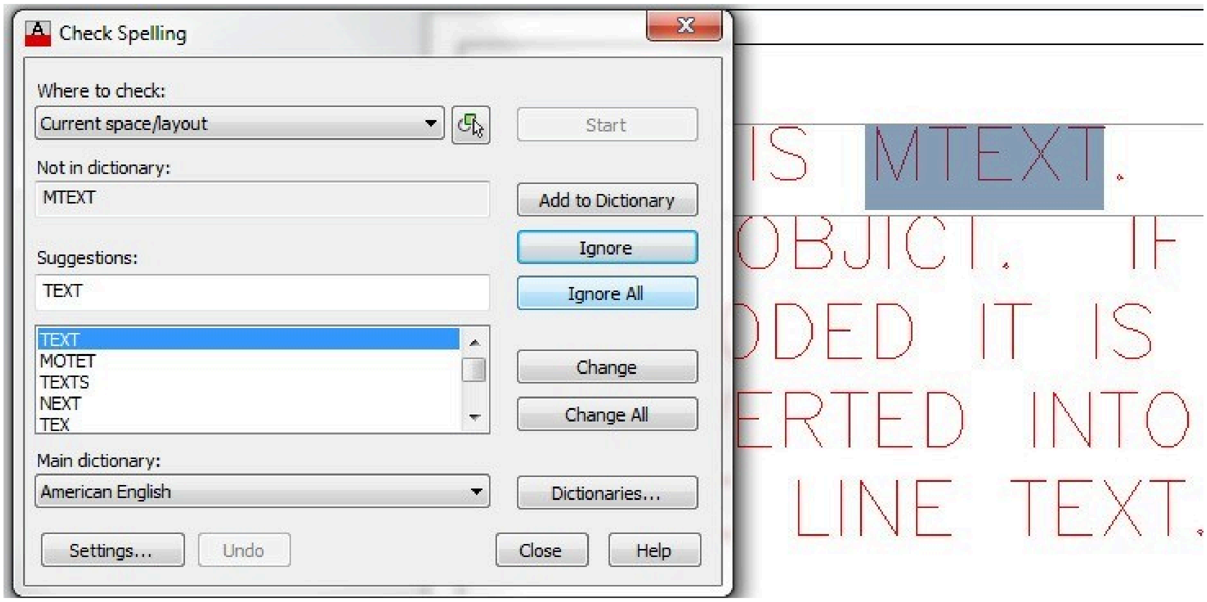
TEXT
MOTET
TEXTS
NEXT
TEX

Main dictionary: American English

Figure Step 3

Step 3

One by one, the dialogue box will display the words it finds that are not in the dictionary and may suggest a replacement. You have the choice to Ignore, Change or to Add the word to the dictionary. In this case, click the Ignore All box. It will ignore all mtext words in the layout. (Figure Step 3)



Check Spelling

Where to check: Current space/layout

Not in dictionary: MTEXT

Suggestions: TEXT

TEXT
MOTET
TEXTS
NEXT
TEX

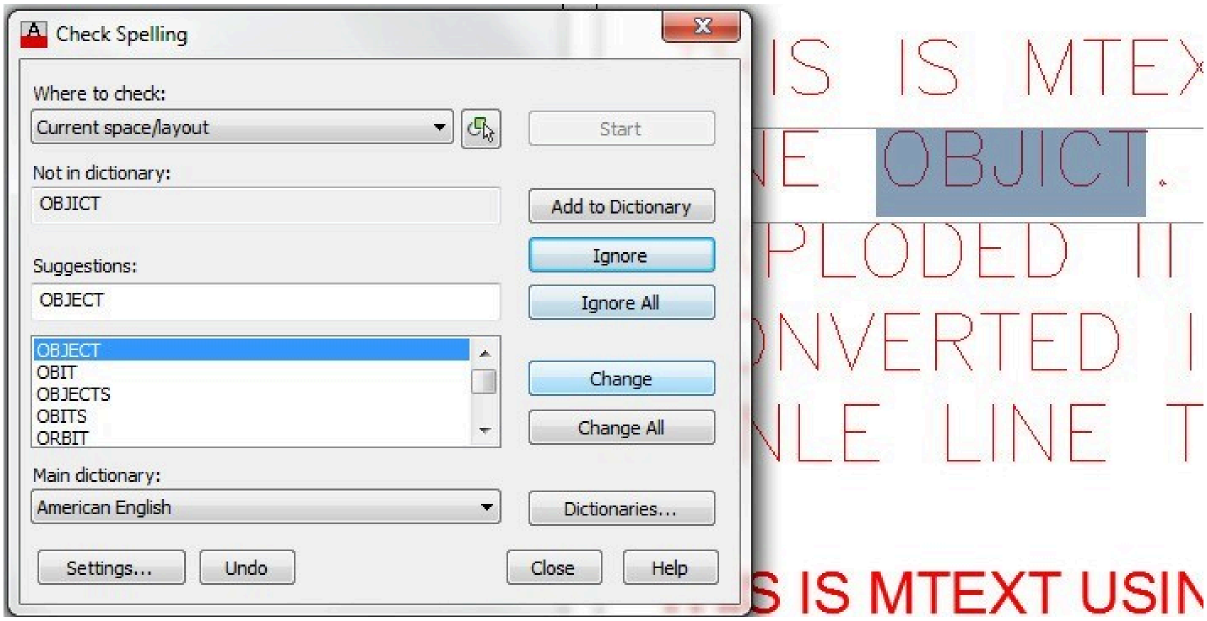
Main dictionary: American English

Start
Add to Dictionary
Ignore
Ignore All
Change
Change All
Dictionaries...
Settings...
Undo
Close
Help
IS MTEXT.
OBJECT. IF
DED IT IS
ERTED INTO
LINE TEXT.

Figure Step 3

Step 4

This spell checker will find the next misspelled word. In this example, it is OBJICT. Click the Change box to correct the spelling to OBJECT. This spell checker will find the next misspelled word. In this example, it is SINLE. Carry on using the spell checker until it completes checking all words. When the spell checker finds ARIAL, click the Add to Dictionary box to add the word. (Figure step 4A. 4B, and 4C).



Check Spelling

Where to check: Current space/layout

Not in dictionary: OBJICT

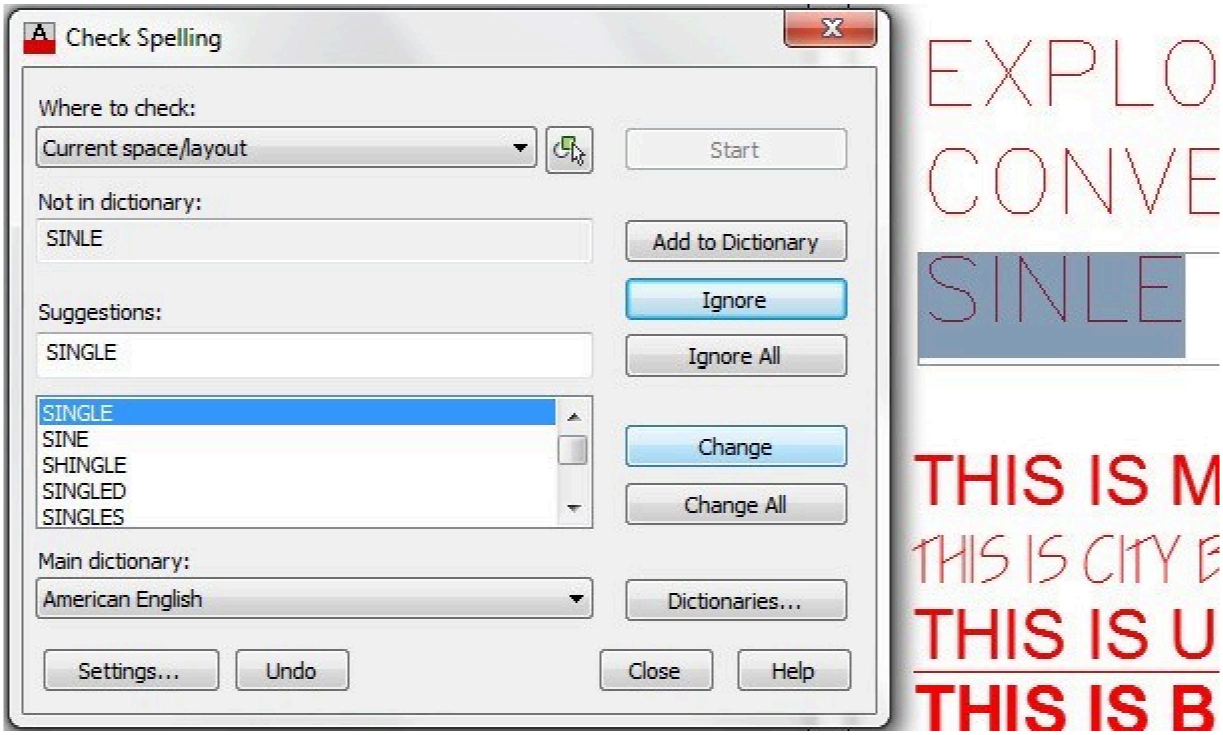
Suggestions: OBJECT

OBJECT
OBIT
OBJECTS
OBITS
ORBIT

Main dictionary: American English

Start
Add to Dictionary
Ignore
Ignore All
Change
Change All
Dictionaries...
Settings...
Undo
Close
Help
IS IS MTEX
NE OBJICT.
PLODED IT
NVERTED I
NLE LINE T
S IS MTEXT USIN

Figure Step 4A



Check Spelling

Where to check: Current space/layout

Not in dictionary: SINLE

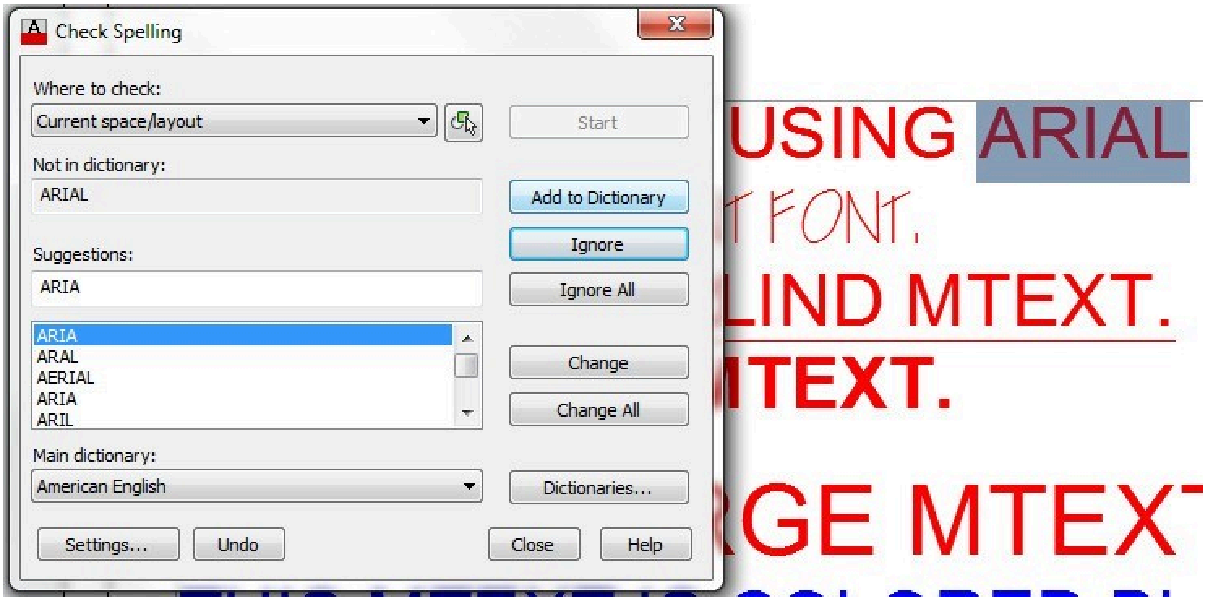
Suggestions: SINGLE

SINGLE
SINE
SHINGLE
SINGLED
SINGLES

Main dictionary: American English

Start
Add to Dictionary
Ignore
Ignore All
Change
Change All
Dictionaries...
Settings...
Undo
Close
Help
EXPLO
CONVE
SINLE
THIS IS M
THIS IS CITY B
THIS IS U
THIS IS B

Figure Step 4B


Check Spelling
Start
Add to Dictionary
Ignore
Ignore All
Change
Change All
Dictionaries...
Settings...
Undo
Close
Help
USING ARIAL
T FONT.
LIND MTEXT.
MTEXT.
RGE MTEX

Where to check: Current space/layout

Not in dictionary: ARIAL

Suggestions: ARIA

ARIA
ARAL
AERIAL
ARIA
ARIL

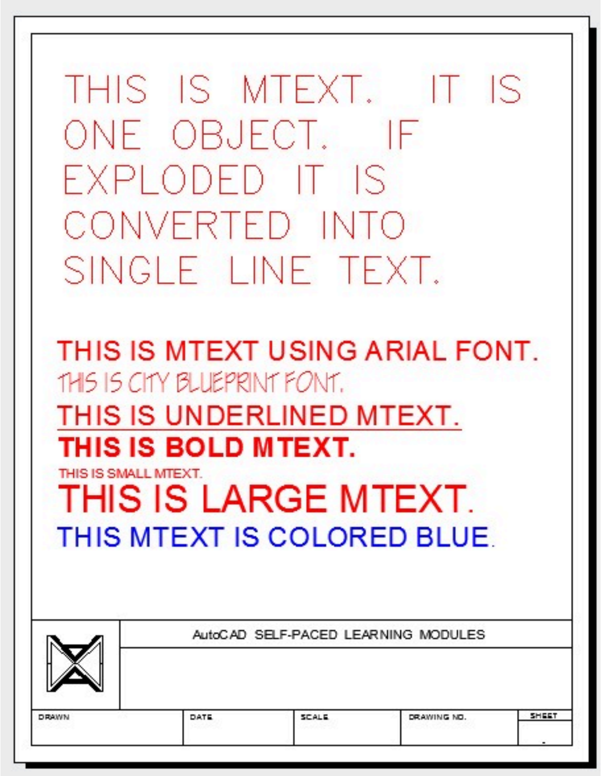
Main dictionary: American English

Figure Step 4C

AUTHOR’S COMMENTS: The words in your spell checker may display in a different order than shown here.

Step 5

Your drawing should now appear as shown in the figure with the spelling errors corrected. (Figure Step 5)



THIS IS MTEXT. IT IS ONE OBJECT. IF EXPLODED IT IS CONVERTED INTO SINGLE LINE TEXT.

THIS IS MTEXT USING ARIAL FONT.

THIS IS CITY BLUEPRINT FONT.

THIS IS UNDERLINED MTEXT.

THIS IS BOLD MTEXT.

THIS IS SMALL MTEXT.

THIS IS LARGE MTEXT.

THIS MTEXT IS COLORED BLUE.

AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO.
SHEET

Figure Step 5

Step 6

Enter the QTEXT command as shown below. (Figure Step 6)

Command: **QTEXT**

Enter mode [ON/OFF] <OFF>: **ON**

Command: **REGEN** Regenerating model

Command:

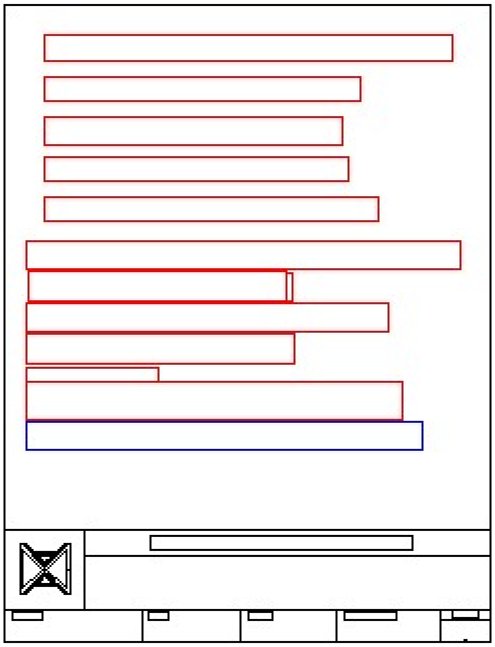


Figure Step 6

AUTHOR’S COMMENTS: After you enter the QTEXT command, you must enter the REGEN command to force the display of the text boxes.

AUTHOR’S COMMENTS: The QTEXT command temporarily replaces the existing text on the layout with rectangular boxes. Since a large amount of text on a drawing can slow down some operations in the drawing process, replacing the text with rectangles will speed it up. Displaying rectangular boxes, at the text’s location, allows the user to see the location of existing text while continuing working on the drawing. When required, simply enter the QTEXT command and turn QTEXT off.

Step 7

Enter the QTEXT and REGEN commands as shown below. (Figure Step 7)

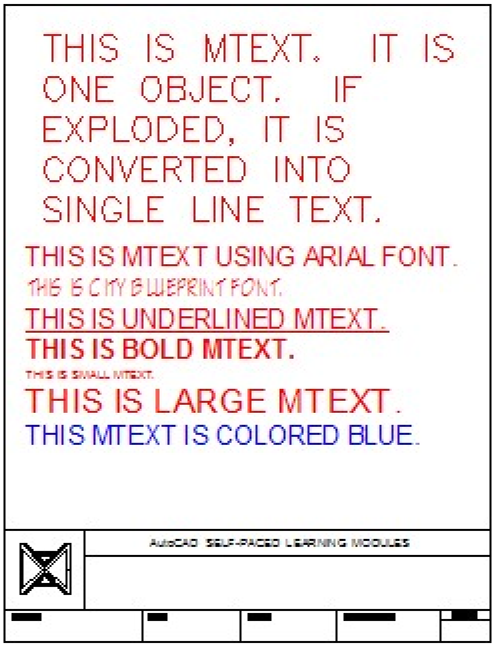
Command: **QTEXT**

Enter mode [ON/OFF] <ON>: **OFF**

Command: **REGEN**

Regenerating model.

Command



THIS IS MTEXT. IT IS ONE OBJECT. IF EXPLODED, IT IS CONVERTED INTO SINGLE LINE TEXT.

THIS IS MTEXT USING ARIAL FONT.

THIS IS CITY BLUEPRINT FONT.

THIS IS UNDERLINED MTEXT.

THIS IS BOLD MTEXT.

THIS IS SMALL MTEXT.

THIS IS LARGE MTEXT.

THIS MTEXT IS COLORED BLUE.

Figure Step 7

Step 8

Using The EXPLODE command, as shown below, explode the mtext object at the top of the drawing only. When exploded, mtext is converted into single line text. Do not explode the lower mtext object. (Figure Step 8)

```
Command: EXPLODE

Select objects: 1 found

Select objects:

Command:
```

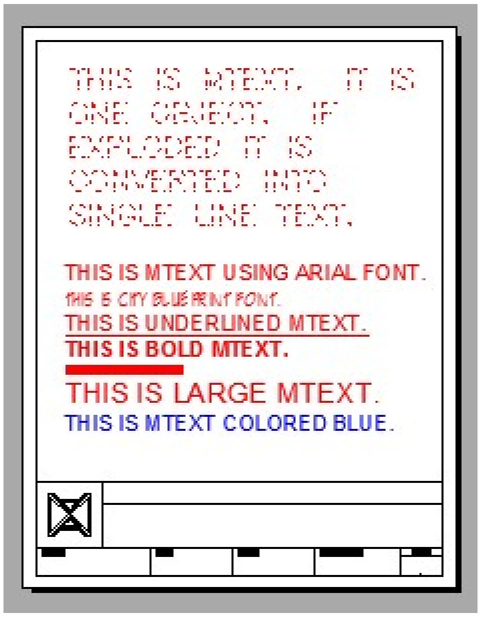
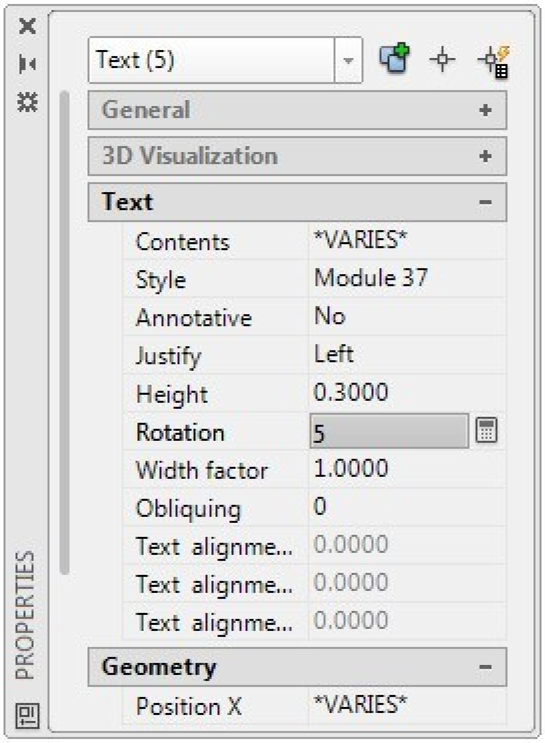

THIS IS MTEXT. IT IS ONE OBJECT. IF EXPLODED IT IS CONVERTED INTO SINGLE LINE TEXT.
THIS IS MTEXT USING ARIAL FONT.
THIS IS CITY BLUEPRINT FONT.
THIS IS UNDERLINED MTEXT.
THIS IS BOLD MTEXT.
THIS IS LARGE MTEXT.
THIS IS MTEXT COLORED BLUE.

Figure Step 8

Step 9

Open the Properties window and then select the five lines of text. While the text lines are selected, set the rotation angle to 5 degrees and the height of 0.3. Using what you learned in Module 19, edit the first two lines of the text to match the figure. (Figure Step 9A and 9B)


PROPERTIES
General
3D Visualization

Text (5)

Text	
Contents	*VARIES*
Style	Module 37
Annotative	No
Justify	Left
Height	0.3000
Rotation	5
Width factor	1.0000
Obliquing	0
Text alignme...	0.0000
Text alignme...	0.0000
Text alignme...	0.0000
Geometry	
Position X	*VARIES*

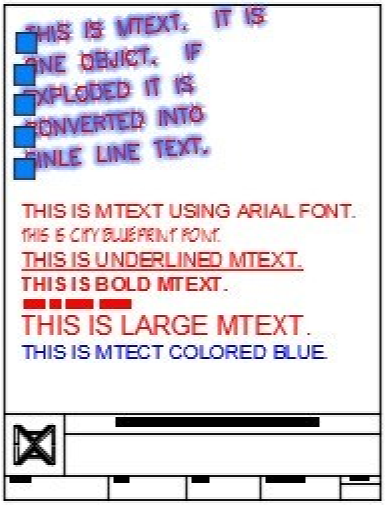
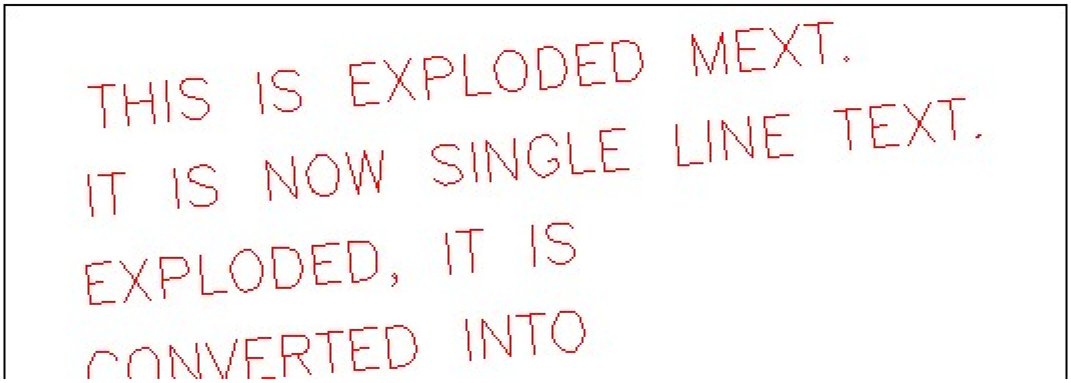

THIS IS MTEXT. IT IS ONE OBJECT. IF EXPLODED IT IS CONVERTED INTO SINLE LINE TEXT.
THIS IS MTEXT USING ARIAL FONT.
THIS IS CITY BLUEPRINT FONT.
THIS IS UNDERLINED MTEXT.
THIS IS BOLD MTEXT.
THIS IS LARGE MTEXT.
THIS IS MTECT COLORED BLUE.

Figure Step 9A

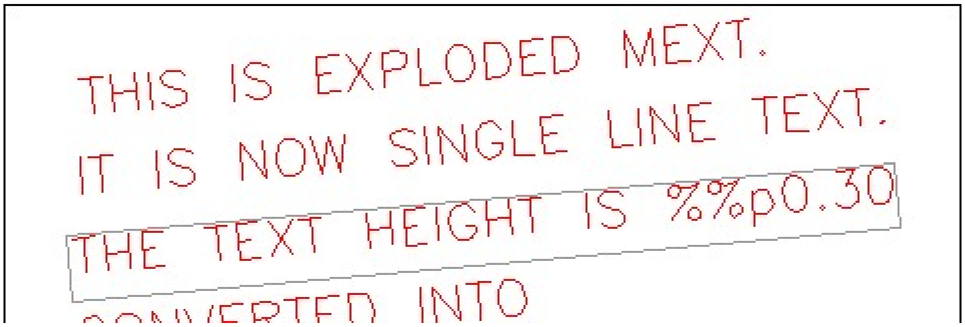


THIS IS EXPLODED MEXT.
IT IS NOW SINGLE LINE TEXT.
EXPLODED, IT IS
CONVERTED INTO

Figure Step 9B

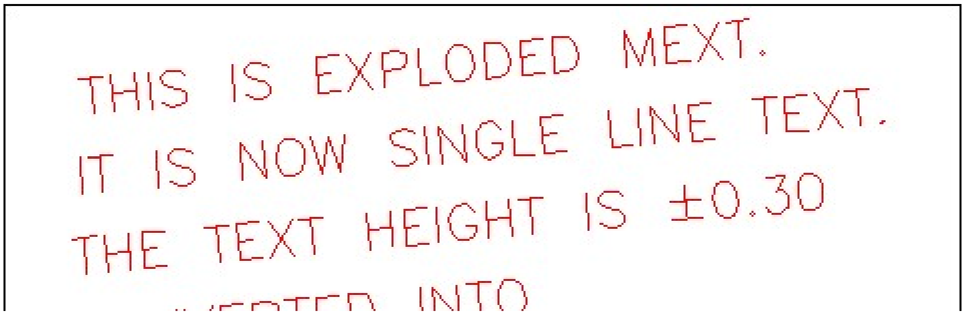
Step 10

Edit the third line of text. Add the special characters ‘ %%p ‘ preceding the number 0.30 and execute the command. Your drawing should appear as shown in the figure. (Figure Step 10A and 10B)



THIS IS EXPLODED MEXT.
IT IS NOW SINGLE LINE TEXT.
THE TEXT HEIGHT IS %%p0.30
INTO

Figure Step 10A

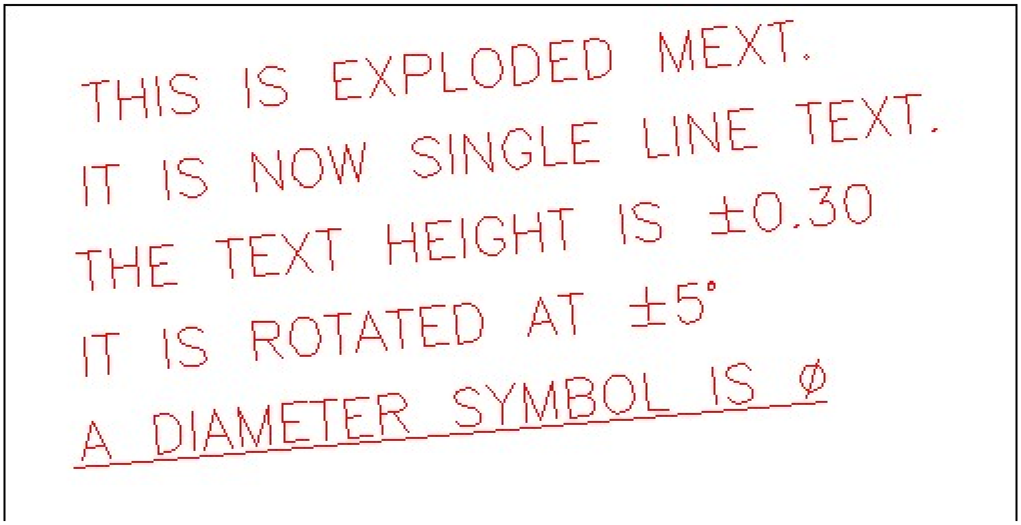


THIS IS EXPLODED MEXT.
IT IS NOW SINGLE LINE TEXT.
THE TEXT HEIGHT IS ±0.30
INTO

Figure Step 10B

Step 11

Using what you just learned and the special character table in Figure 25-3, edit the text to appear as shown in the figure. (Figure Step 11)



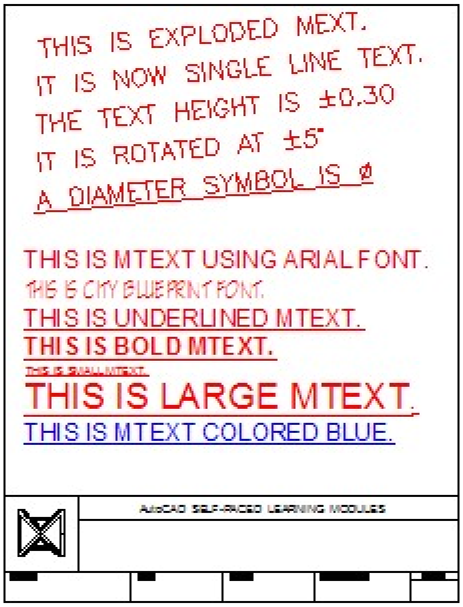
THIS IS EXPLODED MEXT.
IT IS NOW SINGLE LINE TEXT.
THE TEXT HEIGHT IS ±0.30
IT IS ROTATED AT ±5°
A DIAMETER SYMBOL IS Ø

Figure Step 11

AUTHOR'S COMMENTS: If you cannot complete Step 11, see answers below:
IT IS ROTATED AT %%p5%%d%%uA DIAMETER SYMBOL IS %%c%%u

Step 12

Your finished drawing should appear as shown in the figure. (Figure Step 12)



THIS IS EXPLODED MEXT.
IT IS NOW SINGLE LINE TEXT.
THE TEXT HEIGHT IS ±0.30
IT IS ROTATED AT ±5°
A DIAMETER SYMBOL IS ø

THIS IS MTEXT USING ARIAL FONT.
THIS IS CITY BLUEPRINT FONT.
THIS IS UNDERLINED MTEXT.
THIS IS BOLD MTEXT.
THIS IS SMALL MTEXT.
THIS IS LARGE MTEXT.
THIS IS MTEXT COLORED BLUE.

Figure Step 12

Step 13

Save and close the drawing.

MUST KNOW: Mtext can be one or more lines or paragraphs of text that is a single drawing object. It uses the current style properties but within the text object, the font, color and other appearance can vary. Mtext can be exploded with the EXPLODE command. When exploded, mtext will be converted to single line text objects.

USER TIP: While the DDEDIT command can be use to edit mtext, a much faster method is to double click the text. This method will open the Text Formatting dialogue box and display the text in it as soon as the mtext is double clicked.

Key Principles

Key Principles in Module 25

1. Unlike single line text, multiline text is a text object that can contain one or more lines of text and is one drawing object.
2. There are two basic font types available in AutoCAD. They are shape fonts and true type fonts.
3. Shape fonts are vector fonts and true type fonts are raster fonts.
4. Not all characters or symbols are available on a keyboard. These characters are called special characters. To enter them as text, the control codes ‘ %% ‘ must precede the special character.

Lab Exercise 25-1

Time allowed: 45 minutes.

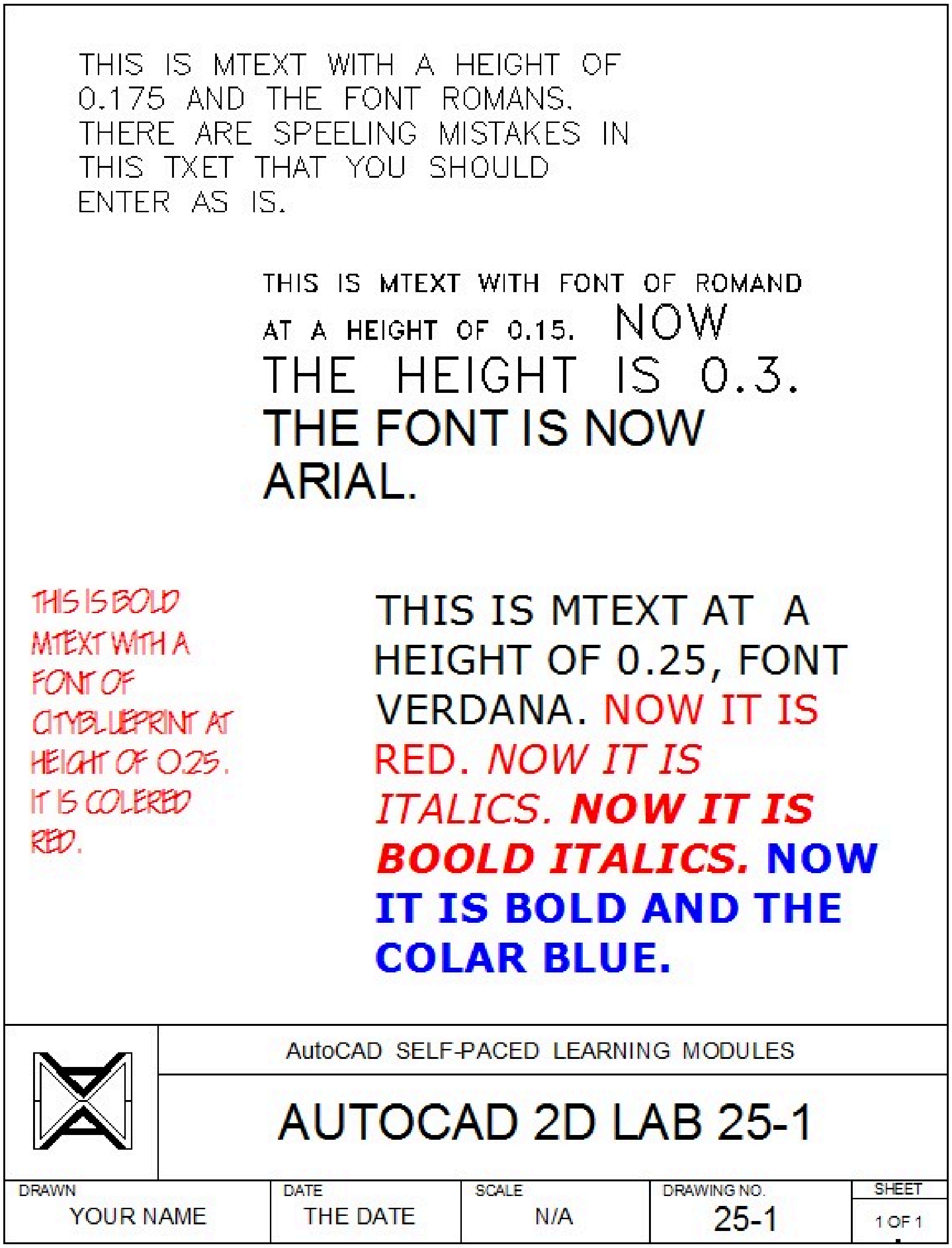
Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 25-1	2D Layout English	Inches

Step 1

Create a text style named: Module Lab 25, font: Romans. Set it as the current text style.

Step 2

Set layer: Text as the current layer. Enable layout: Module Layout A. In Paper space, insert 4 mtext objects as shown in the figure. Leave the spelling errors in the text as you enter it. Locate the text by eye to match the figure the best you can. (Figure step 2)



THIS IS MTEXT WITH A HEIGHT OF 0.175 AND THE FONT ROMANS. THERE ARE SPEELING MISTAKES IN THIS TXET THAT YOU SHOULD ENTER AS IS.

THIS IS MTEXT WITH FONT OF ROMAND AT A HEIGHT OF 0.15. NOW THE HEIGHT IS 0.3. THE FONT IS NOW ARIAL.

THIS IS BOLD MTEXT WITH A FONT OF CITYBLUEPRINT AT HEIGHT OF 0.25. IT IS COLERED RED.

THIS IS MTEXT AT A HEIGHT OF 0.25, FONT VERDANA. NOW IT IS RED. *NOW IT IS ITALICS.* ***NOW IT IS BOOLD ITALICS.*** **NOW IT IS BOLD AND THE COLAR BLUE.**

AutoCAD SELF-PACED LEARNING MODULES

AUTOCAD 2D LAB 25-1

DRAWN	DATE	SCALE	DRAWING NO.	SHEET
YOUR NAME	THE DATE	N/A	25-1	1 OF 1

Figure Step 2

Note: Not all computer systems have the same fonts loaded. This is especially true for true type fonts. If you don't have the font requested in a lab exercise, choose another one of your choice.

Step 3

Complete the titleblock using the AutoCAD book Titleblock Standards shown in Module 20.

Lab Exercise 25-2 Part B

Time allowed: 45 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 25-2	N/A	Inches

Step 1

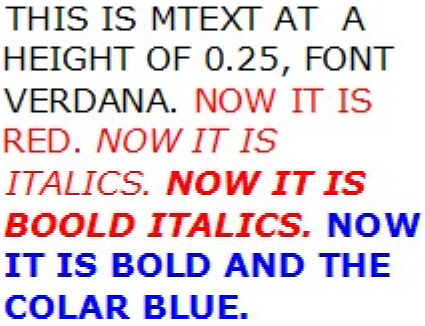
Open the drawing: AutoCAD 2D Lab 25-1. Using the SAVEAS command, save and name it: AutoCAD 2D Lab 25-2.

Step 2

Spell check all text in the layout and correct all spelling errors.

Step 3

Explode the mtext that is shown in Figure Step 3 only.

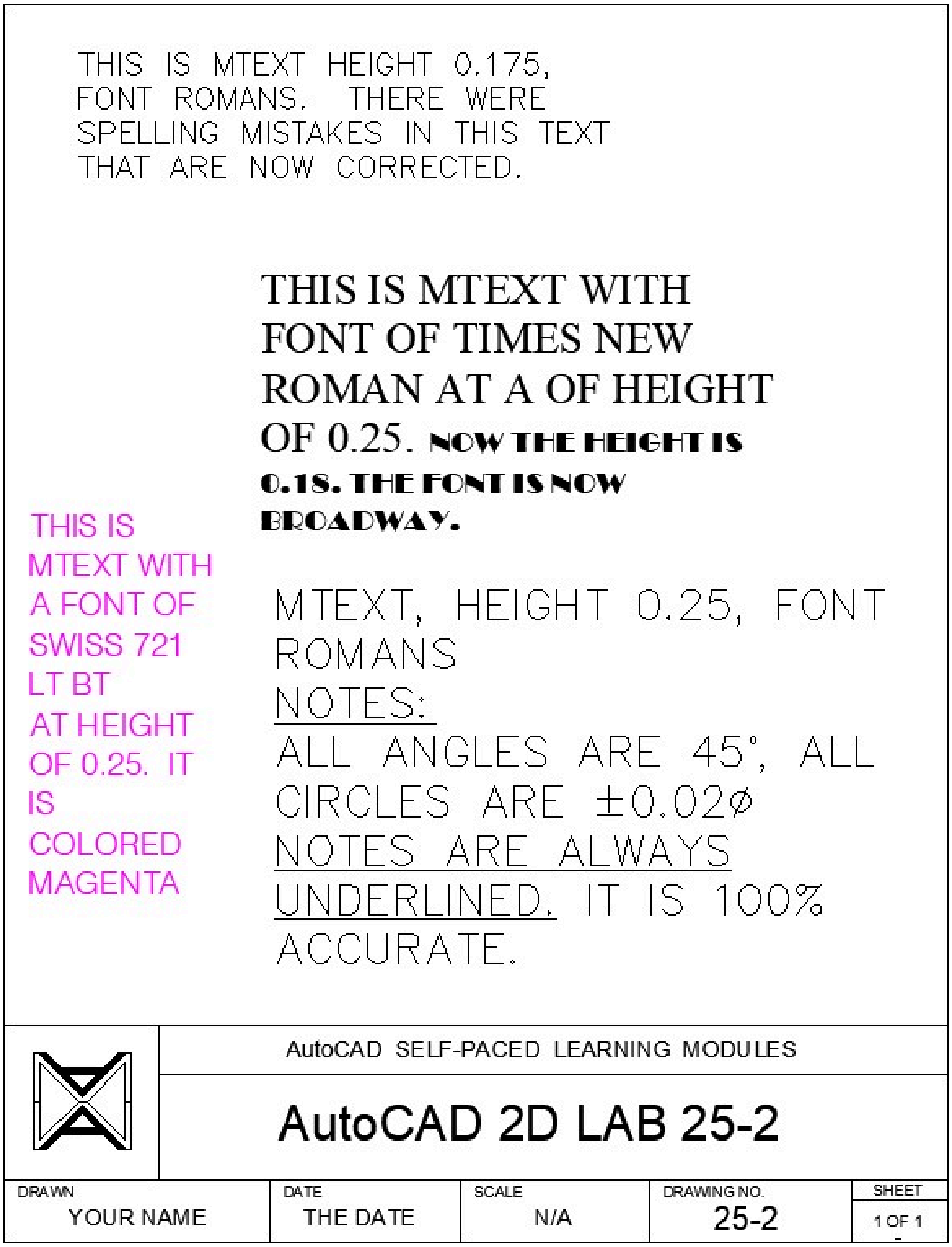


THIS IS MTEXT AT A
HEIGHT OF 0.25, FONT
VERDANA. NOW IT IS
RED. *NOW IT IS*
ITALICS. ***NOW IT IS***
BOOLD ITALICS. **NOW**
IT IS BOLD AND THE
COLAR BLUE.

Figure Step 3

Step 4

Edit the text to appear as shown in Figure Step 4.



THIS IS MTEXT HEIGHT 0.175, FONT ROMANS. THERE WERE SPELLING MISTAKES IN THIS TEXT THAT ARE NOW CORRECTED.

THIS IS MTEXT WITH FONT OF TIMES NEW ROMAN AT A OF HEIGHT OF 0.25. NOW THE HEIGHT IS 0.18. THE FONT IS NOW BROADWAY.

THIS IS MTEXT WITH A FONT OF SWISS 721 LT BT AT HEIGHT OF 0.25. IT IS COLORED MAGENTA

MTEXT, HEIGHT 0.25, FONT ROMANS

NOTES:

ALL ANGLES ARE 45°, ALL CIRCLES ARE ±0.02⌀ NOTES ARE ALWAYS UNDERLINED. IT IS 100% ACCURATE.

AutoCAD SELF-PACED LEARNING MODULES

AutoCAD 2D LAB 25-2

DRAWN	DATE	SCALE	DRAWING NO.	SHEET
YOUR NAME	THE DATE	N/A	25-2	1 OF 1

Figure Step 4

Note: Not all computer systems have the same fonts loaded. This is especially true for true type fonts. If you don’t have the font requested in a lab exercise, choose another one of your choice.

Module 26: Competency Test 5 Open Book

Learning Outcomes

When you have completed this module, you will be able to:

1. Within a three hour time limit, complete a written exam and a lab exercise without the aid of a key.

The AutoCAD 2D book was written with competency based modules. What that means is that you have not completed each module until you have mastered it. The Competency Test module contains multiple choice questions and a comprehensive lab exercise to test your mastery of the set of modules that you completed. There are no answers or keys supplied in a Competency Test module since it is meant to be checked by your instructor. If there are any parts of this module that you have trouble completing, you should go back and reread the module or modules containing the information that you are having trouble with. If necessary, redo as many lab exercises required until you fully understand the material.

If you are Completing this book:

- Without the Aid of an Instructor, complete the written test and the lab exercise.
- In a Classroom with an Instructor, the instructor will give instructions on what to do after this module has been completed.

Multiple Choice Questions

Select the BEST answer.

1. What is the main difference between a mtext object and a regular text object?
   A. Regular text can be exploded, mtext cannot.
   B. Mtext cannot be edited while a regular text object can.
   C. A regular text is a single object and a mtext is multiple objects.
   D. Mtext can have multiple lines while regular text can only be one line.
   E. Mtext can only have one font per object while a regular text can have more than one.
2. On what layer will the object(s) that are arrayed or mirrored, using the ARRAY and MIRROR commands, reside on? Select the best answer.

   - A. On layer Object
   - B. On the current layer
   - C. On the last layer used.
   - D. On the same layer as the objects that were selected in the ARRAY or MIRROR commands.
   - E. They will not be on any layer.

3. What object selection method must be used in the STRETCH command?
   - A. Individual pick
   - B. Crossing window
   - C. Window
   - D. Fence
   - E. You can use any method of the above methods.
4. In a rectangular array, what does a row offset with a positive value mean?
   - A. That the rows in the array are added downward.
   - B. That the rows in the array are added to the right.
   - C. That the rows in the array are added upward.
   - D. That the rows in the array are added to the left.
   - E. It means nothing.
5. What type of object will an exploded mtext be converted to?
   - A. lines and arcs
   - B. xref
   - C. single line text
   - D. block
   - E. A mtext object cannot be exploded.
6. What command is used to edit mtext?
   - A. DDEDIT
   - B. EDIT
   - C. MTEXTEDIT
   - D. TEXTEDIT
   - E. EDITTEXT
7. Which one of the following best describes the STRETCH command?
   - A. It moves the objects that cross the crossing window and stretches the objects that are totally inside the crossing
   - B. It can only stretch an object or objects

   C. It can only stretch an object or objects
   D. It stretches the objects that cross the crossing window and moves the objects that are totally inside the crossing
   E. It can only shorten or lengthen
8. Which one of the following best describes the window polygon method of selecting objects?
   A. Selects all the objects that are totally inside and the ones that cross a window defined by a polygon.
   B. Selects all the objects that are crossed by a line or a series of lines.
   C. Selects all the objects that totally inside and the ones that cross a square or rectangular window defined by two diagonal corners.
   D. Selects all the objects that are totally inside a window defined by a polygon.
   E. Selects all the objects are totally outside a window defined by a polygon.
9. What control code must be added before special characters when entering the special characters into a text or mtext object?
   A. %%
   B. ##
   C. **
   D. @@
   E. $$
10. In a polar array, what does a positive value mean in the Angle to Fill? Select the best answer.
    A. The angle must be between 180 and 360 degrees.
    B. The array will be in the clockwise direction.
    C. The array will be in the counterclockwise direction.
    D. It does not mean anything.
    E. The angle between the arrayed objects.

Lab Exercise 26-1 OPEN BOOK

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 26-1	2D Layout English	Inches

Step 1

On layer: Object, draw the object shown in the figure. Draw it anywhere in Model space. (Figure Step 1)

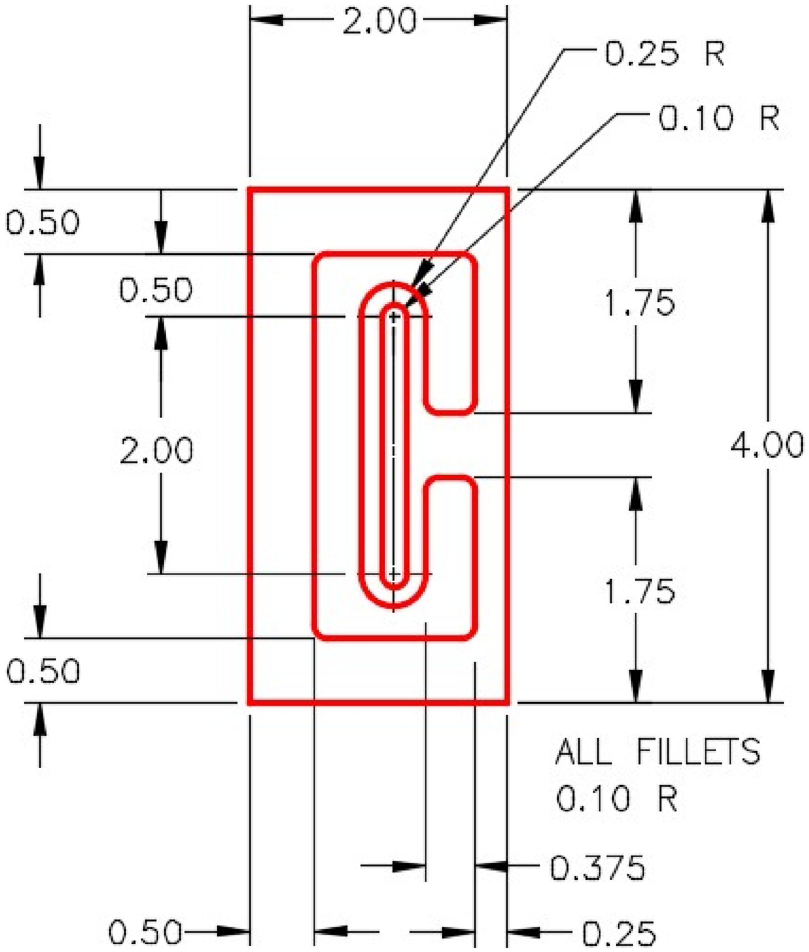

2.00
0.25 R
0.10 R
0.50
0.50
1.75
2.00
4.00
1.75
0.50
ALL FILLETS
0.10 R
0.375
0.50
0.25

Figure Step 1

Step 2

Using the MIRROR command, mirror the object and make the necessary changes to match the figure. (Figure Step 2)

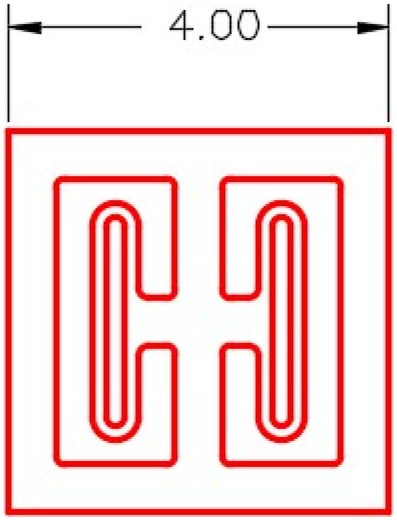


Figure Step 2

Step 3

Complete the object by erasing and extending all lines so they are all continuous in length. This object will be referred to as the original object in this lab exercise.

Step 4

Copy and scale the original object at a scale factor of 0.25 to a location anywhere in Model space. The smaller object (scaled object) will be referred to as the scaled object in this lab exercise. (Figure Step 4)

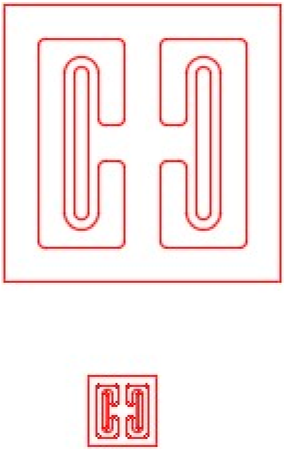


Figure Step 4

AUTHOR'S COMMENTS: Figure Step 4 shows the original and the scaled object.

Step 5

Complete the following steps: (Figure Step 5)

A Copy the scaled object to the coordinates shown. Stretch it to match the dimensioned drawing.

B Copy the scaled object to the coordinates shown. Stretch it to match the dimensioned drawing.

C Copy the scaled object to the coordinates shown. Stretch it to match the dimensioned drawing.

AUTHOR'S COMMENTS: This will take 2 stretches, one from top and one from the bottom.

D Copy the scaled object to the coordinates shown. Stretch it to match the dimensioned drawing.

E Copy the scaled object to the coordinates Stretch it to match the dimensioned shape.

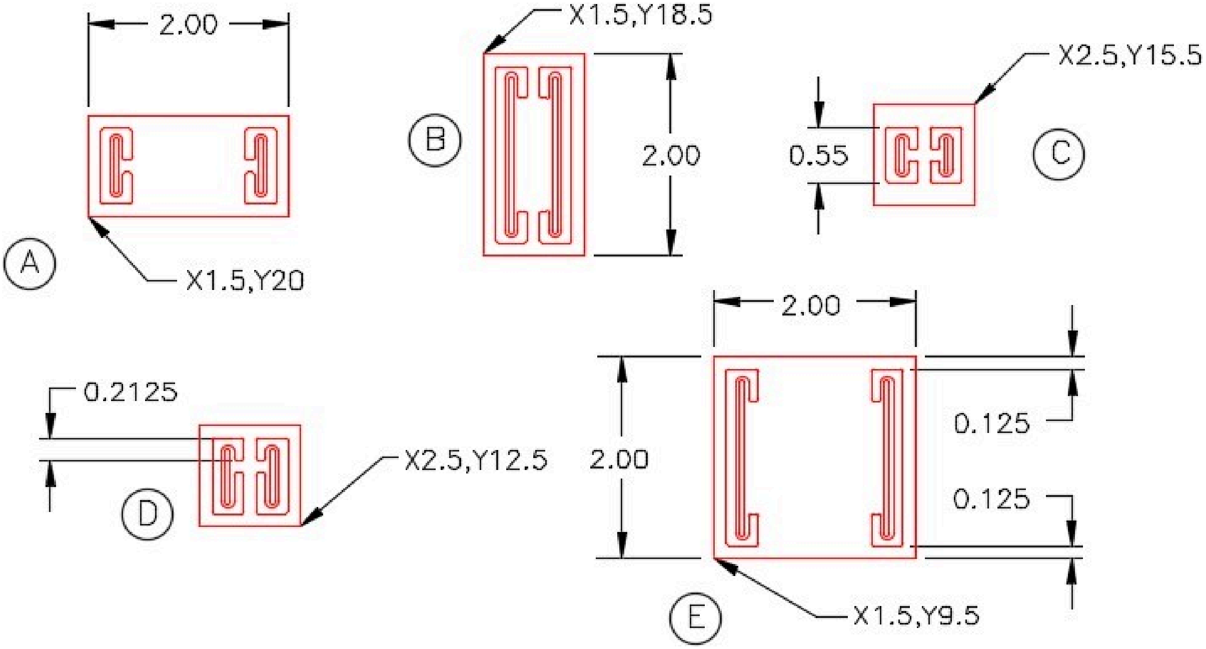

2.00
A
X1.5,Y20
X1.5,Y18.5
B
2.00
X2.5,Y15.5
0.55
C
0.2125
D
X2.5,Y12.5
2.00
2.00
0.125
0.125
E
X1.5,Y9.5

Figure Step 5

Step 6

Complete the following steps: (Figure Step 6)

F Copy the original object to the coordinates Scale it to match the dimensioned drawing.

G Copy the original object to the coordinates shown. Scale it to a scale factor of 5.

H Copy the original object to the coordinates shown. Scale the two outside ones to a scale factor of 0.5 and the center one to fit.

I Copy the original object to the coordinates shown. Rotate it 45 degrees and scale it to match the the figure.

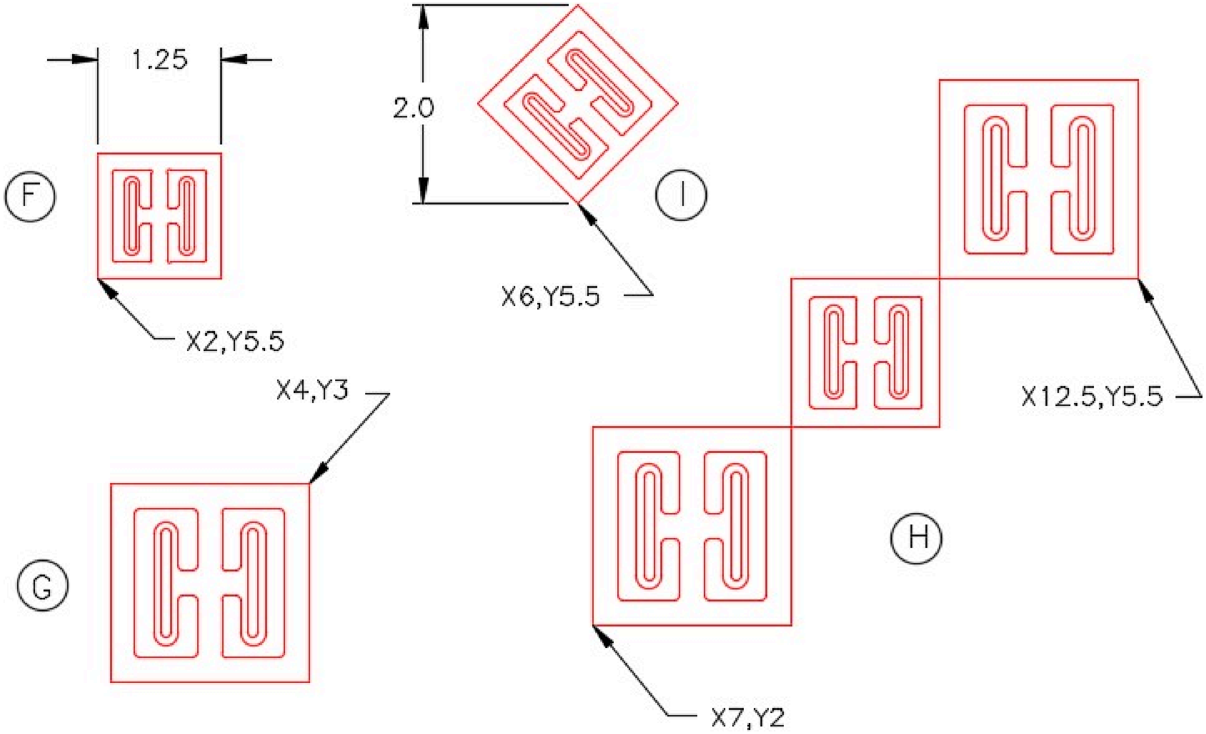


Figure Step 6

Step 7

Complete the following steps: (Figure Step 7A and 7B)

J Draw a 1.4 diameter construction circle at the coordinates shown. Copy the scaled object to the coordinate shown. Array the object to match the figure.

K Draw a 1.4 diameter construction circle at the coordinates shown. Copy the scaled object to the coordinates shown. Array the object to match the figure.

L Copy the original object anywhere on the drawing. Scale the object to match the given dimension. Rotate it 45 degrees. Using 2 polar arrays, complete the diagram shown below. Move the finished objects to the coordinates shown in the figure.

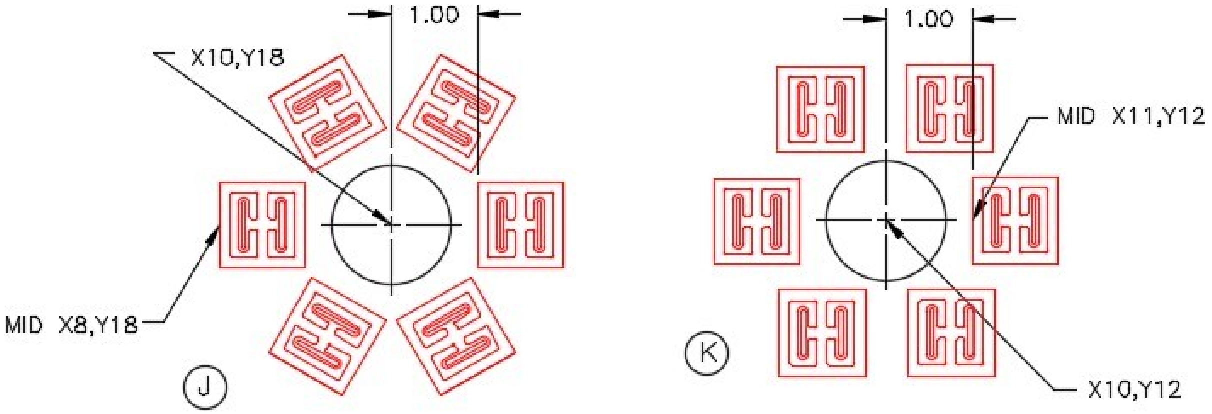


Figure Step 7A

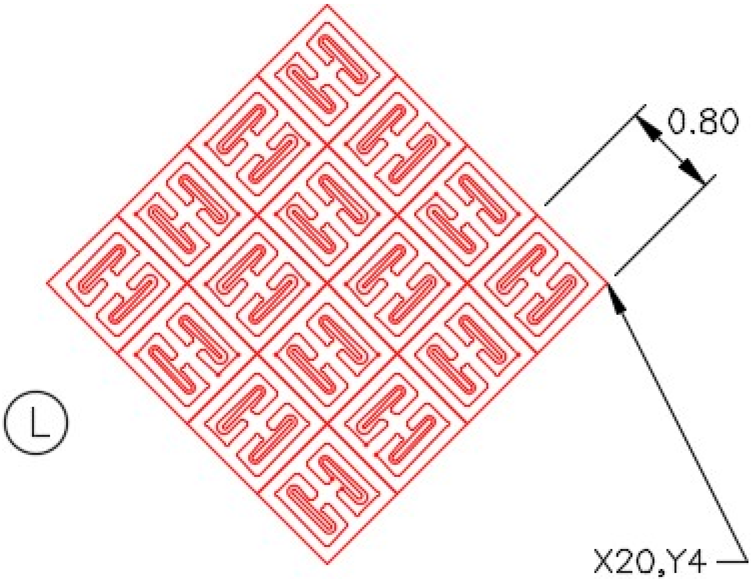
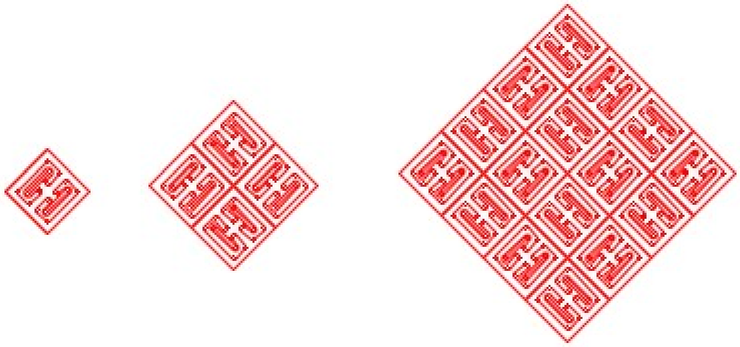


Figure Step 7B



Step 1 *Step 2* *Step 3*

Construction Hint

Step 8

Complete the following: (Figure Step 8A and 8B)

M Draw a construction circle at the coordinates shown. Copy the scaled object to the four locations shown. Execute the ARRAY command, four times, to match the figure.

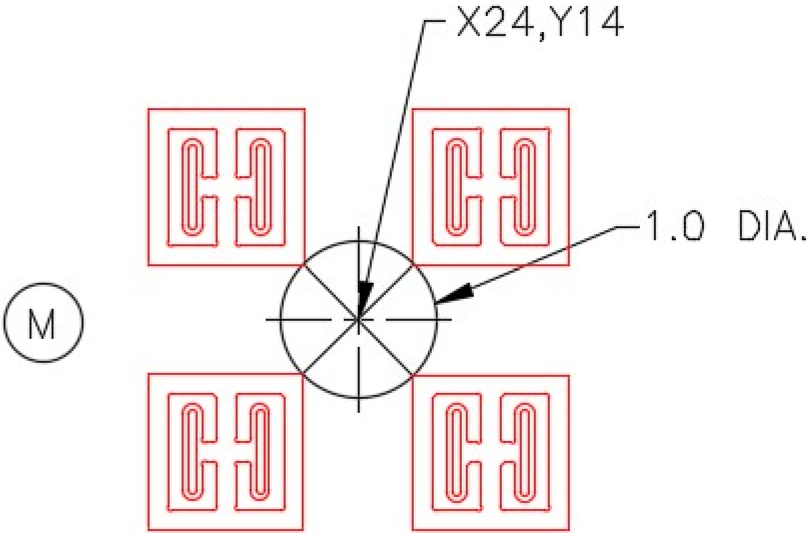


Figure Step 8A

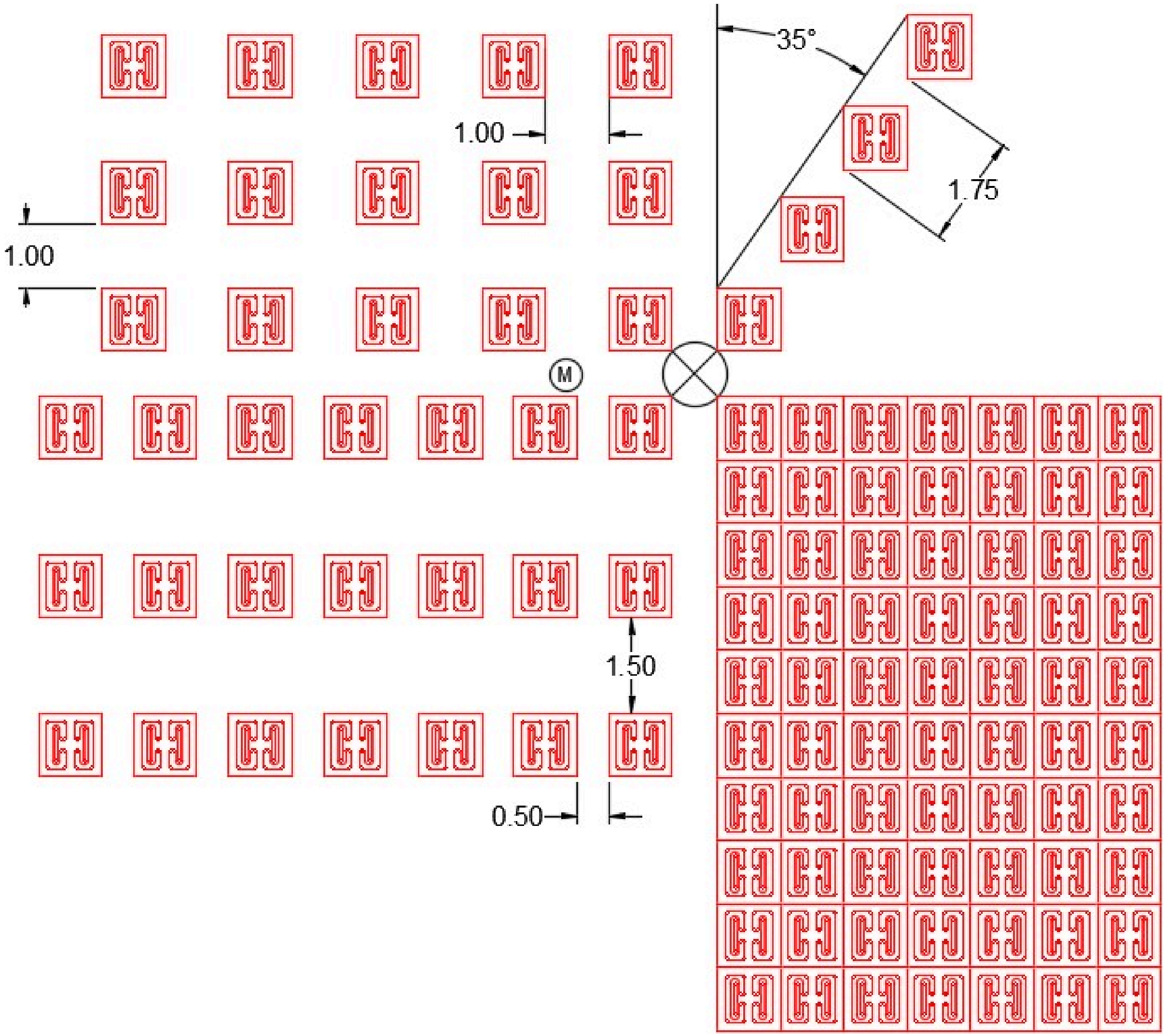
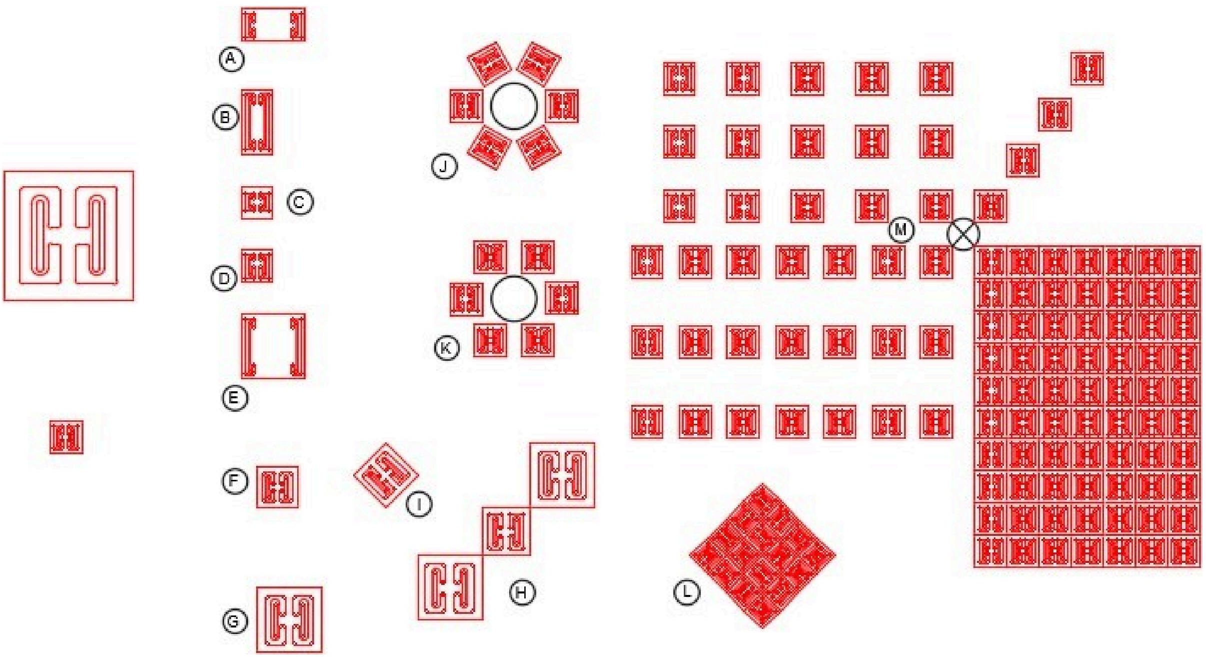
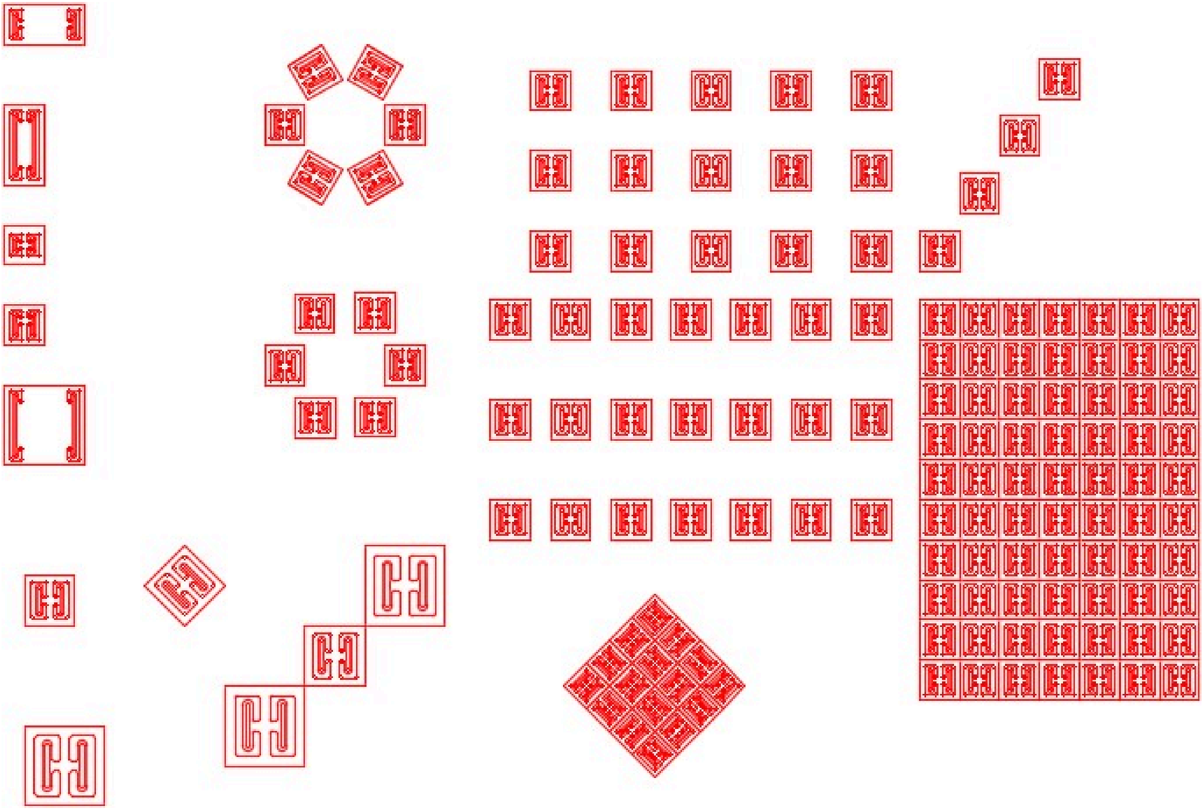

35°
1.75
1.00
1.00
M
1.50
0.50

Figure Step 8B


A
B
C
D
E
F
G
H
I
J
K
L
M

Working Drawing



Completed Drawing

Part 6

Module 27: Multiview Drawings

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe a multiview drawing, the glass box principle, the three standard views, object lines, and hidden lines.
2. From a 3D pictorial of an object, draw a multiview drawing using the three standard views.

NOTE: If you understand multiview drawings, hidden lines, and can draw the three standard views of an object skip this module and go directly to Module 28.

Multiview Drawing

The drafting and design world uses a system of representing a three- dimensional object by drawing two-dimensional views. It is called a *multiview drawing*. To explain this system of drawing, the 3D wireframe model shown in Figure 27-1 will be used. To draw a two- dimensional view of one side of the object, place a imaginary plane parallel to its side and project the view of the model (perpendicular) onto the plane. This is called *orthographic projection*. Imagine the plane to a sheet of glass. See Figure 27-2.

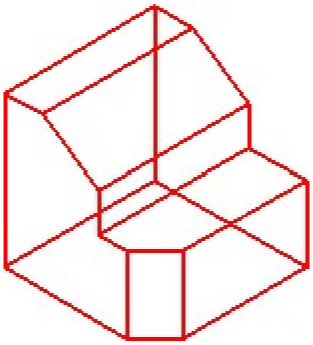


Figure 27-1
Three Dimensional
Wireframe Model

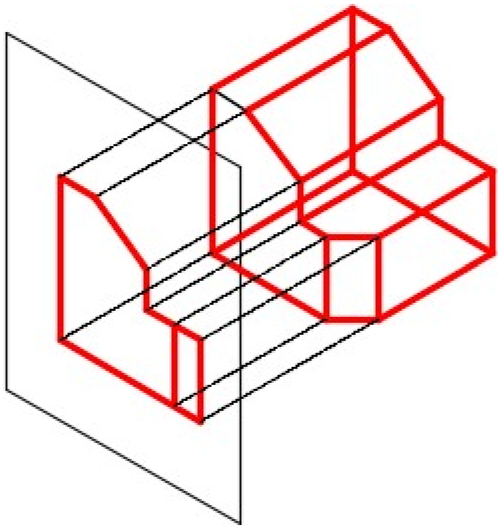


Figure 27-2
Projecting the Two-Dimensional View

The Glass Box Principle

To carry this principal further, place a plane on each side of the object for a total of six planes or sheets of glass to form a glass box. See Figure 27-3. Picture unfolding the glass box onto a flat two-dimensional plane as shown in Figure 27-4.

All six views are now visible at the same time. The six views are named Top, Front, Right Side, Left Side, Rear, and Bottom.

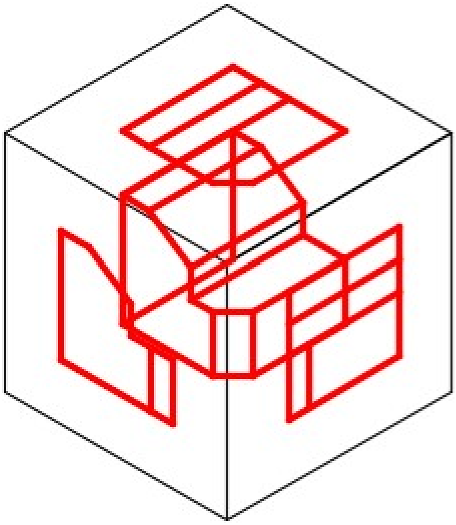


Figure 27-3
The Glass Box Principle

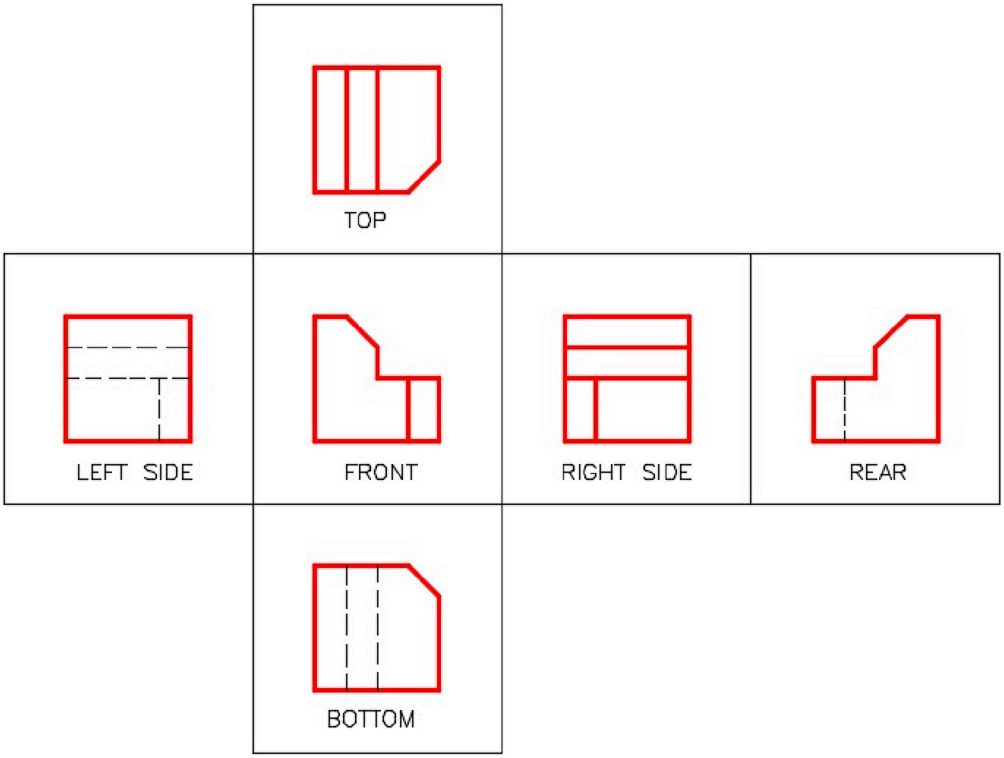

TOP
LEFT SIDE
FRONT
RIGHT SIDE
REAR
BOTTOM

Figure 27-4
The Glass Box Unfolded

The Three Standard Views

In almost all objects, three views are adequate to describe it. In fact, there are many models that only need two views and some that only need one view to describe them.

The three standard views are the *Top*, *Front*, and *Right Side*. They must be drawn in the positions shown in Figure 27-5 and must align.

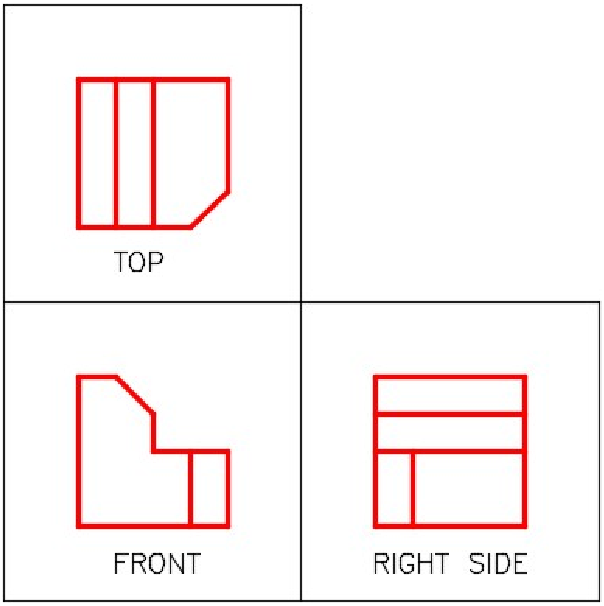


Figure 27-5
The Three Standard Views

Picking the Standard Three Views

The three standard views are always selected as shown in Figure 27-6.

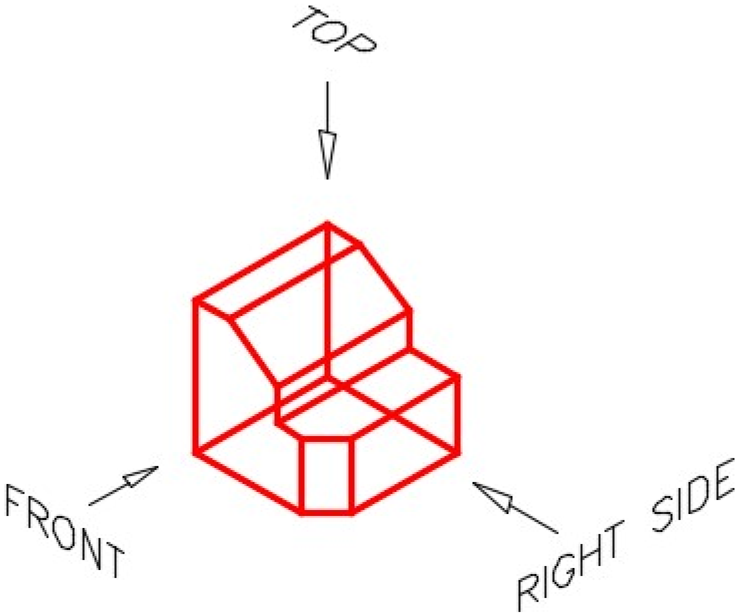


Figure 27-6
Picking the Three Standard Views

Drawing the Views

Usually it is best to draw the view with the most irregular shape first and then project lines to the other two views. For the object in Figure 27-7, the front view should be drawn first and then the top and right side views are projected.

Notice how the views aligned. Figure 27-7 shows two different methods of projecting lines from the top view to the right side view or vise versa.

The distance between the views is not important and can be easily adjusted in AutoCAD using the MOVE command. In most cases, the distances are adjusted to accommodate dimensions and notes.

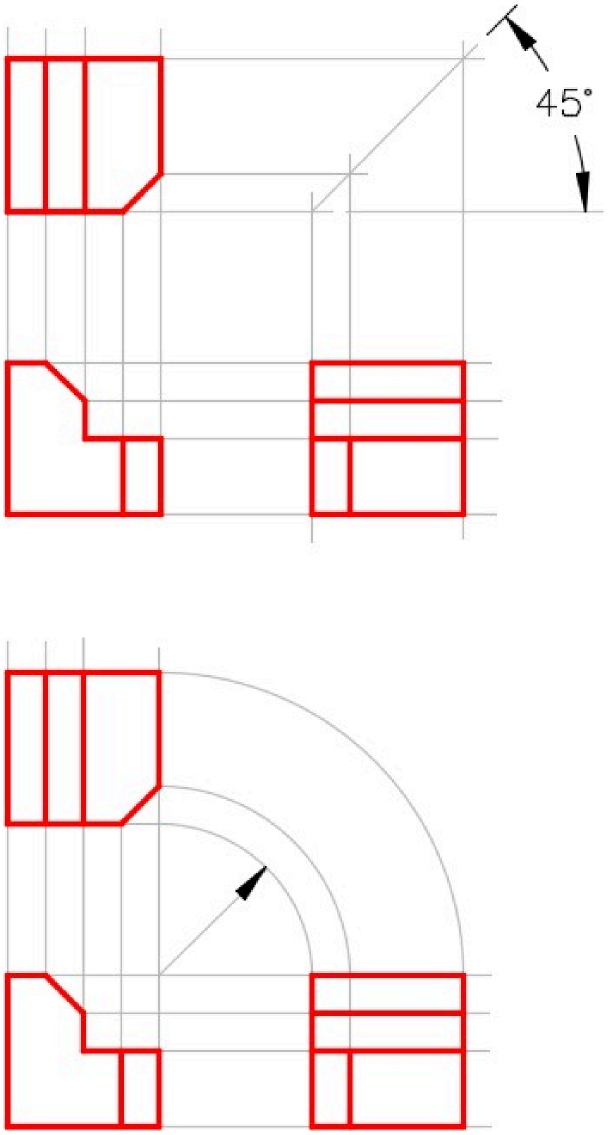


Figure 27-7
Two Methods for View Layout and Alignment

Drafting Lesson: Object and Hidden Lines

Lines and features that can be seen in the views are drawn with continuous or solid lines. They are called *object lines*. Even though they are called object lines, they can be circular in shape. To completely describe a model in a multiview drawing, you must also show all lines or features that are hidden in that view. They are called *hidden lines* and their linetype is dashed. Study the multiview drawing below and take note how the holes going through the object are shown with hidden lines. See Figures 27-8 and 27-9.

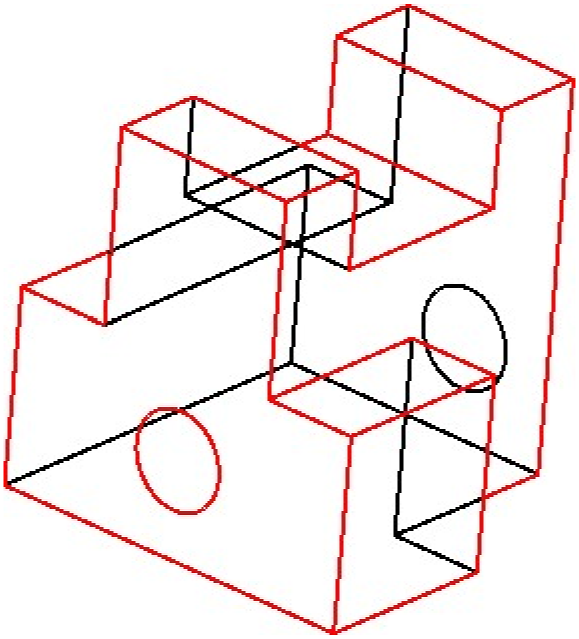


Figure 27-8
The Model

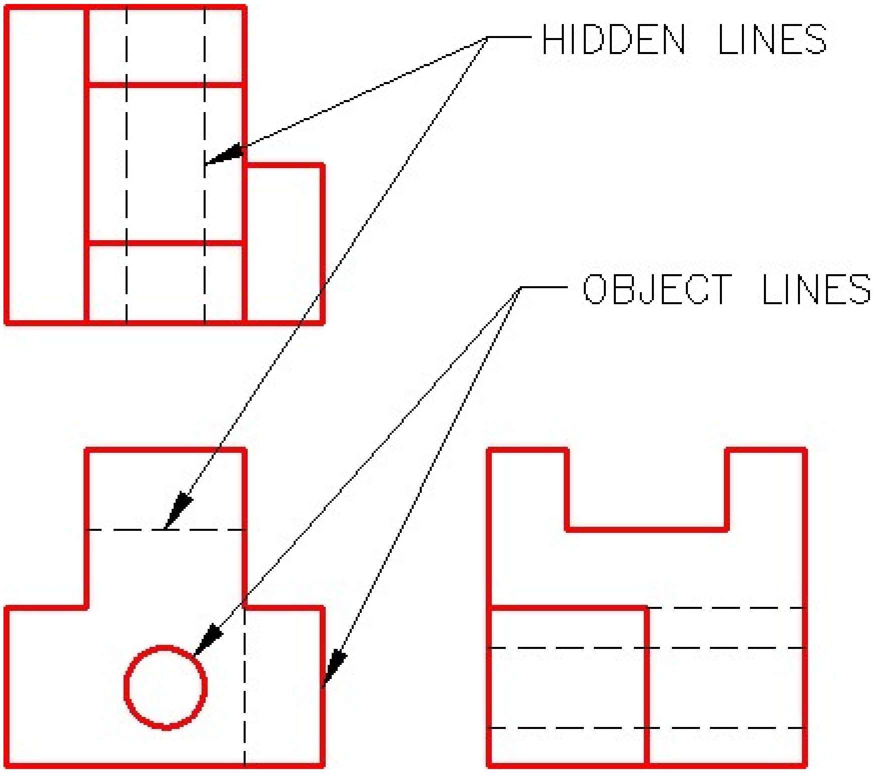

HIDDEN LINES
OBJECT LINES

Figure 27-9
The Multiview Drawing of the Model

WORKALONG: Drawing Multiviews

Step 1

Start a new drawing using the template: 2D Layout English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 27-1.

Step 3

Set the insert units to inches. Open the Insert dialogue box and insert the block: AutoCAD 2D Workalong 27-1. (Figure Step 3)

Step 4

Using The EXPLODE command, as shown below, explode the block you just inserted. Command: **EXPLODE**

Select objects: 1found

(Select the grid)

Select objects:

Command:

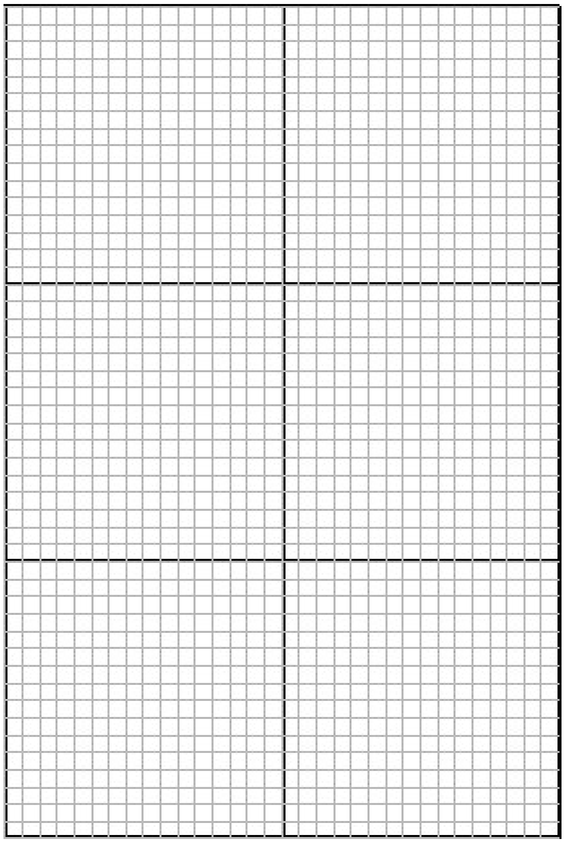
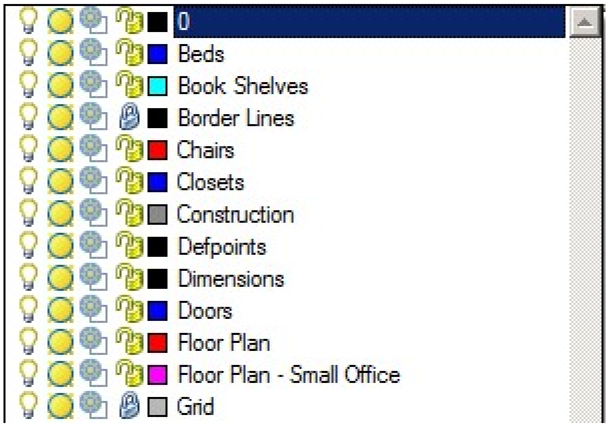


Figure Step 3

Step 5

Lock layers: Grid and Border Lines. (Figure Step 5)



0
Beds
Book Shelves
Border Lines
Chairs
Closets
Construction
Defpoints
Dimensions
Doors
Floor Plan
Floor Plan - Small Office
Grid

Figure Step 5

AUTHOR'S COMMENTS: By locking the Grid and Border Line layers, the drawing objects on those layers cannot be accidentally moved or deleted while you are completing the drawing.

Step 6

Draw in the upper left area. On layer Object, using the isometric drawing as the object to draw, draw the outline of the Front view of the model. (Figure Step 6A and 6B)

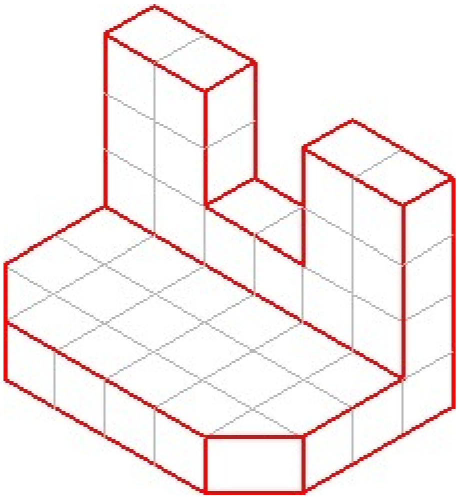


Figure Step 6A

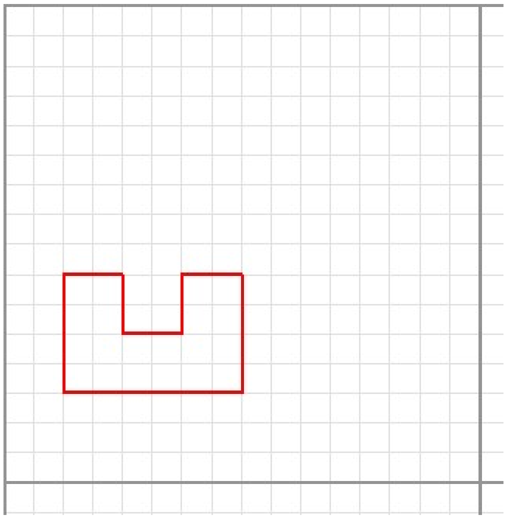


Figure Step 6B

AUTHOR'S COMMENTS: One grid on the isometric drawing is equal to one grid on the drawing. Draw the outline of the view. Do not worry about the lines on the inside of the object.

Step 7

Draw the outline of the Right Side view of the object. (Figure Step 7)

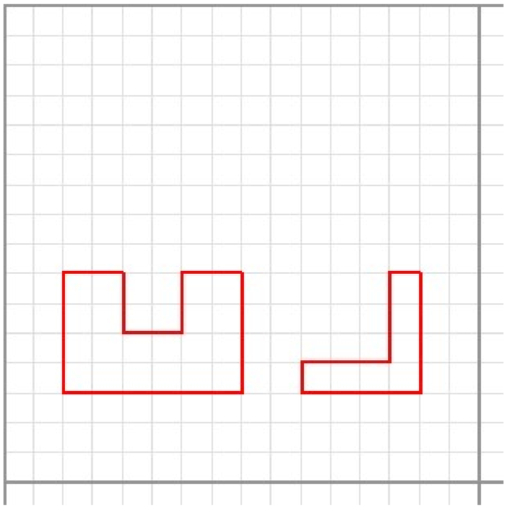


Figure Step 7

AUTHOR'S COMMENTS: The lines of the right side view must line up horizontally with the Front view.

Step 8

Draw the outline of the Top view of the object. (Figure Step 8)

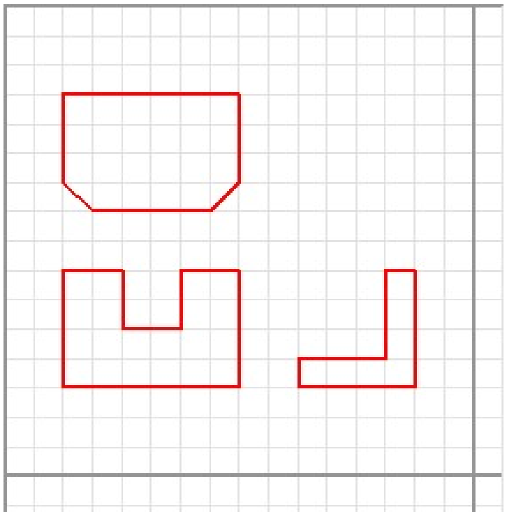


Figure Step 8

AUTHOR'S COMMENTS: The lines of the Top view must line up vertically with the front view.

Step 9

Add the interior object lines in all views. (Figure Step 9)

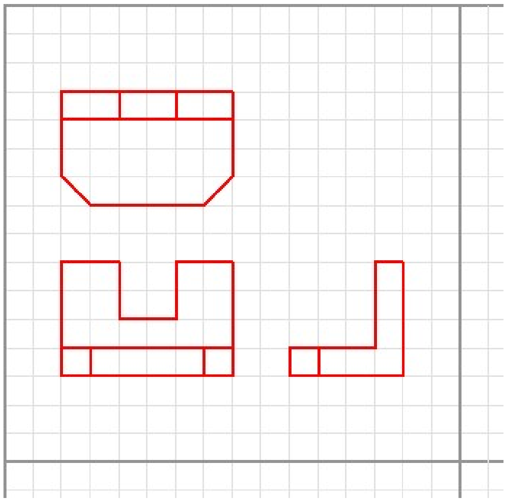


Figure Step 9

Step 10

On layer: Hidden, draw the hidden lines. In this drawing, only one line is required. (Figure Step 10)

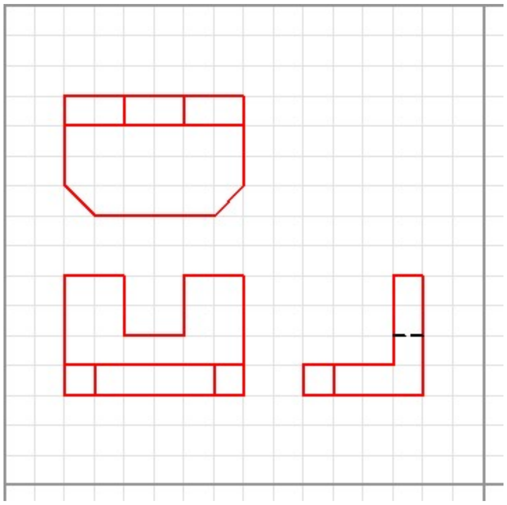


Figure Step 10

Step 11

On layer: Text, add the text 27-1 under the multiview drawing. Use text style: 2D Modules at a height of 0.5 inches. (Figure Step 11)

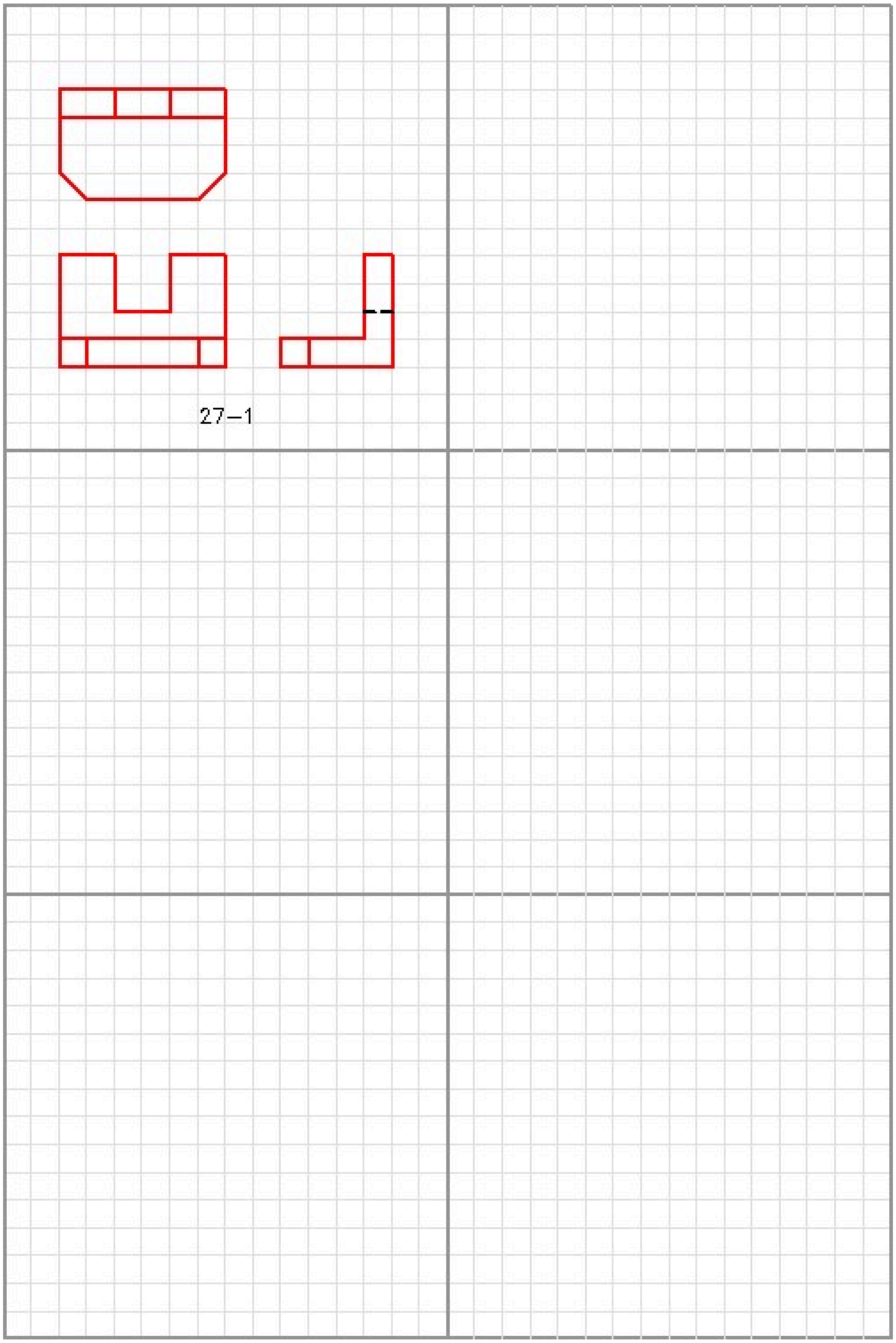


Figure Step 11

Step 12

Save and close the drawing.

Key Principles

Key Principles in Module 27

1. The drafting and design world uses a system of representing a three-dimensional object by drawing two-dimensional views, called a multiview drawing.
2. The three standard views of a multiview drawing are the top, front, and right side.
3. When drawing a multiview drawing, the top, front and right side views of an object must be drawn in the correct position and must The distance between the views is not important.

Lab Exercise 27-1

Time allowed: 40 minutes.

Step 1

Open drawing: AutoCAD 2D Workalong 27-1. Using the SAVEAS command, save with the name: AutoCAD 2D Lab 27-1. (Figure Step 1)

Step 2

On layer: Object and Hidden, draw a multiview drawing using the standard three views of each of the five objects in Figure 27-2 to 27-6.

Step 3

Check your answer in Figure 27-7. Do not look at the answer until you have completed your drawings.

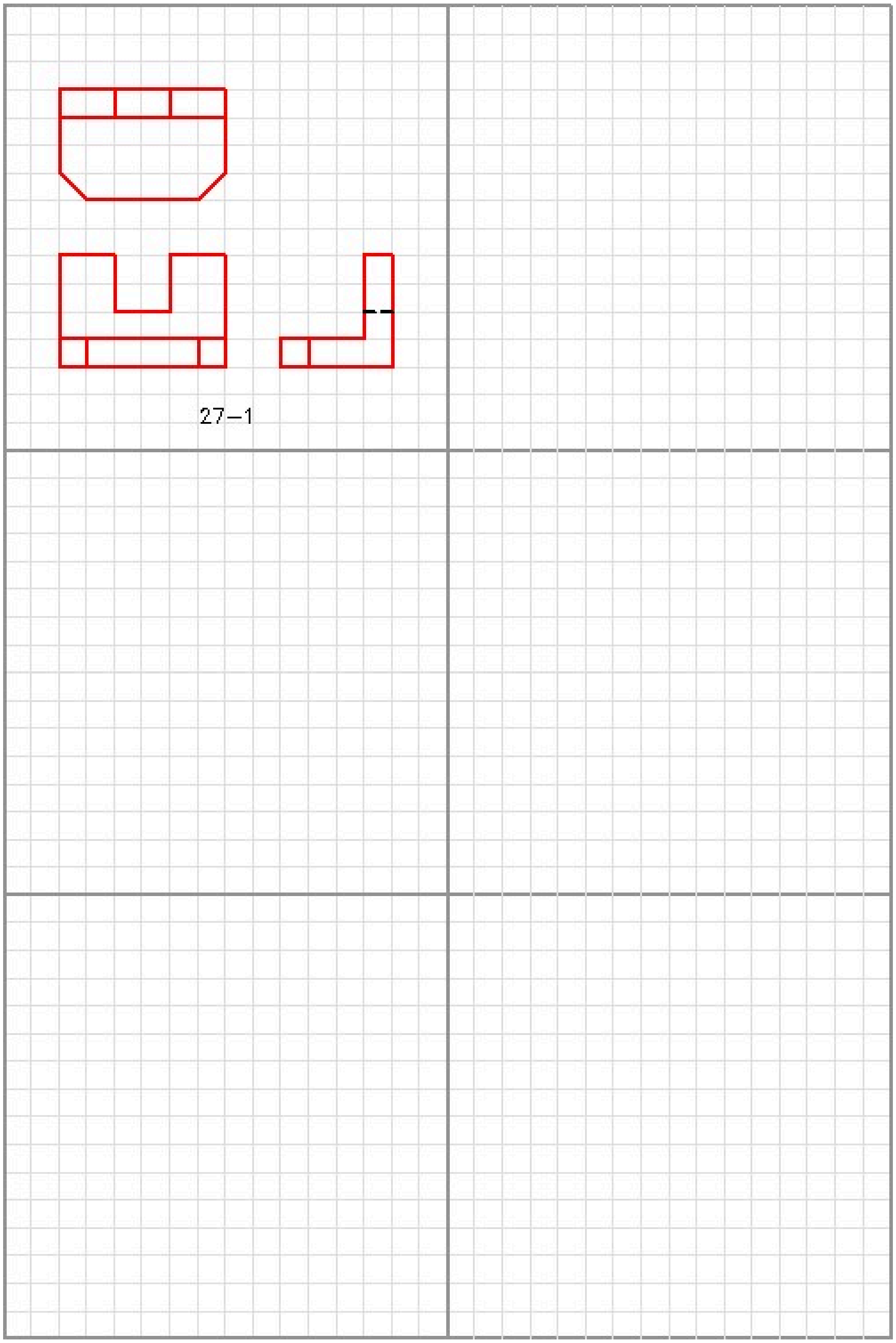
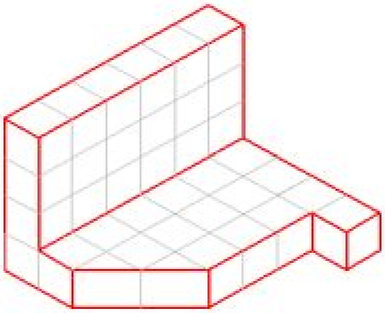
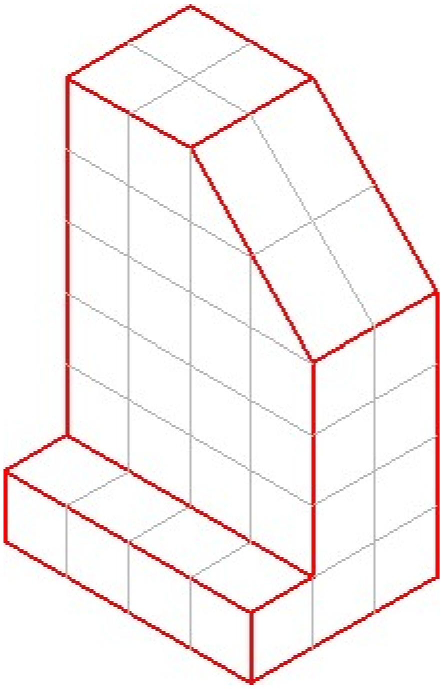


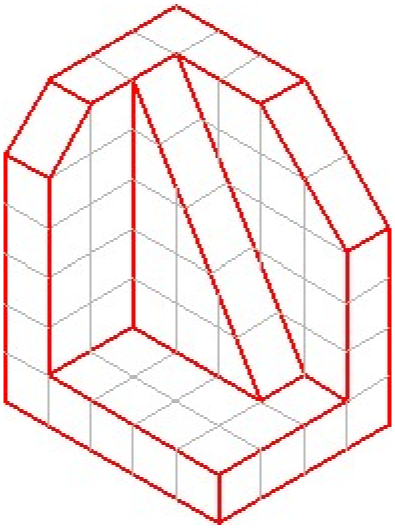
Figure Step 1



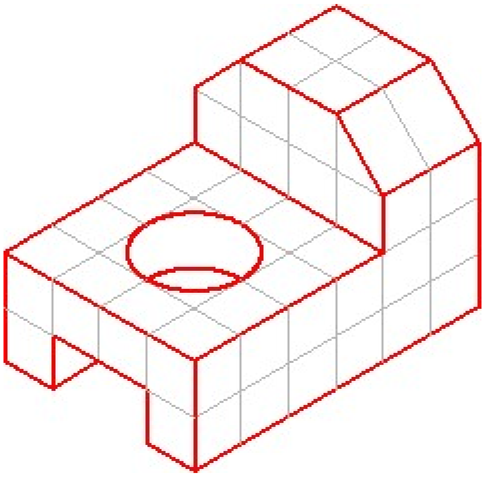
27-2



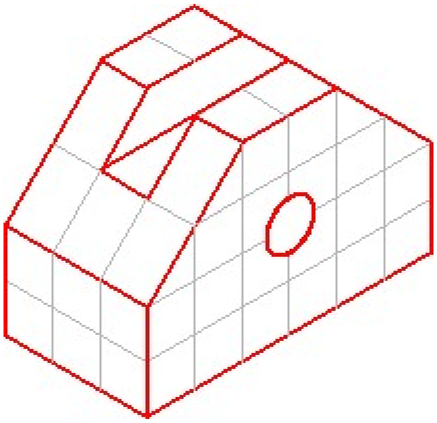
27-3



27-4



27-5



27-6

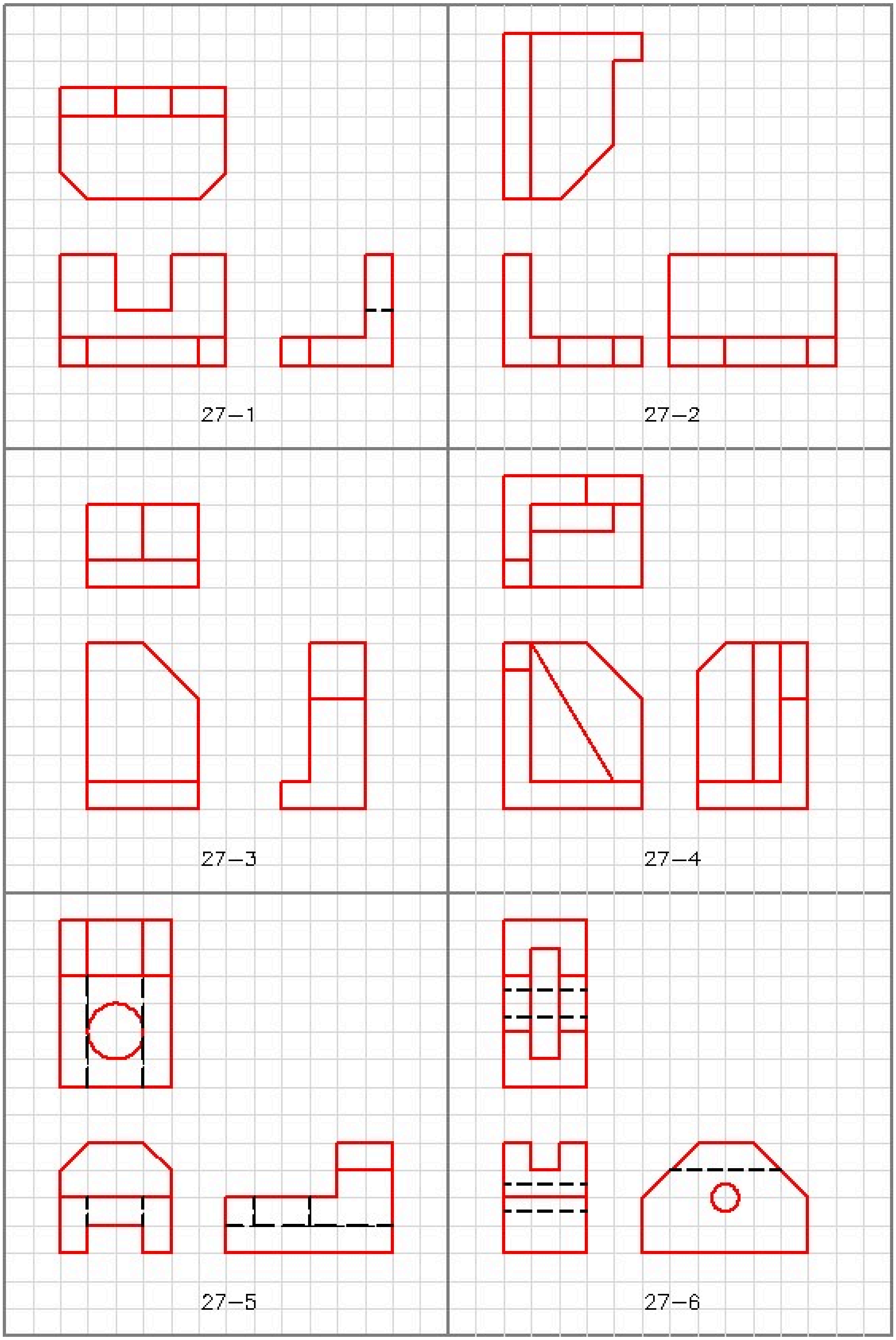

27–1
27–2
27–3
27–4
27–5
27–6

Figure 27-7 Answers

Module 30: Regular Polygons

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe a regular polygon and the methods used to construct them.
2. Apply the POLYGON command to create regular polygons using three methods.
3. Apply the ELLIPSE command to create ellipses.
4. Apply the RECTANGLE command to create rectangles.
5. Apply the AREA command to measure areas and perimeters.
6. Describe how to work with and add and subtract angles with degrees minutes and seconds.

Drawing Regular Polygons

A *regular polygon* is a closed polyline having between 3 and 1,024 equal-length sides. Regular polygons can be drawn using the POLYGON command. Using the POLYGON command to create polygons is a fast and simple way to draw squares, equilateral triangles, hexagons, octagons, and any other regular polygon. See Module 14.

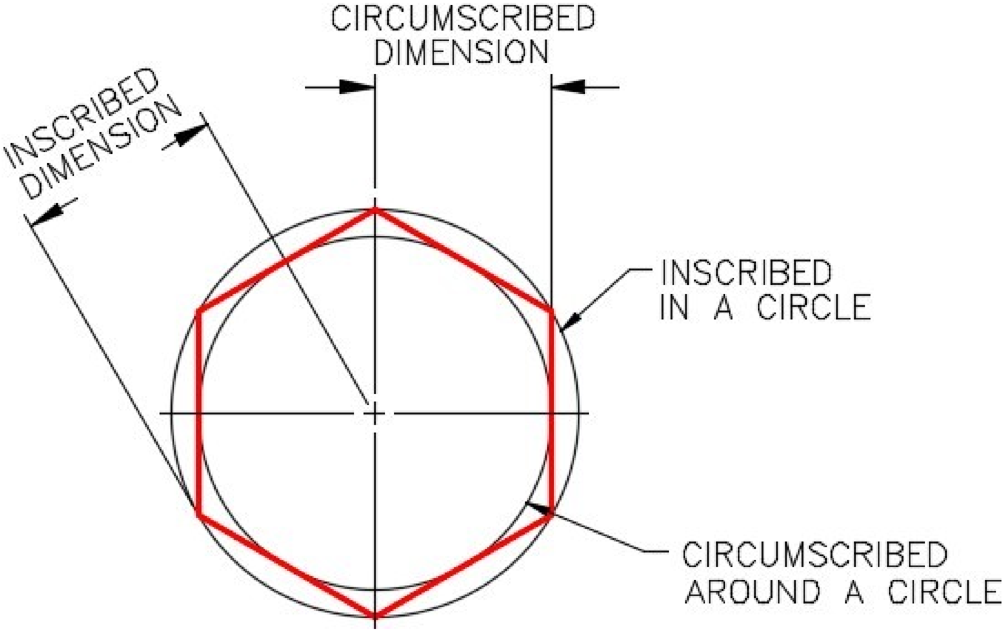

CIRCUMSCRIBED DIMENSION
INSCRIBED DIMENSION
INSCRIBED IN A CIRCLE
CIRCUMSCRIBED AROUND A CIRCLE

Figure 30-1
A Regular Polygon

A polygon drawn using the POLYGON command constructs a closed pline. Polygons can be constructed using the inscribed, circumscribed, or the edge method. See Figure 30-1 and 30-2. Since the polygon drawn by the POLYGON command is a polyline, it is one object in AutoCAD. It can be exploded into individual lines using the EXPLODE command.

Even though a square can be constructed using the POLYGON command, they are easier to draw using the RECTANGLE command.

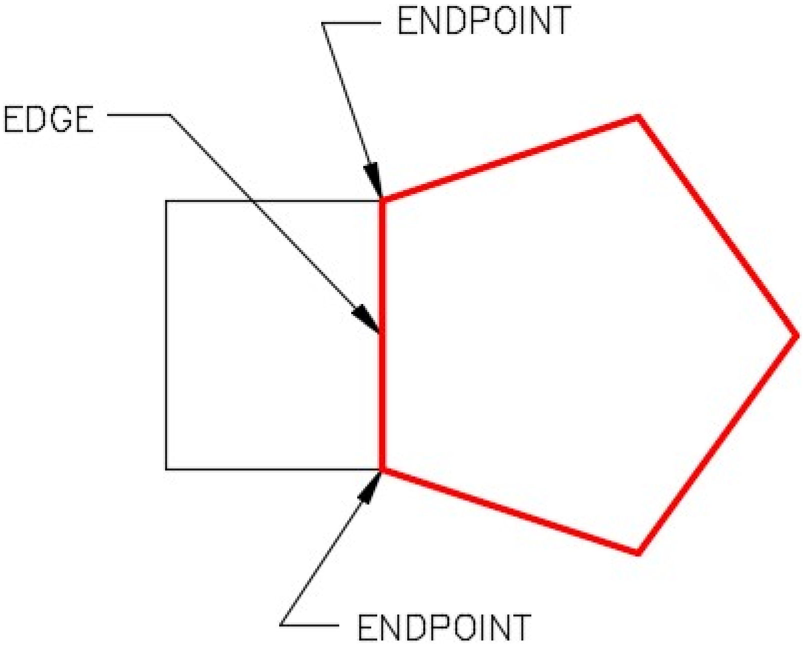
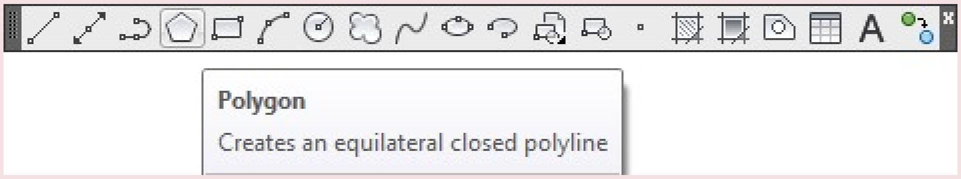


Figure 30-2
Drawing a Regular Polygon – Edge Method

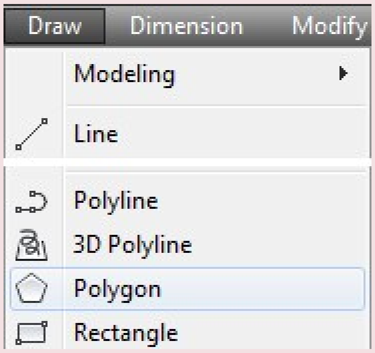
AutoCAD Command: POLYGON

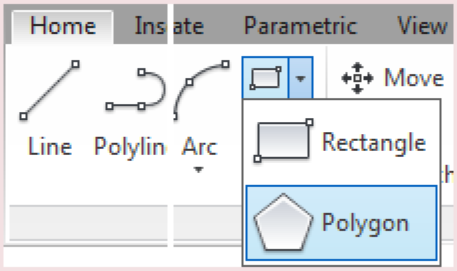
The POLYGON command is used to create closed polylines to form regular polygons that have between 3 to 1024 sides.

Shortcut: **POL**



Polygon
Creates an equilateral closed polyline

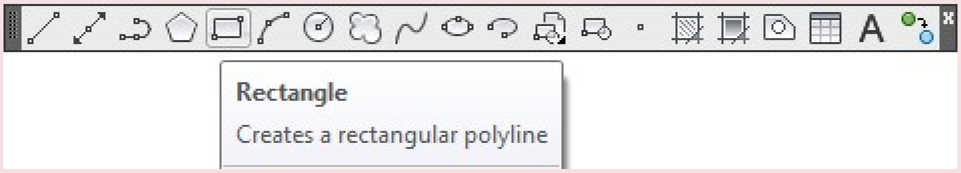

Draw
Dimension
Modify
Modeling
Line
Polyline
3D Polyline
Polygon
Rectangle

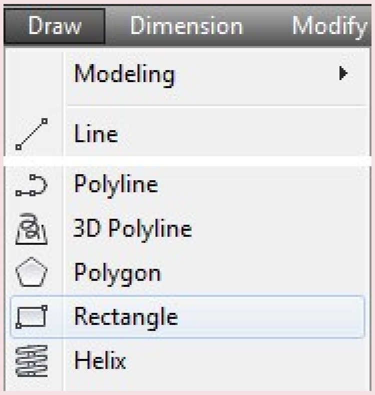

Home
Ins
ate
Parametric
View
Move
Line
Polylin
Arc
Rectangle
Polygon

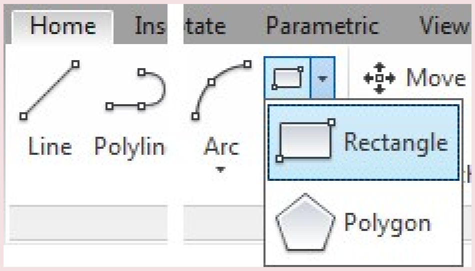
AutoCAD Command: RECTANGLE

The RECTANGLE command is used to construct a 4 sided polygon using a closed polyline in the shape of a rectangle or a square.

Shortcut: **REC**


Rectangle
Creates a rectangular polyline

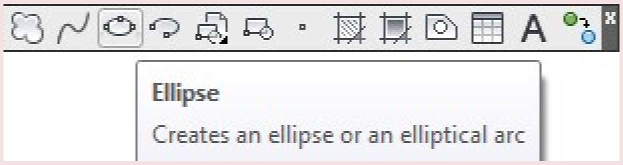

Draw
Dimension
Modify
Modeling
Line
Polyline
3D Polyline
Polygon
Rectangle
Helix


Home
Ins
tate
Parametric
View
Line
Polylin
Arc
Move
Rectangle
Polygon

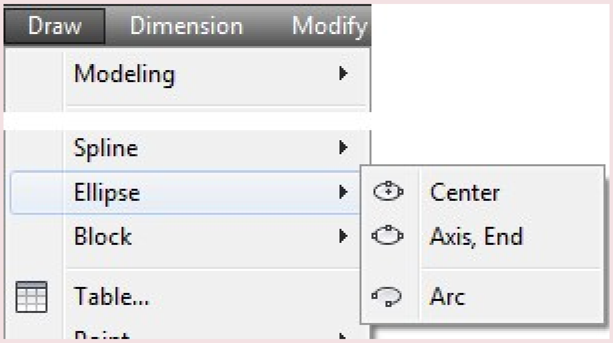
AutoCAD Command: ELLIPSE

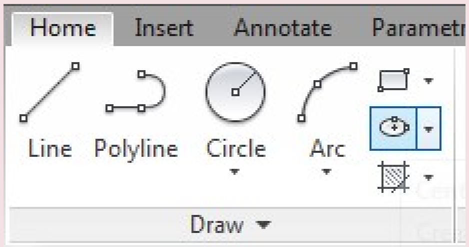
The ELLIPSE command is used to construct ellipses or elliptical arcs.

Shortcut: **EL**


Ellipse

Creates an ellipse or an elliptical arc


Draw
Dimension
Modify
Modeling
Spline
Ellipse
Block
Table...
Center
Axis, End
Arc


Home
Insert
Annotate
Parametr
Line
Polyline
Circle
Arc
Draw

Geometry Lesson: Ellipse

An *ellipse* is determined by two axes that define its length and width. The longer axis is called the *major axis*, and the shorter one is the *minor axis*. The center of the ellipse is located at the intersection of the axes. The rotation angle of the ellipse determines length of the minor axis compared to the length of the major axis. See Figure 30-3.

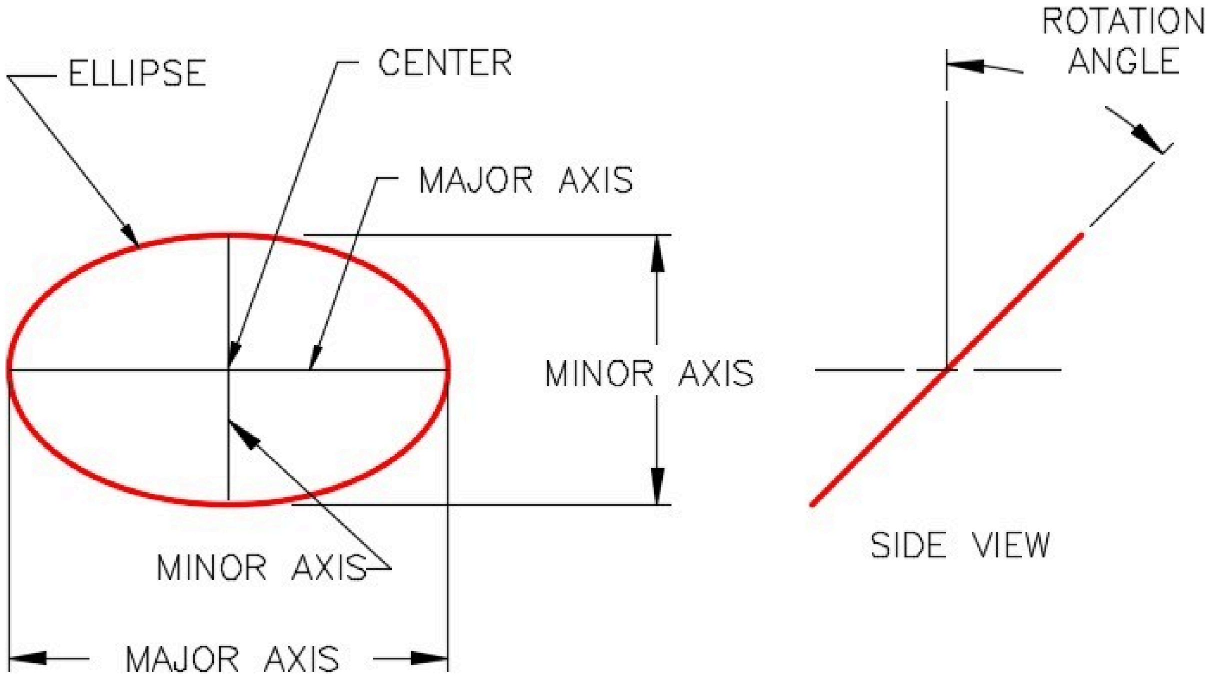

ELLIPSE
CENTER
MAJOR AXIS
MINOR AXIS
MINOR AXIS
MAJOR AXIS
ROTATION ANGLE
SIDE VIEW

Figure 30-3
An Ellipse

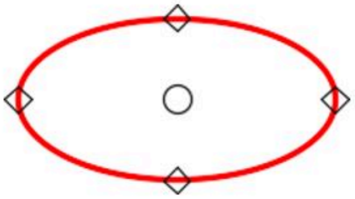
Object Snap Locations - Ellipses				
Mode	**Abbreviations**	**Icon**	**Marker**	**The AutoCAD Object**
Center	Cen	⊙	○	
Quadrant	Quad	◌	◇	

Figure 30-4
Object Snap Location for Ellipses

WORKALONG: Using the POLYGON, RECTANGLE and ELLIPSE Commands

Step 1

Start a new drawing using the template: 2D Layout English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 30-1.

Step 3

On layer: Construction, draw two horizontal and 3 vertical lines as shown in the dimensioned drawing. Where you start the drawing is not important. (Figure Step 3A and 3B)

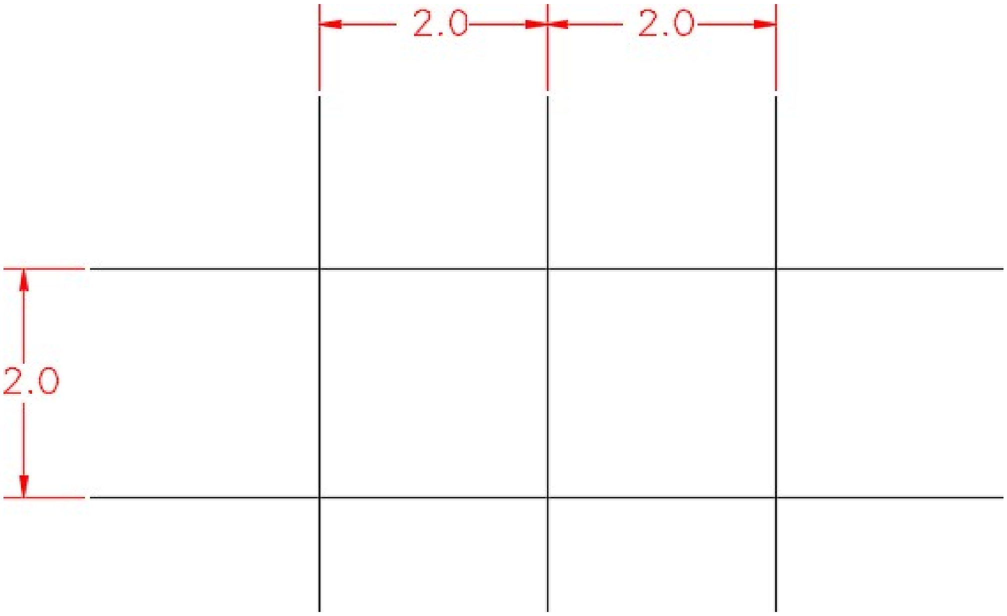


Figure Step 3A

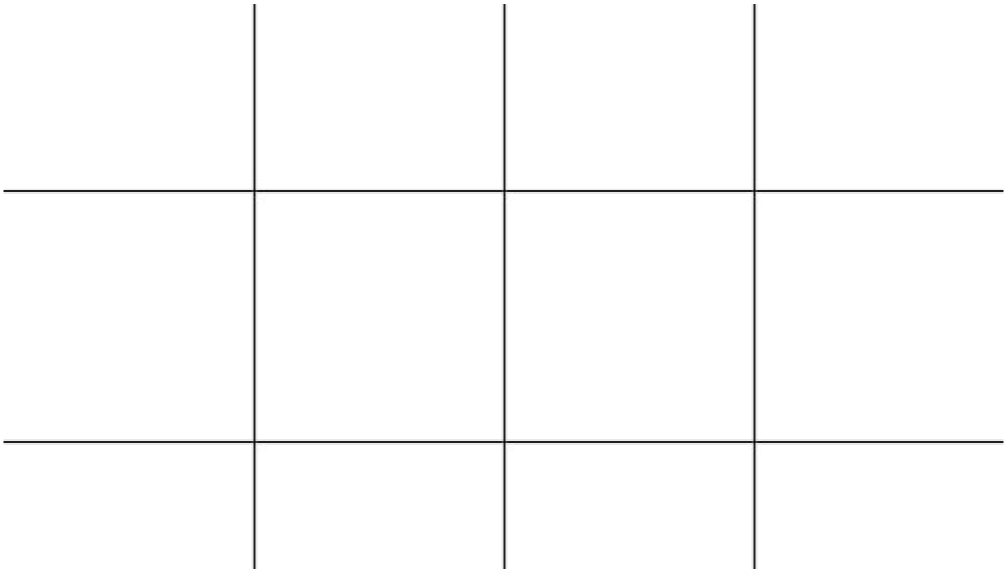


Figure Step 3B

Step 4

Set layer: Object as the current layer. Enter the POLYGON command, as shown below, to draw a hexagon with its center at the intersection of the top and left line. (Figure Step 4A and 4B)

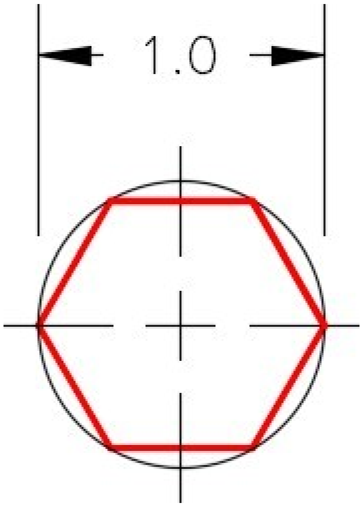


Figure Step 4A

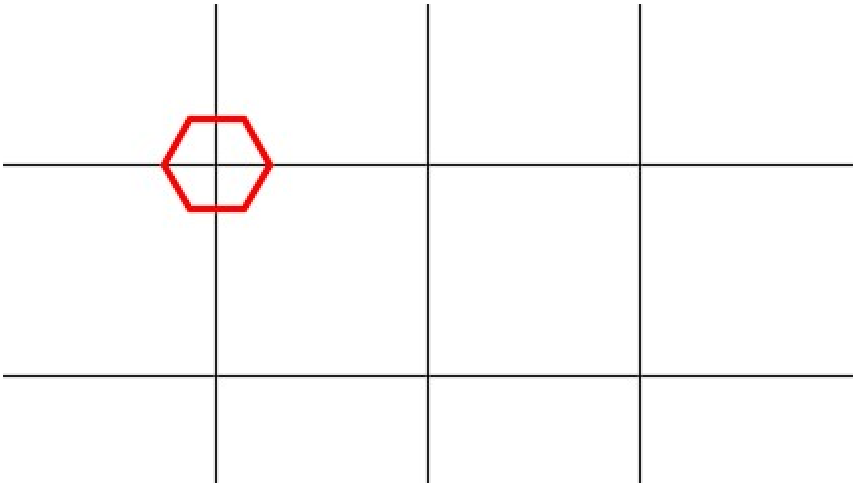


Figure Step 4B

AUTHOR'S COMMENTS: An inscribed polygon is dimensioned between two opposite points on the polygon. An circumscribed polygon is dimensioned between opposite sides.

Step 5

Enter the POLYGON command, as shown below, to draw the hexagon with its center at the intersection of the top and the center line. (Figure Step 5A and 5B)

Command: **POL**

Enter number of sides <8>: **6**

(Enter the number of sides of the polygon.)

Specify center of polygon or [Edge]: *(int)*

(Snap to the intersection of the two lines.)

Enter an option [Inscribed in circle/Circumscribed about circle] <C>: **I**

(This is an inscribed polygon since it is dimensioned to a corner.)

Specify radius of circle: **@0.5<0**

(Radius of the inscribed circle.)

Command:

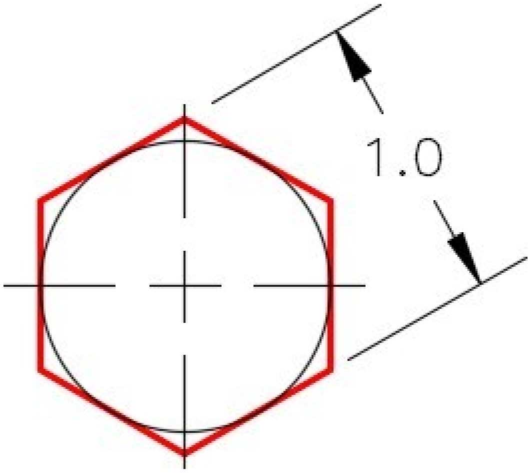


Figure Step 5A

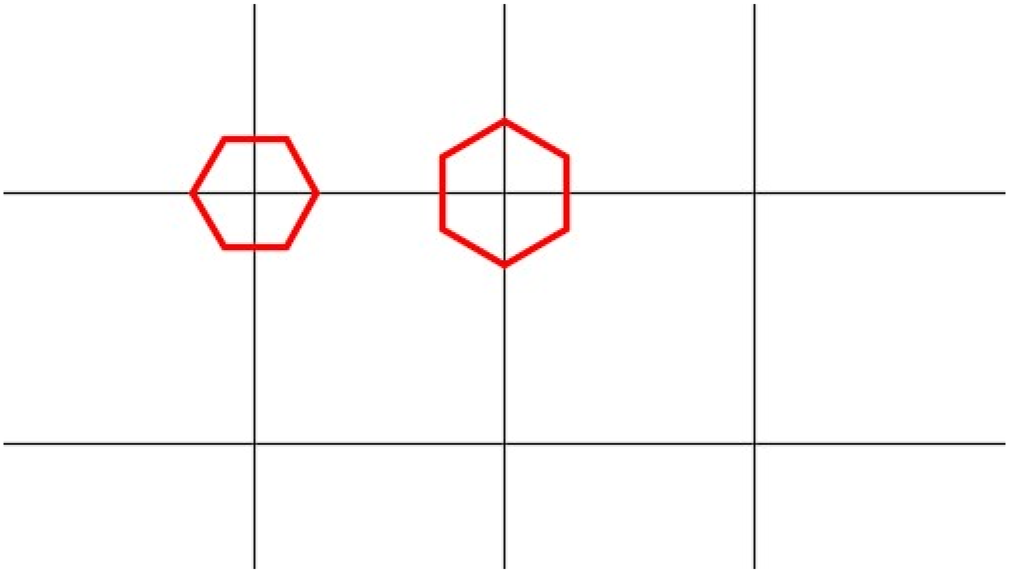


Figure Step 5B

AUTHOR'S COMMENTS: An inscribed polygon is dimensioned between two opposite points on the polygon. An circumscribed polygon is dimensioned between opposite sides.

Step 5

Enter the POLYGON command, as shown below, to draw the hexagon with its center at the intersection of the top and the center line. (Figure Step 5A and 5B)

Command: **POL**

Enter number of sides <8>: **6**

(Enter the number of sides of the polygon.)

Specify center of polygon or [Edge]: *(int)*

(Snap to the intersection of the two lines.)

Enter an option [Inscribed in circle/Circumscribed about circle] <I>: **C**

(This is a circumscribed polygon.)

Specify radius of circle: **@0.5<0**

(Radius of the circumscribed circle.)

Command:

Step 6

Enter the POLYGON command, as shown below, to draw a hexagon with its center at the intersection of the top and the right line. (Figure Step 6A and 6B)

Command: **POL**

Enter number of sides <8>: **6**

Specify center of polygon or [Edge]: *(int)*

Enter an option [Inscribed in circle/Circumscribed about circle] <I>: **I**

Specify radius of circle: **@0.5<90**

(Radius and location of the corner of the polygon.)

Command:

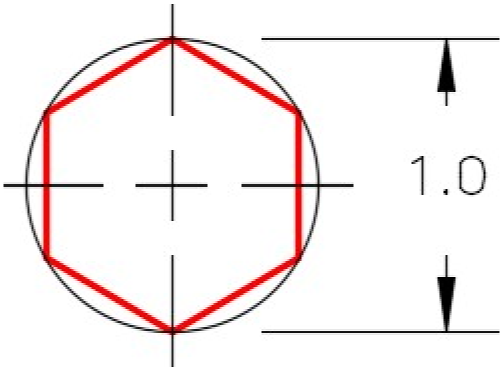


Figure Step 6A

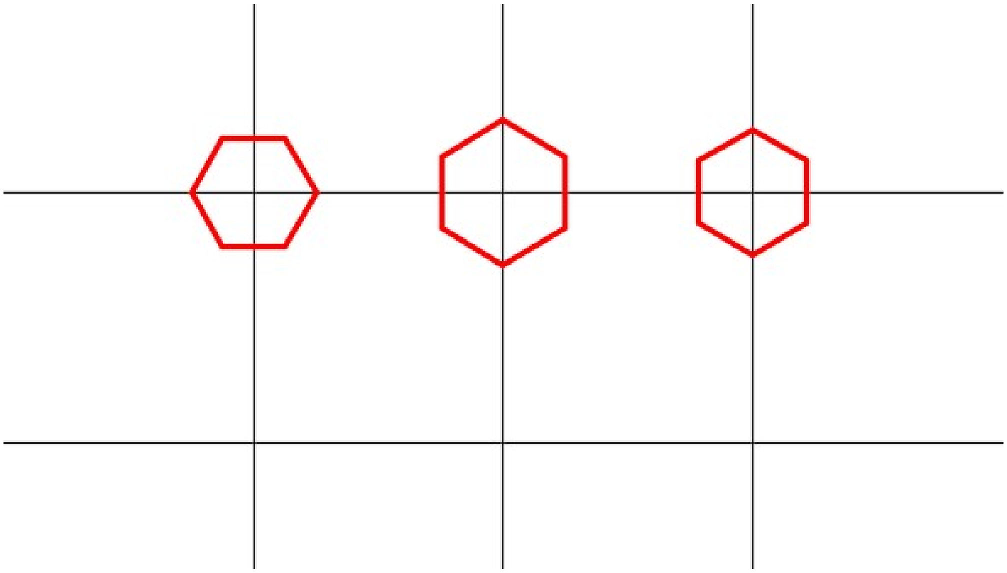


Figure Step 6B

AUTHOR'S COMMENTS: Note how the corner of a polygon can be located by entering a coordinate to locate the radius.

Step 7

Using what you learned in Steps 4 to 6, draw the three regular polygons and locate them as shown in the figure. (Figure Step 7A and 7B)

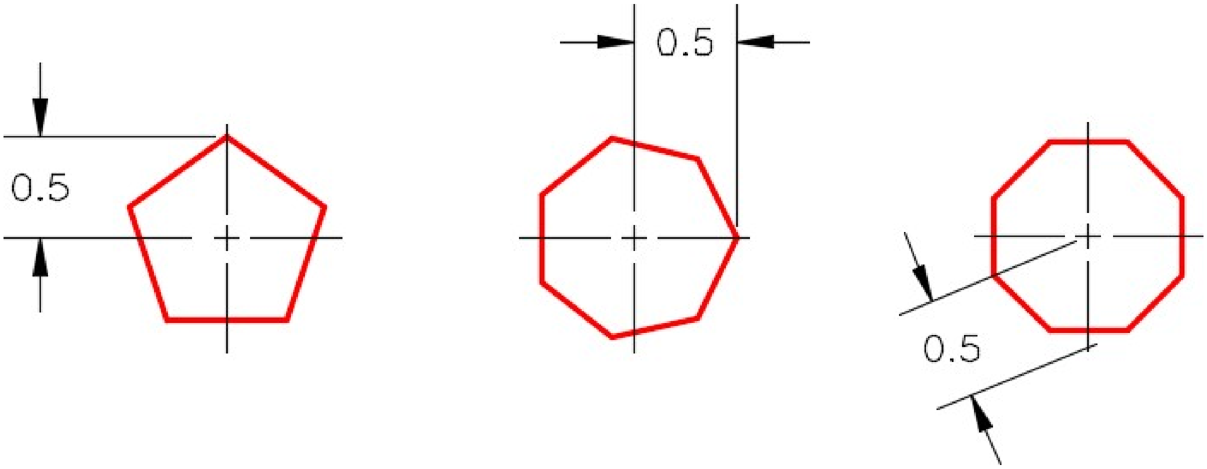


Figure Step 7A

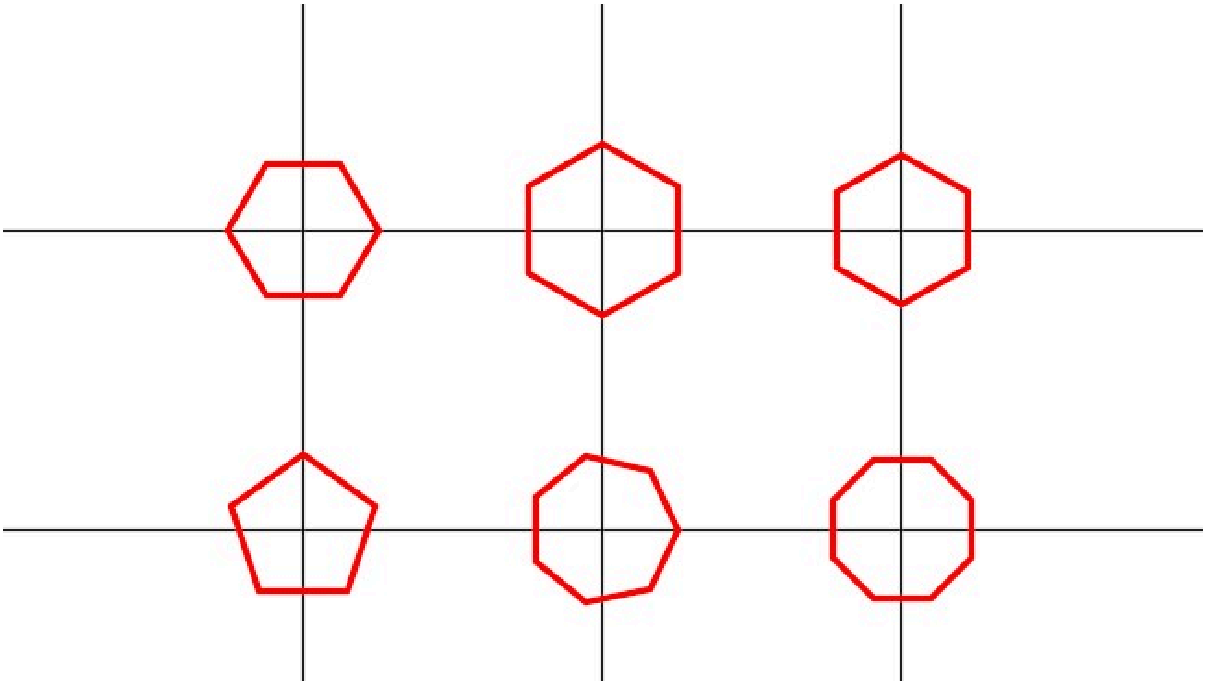


Figure Step 7B

Step 8

On layer: Object, enter the RECTANGLE command, as shown below, to draw the rectangle. (Figure Step 8)

Command: **RECTANGLE**

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **1,2.5**

Specify other corner point or [Area/Dimensions/Rotation]: **7.5,6**

Command:

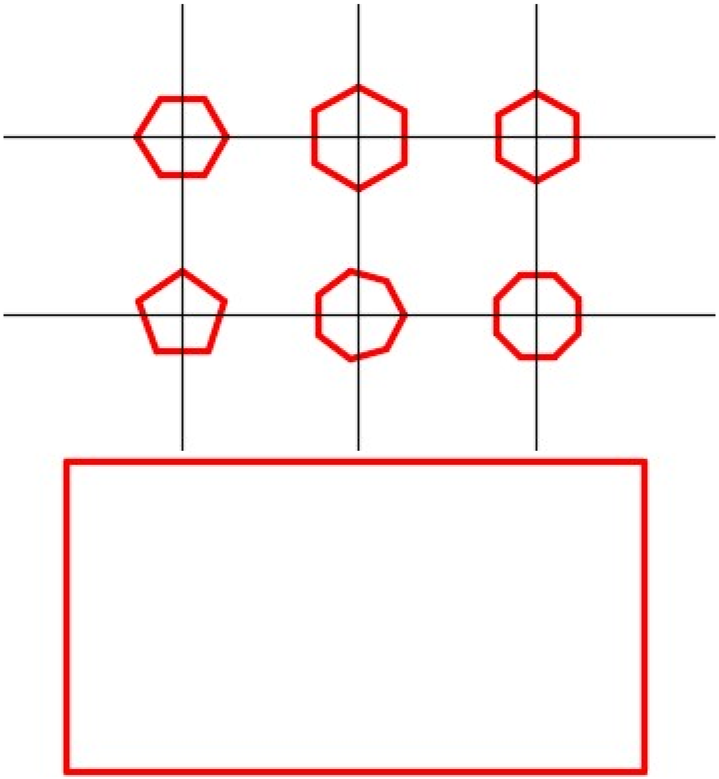


Figure Step 8

Step 9

On layer: Construction, draw diagonal lines by snapping from corner to corner of the rectangle. (Figure Step 9)

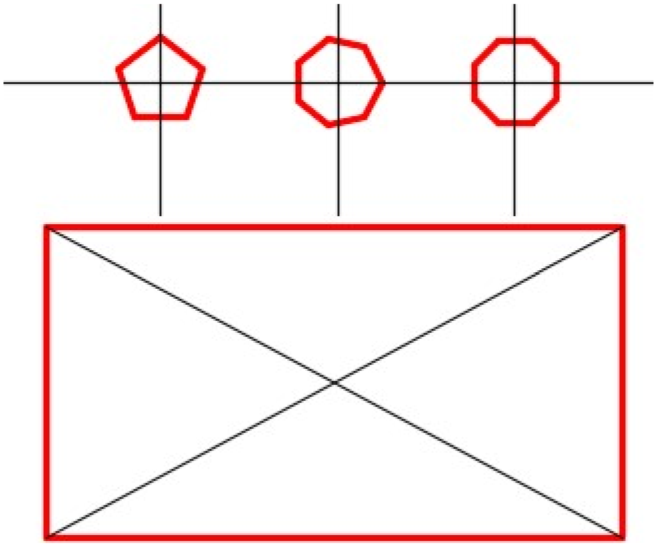


Figure Step 9

Step 10

On layer: Object, using the figure as a reference, enter the ELLIPSE command, as shown below, to draw the ellipse. (Figure Step 10A and 10B)

Command: **ELLIPSE**

Specify axis endpoint of ellipse or [Arc/Center]: **C**

Specify center of ellipse: *(int)*

(Snap to the intersection of the two diagonal lines to located the center of the ellipse.)

Specify endpoint of axis: **@2,0**

Specify distance to other axis or [Rotation]: **@0,1**

Command:

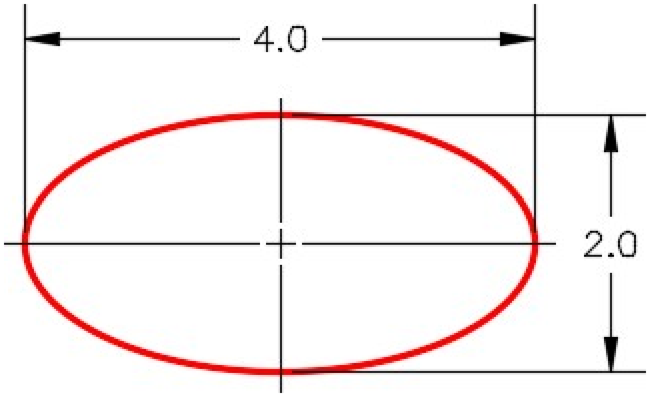


Figure Step 10A

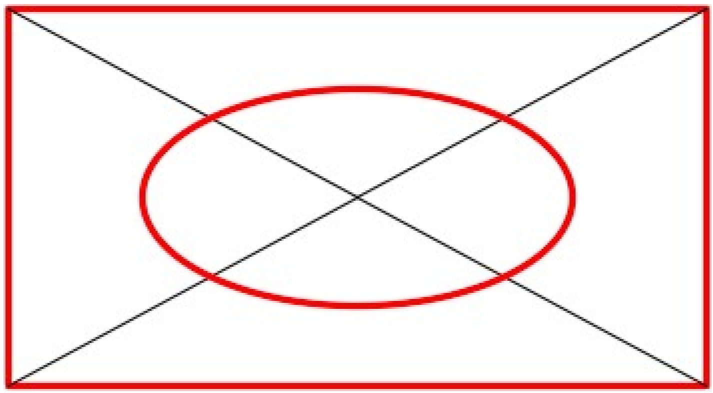


Figure Step 10B

Step 11

Trim the construction lines inside the ellipse. (Figure step 11)

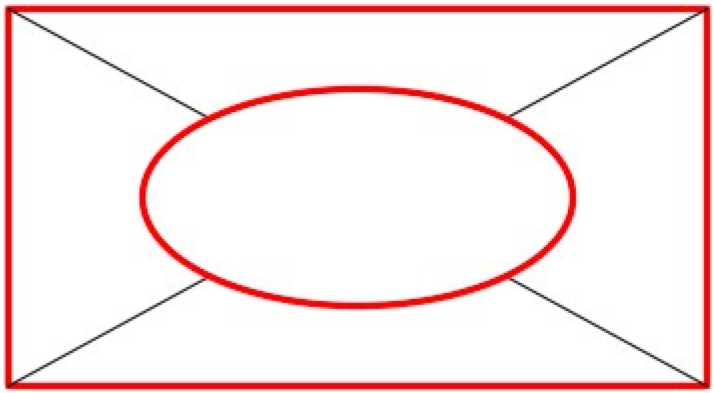


Figure Step 11

Step 12

Using what you learned earlier in this workalong, on layer: Object, draw the four polygons shown in the figure. Locate the centers of the polygons at the midpoint of each construction line. (Figure Step 12A and 12B)

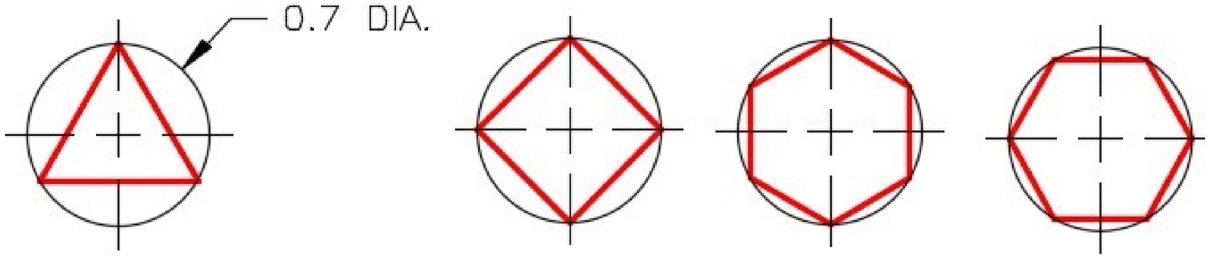


Figure Step 12A

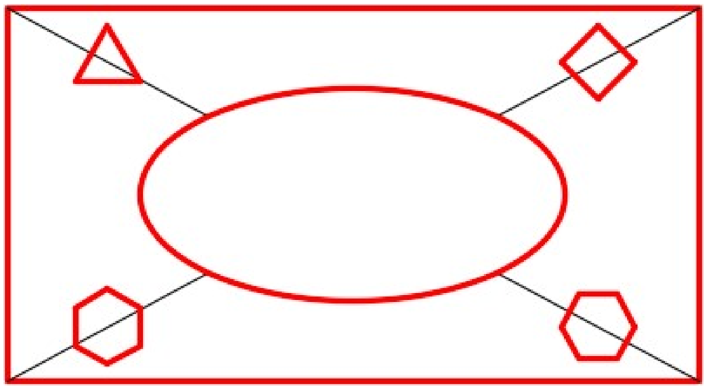


Figure Step 12B

Step 13

Turn layer: Construction off. Enter the POLYGON command, as shown below, to draw a hexagon using the Edge method. (Figure 13A and 13B)

```
Command: POL

POLYGON Enter number of sides <8>: 6

Specify center of polygon or [Edge]: E

Specify first endpoint of edge: (end) P1

Specify second endpoint of edge: (end) P2

Command:
```

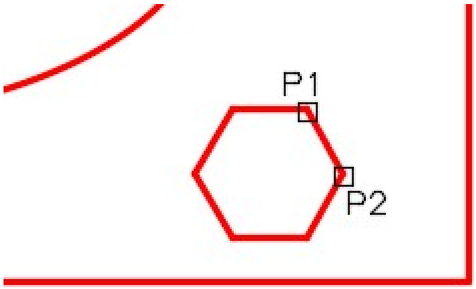


Figure Step 13A

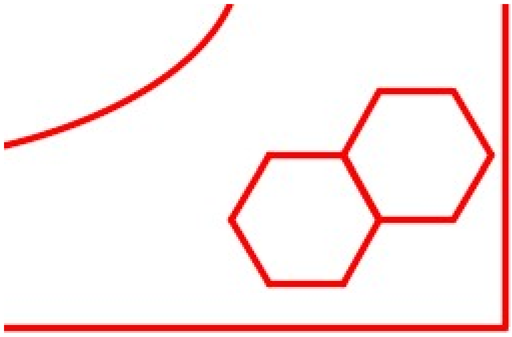


Figure Step 13B

Step 14

Explode both polygons and delete the lines they share. (Figure Step 14)

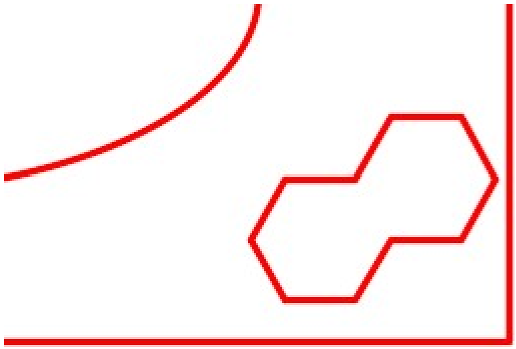
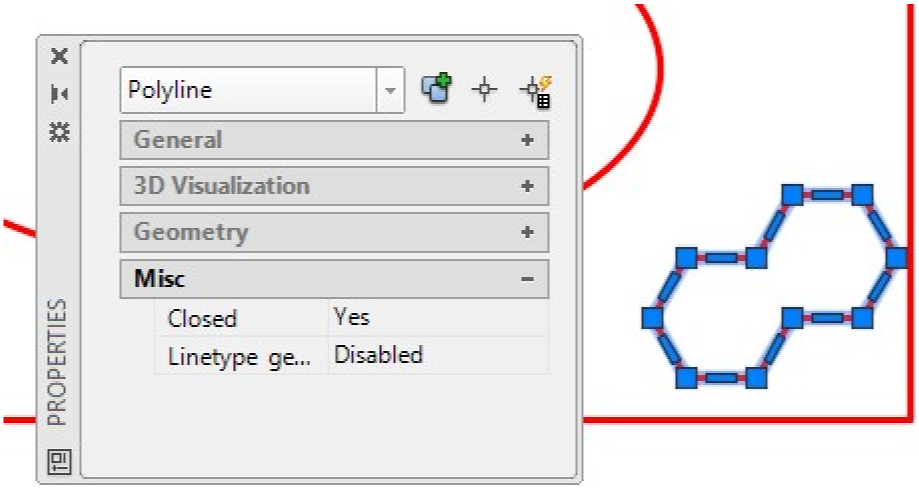


Figure Step 14

Step 15

Using what you learned in Module 29, use the PEDIT command to create a closed polyline. Using the Properties window, confirm that the polygon you just created is closed. (Figure Step 15)



Polyline

Closed: Yes

Linetype ge...: Disabled

Figure Step 15

Step 16

The bottom section of your drawing should appear as shown in the figure. (Figure Step 16)

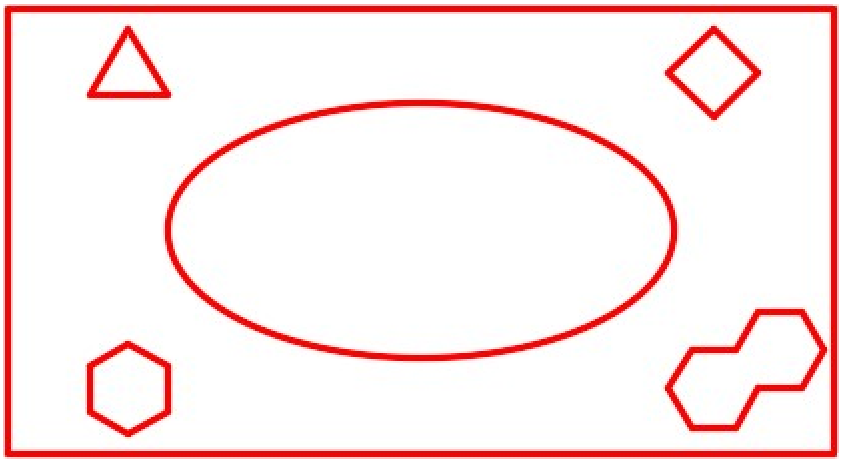
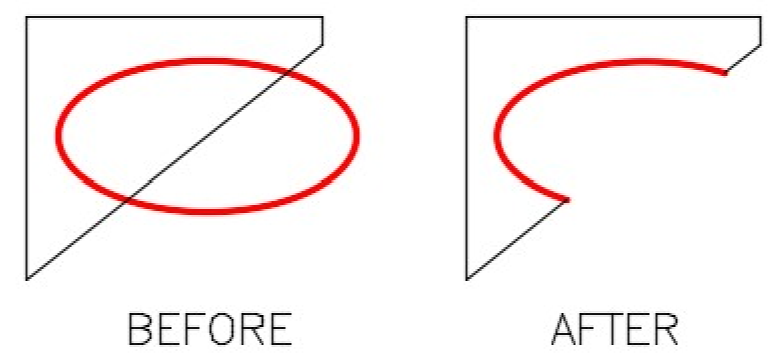


Figure Step 16

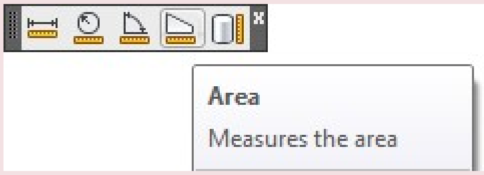
USER TIP: While the ELLIPSE command can be used to draw elliptical arcs, it is much easier to draw full ellipses and then trim or break them to the desired size.


BEFORE
AFTER

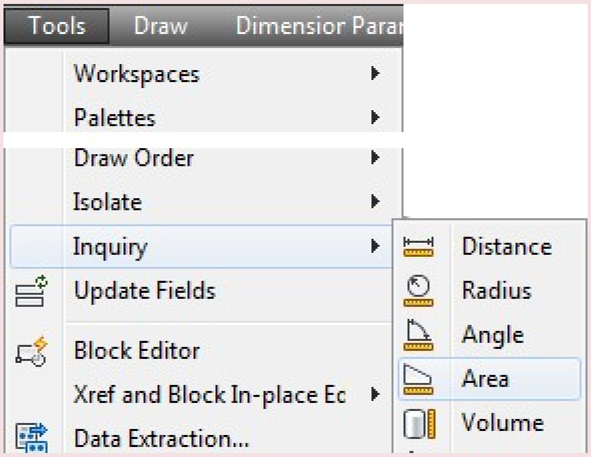
AutoCAD Command: AREA

The AREA command is used to find the area or perimeter of a series of coordinates, a closed polyline, circle or ellipse.

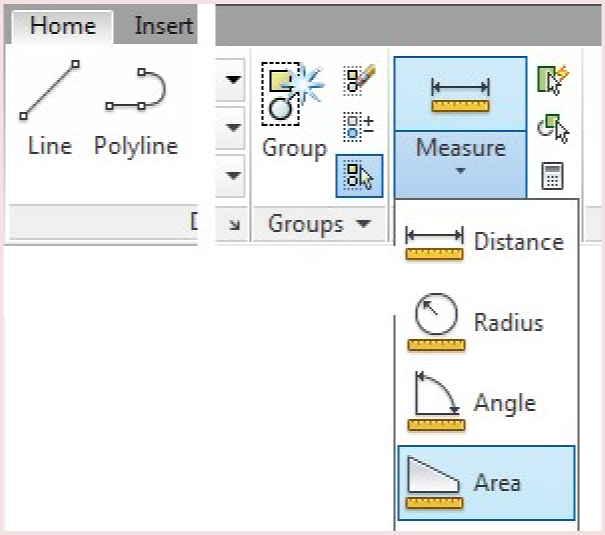
Shortcut: **none**


Area
Measures the area

Measurement Tools Toolbar


Tools
Draw
Dimension Para
Workspaces
Palettes
Draw Order
Isolate
Inquiry
Update Fields
Block Editor
Xref and Block In-place Ed
Data Extraction...
Distance
Radius
Angle
Area
Volume

Tools Pull-down


Home
Insert
Line
Polyline
Group
Groups
Measure
Distance
Radius
Angle
Area

Home Ribbon

MUST KNOW: The EXPLODE command converts an open or closed polyline back into their individual line objects. When exploded, polylines will lose their assigned width.

WORKALONG: Using the AREA Command

Step 1

Open the drawing: AutoCAD 2D Workalong 30-1. (Figure Step 1)

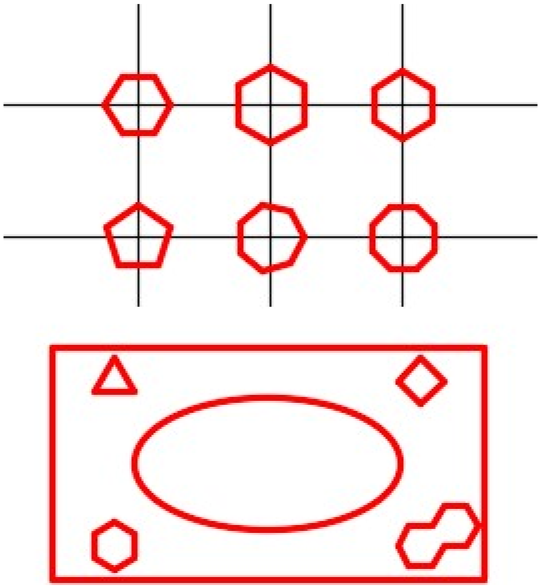


Figure Step 1

Step 2

Enter the AREA command, as shown below, to measure the area of the rectangle using the Corner Point method. (Figure Step2)

Command: **AREA**

Specify first corner point or [Object/Add/Subtract]: *(end)* **P1**

Specify next corner point or press ENTER for total: *(end)* **P2**

Specify next corner point or press ENTER for total: *(end)* **P3**

Specify next corner point or press ENTER for total: *(end)* **P4**

Specify next corner point or press ENTER for total: *(press Enter)*

(You do not have to snap back to the start point. The AREA command will go back to the first point and calculate the area and perimeter.)

Area = 22.7500, Perimeter = 20.0000

Command:

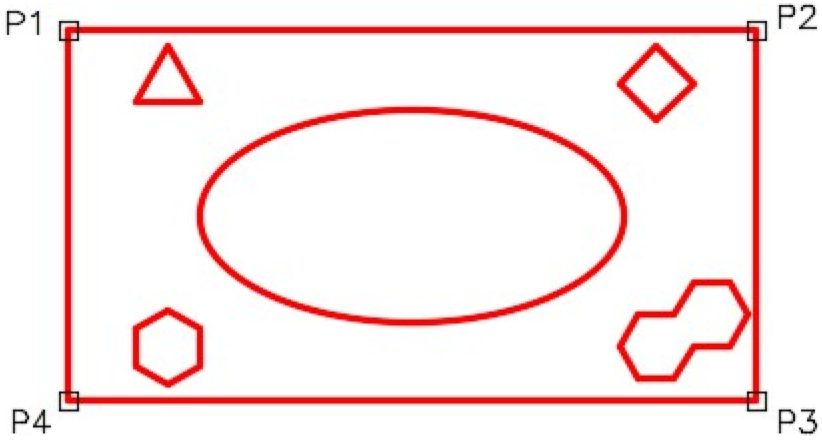


Figure Step 2

AUTHOR'S COMMENTS: This is an example of using the point method of finding the area and the perimeter of a series of coordinate locations. The corner point method only works for an area containing lines. It does not work if any one of the sides is an arc.

Step 3

Enter the AREA command, as shown below, to measure the area and perimeter of the rectangle using the Object method. (Figure Step 3)

Command: **AREA**

Specify first corner point or [Object/Add/Subtract]: **O**

(Using the Object method.)

Select objects: **P5**

(Select the object. It must be a closed polygon, a circle or an ellipse.)

Area = 22.7500, Perimeter = 20.0000

(This answer will match the results found in Step 2)

Command:

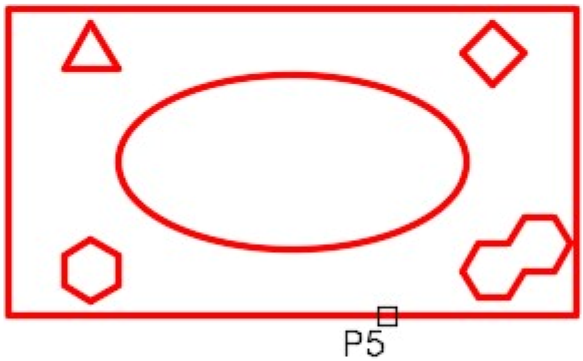


Figure Step 3

Step 4

Enter the AREA command, as shown below, to measure the area of the rectangle subtracting objects from the overall area. (Figure Step 4)

Command: **AREA**

Specify first corner point or [Object/Add/Subtract]: **A**

(Specifying you want to be in Add mode.)

Specify first corner point or [Object/Subtract]: **O**

(ADD mode) Select objects: **P6**

(Add in the rectangle area.)

Area = 22.7500, Perimeter = 20.0000

Total area = 22.7500

(ADD mode) Select objects:

(Press enter to end the Add mode)

```
Specify first corner point or [Object/Subtract]: S

(Beginning the Subtract mode.)

Specify first corner point or [Object/Add]: O

(SUBTRACT mode) Select objects: P7

Area = 0.1591, Perimeter = 1.8187

Total area = 22.5909

(SUBTRACT mode) Select objects: P8

Area = 0.3183, Perimeter = 2.1000

Total area = 22.2726

(SUBTRACT mode) Select objects: P9

Area = 6.2832, Perimeter = 9.6884

Total area = 15.9894

(SUBTRACT mode) Select objects: P10
```

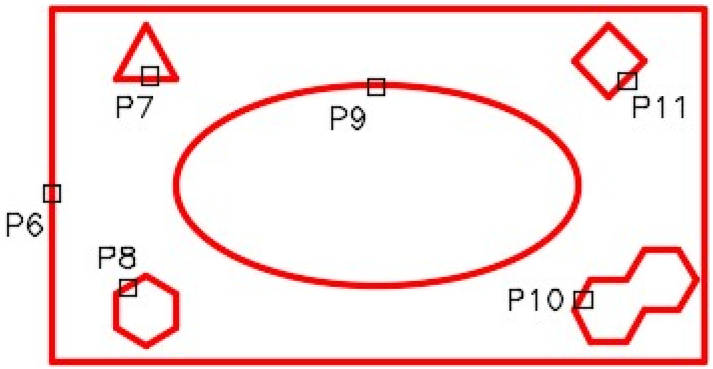


Figure Step 4

```
Area = 0.6365, Perimeter = 3.5000

Total area = 15.3529

(SUBTRACT mode) Select objects: P11

Area = 0.2450, Perimeter = 1.9799

Total area = 15.1079

(SUBTRACT mode) Select objects:
```

(Press Enter to end the Subtract mode.)

Total area = 15.1079

(The area of the overall object minus the 5 inner objects is 15.1079 square inches.)

Command:

Step 5

Enter the AREA command, as shown below, to measure and add the areas of three individual objects. (Figure Step 5)

Command: **AREA**

Specify first corner point or [Object/Add/Subtract]: **A**

(Enter the Add mode.)

Specify first corner point or [Object/Subtract]: **O**

(ADD mode) Select objects: **P12**

Area = 0.1591, Perimeter = 1.8187

Total area = 0.1591

(ADD mode) Select objects: **P13**

Area = 0.2450, Perimeter = 1.9799

Total area = 0.4041

(ADD mode) Select objects: **P14**

Area = 0.6365, Perimeter = 3.5000

Total area = 1.0407

(The total area of the three objects combined is 1.0407 square inches.)

Command:

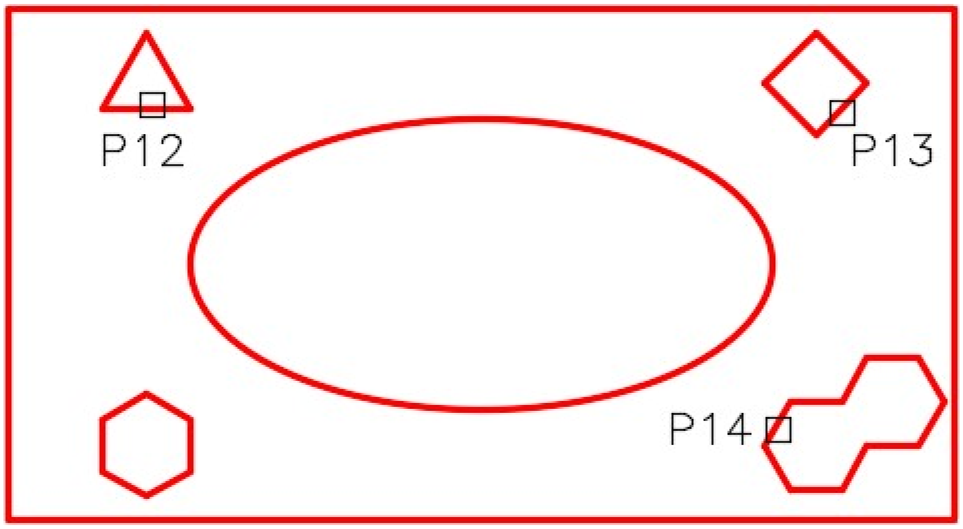
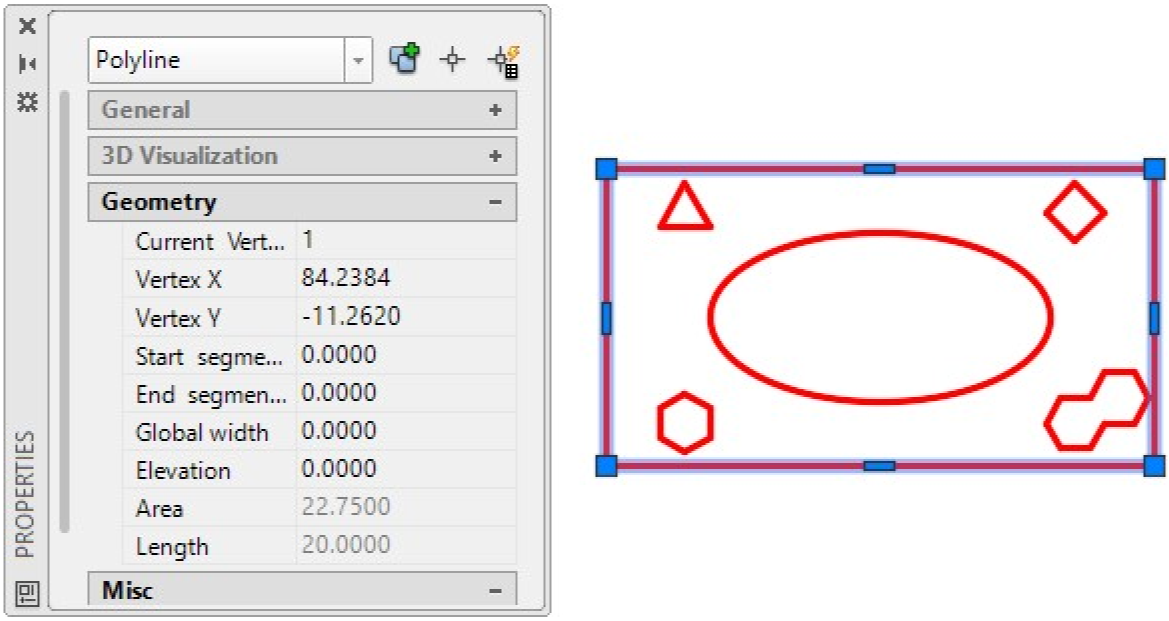


Figure Step 5

AUTHOR'S COMMENTS: The area and perimeter can also be obtained by using the Properties window. The areas to be measured must be closed polygons.

Step 6

Open the Properties window and select the large rectangle. The area and the perimeter (length) will display. (Figure Step 6)

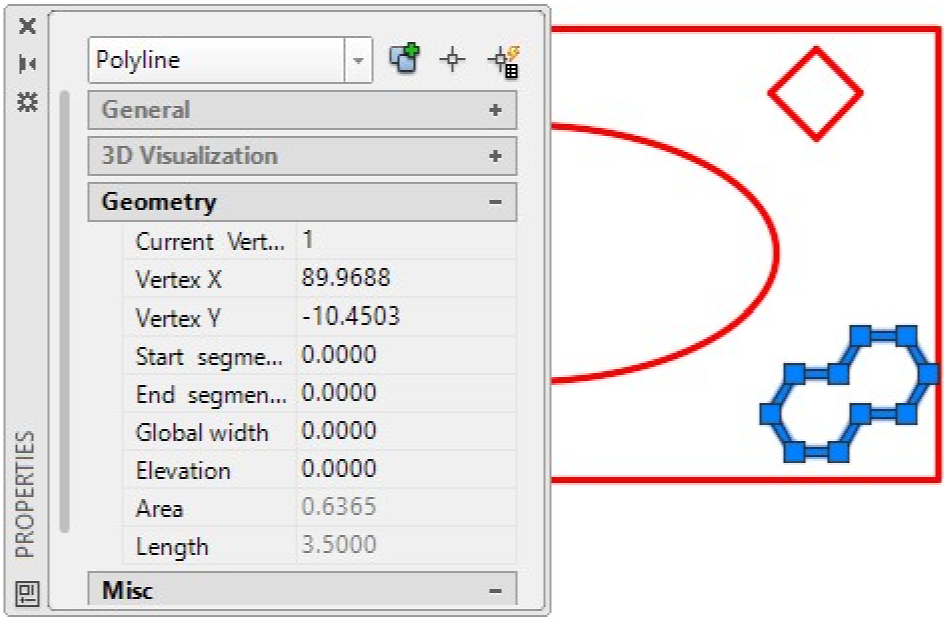

PROPERTIES
Polyline
General
3D Visualization
Geometry
Misc

Current Vert...	1
Vertex X	84.2384
Vertex Y	-11.2620
Start segme...	0.0000
End segmen...	0.0000
Global width	0.0000
Elevation	0.0000
Area	22.7500
Length	20.0000

Figure Step 6

Step 7

Unselect the rectangle and select the object in the bottom right. The area and the perimeter (length) will display. (Figure Step 7)



Current Vert...	1
Vertex X	89.9688
Vertex Y	-10.4503
Start segme...	0.0000
End segmen...	0.0000
Global width	0.0000
Elevation	0.0000
Area	0.6365
Length	3.5000

Figure Step 7

Geometry Lesson: Degrees Minutes and Seconds

Since a full degree is not always precise enough to measure an angle, you must be able to work to a more precise measurement. Degrees are broken down to smaller and more precise units of *minutes* and *seconds*.

1 degree is divided into 60 minutes and 1 minute is divided into 60 seconds:

1 degree = 60 minutes

1 minute = 60 seconds

Therefore, 1 degree = 3600 seconds.

Degrees are written 1E, minutes are written 01′ and seconds 01″.

Combined: 1<01’01”.

Here are some examples of adding and subtracting angles with degree minutes and seconds. Study them and then solve problems A to F below.

①

$$\begin{array}{r} 54^\circ 23' 56'' \\ +\ 34^\circ 51' 12'' \\ \hline 88^\circ 74' 68'' \end{array} \quad = 88^\circ 75' 08'' \quad = 89^\circ 15' 08''$$

②

$$\begin{array}{r} 180^\circ 00' 00'' \\ -\ 34^\circ 51' 12'' \\ \hline \end{array}$$

180 DEGREES IS THE
SAME AS 179°59'60".

$$\begin{array}{r} 179^\circ 59' 60'' \\ -\ 34^\circ 51' 12'' \\ \hline 145^\circ 08' 48'' \end{array}$$

③

$$\begin{array}{r} 32^\circ 12' 08'' \\ -\ 14^\circ 51' 34'' \\ \hline \end{array} \quad = \quad \begin{array}{r} 31^\circ 71' 68'' \\ -\ 14^\circ 51' 34'' \\ \hline 17^\circ 20' 34'' \end{array}$$

Adding and Subtracting Degrees Minutes Seconds

Solve the following problems of adding and subtracting degrees, minutes and seconds. The answers are shown below.

Ⓐ

$$\begin{array}{r} 62^\circ 51' 12'' \\ +\ 42^\circ 31' 57'' \\ \hline \end{array}$$

Ⓑ

$$\begin{array}{r} 12^\circ 25' 47'' \\ +\ 32^\circ 41' 38'' \\ \hline \end{array}$$

Ⓒ

$$\begin{array}{r} 218^\circ 19' 45'' \\ +\ 38^\circ 22' 59'' \\ \hline \end{array}$$

Ⓓ

$$\begin{array}{r} 68^\circ 56' 11'' \\ -\ 42^\circ 31' 57'' \\ \hline \end{array}$$

Ⓔ

$$\begin{array}{r} 180^\circ 00' 00'' \\ -\ 32^\circ 41' 38'' \\ \hline \end{array}$$

Ⓕ

$$\begin{array}{r} 218^\circ 19' 45'' \\ -\ 112^\circ 59' 59'' \\ \hline \end{array}$$

Answers to Adding and Subtracting Degrees Minutes Seconds:

A 105E23’09”

B 45E07’25”

C 256E42’44”

D 26E24’14”

E 147E18’22”

F 105E19’46”

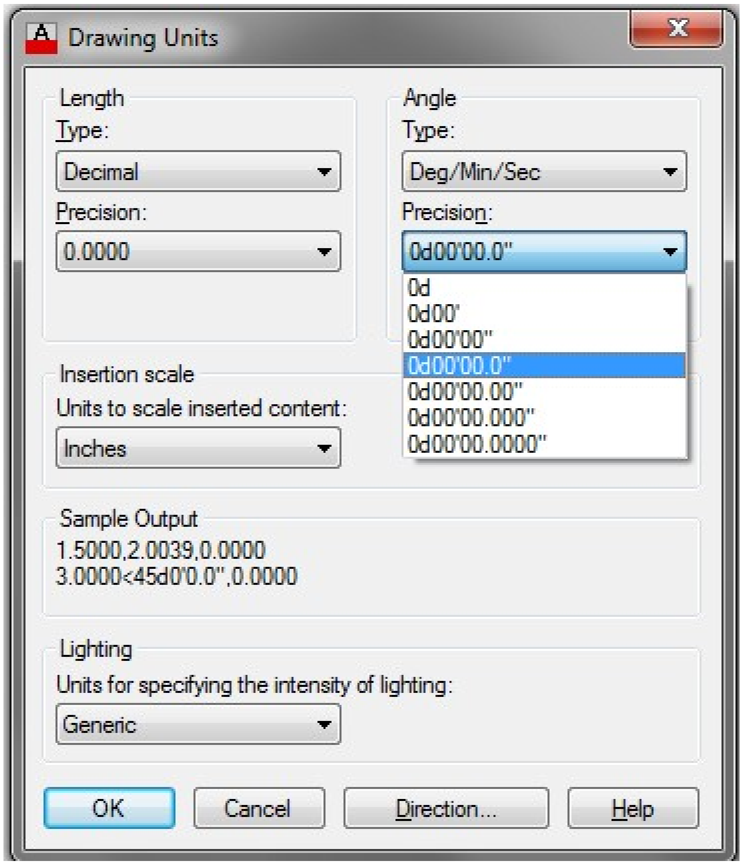
Working in Degrees Minutes and Seconds

Up to this point in the book, all of the angles have been entering using degrees and decimals of a degree. The values of degrees can also be entered using degree/minutes/seconds.

The Drawing Units dialogue box must be set to report angular units and precision in degree/minutes/seconds. The Angle Type must be set to Deg/Min/Sec and the Precision set to the value that you want it reported. See Figure 30-5.

Since there is no degree symbol on the keyboard, enter the letter ‘ d ‘ after the number when entering degrees in response to an AutoCAD prompt.

For example: 34d



Drawing Units

Length

Type: Decimal

Precision: 0.0000

Angle

Type: Deg/Min/Sec

Precision: 0d00'00.0"

Insertion scale

Units to scale inserted content: Inches

Sample Output

1.5000,2.0039,0.0000
3.0000<45d0'0.0",0.0000

Lighting

Units for specifying the intensity of lighting: Generic

Figure 30-5
Setting the Angle Type and Precision

To specify the number of minutes, enter a ‘ (single quote) after the number and to enter seconds, enter a ” (double quote) after the number. See Figure 30-6.

For example: 34d56’32” Command: **LINE**

```
Specify first point: 2,2

Specify next point or [Undo]: @2<34d56’32”

Specify next point or [Undo]:

Command:
```

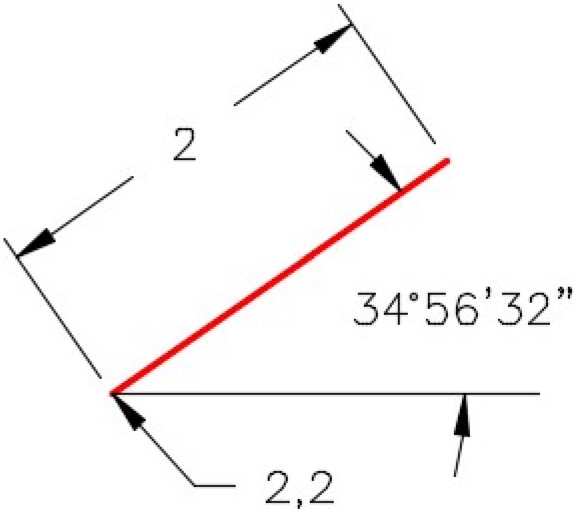


Figure 30-6
An Angle Using Deg/Min/Sec

Object Snap Locations - Geometric Center

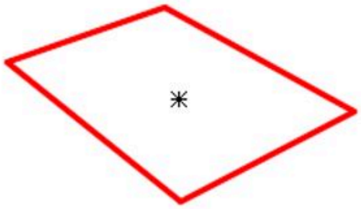
Mode	Abbreviations	Icon	Marker	The AutoCAD Object
Geometric Center	N/A	○	✳	

Figure 30-7
Object Snap Location for Geometric Center
Note: Geometric Center osnap mode snaps to the centroid of any closed polyline.

Key Principles

Key Principles in Module 30

1. A regular polygon is a closed polyline having between 3 and 1,024 equal-length sides and is drawn using the POLYGON command.
2. The EXPLODE command converts a polygon made from an open or closed polyline back into its individual line objects. When exploded, polylines will lose their assigned widths.

Lab Exercise 30-1

Time allowed: 60 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 30-1	2D Layout English	Inches

Step 1

On layer: Object, draw the object shown in the figures. (Figure Step 1A and 1B)

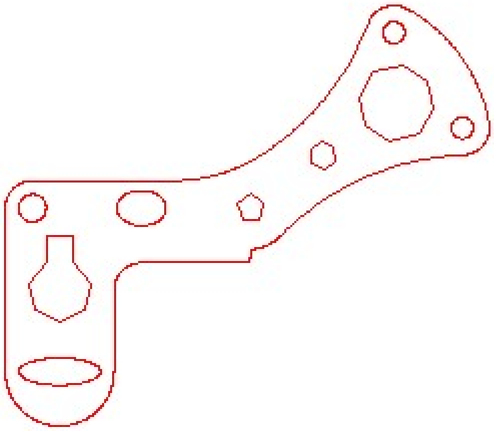


Figure Step 1A
Completed Drawing

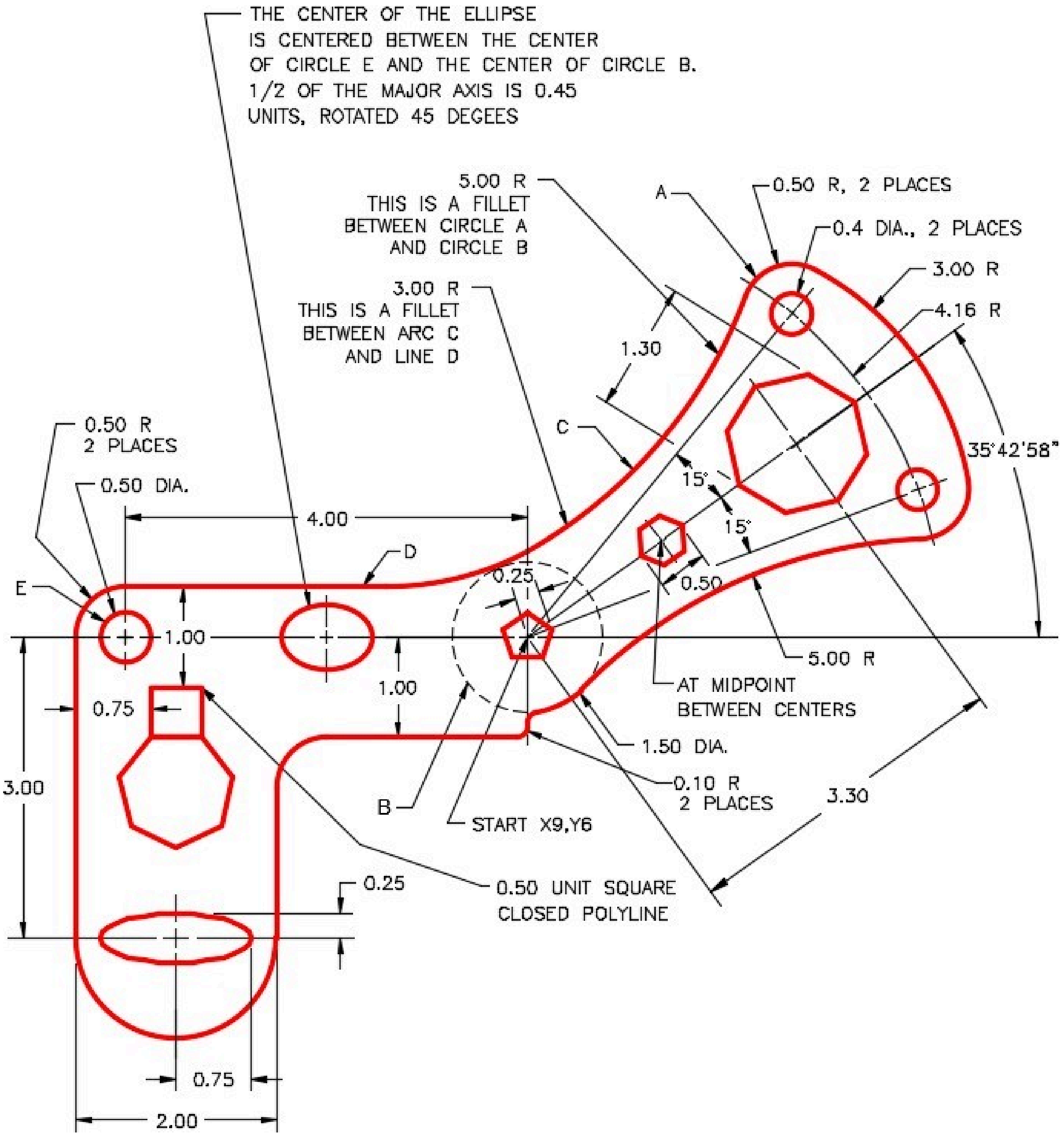

THE CENTER OF THE ELLIPSE IS CENTERED BETWEEN THE CENTER OF CIRCLE E AND THE CENTER OF CIRCLE B. 1/2 OF THE MAJOR AXIS IS 0.45 UNITS, ROTATED 45 DEGEES
5.00 R THIS IS A FILLET BETWEEN CIRCLE A AND CIRCLE B
A
0.50 R, 2 PLACES
0.4 DIA., 2 PLACES
3.00 R
4.16 R
3.00 R THIS IS A FILLET BETWEEN ARC C AND LINE D
1.30
C
35°42'58"
0.50 R 2 PLACES
0.50 DIA.
15°
15°
4.00
D
0.25
0.50
E
1.00
5.00 R
AT MIDPOINT BETWEEN CENTERS
1.00
0.75
1.50 DIA.
3.00
0.10 R 2 PLACES
3.30
B
START X9,Y6
0.50 UNIT SQUARE CLOSED POLYLINE
0.25
0.75
2.00

Figure Step 1B

Step 2

On layer: Boundary, using the PEDIT command, create a closed polygon of the outside outline of the object. (Figure Step 2)

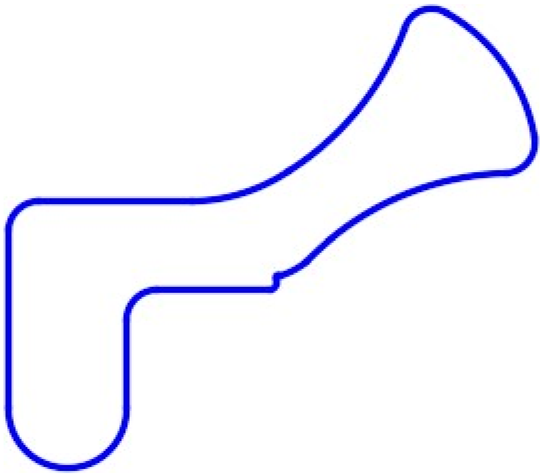


Figure Step 2

Step 3

To the precision of 4 decimal places, find the area and perimeter of the closed polygon created in Step 2.

A Area = ____________________

B Perimeter = ____________________

Step 4

Change the square and the heptagon to appear as shown in the figure. On layer Boundary, using the BOUNDARY command, create a closed polygon of the new shape. (Figure Step 4)

Step 5

To the precision of 6 decimal places, find the area and perimeter of the closed polygon created in Step 4.

A Area = ____________________

B Perimeter = ____________________

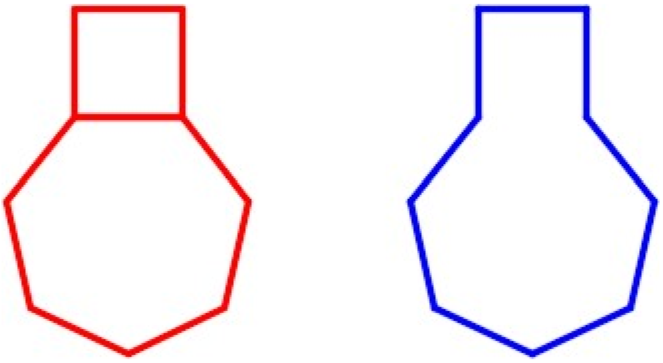


Figure Step 4

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hint. If you get stuck and cannot complete it on your own, use the following hint to help you.

Hint 1

To draw the 3 radius tangent arc on the top right side of the object, you must draw a tangent circle and then trim it. The FILLET command cannot be used as it will draw the fillet on the inside. Use the CIRCLE command and the TTR option to insert the tangent circle and then trim it. (Figure Hint 1)

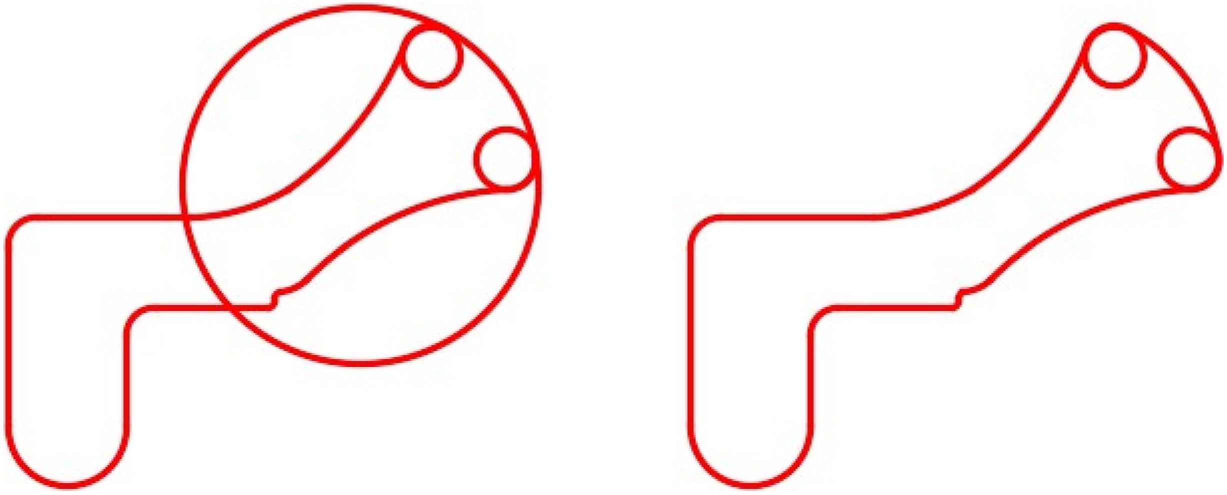


Figure Hint 1

Lab Exercise 30-2

Time allowed: 60 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 30-2	2D Layout English	Feet

Step 1

Start a new drawing using the template shown above. (Figure Step 1)

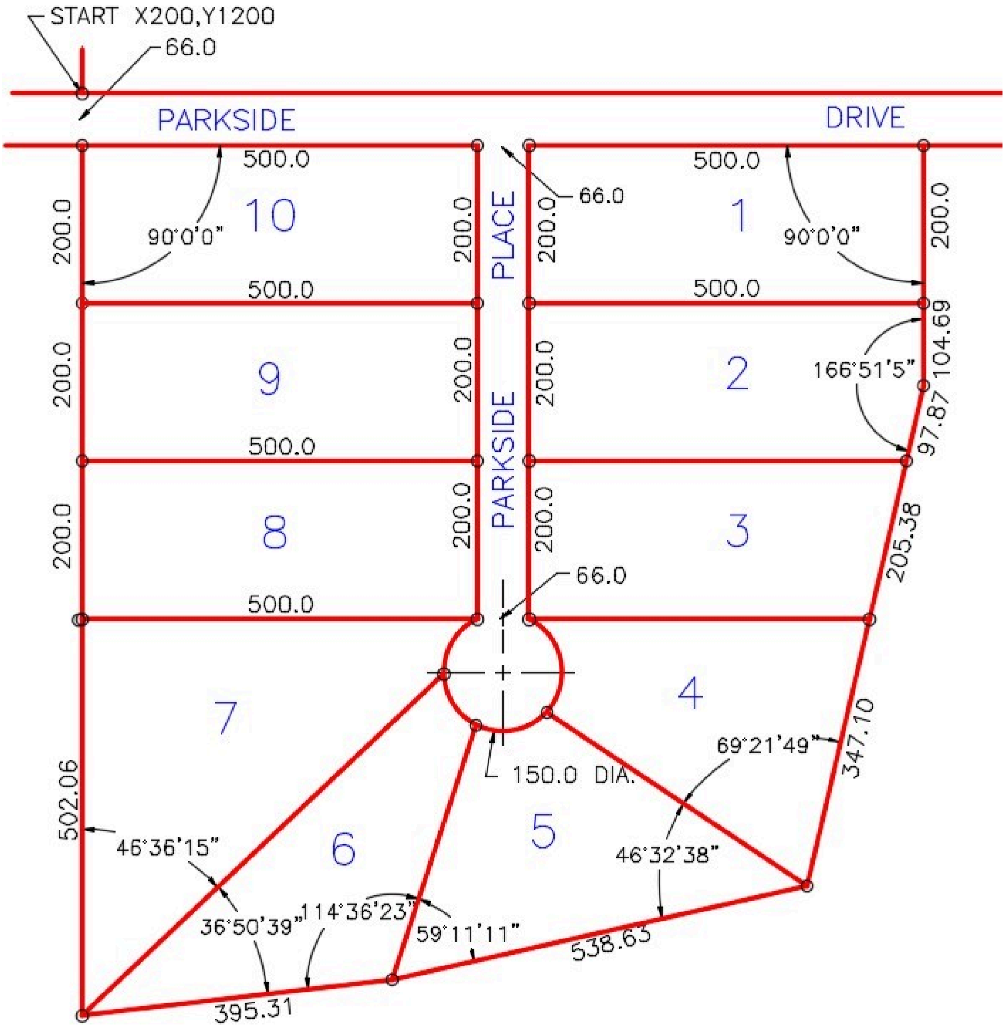
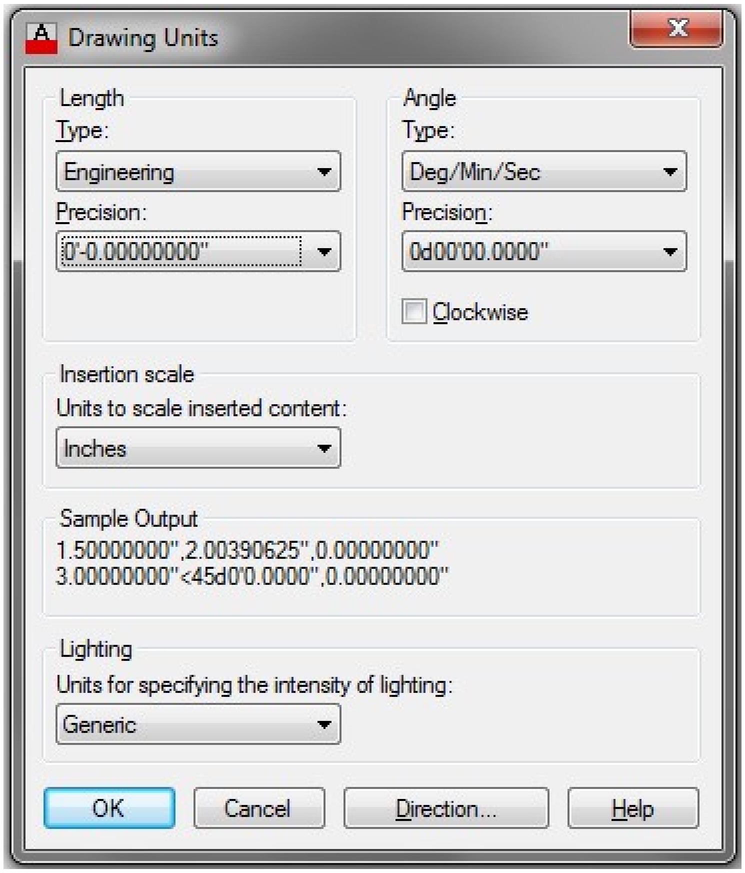

START X200,Y1200
66.0
PARKSIDE
DRIVE
500.0
500.0
10
1
90°0'0"
90°0'0"
200.0
200.0
200.0
200.0
PLACE
66.0
500.0
500.0
9
2
200.0
200.0
200.0
200.0
166°51'5"
104.69
97.87
500.0
PARKSIDE
8
3
200.0
200.0
200.0
205.38
66.0
500.0
7
4
150.0 DIA.
69°21'49"
347.10
502.06
46°36'15"
6
5
46°32'38"
36°50'39"
114°36'23"
59°11'11"
538.63
395.31

Figure Step 1
All Dimensions in are Feet

Step 2

Using the UNITS command, set the Length type to Engineering, the Angle type to Deg/Min/Sec and Units to scale inserted content: to Inches, as shown in the figure. (Figure Step 2)



Drawing Units

Length

Type: Engineering

Precision: 0'-0.00000000"

Angle

Type: Deg/Min/Sec

Precision: 0d00'00.0000"

☐ Clockwise

Insertion scale

Units to scale inserted content: Inches

Sample Output

1.50000000",2.00390625",0.00000000"
3.00000000"<45d0'0.0000",0.00000000"

Lighting

Units for specifying the intensity of lighting: Generic

Figure Step 2

AUTHOR’S COMMENTS: You are going to be working in the units of feet and decimals of a foot in this drawing. Keep in mind that AutoCAD always wants to work in inches, therefore, all numbers that you input must be followed by the foot mark character. i.e. 500.0′

Step 3

On layer: Lot Lines, draw the first line of the model, as shown below: (Figure Step 3A and 3B)

Command: **LINE**

Specify first point: **200.0′,1200.0‘**

(Make sure that all numbers are followed by the ‘ character.)

Specify next point or [Undo]: **@1200.0′,0′**

```
Specify next point or [Undo]:

Command: ZOOM

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: E

Regenerating model.

Command:
```

Figure Step 3A

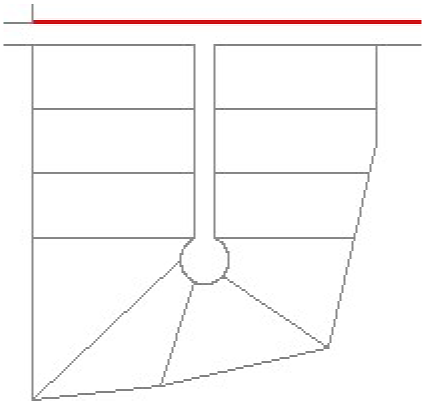


Figure Step 3B

AUTHOR'S COMMENTS: The first line you draw will not appear in the current zoomed state of the drawing. After you draw the line, you have to enter the ZOOM Extents command so that you can view it and complete the drawing.

Step 4

On layer: Lot Lines, complete the model using the dimensioned drawing shown in Step 1. The input must be is feet and decimals of a foot. (Figure Step 4)

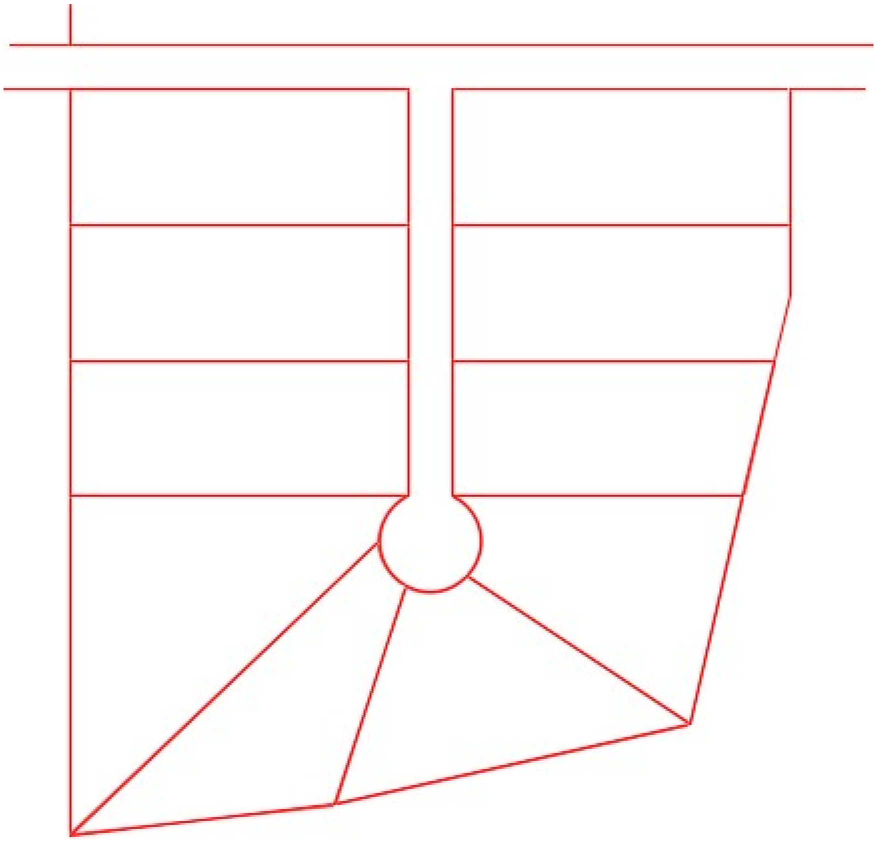


Figure Step 4

AUTHOR'S COMMENTS MINI-SURVEY PLAN LESSON: The length of the lots, on survey maps, are dimensioned by placing the length dimension parallel and along the lot line it is referring to. The length is measured from corner of the lot to corner of the lot. In Figure 30-8, the distance is 500.0 feet. The pin, small circles at the ends, have nothing to do with the dimensions. They only show where the surveyor has placed a metal pin in the ground when the survey was completed. When you draw the map, draw all the lines and then insert 10 feet diameter circles snapping their centers at the ends of the lines.

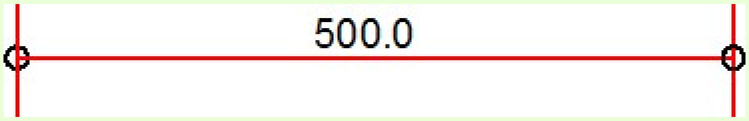


Figure 30-8

Step 5

On layer: Lot Pins, draw circles, with 10 foot diameters, to indicate the location of the survey pins. Hint: Draw one circle and copy it to the other locations. (Figure Step 5)

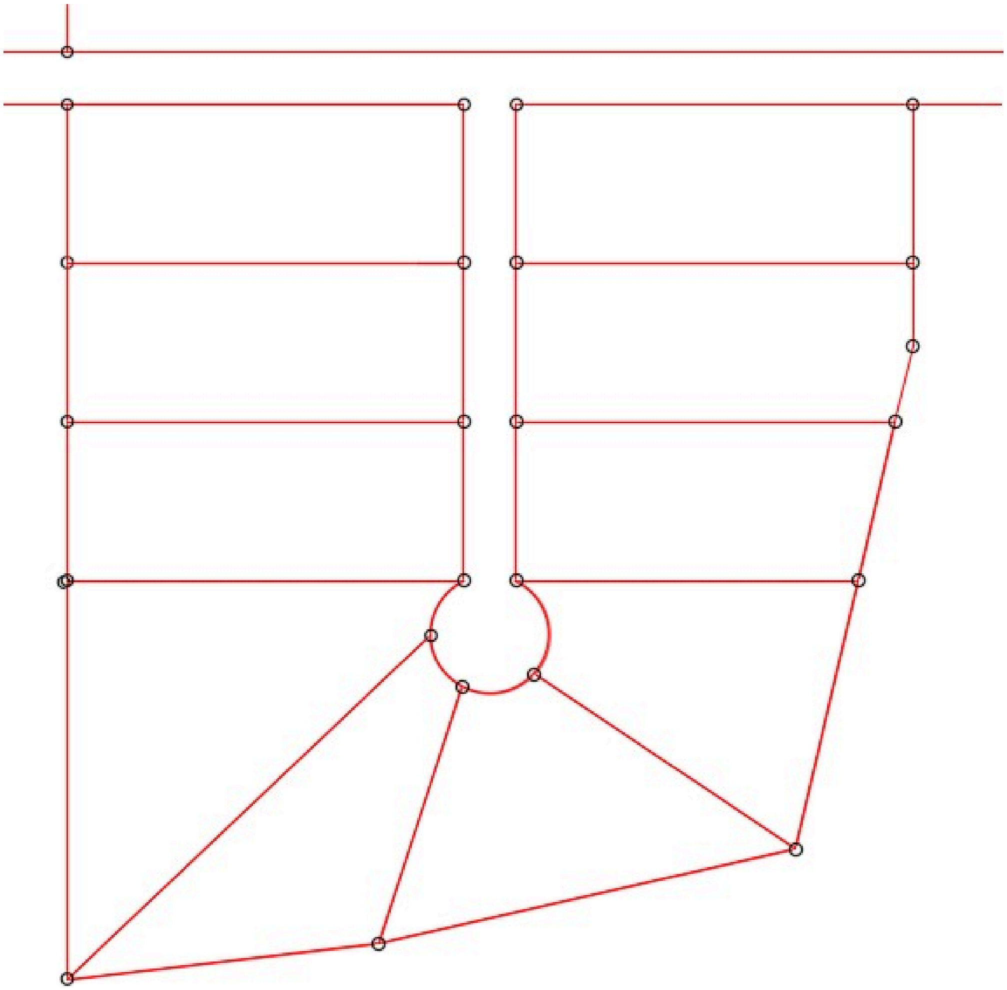


Figure Step 5

Step 6

With INSUNITS set to Inches, insert the key to check your model for accuracy.

Step 7

Enable layout: Module Layout C. On layer: Viewport, create a viewport as shown in the figure. Set the scale to 1:1200 or 0.000833 and lock the display. (Figure Step 7)

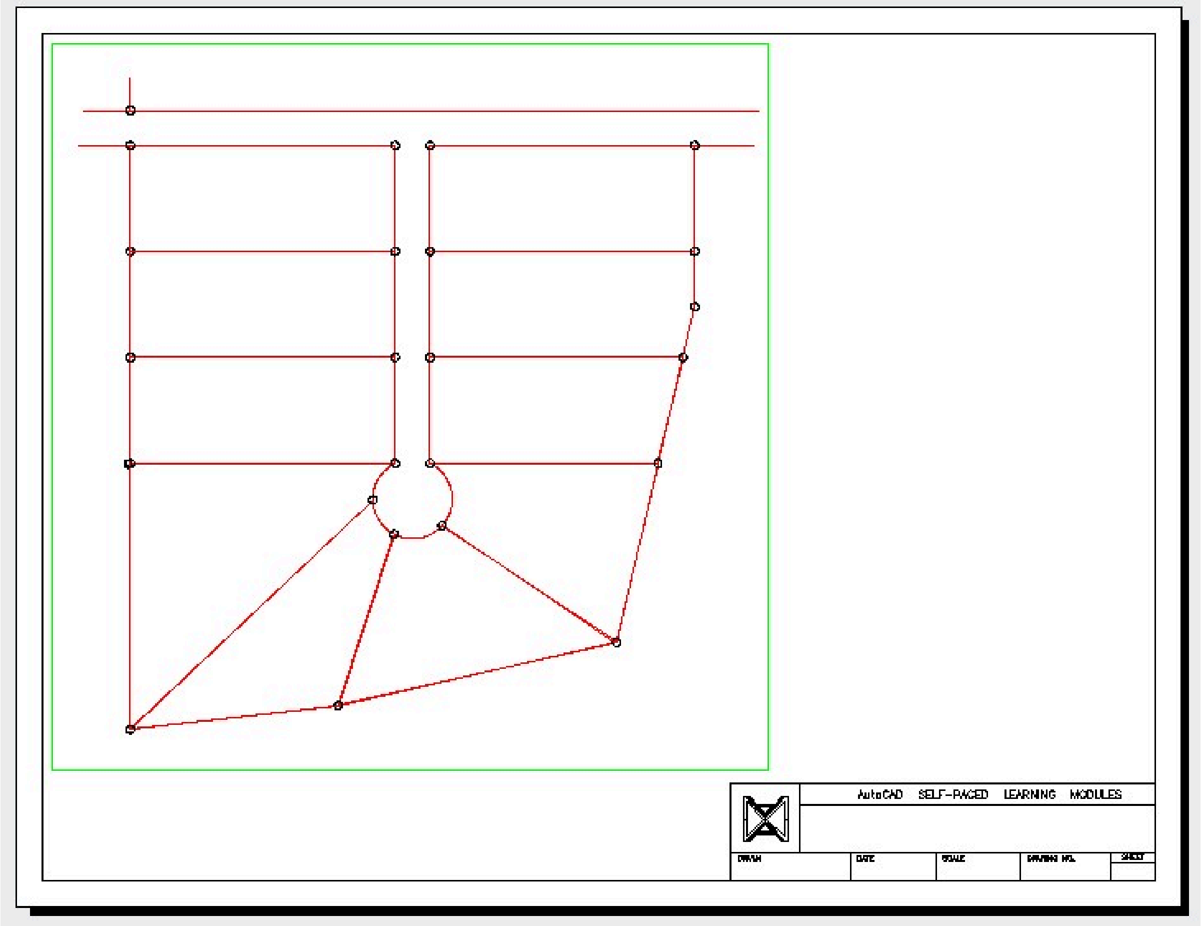
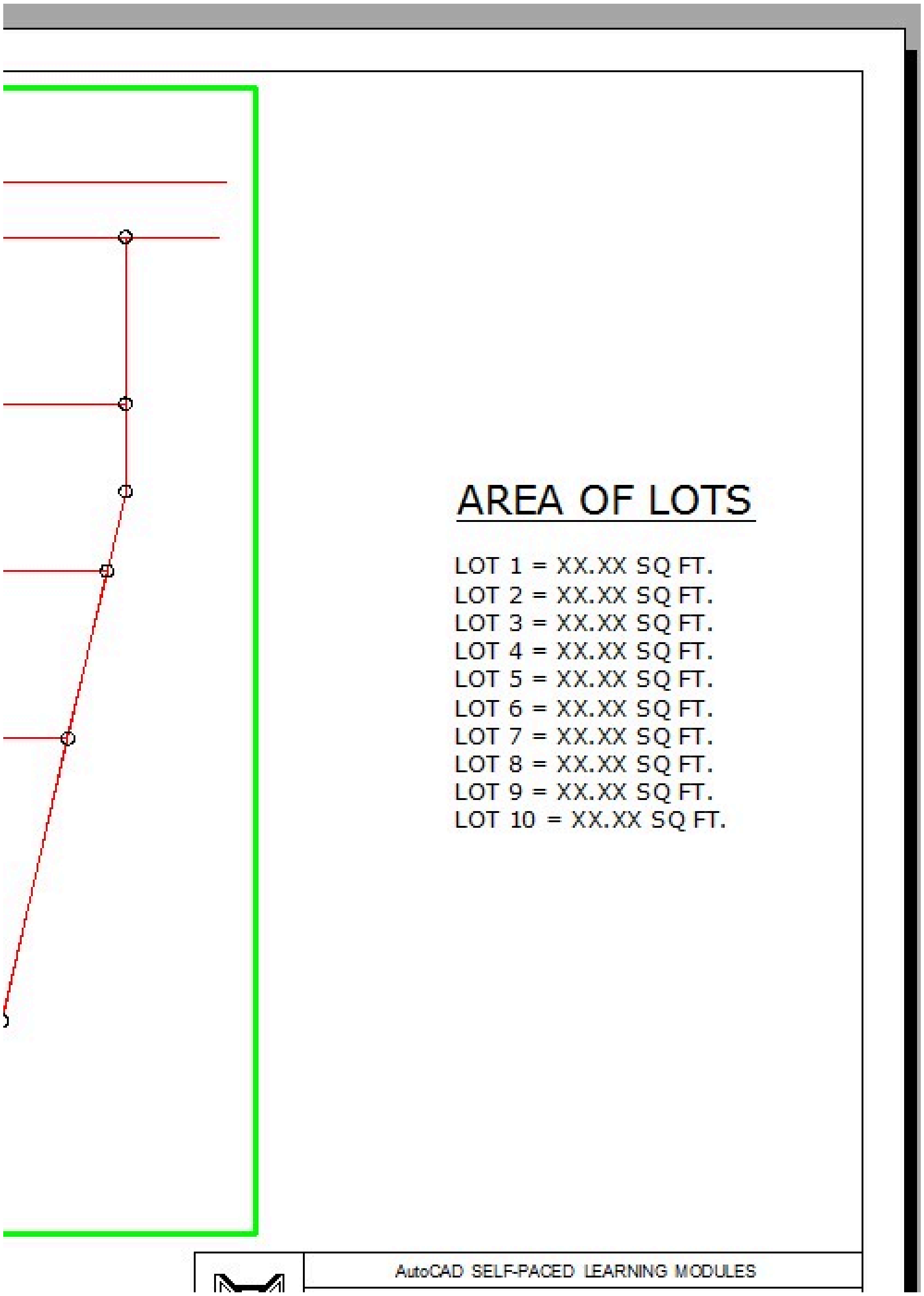

AutoCAD SELF-PACED LEARNING MODULES
DRAWN
DATE
SCALE
DRAWING NO.
SHEET

Figure Step 7

Step 8

Using the AREA command, find the area of each of the 10 lots. On layer: Text, in paper space, insert the text as shown in the figure. Edit the text with the applicable area. Fill in the titleblock. (Figure Step 8)



AREA OF LOTS

LOT 1 = XX.XX SQ FT.
LOT 2 = XX.XX SQ FT.
LOT 3 = XX.XX SQ FT.
LOT 4 = XX.XX SQ FT.
LOT 5 = XX.XX SQ FT.
LOT 6 = XX.XX SQ FT.
LOT 7 = XX.XX SQ FT.
LOT 8 = XX.XX SQ FT.
LOT 9 = XX.XX SQ FT.
LOT 10 = XX.XX SQ FT.

Figure Step 8

Step 9

In Paper space, on layer: Lot Dimensions, insert the dimensions of the lot lines as shown in the figure. On layer: Lot Labels, insert text of the street name and the lot numbers. Create your own text style, pick the font and select the text height. (Figure Step 9)

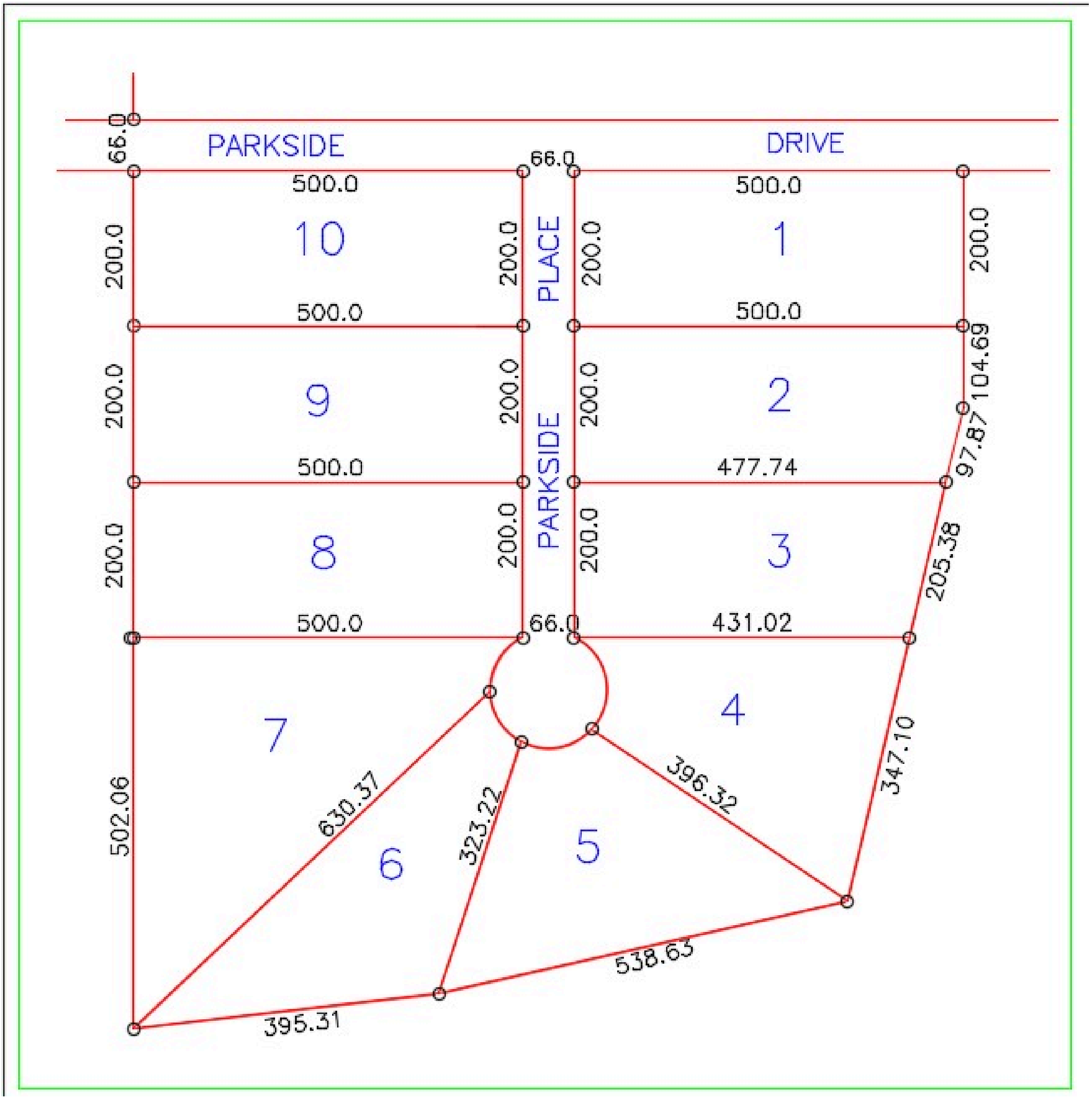

PARKSIDE
DRIVE
66.0
66.0
500.0
500.0
10
1
200.0
200.0
200.0
PLACE
200.0
500.0
500.0
104.69
9
2
200.0
200.0
200.0
200.0
500.0
477.74
97.87
PARKSIDE
8
3
200.0
200.0
200.0
205.38
500.0
66.0
431.02
7
4
347.10
502.06
630.37
323.22
396.32
6
5
538.63
395.31

Figure Step 9

Step 10

Turn layer: Viewport off to complete the drawing. (Figure Step 10)

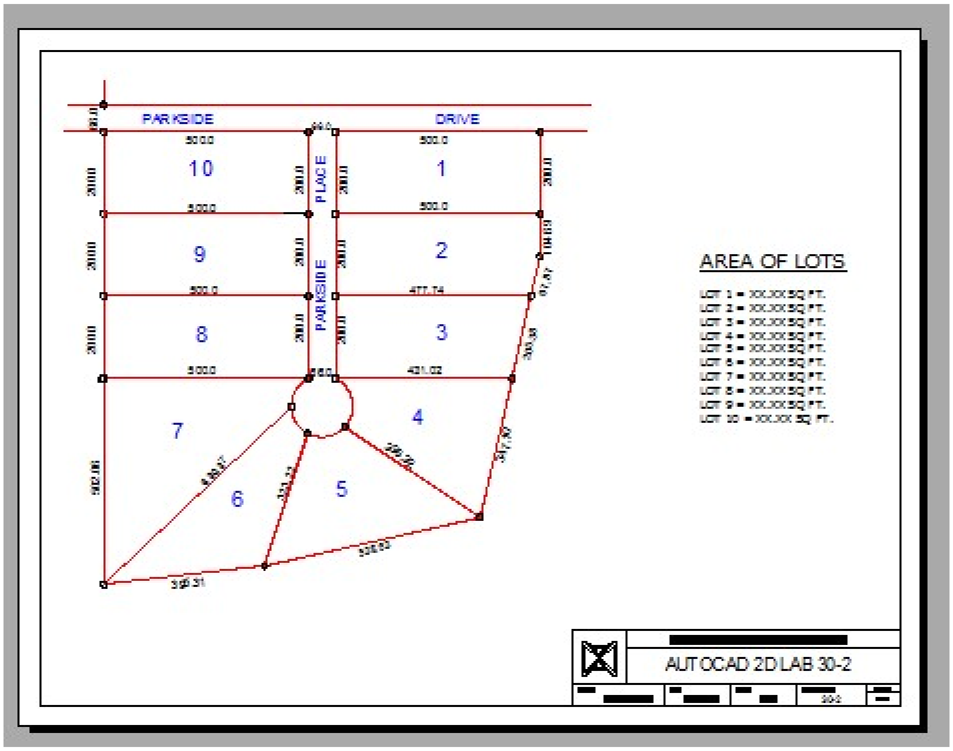

PARKSIDE
DRIVE
PLACE
PARKSIDE
10
1
9
2
8
3
7
4
6
5
500.0
200.0
477.74
431.02
AREA OF LOTS
LOT 1 = XX.XX SQ FT.
LOT 2 = XX.XX SQ FT.
LOT 3 = XX.XX SQ FT.
LOT 4 = XX.XX SQ FT.
LOT 5 = XX.XX SQ FT.
LOT 6 = XX.XX SQ FT.
LOT 7 = XX.XX SQ FT.
LOT 8 = XX.XX SQ FT.
LOT 9 = XX.XX SQ FT.
LOT 10 = XX.XX SQ FT.
AUTOCAD 2D LAB 30-2

Figure Step 10

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

To draw the arc in the center, first draw two 150 foot diameter circles on layer: Construction. Use the ends of the lot lines as the center locations. Where the circles intersect is the center of the 150 foot diameter arc. Draw a circle and then trim it. (Figure Hint 1A and 1B).

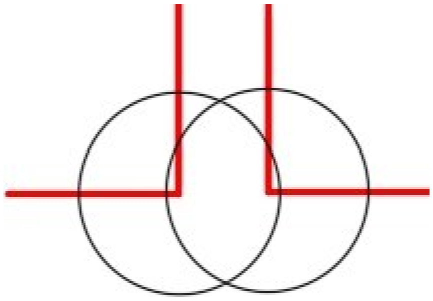


Figure Hint 1A

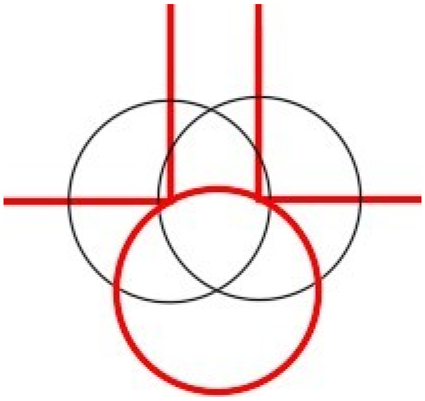


Figure Hint 1B

Hint 2

Figure Hint 2A to 2F will help you calculate the angles.

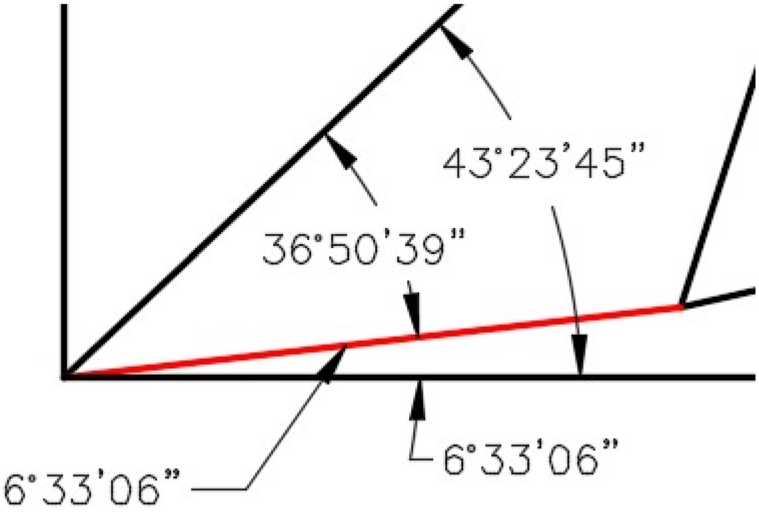

43°23'45"
36°50'39"
6°33'06"
6°33'06"

Figure Hint 2A

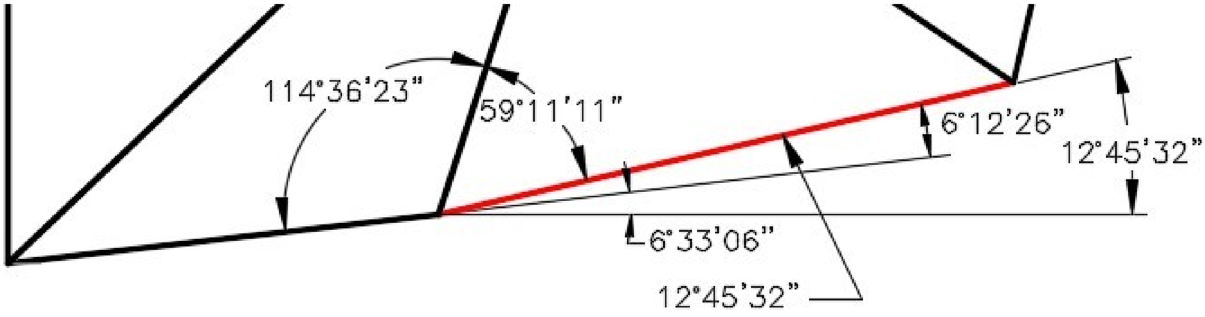

114°36'23"
59°11'11"
6°12'26"
12°45'32"
6°33'06"
12°45'32"

Figure Hint 2B

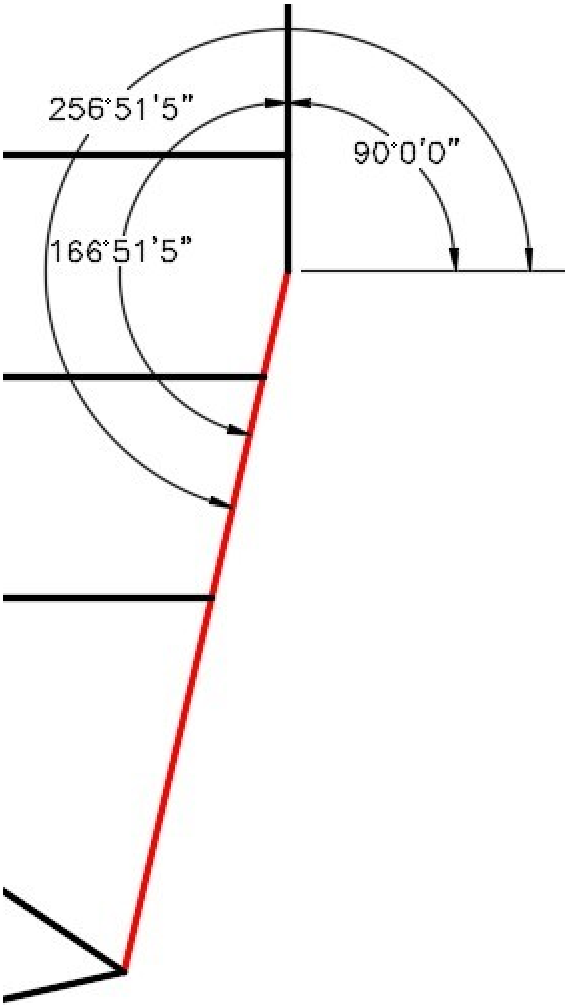

256°51'5"
90°0'0"
166°51'5"

Figure Hint 2C

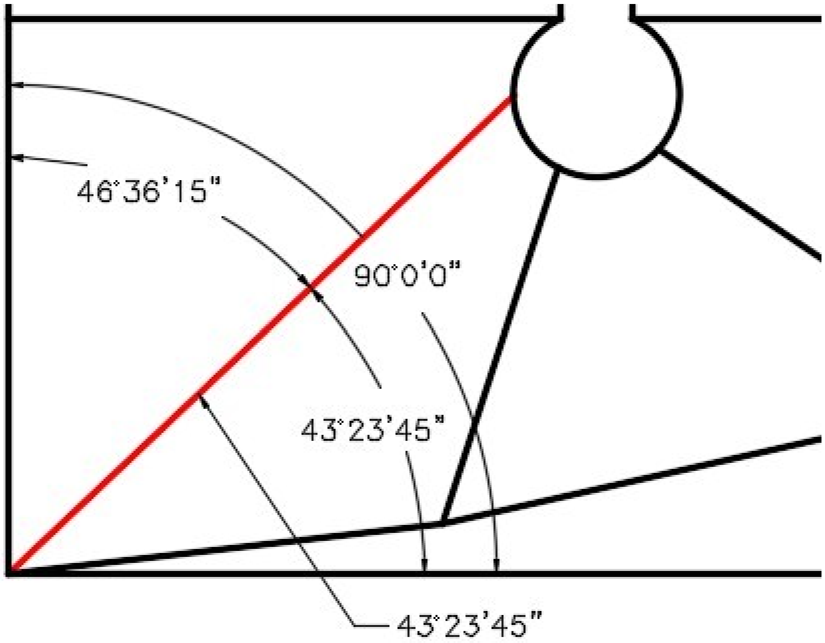

46°36'15"
90°0'0"
43°23'45"
43°23'45"

Figure Hint 2D

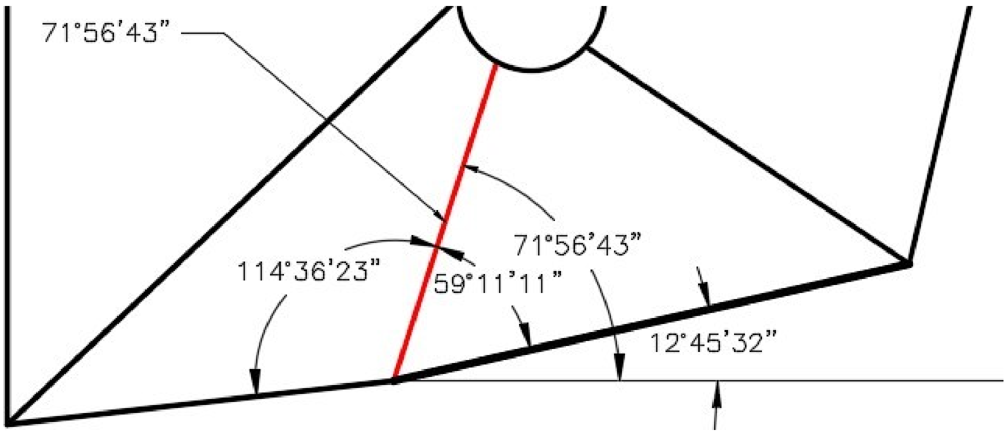

71°56'43"
71°56'43"
114°36'23"
59°11'11"
12°45'32"

Figure Hint 2E

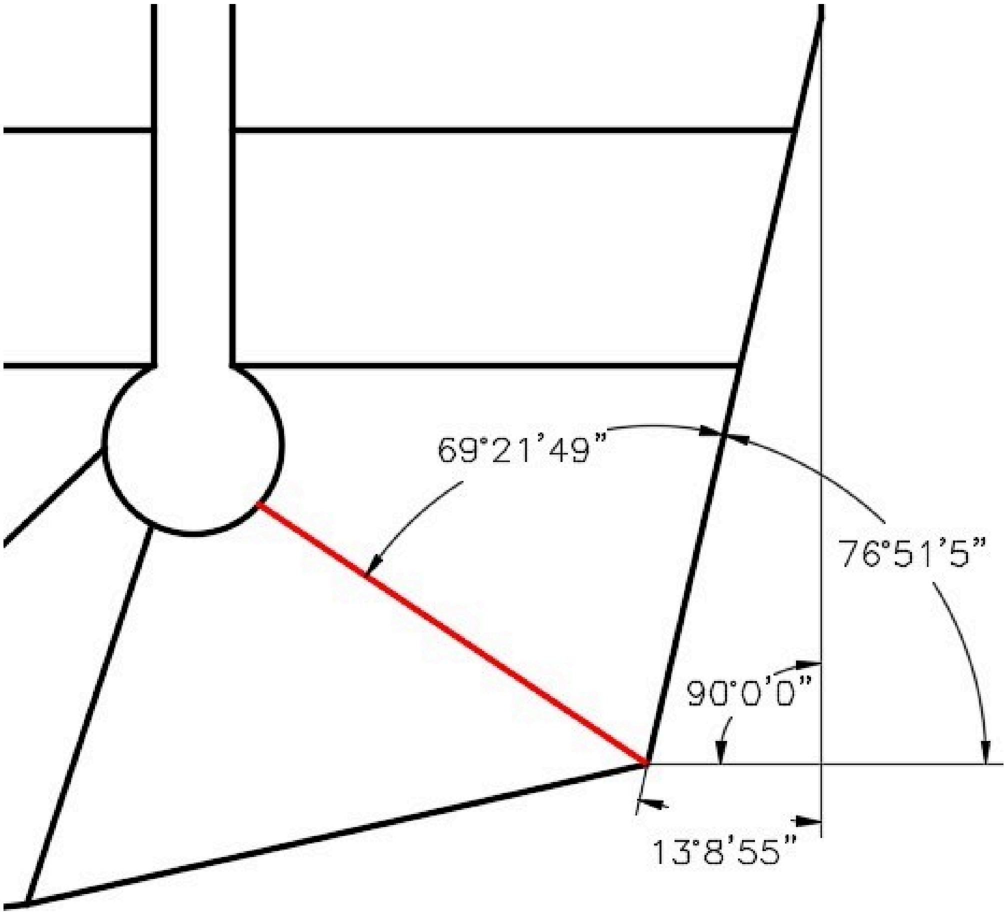

69°21'49"
76°51'5"
90°0'0"
13°8'55"

Figure Hint 2F

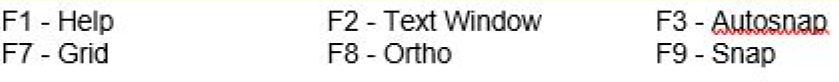
Module 28: Grids and Snap

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe grids and snap and explain how they are used in a drawing.
2. Apply the commands GRID and SNAP to create a drawing.
3. Create a three view multiview drawing from a 3D wireframe model.
4. Describe how to create a three point arc by snapping to two endpoints and a midpoint.

USER TIP: Whenever possible, use the keyboard function keys to toggle features rather than using the Status bar or the applicable commands. Using a function key to toggle a feature is a much faster method and will save the wear and tear on your mouse arm. The function keys that you have learned to this point in the book are:



F1 - Help
F2 - Text Window
F3 - Autosnap
F7 - Grid
F8 - Ortho
F9 - Snap

Using Grids and Snap

Grids are a series of dots or lines that display and overlay the Graphic window in a rectangular or square shape similar to overlaying a paper drawing with grid paper. Grids do not plot and are not part of the drawing's database, they are simply a drawing aid.

The grid can be set to display using dots or lines. See Figures 28-1 and 28-2. The grid's density can be adjusted and their display can be enabled or disabled, when required. The horizontal and vertical spacing between the grid dots or lines can be different, but by default are equal. The grid display can be toggled with the F7 key, the icon on the Status bar, with the GRID command, or by using the Drafting Settings dialogue box. The size of the rectangle that the grid fills can be set to the size of the drawing limits or it can be set to fill Model space. Grids can also be displayed in a layout in Paper space.

Figure 28-1
A Grid Using a Dotted Grid

The *snap* feature works in conjunction with grids. If snap is enabled, the crosshairs snaps only to grid dots, similar to object snap mode. The grid density and the snap density do not have to be set at the same value. Therefore, the crosshairs could snap to locations between the displayed grid dots. The snap can be toggled using the F9 key, with the icon on the Status bar, using the SNAP command, or with the Drafting Settings dialogue box. If snap is enabled, it snaps to the snap density even if the grid display is disabled.

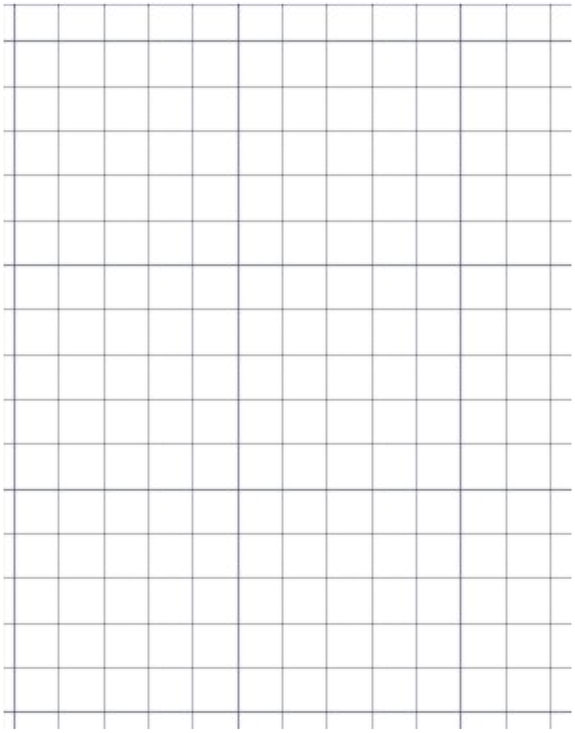
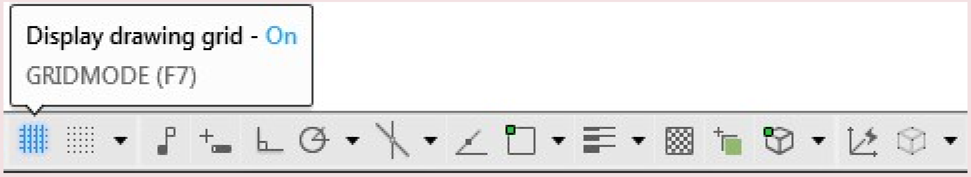


Figure 28-2
A Grid Using a Lined Grid

AutoCAD Command: GRID

The GRID command is used to display a series of dots or lines in either an equal or unequal horizontal and vertical spacing. The function key F7 toggles grid mode.

Shortcut: **F7**

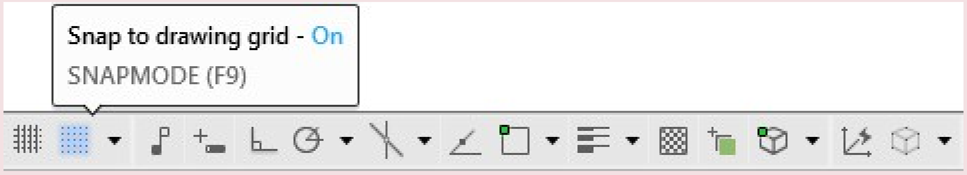


Display drawing grid - On
GRIDMODE (F7)

AutoCAD Command: SNAP

The SNAP command is used to control and enable the snap feature. The function key F9 toggles snap mode.

Shortcut: **F9**



Snap to drawing grid - On
SNAPMODE (F9)

WORKALONG: Using Grids and Snaps

Step 1

Start a new drawing using the template: 2D Layout English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 28-1.

Step 3

Open the Drafting Setting dialogue box by right clicking the Snap icon in the Status bar and clicking Snap Settings. (Figure Step 3)

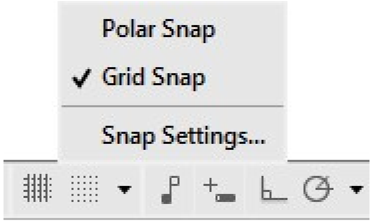
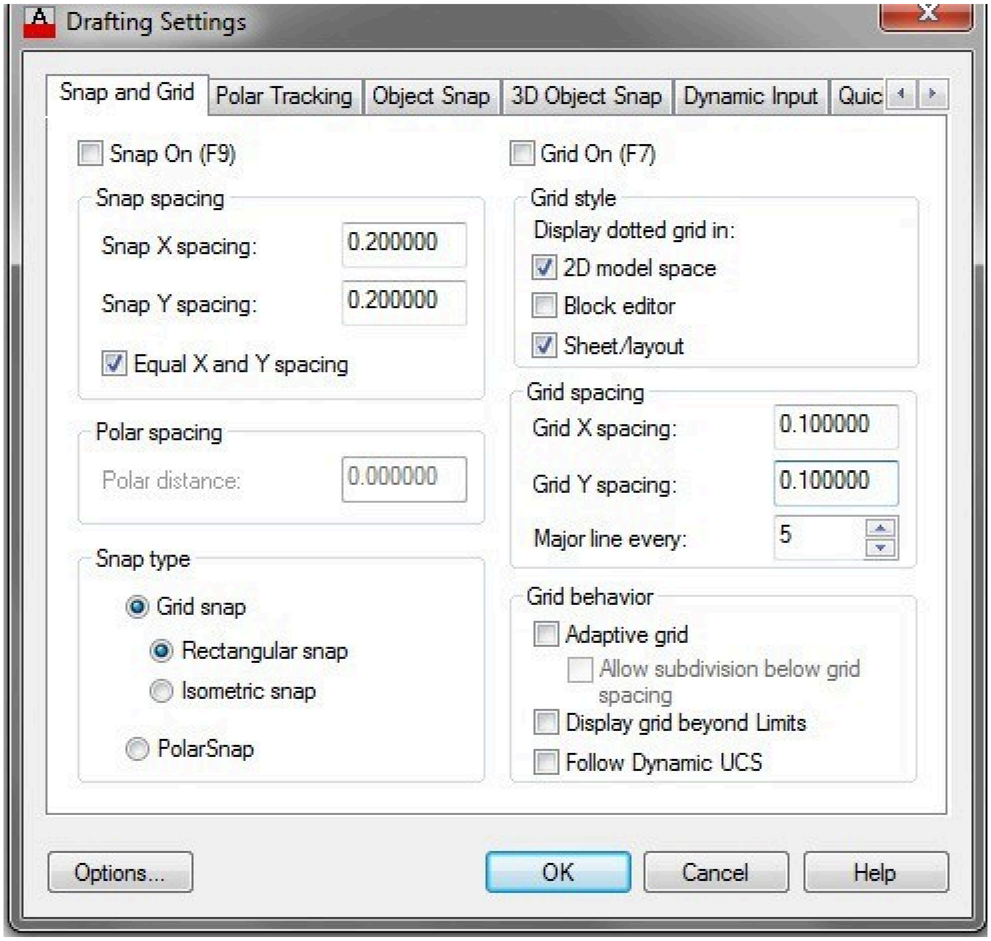

Polar Snap
✓ Grid Snap
Snap Settings...

Figure Step 3

Step 4

In the Drafting Setting dialogue box, enable the Snap and Grid tab. Set the X and Y snap and grid spacings to 0.2 and 0.1 respectfully. Check all of the other settings and change them, if necessary, to match the figure. (Figure Step 4)


Drafting Settings
Snap and Grid
Polar Tracking
Object Snap
3D Object Snap
Dynamic Input
Quic
Options...
OK
Cancel
Help

☐ Snap On (F9)

Snap spacing
Snap X spacing: 0.200000
Snap Y spacing: 0.200000
☑ Equal X and Y spacing

Polar spacing
Polar distance: 0.000000

Snap type
● Grid snap
● Rectangular snap
○ Isometric snap
○ PolarSnap

☐ Grid On (F7)

Grid style
Display dotted grid in:
☑ 2D model space
☐ Block editor
☑ Sheet/layout

Grid spacing
Grid X spacing: 0.100000
Grid Y spacing: 0.100000
Major line every: 5

Grid behavior
☐ Adaptive grid
☐ Allow subdivision below grid spacing
☐ Display grid beyond Limits
☐ Follow Dynamic UCS

Figure Step 4

AUTHOR'S COMMENTS: Ensure that you set all items in your dialogue box to match the figure.

Step 5

Enter the LIMITS command, as shown below, to set the limits to 8 1/2 X 11 inches.

```
Command: LIMITS

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000,0.0000>:

Specify upper right corner <12.0000,9.0000>: 8.5,11

Command:
```

Step 6

Press F7 to enable the grid display and your drawing should appear similar to the figure. (Figure Step 6)

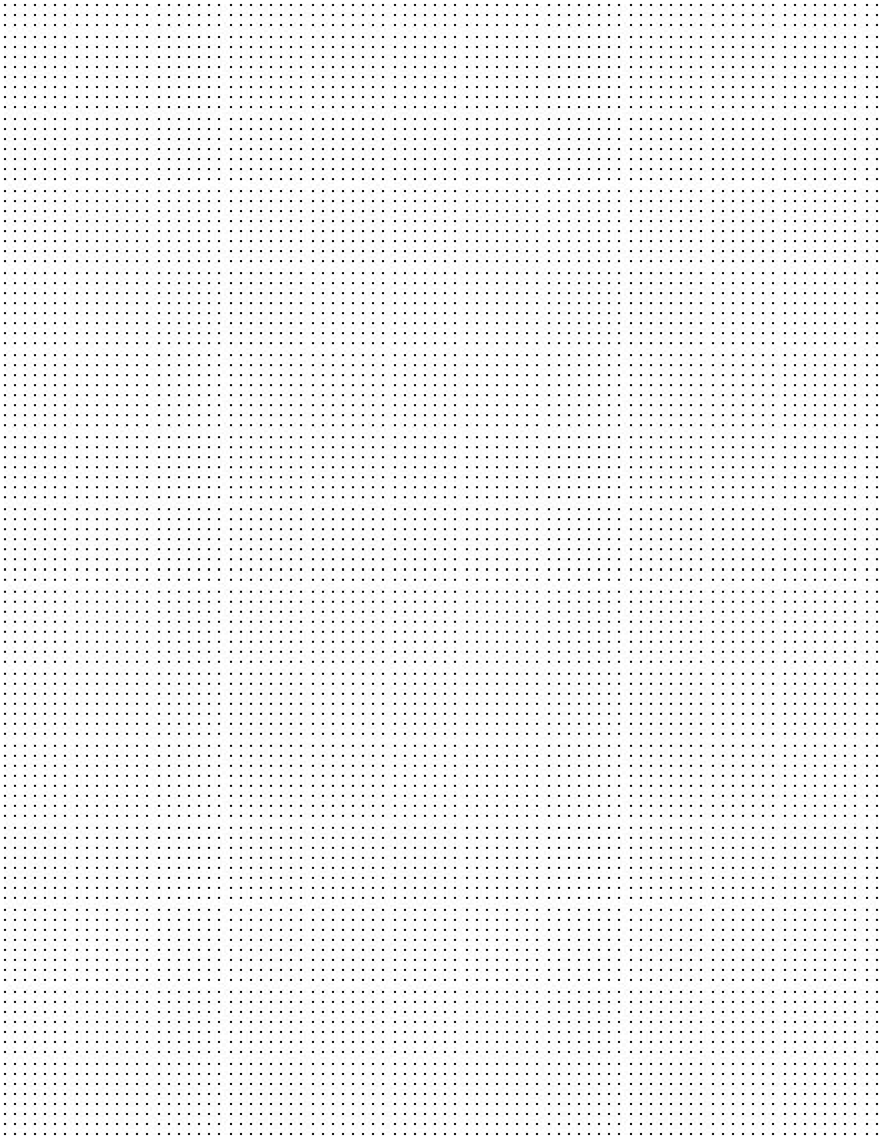
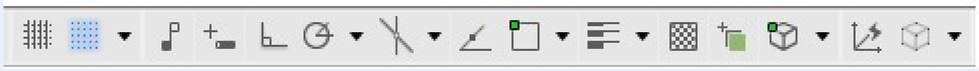
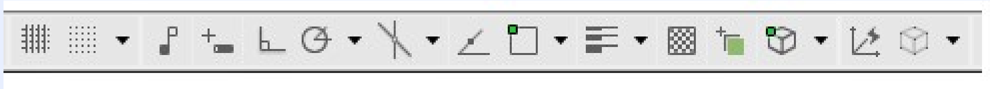


Figure Step 6

MUST KNOW: If you find your crosshairs jumping around your screen and hard to control, check to see if the snap icon is enabled on the Status bar. Some menu commands will toggle snap mode without you knowing it. Disable snap whenever you are not using it.



Snap is enabled



Snap is disabled

Keep in mind that you can toggle the snap with F9 or by clicking the icon on the Status bar.

Step 7

Press F9 to enable the snap function.

Step 8

On layer: Construction, enter the LINE command and draw lines as shown in the figure. Locate and draw the lines by eye. Try to match the figure the best you can. (Figure Step 8)

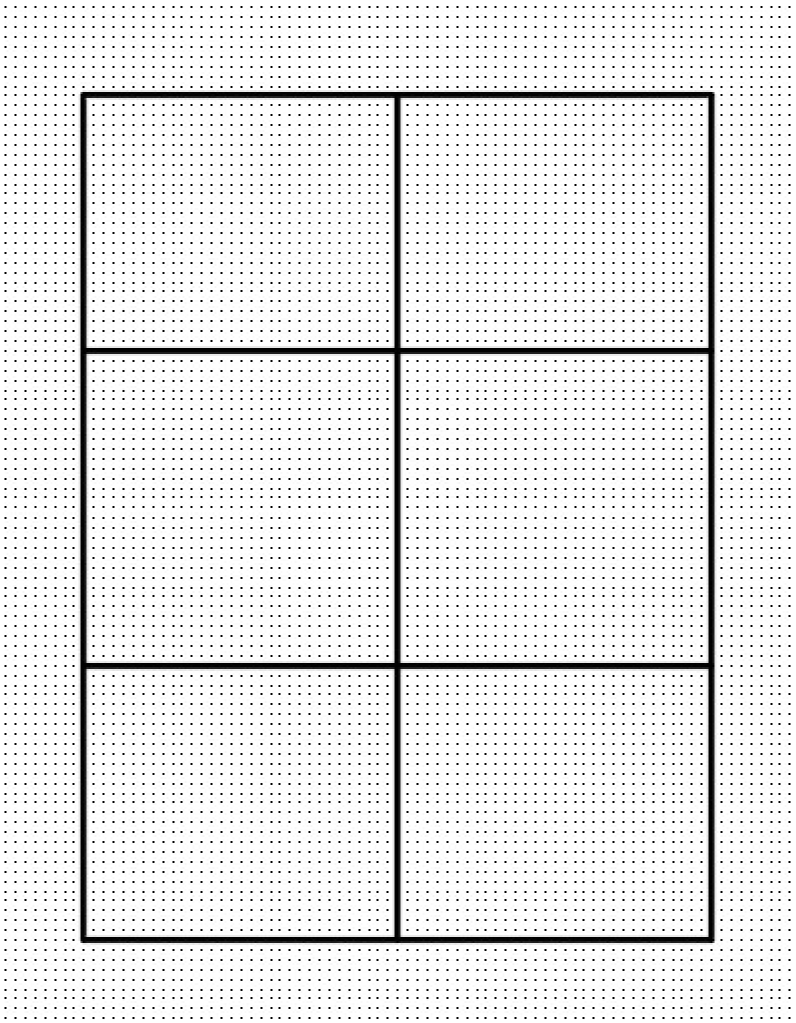


Figure Step 8

AUTHOR'S COMMENTS: When snap is enabled, specifying locations in commands will jump or snap to the snap grid only. You cannot draw to locations between the snap spacing.

AUTHOR'S COMMENTS: The snap is set to 0.2 and grid spacing is set to 0.1. That means the grid will display a dot every 0.1 inches and will snap to the every other grid dot.

AUTHOR'S COMMENTS: Since you are drawing a simple electric schematic diagram in this workalong, you do not have to be concerned with exact sizes as you have being doing to this point in the book. Schematic diagrams are not real objects, therefore they do not have to be drawing with specified sizes.

Step 9

On layer: Object, use the CIRCLE and LINE commands to draw the 5 circles and three lines to form the triangle. Use the detailed figure (Figure Step 9B) to size the circles and the triangle. Snap to grid dots to draw all objects. (Figure Step 9A and 9B)

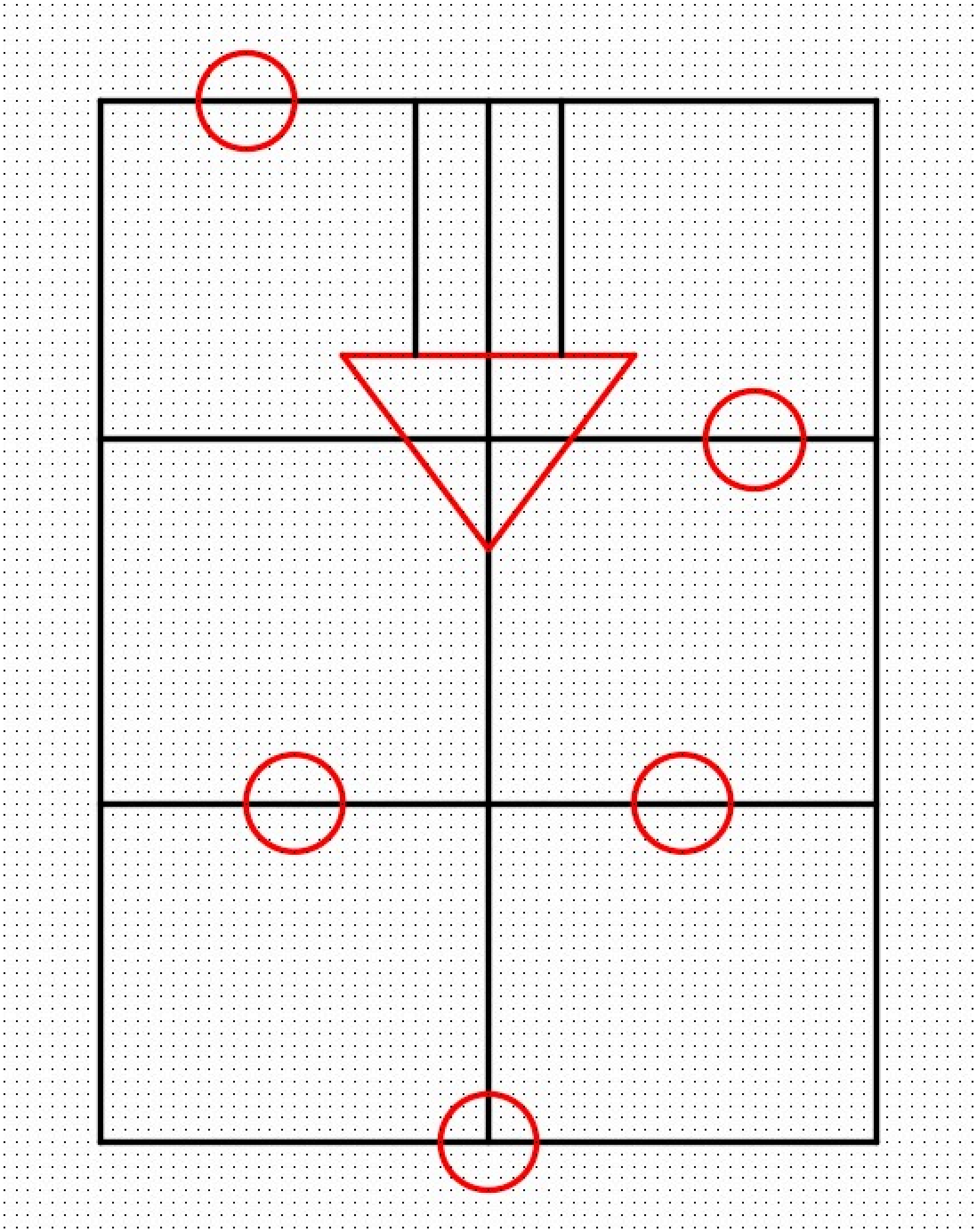


Figure Step 9A

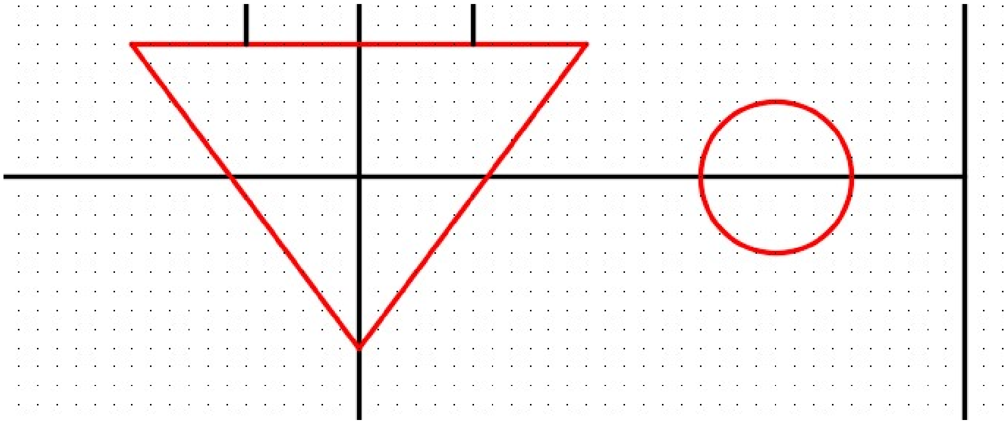


Figure Step 9B

Step 10

On layer: Object, use the LINE command to draw the lines as shown in the figure. Snap to the grid dots for all lines. (Figure Step 10)

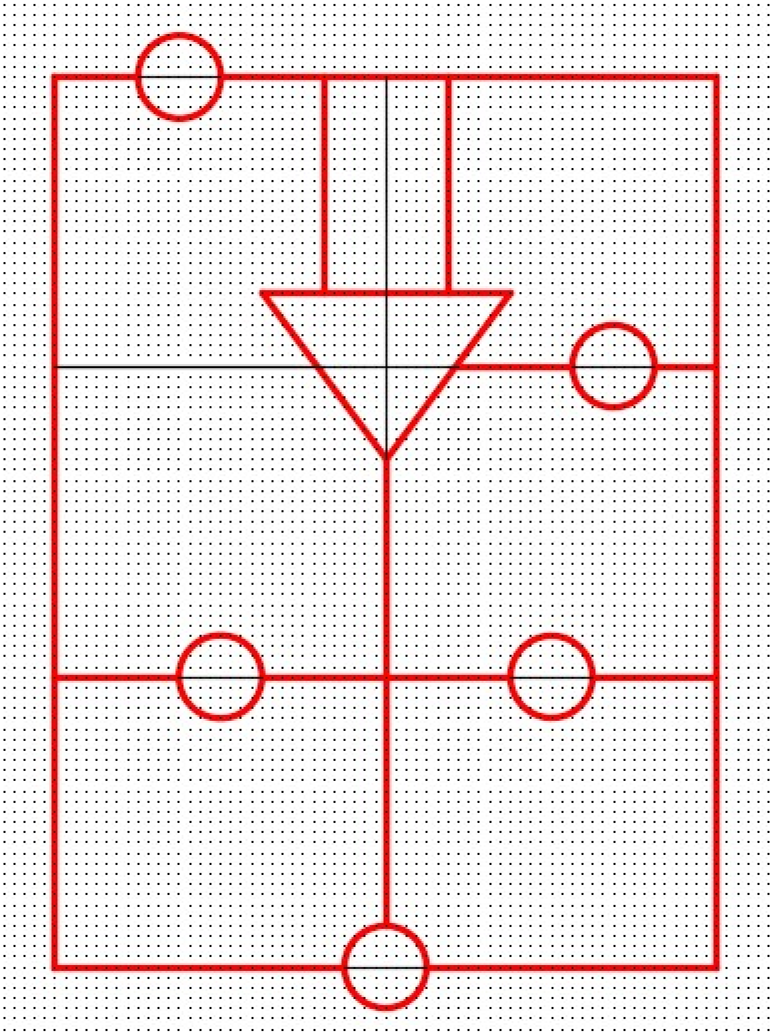


Figure Step 10

Step 11

Turn layer: Construction off and trim the any necessary lines. (Figure Step 11)

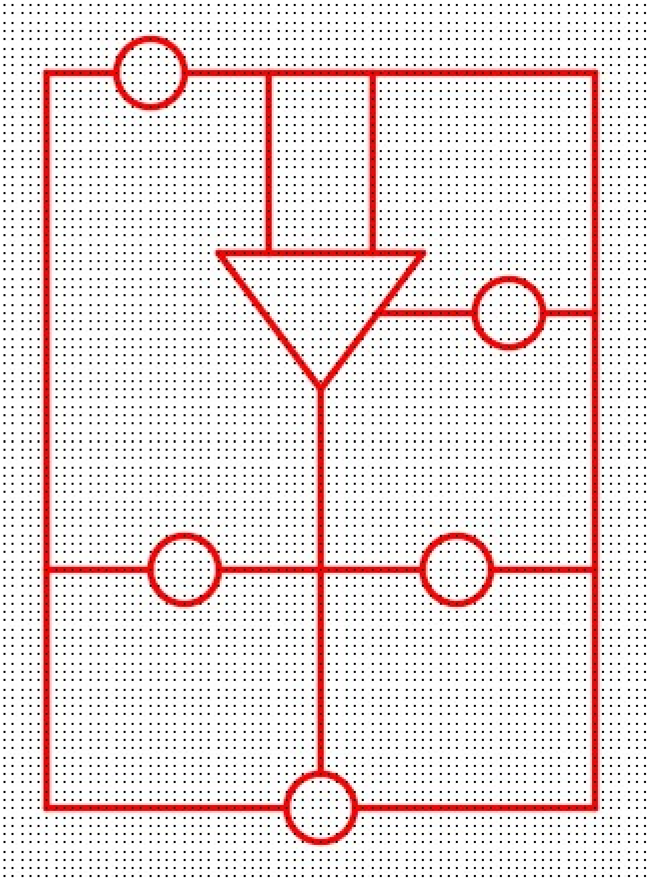


Figure Step 11

Step 12

Press F7 and F9 to disable the grid and the snap functions. (Figure Step 12)

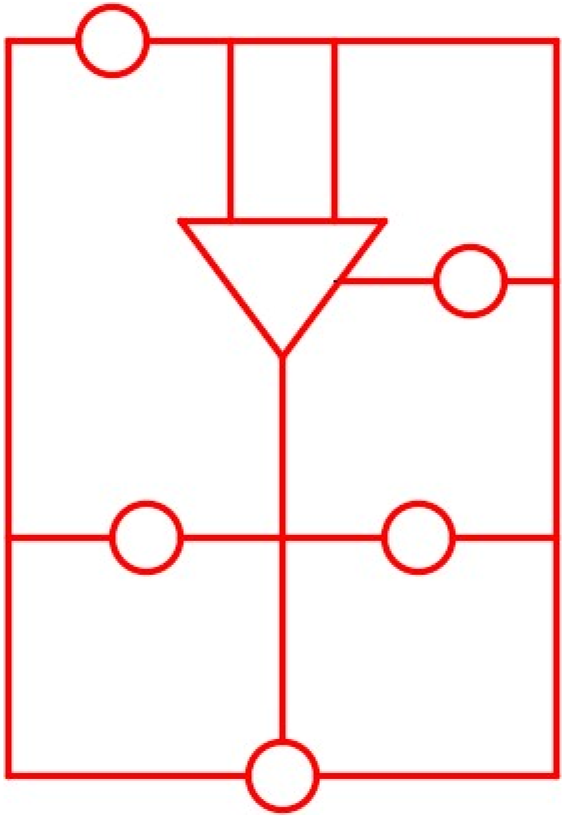


Figure Step 12

Step 13

On layout: Module Layout A, create a viewport on layer: Viewport. Set the scale of the viewport to 0.75:1 and lock the display. Complete the titleblock using the AutoCAD book Titleblock Standards shown in Module 20. (Figure Step 13)

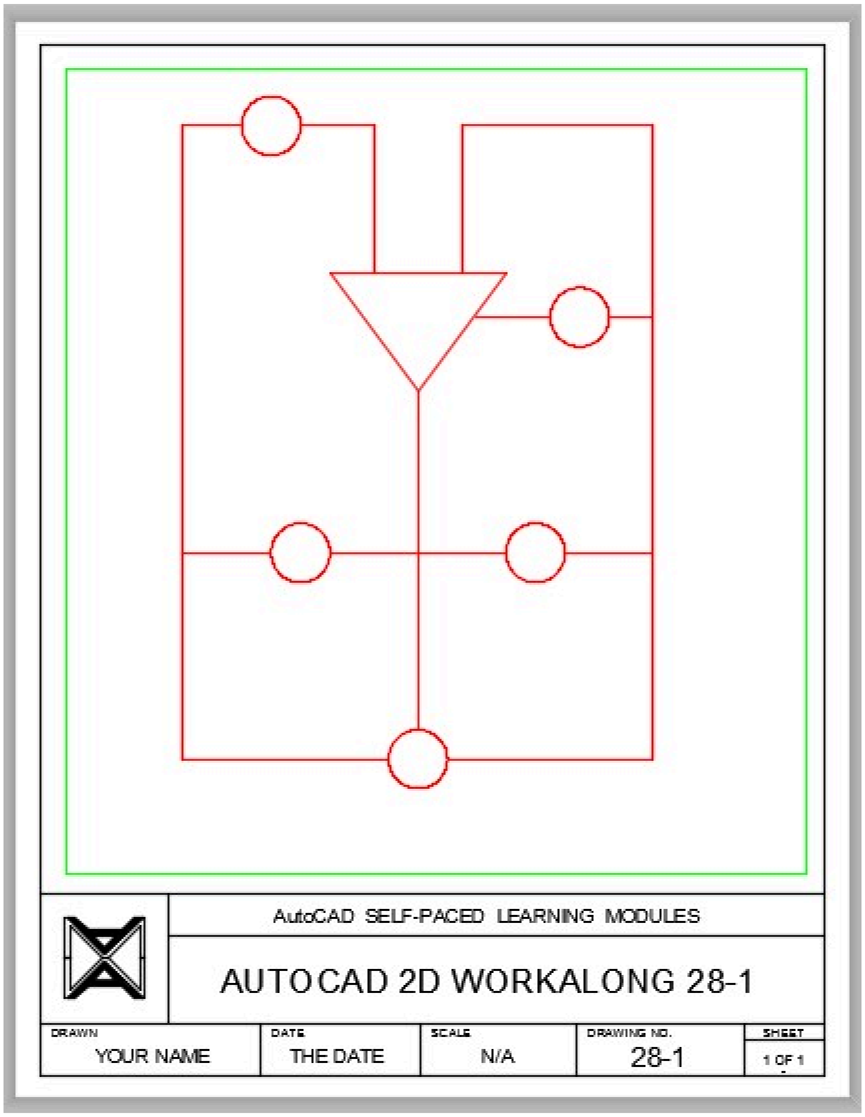

AutoCAD SELF-PACED LEARNING MODULES
AUTOCAD 2D WORKALONG 28-1
DRAWN
YOUR NAME
DATE
THE DATE
SCALE
N/A
DRAWING NO.
28-1
SHEET
1 OF 1

Figure Step 13

Step 14

Turn layer: Viewport off. (Figure Step 14)

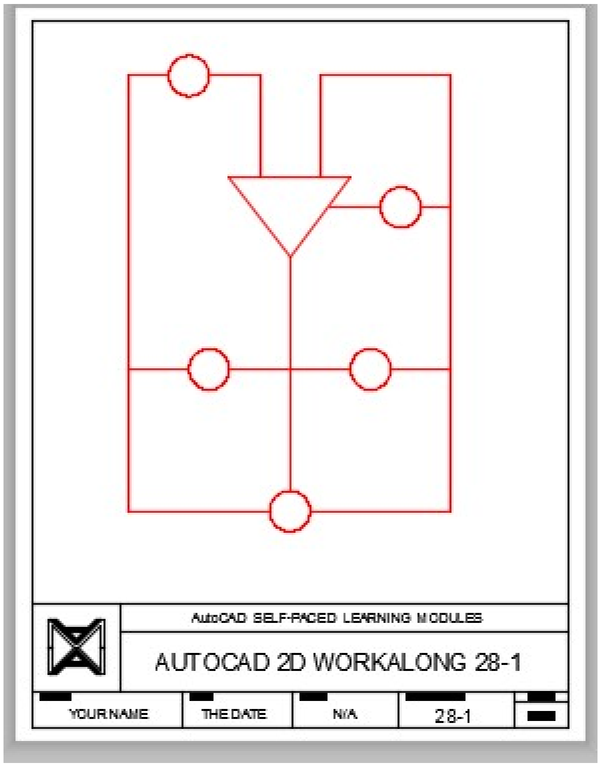

AutoCAD SELF-PACED LEARNING MODULES
AUTOCAD 2D WORKALONG 28-1
YOUR NAME
THE DATE
N/A
28-1

Figure Step 14

Step 15

Save and close the drawing.

WORKALONG: Using Ortho Mode as an Aid to Draw a Multiview Drawing

Step 1

Start a new drawing using the template: 2D Layout Metric.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 28-2.

Step 3

Set layer: Object as the current layer. Study the figures. (Figure Step 3A, 3B, and 3C)

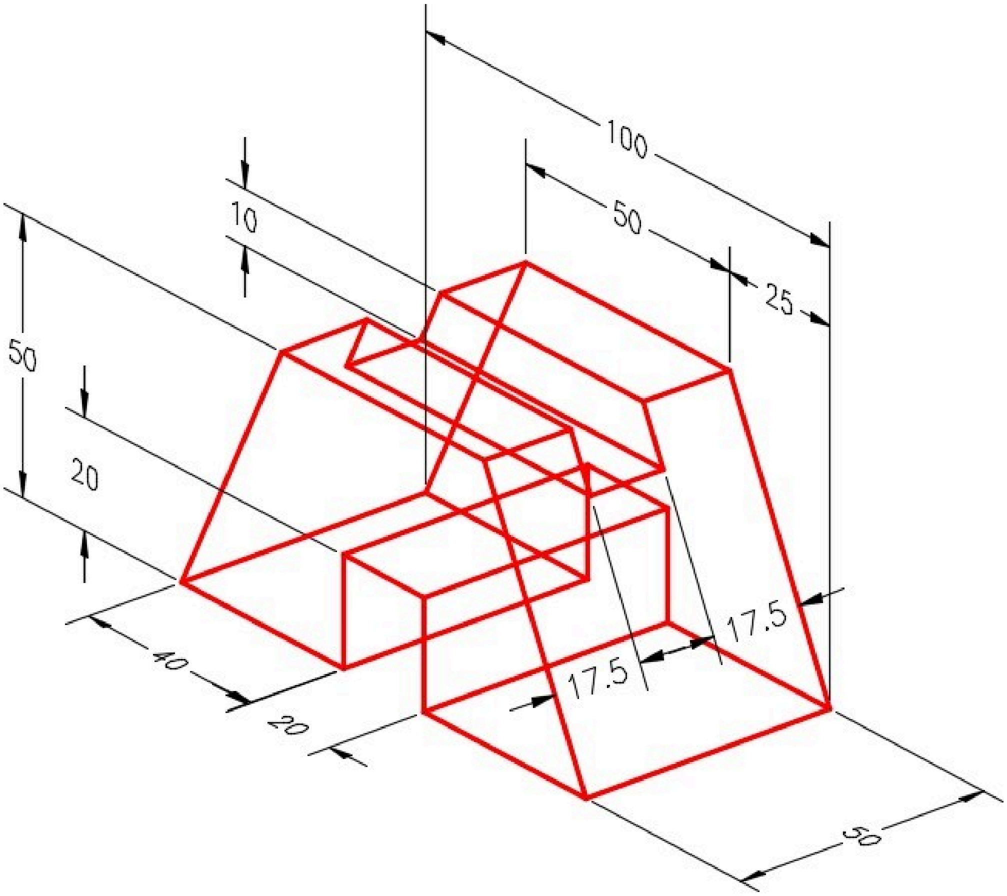


Figure Step 3A
Dimensioned Wireframe Model

AUTHOR'S COMMENTS: The three figures are a 3D wireframe model, a solid model, and a multiview drawing of the same object. In this workalong, you will be drawing the multiview drawing of the Front, Top, and Right Side views of this object using Ortho mode to assist you.

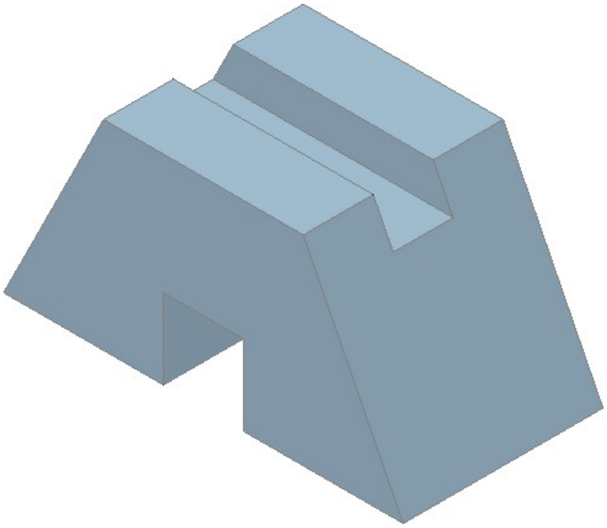


Figure Step 3B
Solid Model

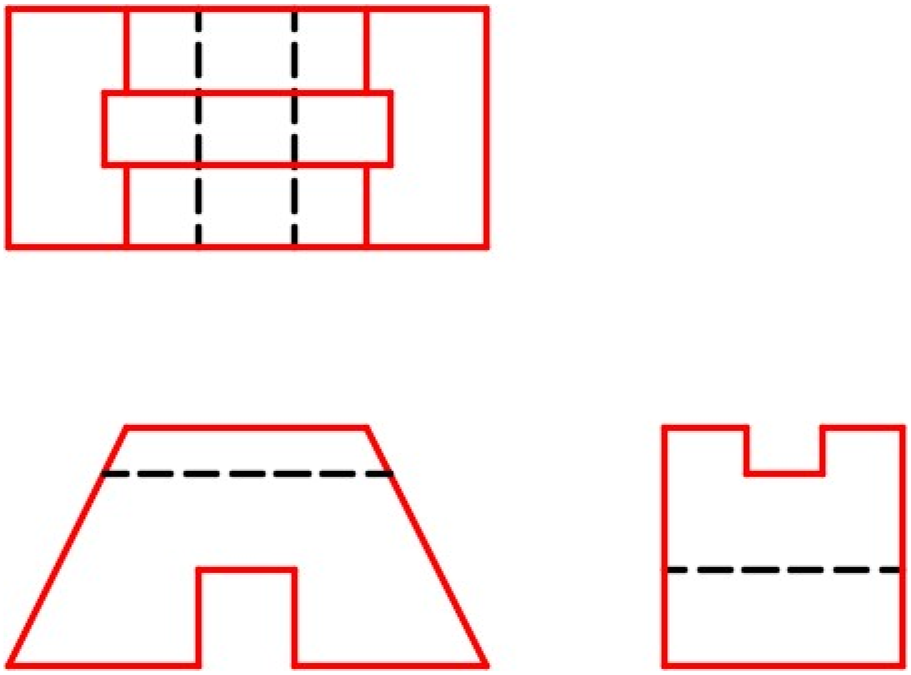


Figure Step 3C
Multiview Drawing

AUTHOR’S COMMENTS: When Ortho mode is enabled, you can shortcut coordinate input when drawing lines using the LINE command. You are only required to enter the distance in the direction of current axis. This method allow you to draw faster if the lines you are drawing are horizontal and vertical. For example, If you wanted to draw a horizontal line in the positive X direction, enable ortho mode and then move the crosshairs in the positive X direction from the lastpoint and then enter the length of the line.

Step 4

Enable Ortho mode by pressing F8. Draw the Front view by entering the LINE command shown below. (Figure Step 4A to 4H)

Command: **L**

Specify first point: **100,100**

(Move the crosshairs in the positive X direction.)

Specify next point or [Undo]: **40**

(Note how you only have to enter the X distance. As long as Ortho mode is enabled, you are not required to enter the @.)

(See Figure Step 4A)

(Move the crosshairs in the positive Y direction.)

Specify next point or [Undo]: **20**

(See Figure Step 4B)

(Move the cursor in the positive X direction.)

Specify next point or [Undo]: **20**

(See Figure Step 4C)

(Move the crosshairs in the negative Y direction.)

Specify next point or [Undo]: **20**

(See Figure Step 4D)

(Move the crosshairs in the positive X direction.)

Specify next point or [Undo]: **40**

(See Figure Step 4E)

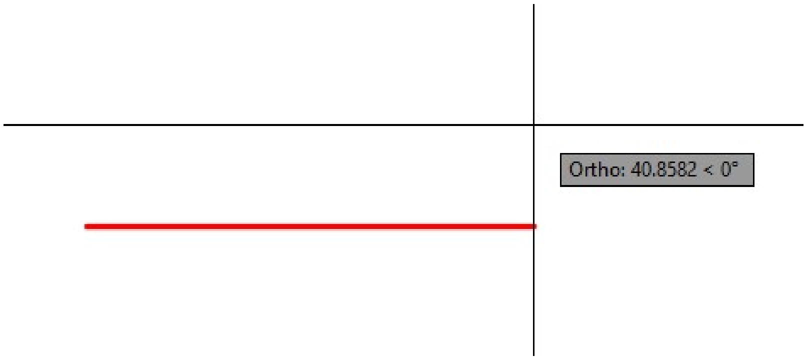

Ortho: 40.8582 < 0°

Figure Step 4A

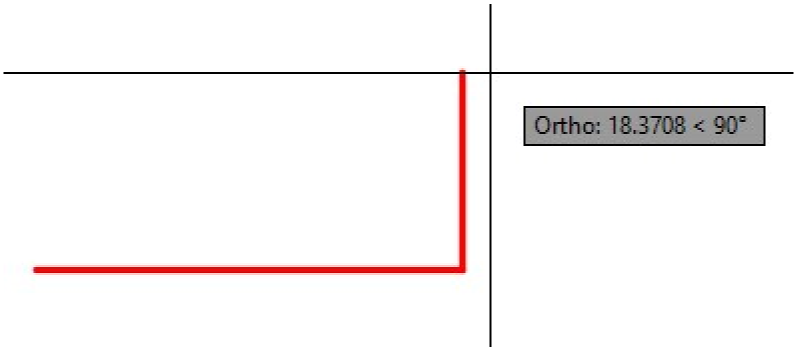

Ortho: 18.3708 < 90°

Figure Step 4B

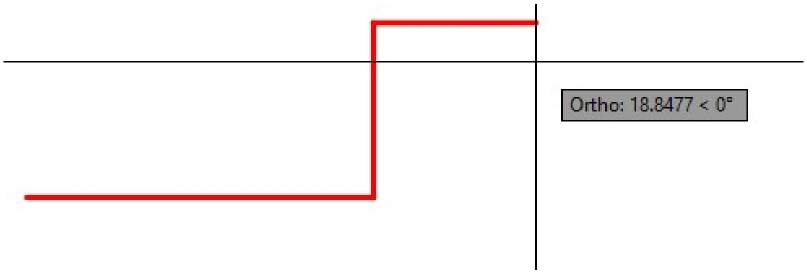

Ortho: 18.8477 < 0°

Figure Step 4C

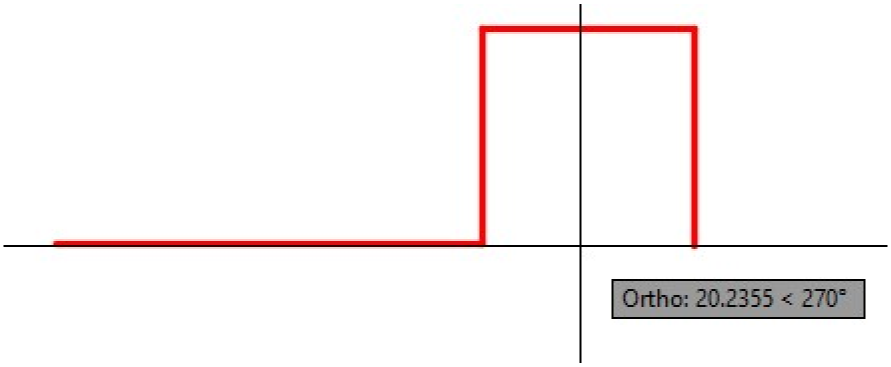

Ortho: 20.2355 < 270°

Figure Step 4D

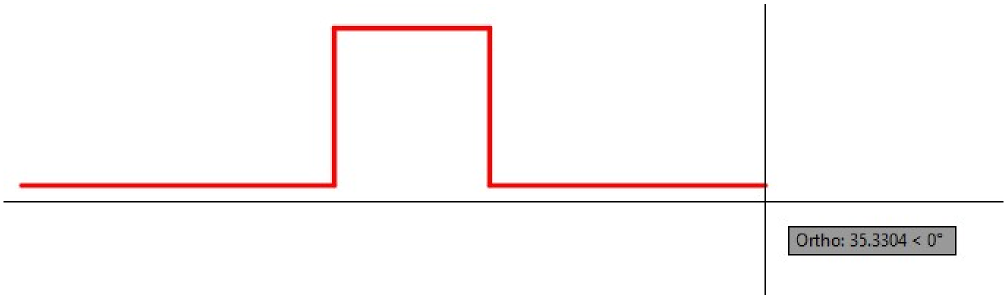


Figure Step 4E

Specify next point or [Undo]: **@-25,50**

(The crosshairs position is not important when you enter exact coordinates. Ortho mode will be ignored when both coordinates are entered. See Figure Step 4F)

(Move the crosshairs in the negative X direction.)

Specify next point or [Undo]: **50**

(See Figure Step 4G)

Specify next point or [Close/Undo]: **C**

(Close the line to the start point. Figure Step 4H)

Specify next point or [Close/Undo]:

Command:

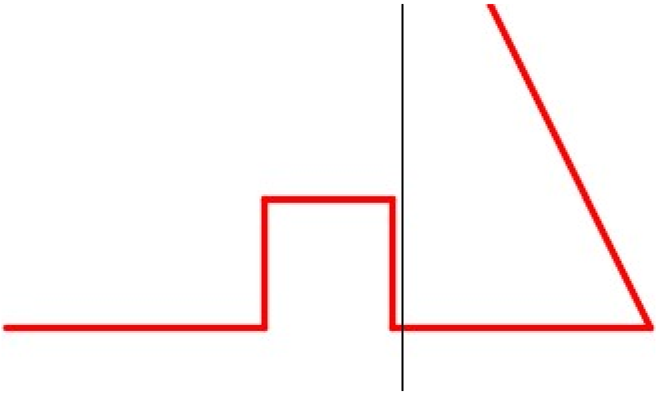


Figure Step 4F

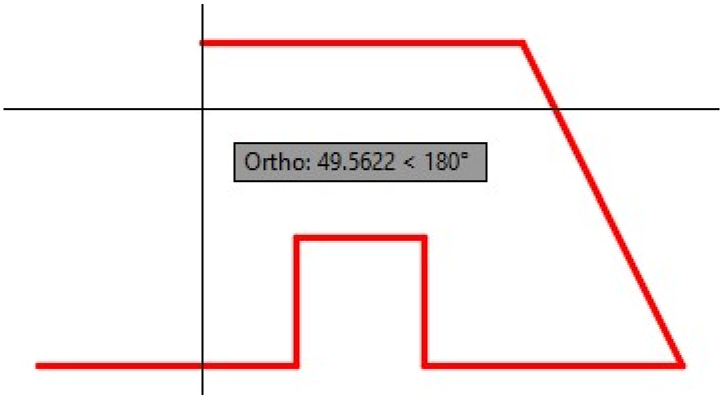

Ortho: 49.5622 < 180°

Figure Step 4G

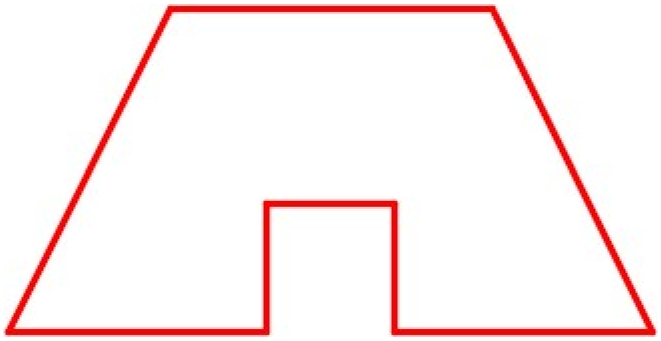


Figure Step 4H

Step 5

On layer: Construction, with Ortho mode and Osnap mode enabled, use the LINE command to draw a vertical line by snapping the first point to the end of the bottom line. The length of the line is not important. (Figure Step 5)

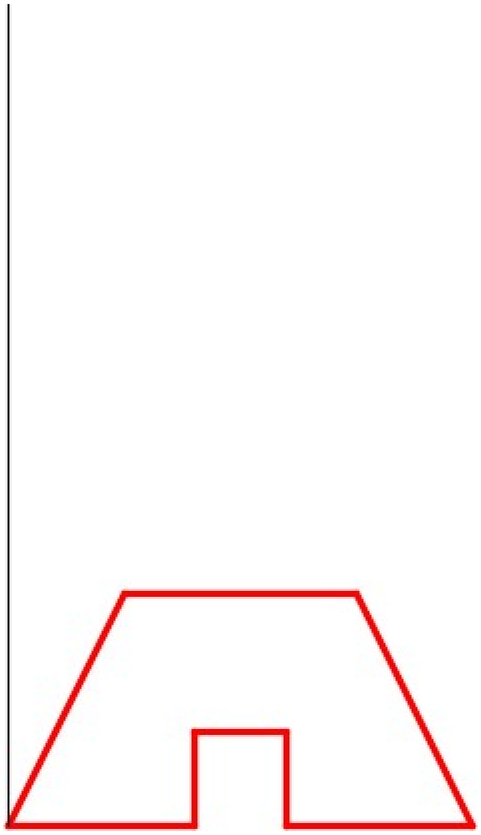


Figure Step 5

Step 6

Using what you just learned, repeat the same thing five additional times to draw vertical lines from each endpoint on the front view. (Figure Step 6)

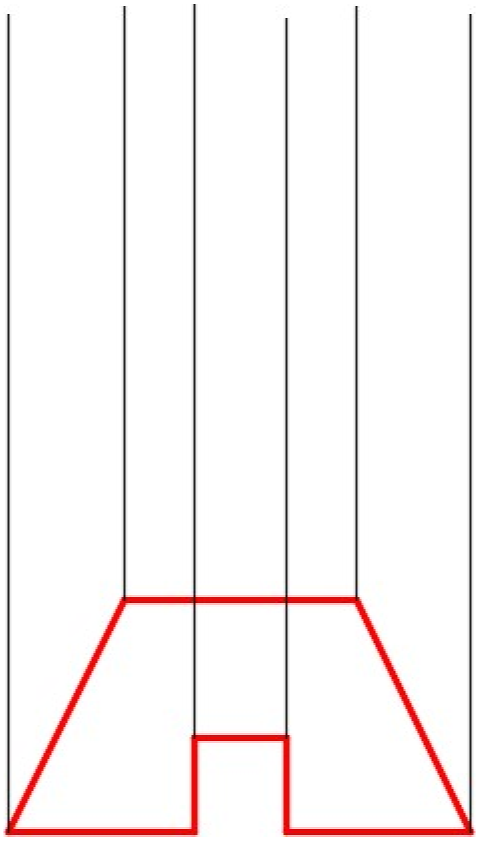


Figure Step 6

Step 7

Using what you just learned, repeat the same thing by drawing horizontal lines from the three endpoints on the Front view. Guess at the length of the lines. (Figure Step 7)

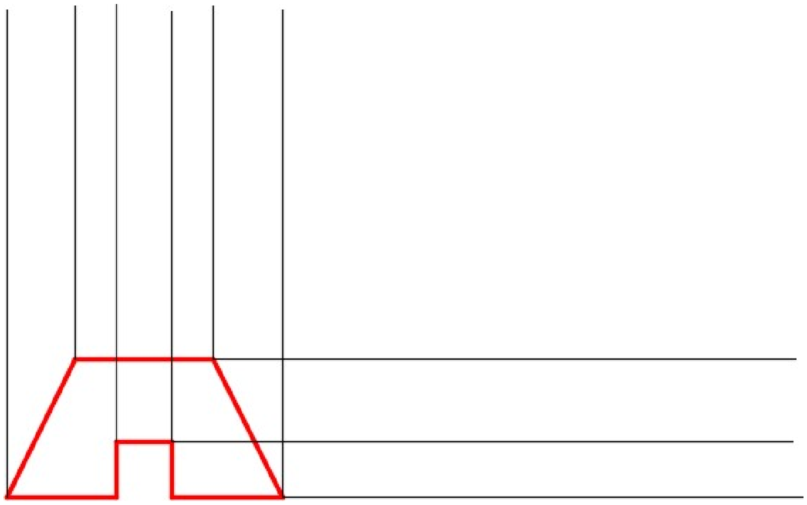


Figure Step 7

Step 8

Draw a 45 degree line starting from the intersection of the construction line on the far right and the construction line on the top as shown in the figure. The line should be a least 200 units long. (Figure Step 8)

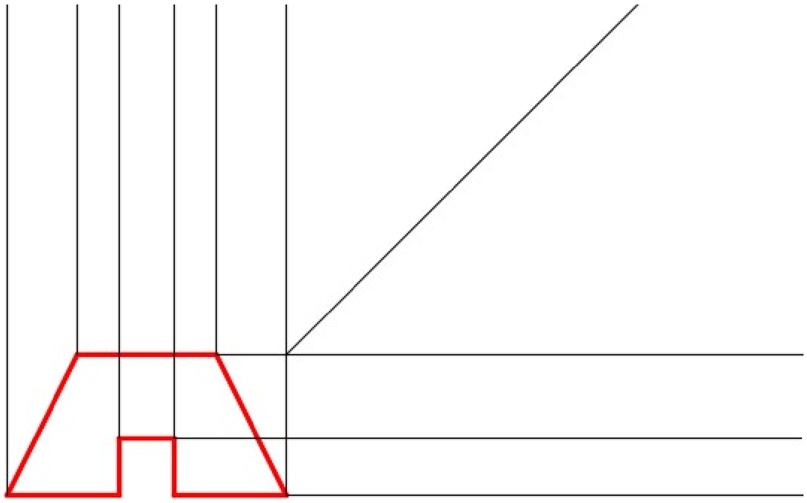


Figure Step 8

Step 9

Using Ortho mode, draw a horizontal line as shown in the figure. The location and length of line is not important. Try to match the figure as close as you can by eye. (Figure Step 9)

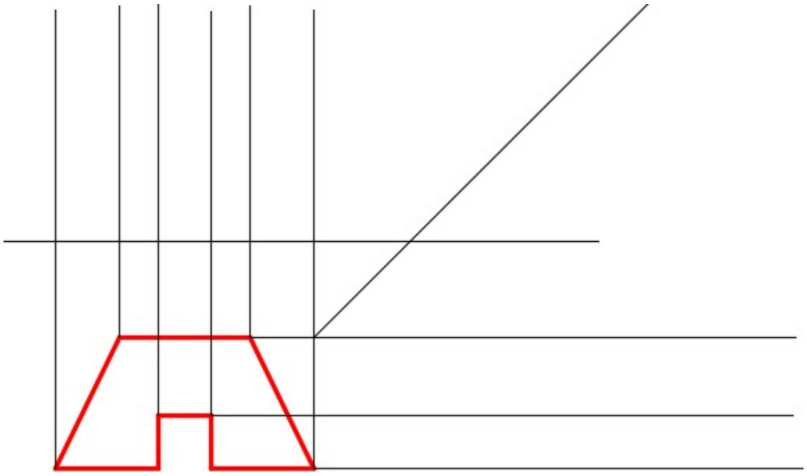


Figure Step 9

Step 10

With Ortho mode enabled, draw a vertical line from the intersection of the inclined line and the line you drew in Step 9. (Figure Step 10)

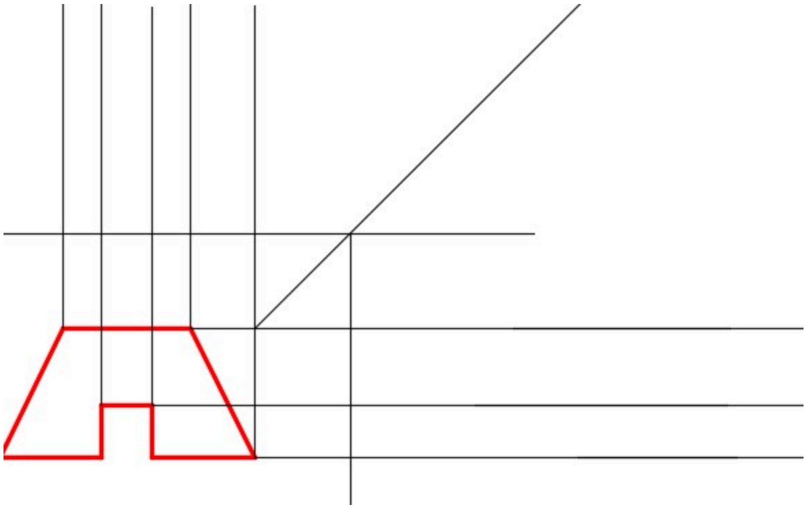


Figure Step 10

Step 11

Using the OFFSET command, offset the line you drew in Step 9, 17.5 millimeters up. Repeat using 15 and 17.5 millimeters. (Figure step 11)

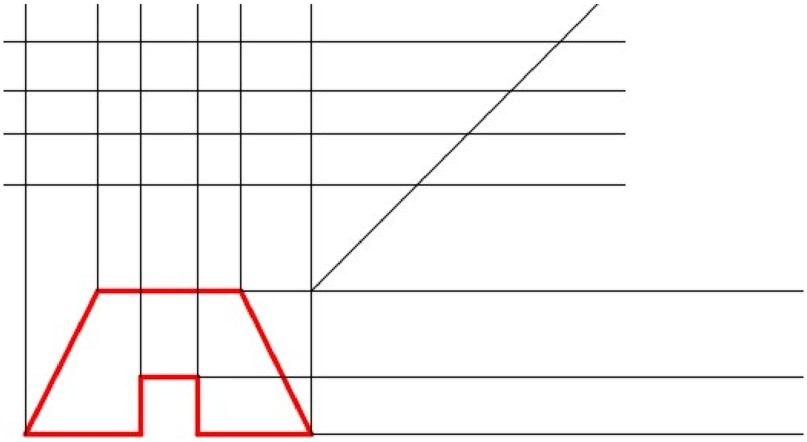


Figure Step 11

Step 12

With Ortho mode enabled, draw vertical lines from the intersection of the horizontal lines and the inclined line. Ensure that you start each line by snapping to the intersection of the lines. (Figure Step 12)

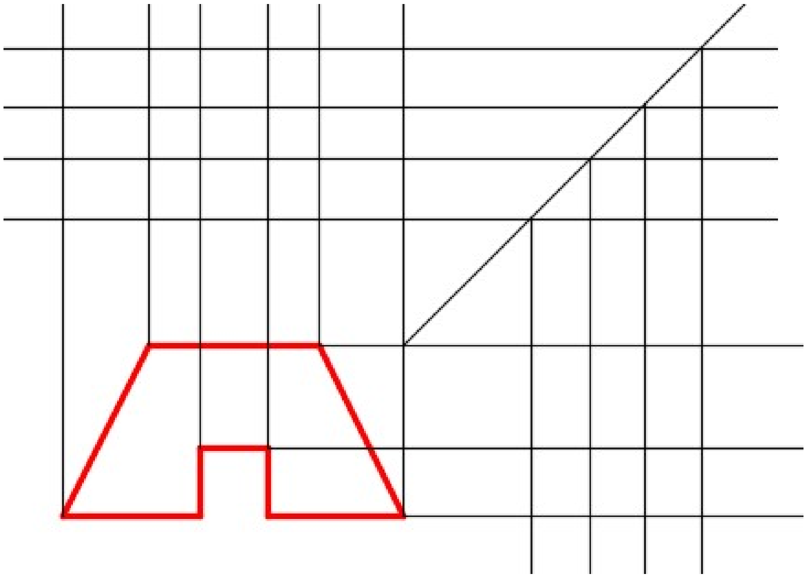


Figure Step 12

Step 13

Trim the lines in the top and Right Side views as shown in the figure. (Figure Step 13)

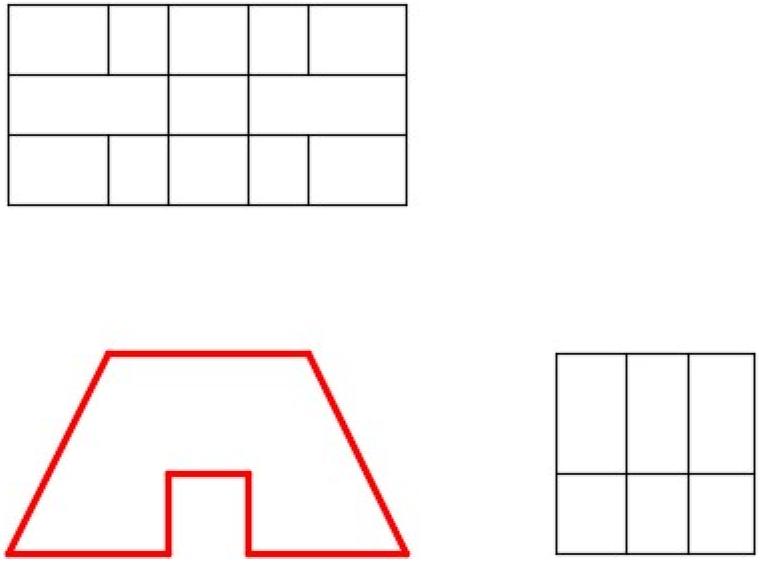


Figure Step 13

Step 14

In the Right Side view, offset the top line 10 millimeters down. With Ortho mode enabled, extend the line across the Front view. (Figure Step 14)

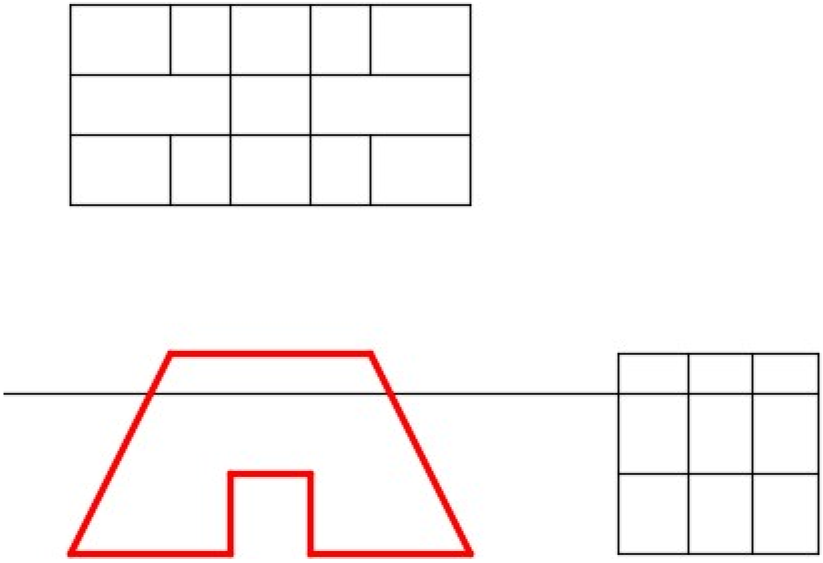


Figure Step 14

Step 15

From the point where the line you drew in Step 14 intersect the incline lines in the Front view, use Ortho mode and draw the lines vertical across the Top view. (Figure Step 15)

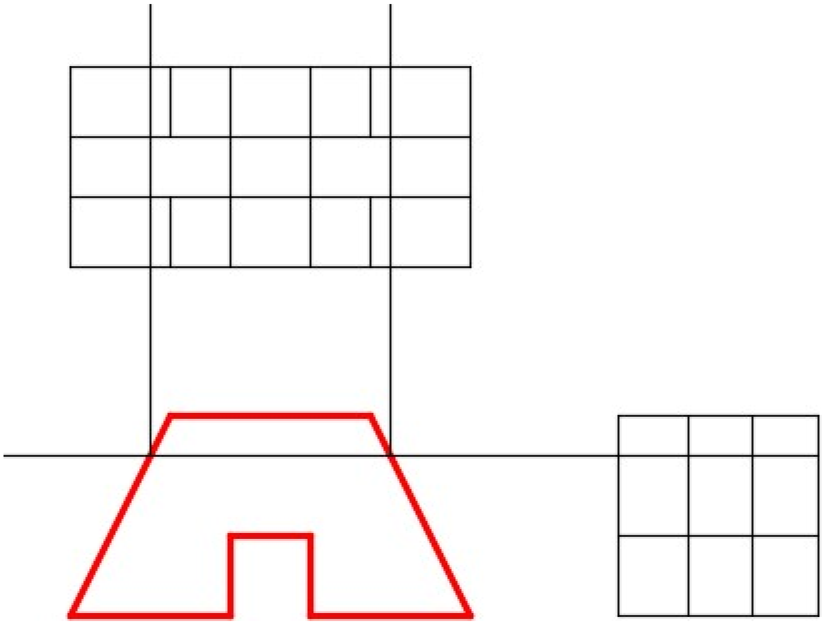


Figure Step 15

Step 16

Trim the lines in the Top and Right Side views to match the figure. (Figure Step 16)

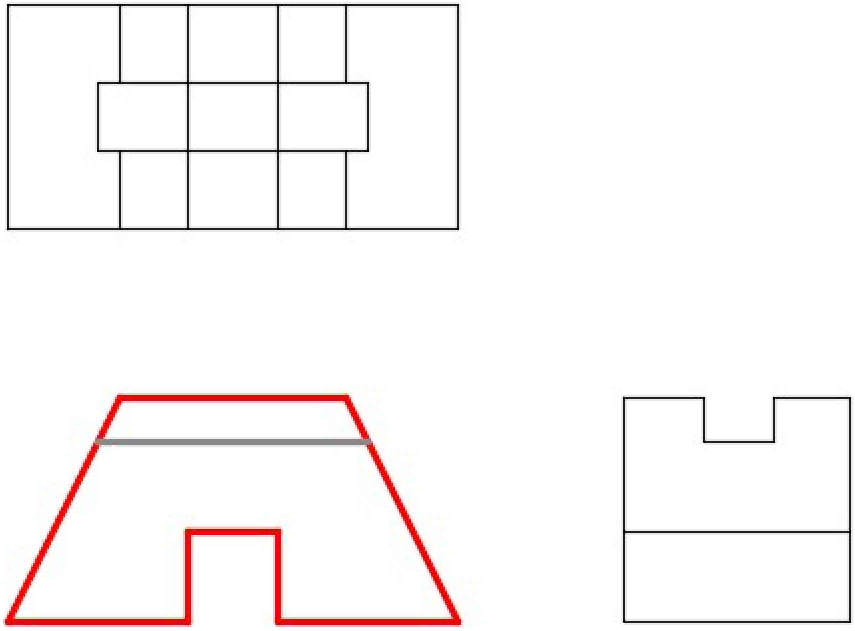


Figure Step 16

Step 17

Change the object lines to layer: Object and the hidden lines to layer: Hidden. (Figure Step 17)

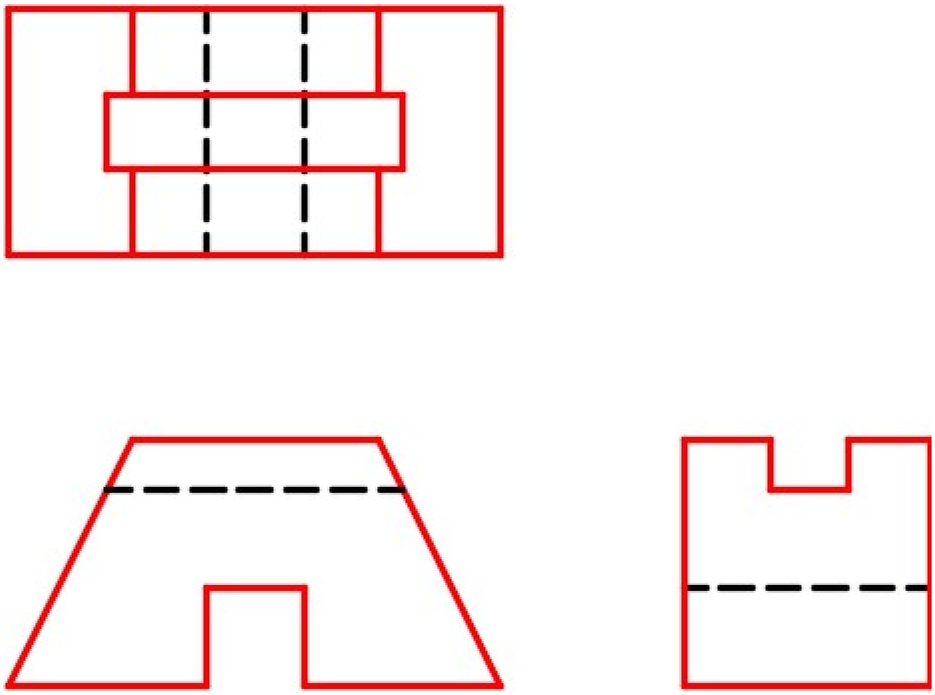


Figure Step 17

Step 18

Save and close the drawing.

Geometry Lesson: Drawing a Three Point Arc

When the radius is not given for an arc, it can be drawn using with three known points on the arc. See Figure 28-3.

Step 1

Draw the construction lines to match the given dimensions.

Step 2

Enter the ARC command as shown below:

Command: **ARC**

Specify start point of arc or [Center]: *(end)* **P1**

Specify second point of arc or [Center/End]: *(mid)* **P2**

Specify end point of arc: *(end)* **P3**

Command:

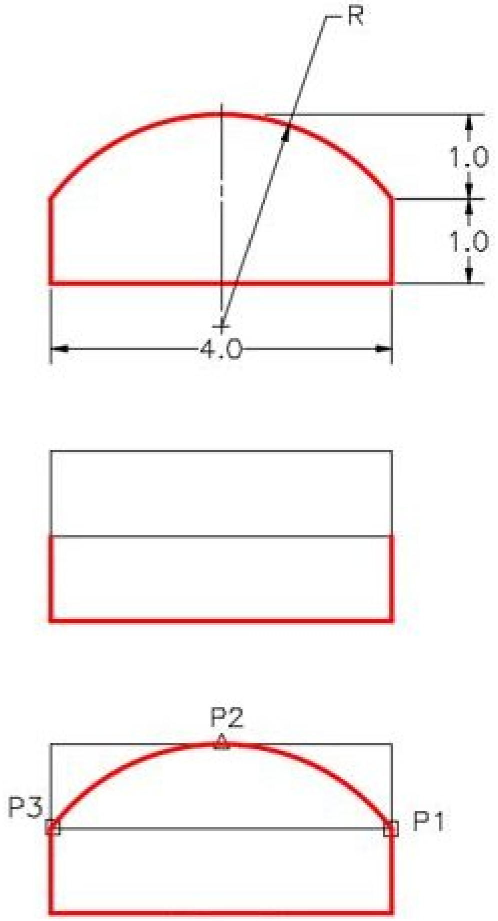


Figure 28-3
Three Point Arc

Key Principles

Key Principles in Module 28

1. If set, the grid always fills the rectangle or square shape defined by the limits of the drawing.
2. The grid display and the snap density do not have to be set to the same size.
3. The grid display is toggled with F7.
4. Snap mode is toggled with F9.
5. When Ortho mode is enabled, coordinate input can be shortcut when drawing lines. You only have to enter the distance along the direction of current axis from the lastpoint.

Lab Exercise 28-1

Time allowed: 45 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 28-1	2D Layout English	Inches

Step 1

Using the figure, draw a multiview drawing of the object. Draw the Top, Front, and Right Side views. (Figure Step 1)

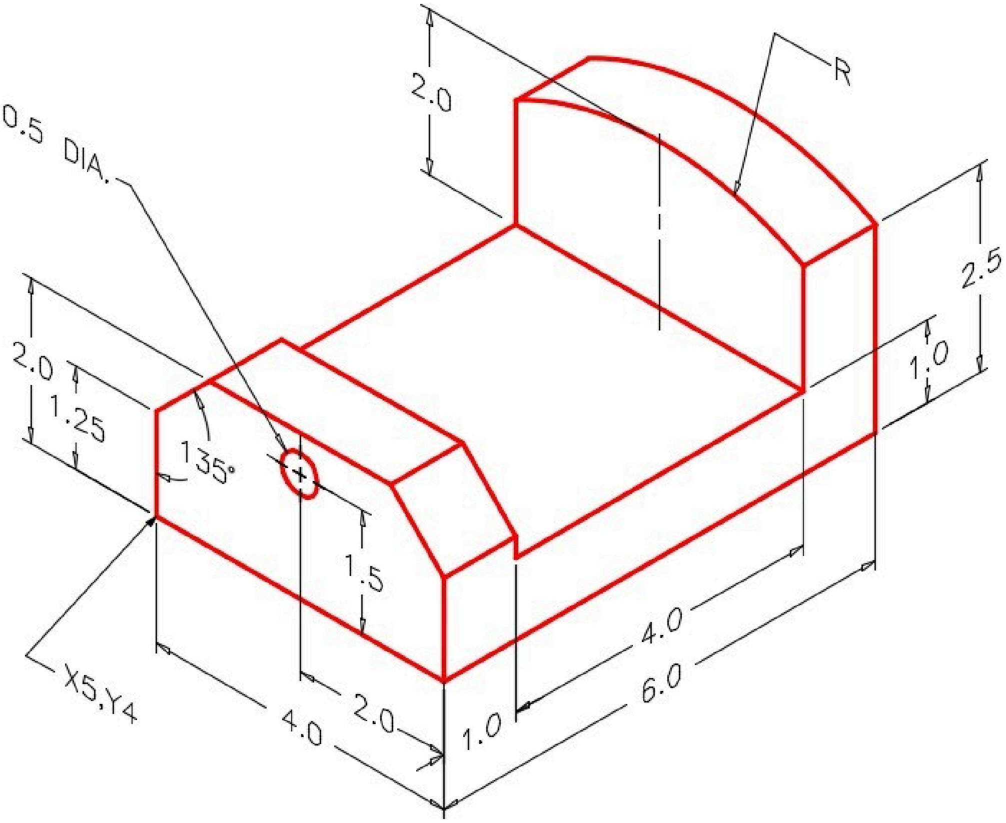

0.5 DIA.
2.0
R
2.5
1.0
2.0
1.25
135°
1.5
X5,Y4
4.0
2.0
1.0
4.0
6.0

Figure Step 1

Step 2

Space the views 2 inches apart. (Figure Step 2)

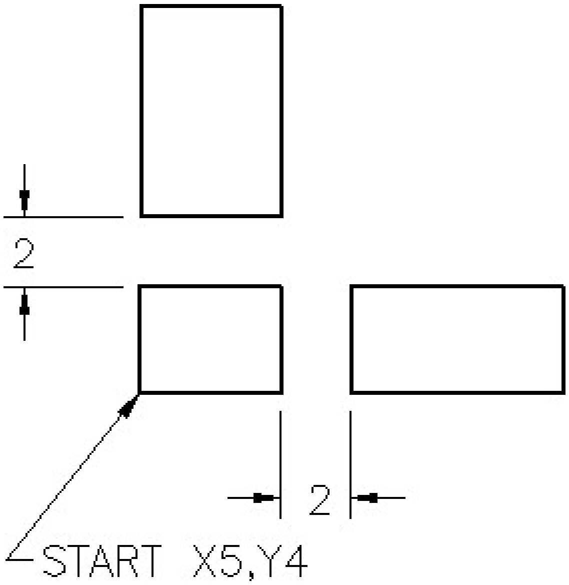


Figure Step 2
Location and Layout of Views

Step 3

Draw all construction lines on layer: Construction, the object lines on layer: Object, and the hidden lines on layer: Hidden.

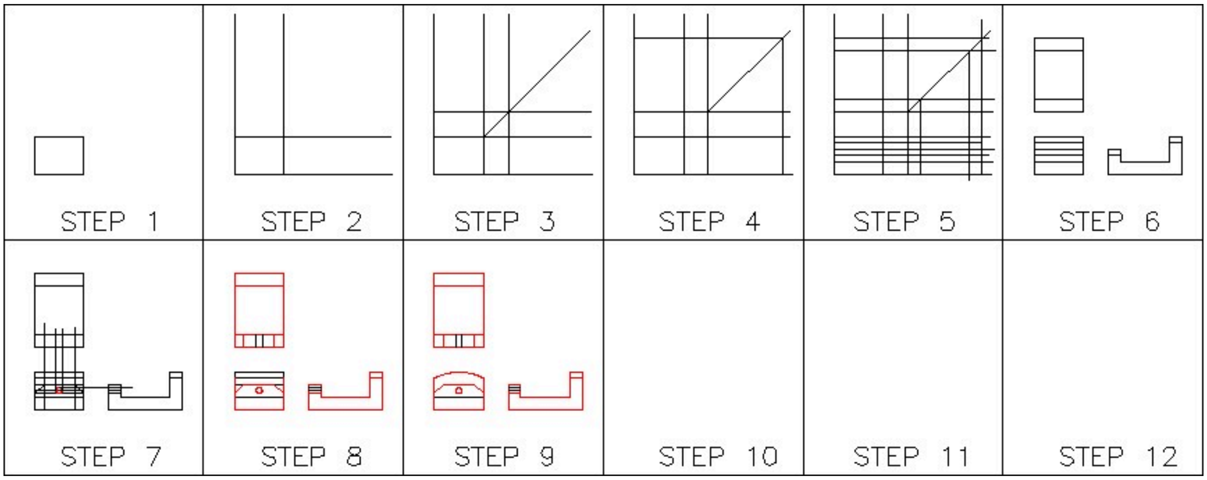
Step 4

Check your drawing with the key.

Step 5

Save and close the drawing.

AUTHOR'S CONSTRUCTION TECHNIQUES: The following steps are the construction technique suggested by the author to help you learn how to construct objects using AutoCAD. It is only the suggested method and if you can complete the drawing accurately using a different construction technique, that is what is important. You may want to compare your construction technique with the authors.


STEP 1
STEP 2
STEP 3
STEP 4
STEP 5
STEP 6
STEP 7
STEP 8
STEP 9
STEP 10
STEP 11
STEP 12

AUTHOR’S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hint. If you get stuck and cannot complete it on your own, use the following hint to help you.

Hint 1

Draw the arc in the Front view using the User Tip at the start of this module. (Figure Hint 1)

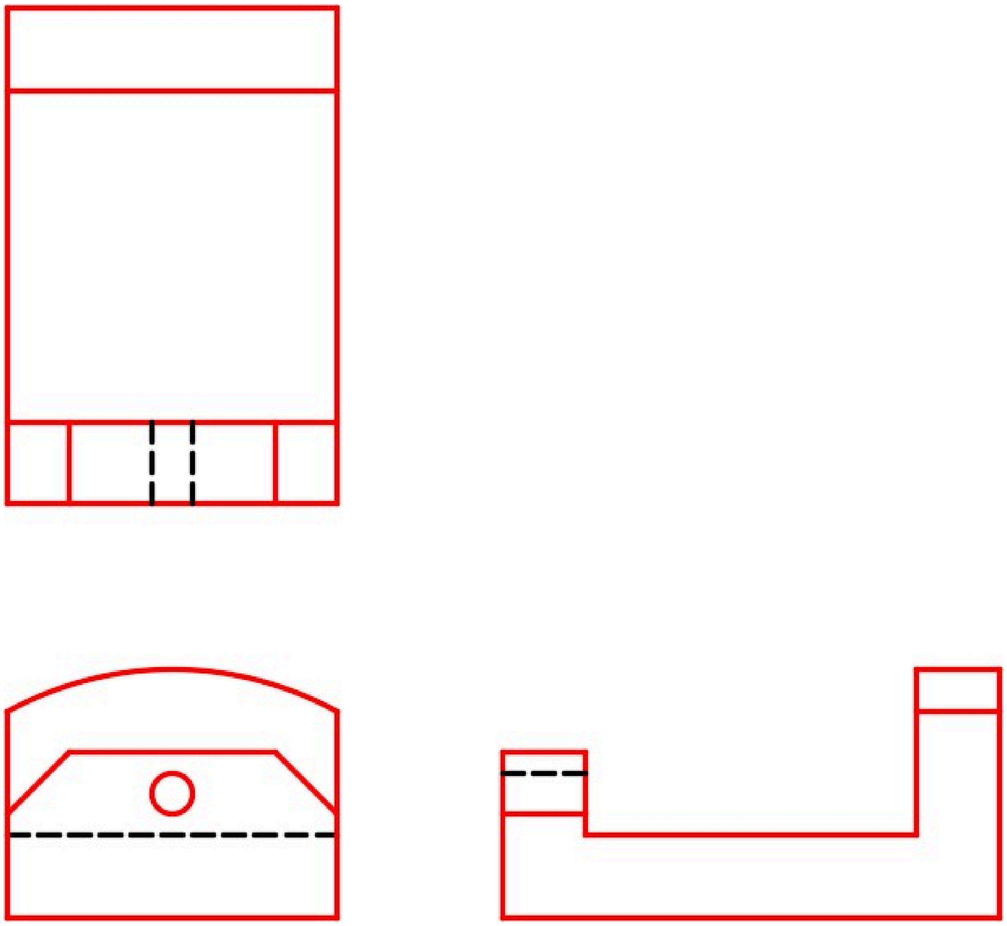


Figure Hint 1

Lab Exercise 28-2

Time allowed: 45 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 28-2	2D Layout English	Inches

Step 1

Using the figure, draw a multiview drawing of the object. Draw the Top, Front and Right Side views. (Figure Step 1),

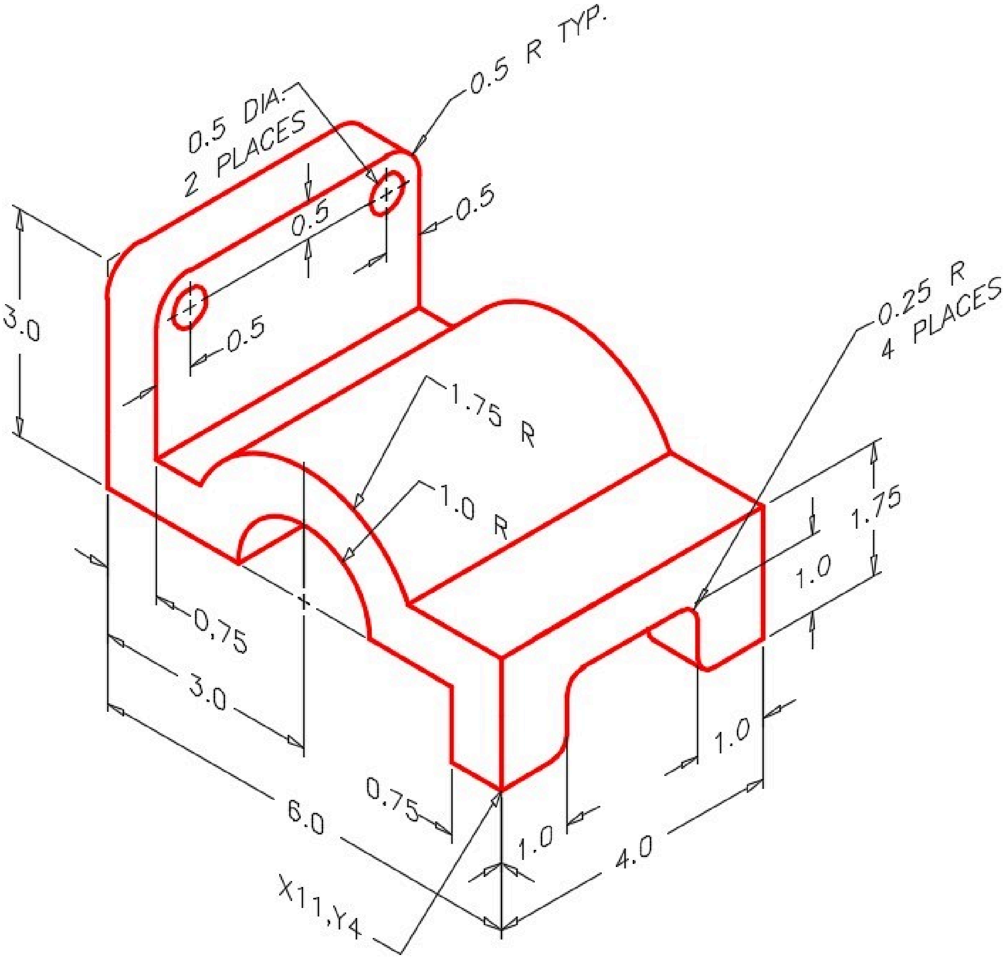

0.5 R TYP.
0.5 DIA.
2 PLACES
0.5
0.5
0.5
3.0
0.5
0.25 R
4 PLACES
1.75 R
1.0 R
1.75
1.0
0.75
3.0
6.0
0.75
1.0
1.0
4.0
X11,Y4

Figure Step 1

Step 2

Space the views 2 inches apart. (Figure Step 2)

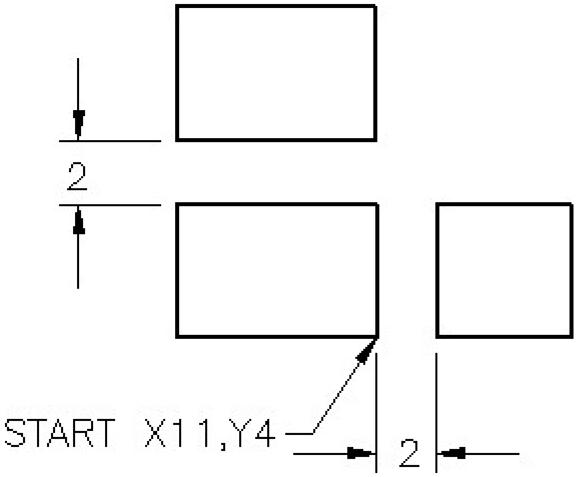


Figure Step 2
Location and Layout of Views

Step 3

Draw all construction lines on layer: Construction, the object lines on layer: Object, and the hidden lines on layer: Hidden.

Step 4

Check your drawing with the key.

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hint. If you get stuck and cannot complete it on your own, use the following hint to help you.

Hint 1

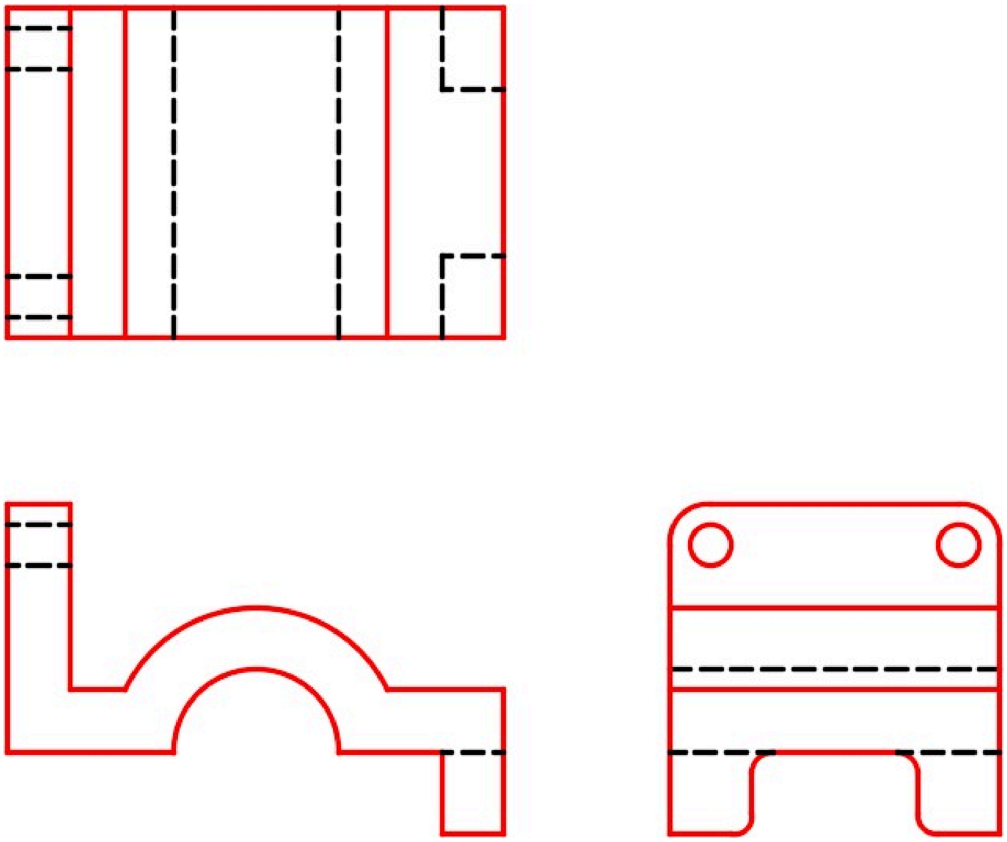


Figure Hint 1

Module 29: Polylines

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe a polyline, a 2D polyline, an open polyline, and a closed polyline.
2. Explain how to create, close, and edit a polyline.
3. Describe a donut.
4. Apply the PLINE, PEDIT, DONUT, FILL, EXPLODE, and BOUNDARY commands.
5. Explain how to work in feet and inches.

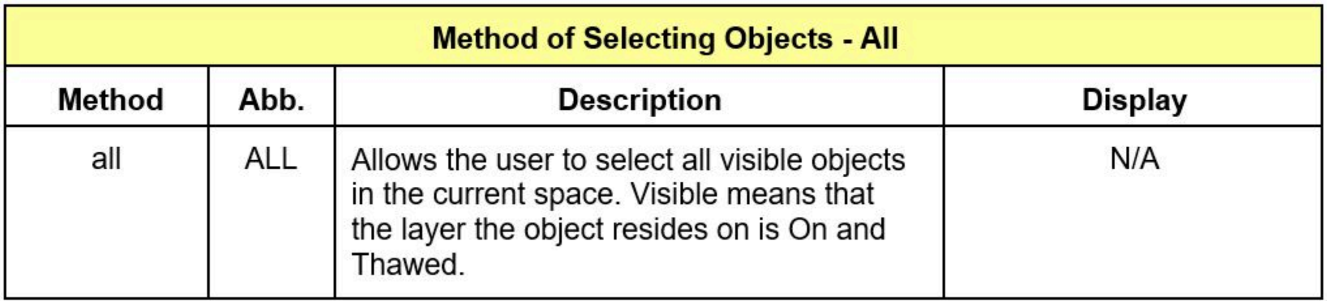
Polylines

A *polyline* is a series of line and/or arc segments connected to create a single drawing object. Polylines are created using the PLINE or the PEDIT command from existing straight line segments, arc segments, or a combination of the two. A polyline, usually called a *pline*, can be edited with the PEDIT command or the Properties window. A polyline can be given a width or can be converted back to the individual drawing object segments by exploding it with the EXPLODE command.

2D or Lightweight Polyline

Since the release of AutoCAD 14, polylines have been created as *lightweight polylines*. Sometimes called a *2D polyline*, a lightweight polyline is greatly improved over the older heavyweight or *3D polyline* which was used before AutoCAD 14. Using 2D polylines allows AutoCAD to regenerate the drawing faster, which in turn allows you to work faster.

In this module, only 2D polylines are taught. 3D polylines are taught in the AutoCAD 3D book. In the AutoCAD 2D book, the terms pline or polyline refers to a 2D polyline. A 3D polyline is called a 3D polyline.

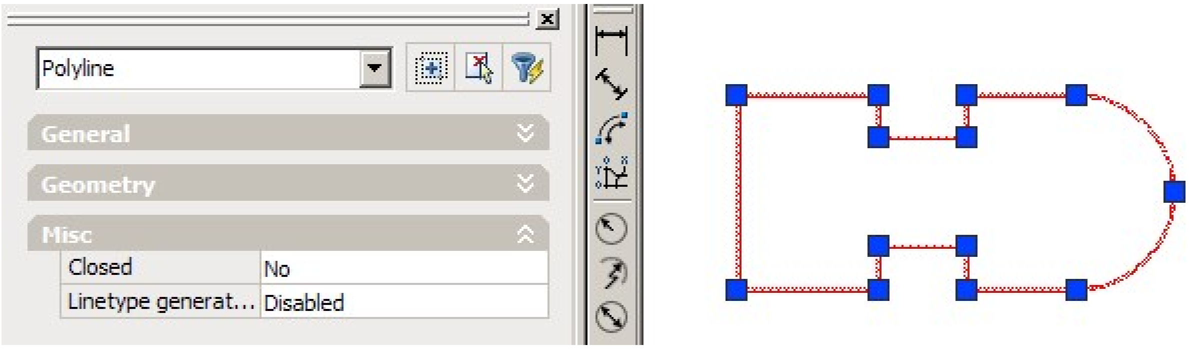


Method of Selecting Objects - All			
Method	**Abb.**	**Description**	**Display**
all	ALL	Allows the user to select all visible objects in the current space. Visible means that the layer the object resides on is On and Thawed.	N/A

Figure 29-1
Selecting Objects – All

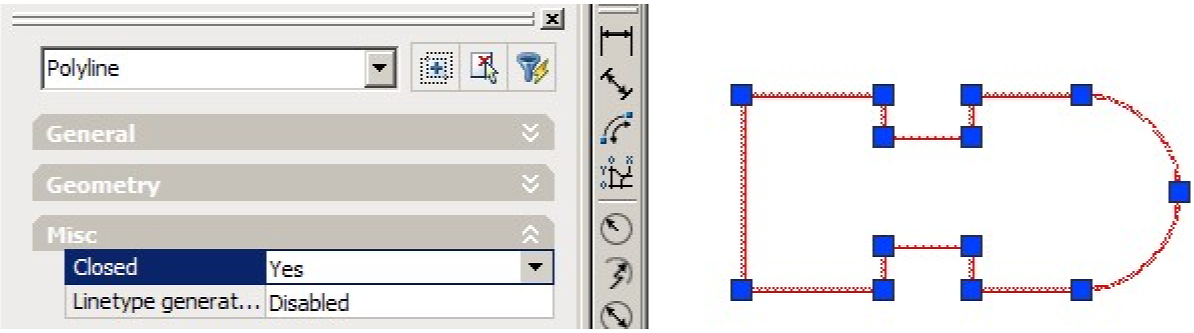
An Open Verses a Closed Polyline

A *closed polyline* creates a polygon. A closed polyline or a *polygon* has many different uses in an AutoCAD drawing but may appear exactly the same as an *open polyline*. It is important for you to be able to verify if a polyline is opened or closed. Polygons are assigned a Closed property. To check or change the Closed polyline property, use the Properties window. Figure 29-2 shows the selected polyline as open. On the other hand, Figure 29-3 shows how the polyline property has been changed to close the polyline.



Closed: No
Linetype generat…: Disabled

Figure 29-2
An Open Polyline



Closed: Yes
Linetype generat…: Disabled

Figure 29-3
A Closed Polyline

Creating Polylines from Existing Objects

Polylines can be drawn from scratch, using the PLINE command, or they can be created from existing objects using the PEDIT or the BOUNDARY command. The PEDIT command is used to convert existing lines and arcs into polylines and the BOUNDARY command will create a closed polyline object using enclosed areas of existing lines, arcs, and/or polylines.

Editing Polylines

To edit a polyline, use the PEDIT command or the Properties window. The AutoCAD 2D book does not cover all of the editing features of PEDIT as they are every extensive. The AutoCAD 3D book covers some of the more advanced options.

A Vertex

Each endpoint of a line or an arc segment in a polyline is called a *vertex*. A polyline changes direction at each vertex. See Figure 29-4.

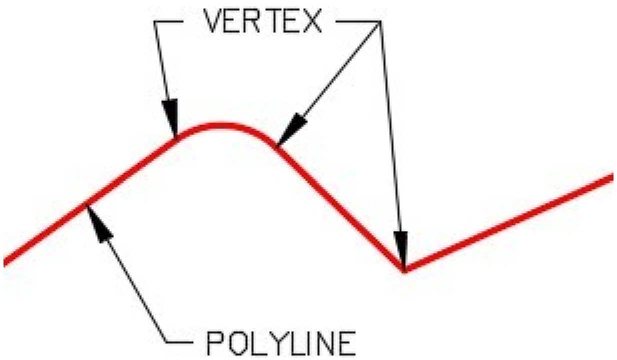
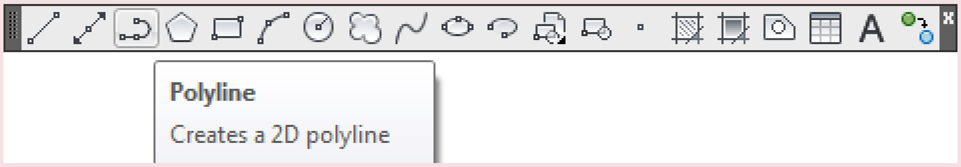


Figure 29-4
A Vertex

AutoCAD Command: PLINE

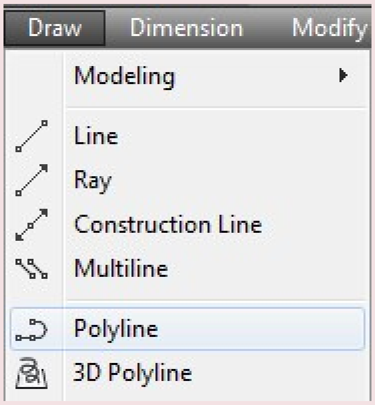
The PLINE command is used to create a connected sequence of line and/or arc segments to form a single object. A closed polyline is a polygon.

Shortcut: **PL**

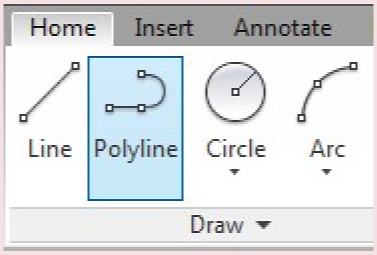


Polyline
Creates a 2D polyline

Draw Toolbar


Draw
Dimension
Modify
Modeling
Line
Ray
Construction Line
Multiline
Polyline
3D Polyline

Draw Pull-down

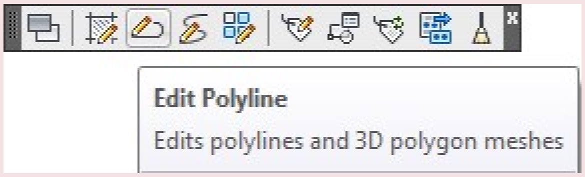

Home
Insert
Annotate
Line
Polyline
Circle
Arc
Draw

Home Ribbon

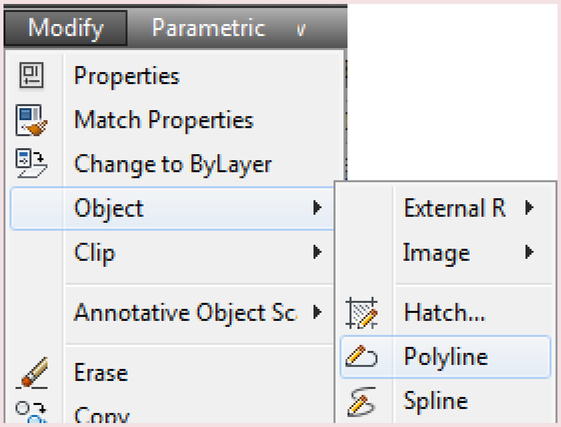
AutoCAD Command: PEDIT

The PEDIT command is used to edit existing polylines or create new polylines.

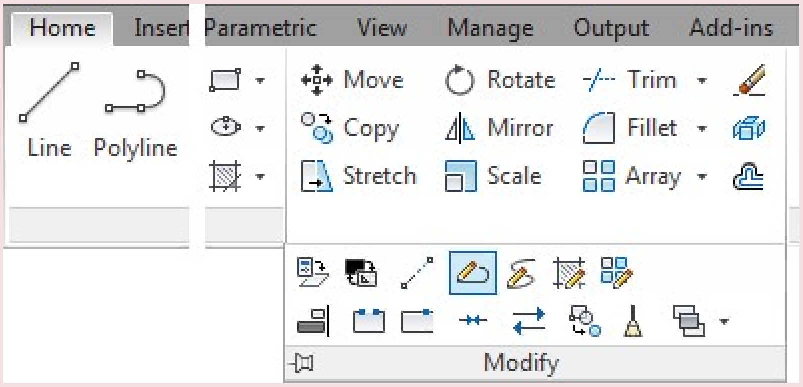
Shortcut: **PE**


Edit Polyline
Edits polylines and 3D polygon meshes

Modify II Toolbar


Modify
Parametric
Properties
Match Properties
Change to ByLayer
Object
Clip
Annotative Object Sc
Erase
Copy
External R
Image
Hatch...
Polyline
Spline

Modify Pull-down


Home
Insert
Parametric
View
Manage
Output
Add-ins
Line
Polyline
Move
Rotate
Trim
Copy
Mirror
Fillet
Stretch
Scale
Array
Modify

Home Ribbon

USER TIP: Try to avoid using the PLINE command to create polylines. They are much easier to create by drawing the individual line and arc objects using the LINE and ARC commands. Make sure that they connect exactly at their endpoints. Using either the PEDIT or BOUNDARY command, create the desired polyline using the lines and arcs you created.

WORKALONG: Using the PLINE and PEDIT Commands

Step 1

Start a new drawing using the template: 2D Layout English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 29-1.

Step 3

On layer: Object, enter the PLINE command, as shown below, to draw a square 2 inches by 2 inches with a closed 2D polyline. (Figure Step 3)

```
Command: PLINE

Specify start point 1,8

Current line-width is 0.0000

(The width should be 0.)

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: @2,0

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: @0,2

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: @-2,0

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: C

(Closing the pline.)

Command:
```

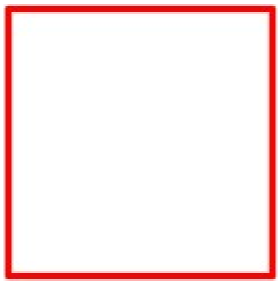


Figure Step 3

Step 4

Without entering a command, select the polyline you drew in Step 3. Note how it is one drawing object. (Figure Step 4)

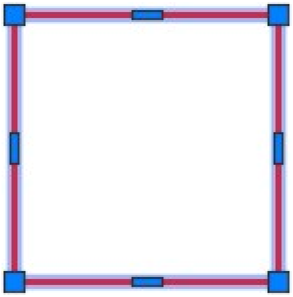


Figure Step 4

Step 5

Press Esc to unselect the pline. Enter the PLINE command, as shown below, to draw another polyline inside the first one. In this command, you will be assigning a width of 0.1 to the pline. (Figure Step 5A and 5B)

Command: **PLINE**

Specify start point: *(mid)* **P1**

Current line-width is 0.0000

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **W**

Specify starting width <0.000>: **0.1**

(Setting the width of the polyline to 0.1.)

Specify ending width <0.1000>:

(Press enter to set both ends of the pline to the same width.)

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: *(mid)* **P2**

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: *(mid)* **P3**

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: *(mid)* **P4**

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **C**

Command:

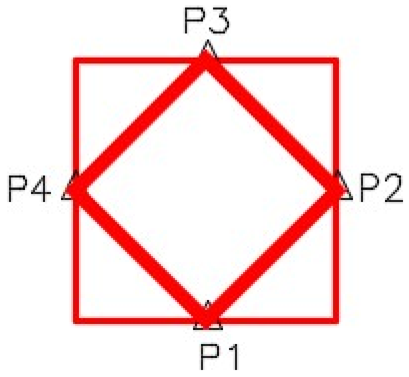


Figure Step 5A

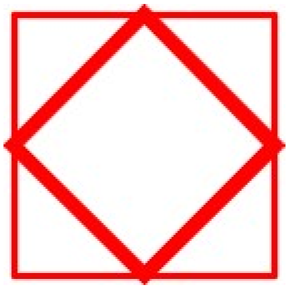


Figure Step 5B

AUTHOR'S COMMENTS: Note how a pline can have a 0 width or be assigned a width.

Step 6

Enter the PEDIT command, as shown below, to change the width of an existing pline. (Figure Step 6A and 6B)

Command: **PEDIT**

Select polyline or [Multiple]: **P5**

Enter an option [Open/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: **W**

(Select the Width option.)

Specify new width for all segments: **0.05**

(Set the width to 0.05 units.)

Enter an option [Open/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]:

Command:

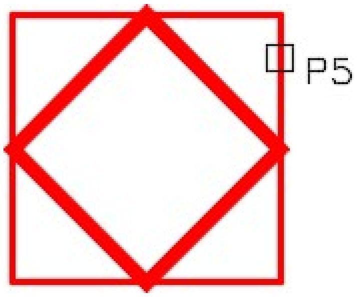


Figure Step 6A

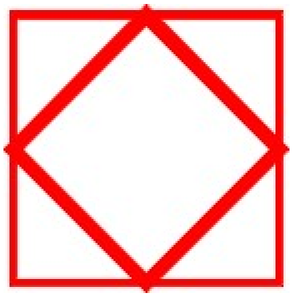
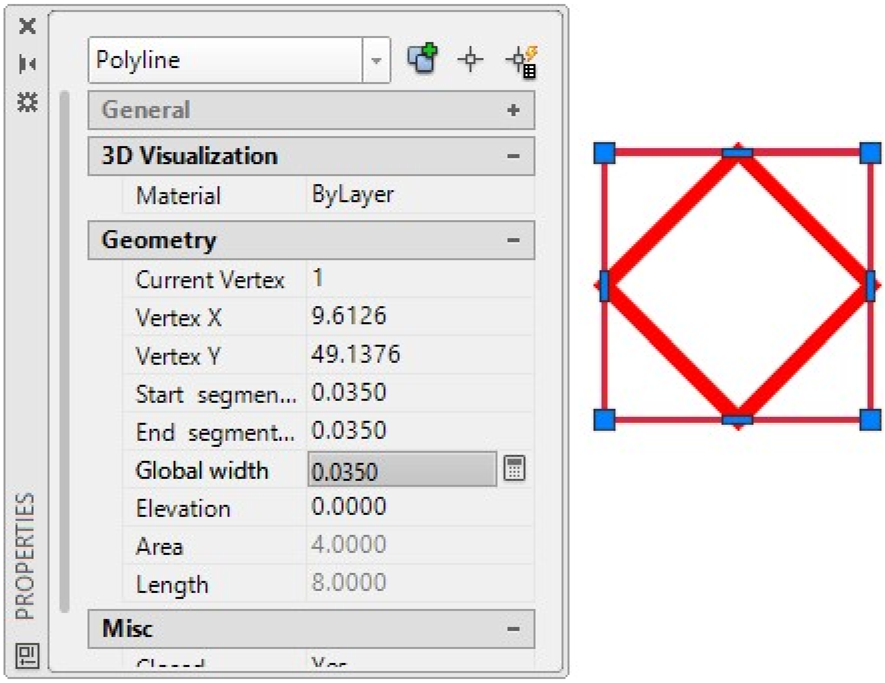


Figure Step 6B

Step 7

Open the Properties window. Without entering a command, select the outside polyline. In the Properties window, change the Global width property to 0.035. (Figure Step 7)



Material: ByLayer
Current Vertex: 1
Vertex X: 9.6126
Vertex Y: 49.1376
Start segmen...: 0.0350
End segment...: 0.0350
Global width: 0.0350
Elevation: 0.0000
Area: 4.0000
Length: 8.0000

Figure Step 7

Step 8

Enter the PLINE and PEDIT commands, as shown below, to create an open polyline and then close it. (Figure Step 8A and 8B)

Command: **PLINE**

Specify start point: **5,8** Current line-width is **0.04**

(Set the width to 0.04)

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **@2,0**

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **@0,2**

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **@-2,0**

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]:

(This is an open pline.)

Command: **PEDIT**

Select polyline or [Multiple]: **P6**

Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: **C**

(Closing the polyline. Note how the last section has been added by this command.)

Enter an option [Open/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]:

Command:

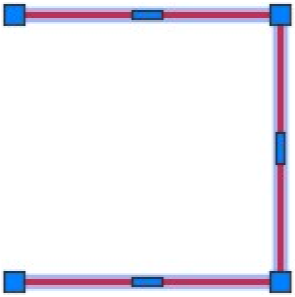


Figure Step 8A

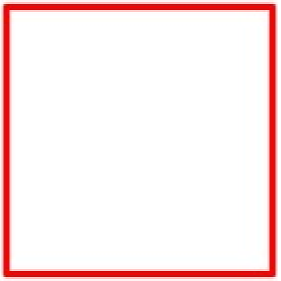


Figure Step 8B

Step 9

Your drawing should now appears a shown in the figure. (Figure Step 9)

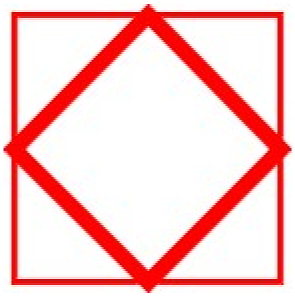
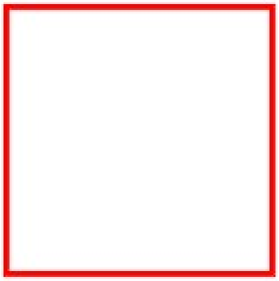


Figure Step 9



Step 10

On layer: Object, draw the object shown in the figure. Using what you learned earlier in the book, draw it using lines and circles. Trim the circles to complete it. Your drawing should now appear as shown in the figure. (Figure Step 10A and 10B)

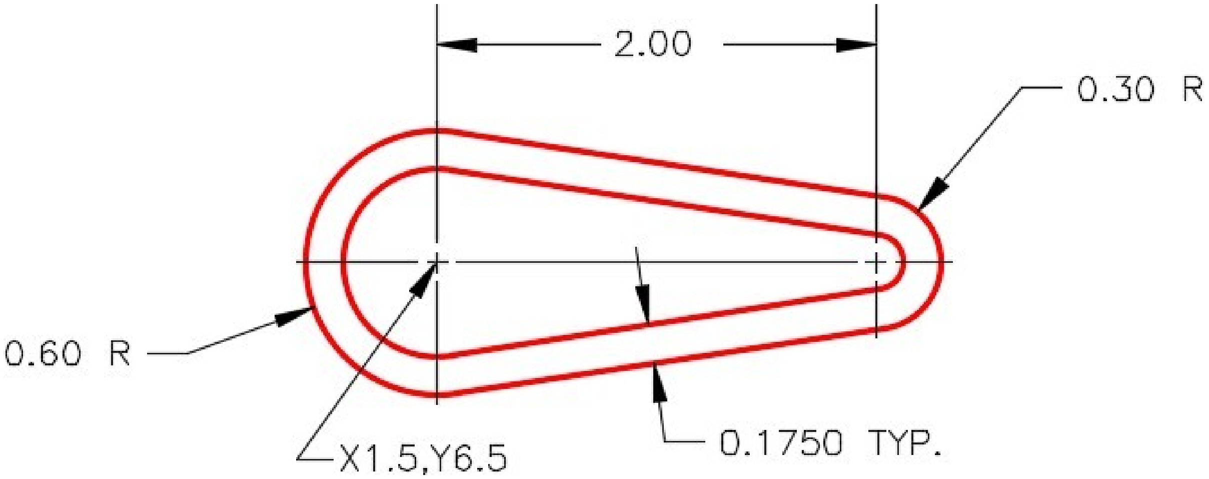


Figure Step 10A

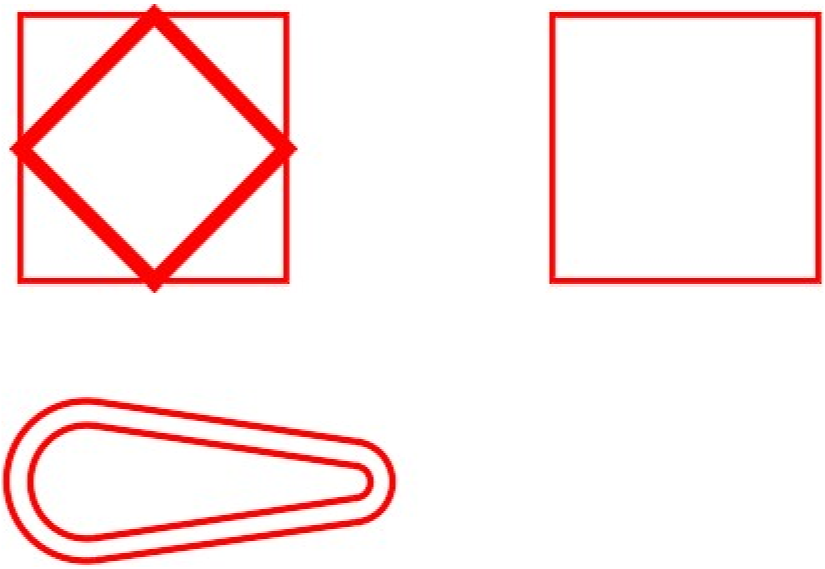


Figure Step 10B

AUTHOR'S COMMENTS: Ensure that you draw the lines tangent at both ends. If you are not sure how to do that, see Module 12. Trim the circles to complete it. See Figure 29-5.

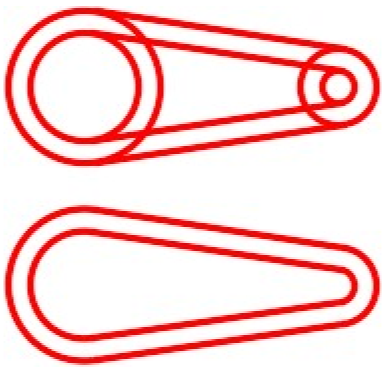


Figure 29-5

Step 11

Enter the PEDIT command, as shown below, to create a polyline using existing objects. (Figure Step 11A and 11B)

Command: **PEDIT**

Select polyline or [Multiple]: **P7**

(Select one of the objects.)

Object selected is not a polyline

Do you want to turn it into one? <Y> **Y**

Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: **J**

(Enter the Join option.)

Select objects: **P8** 1 found

Select objects: **P9** 1 found, 2 total

Select objects: **P10** 1 found, 3 total

Select objects:

3 segments added to polyline

(Joining the 4 objects to form one pline.)

Enter an option [Open/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]:

Command:

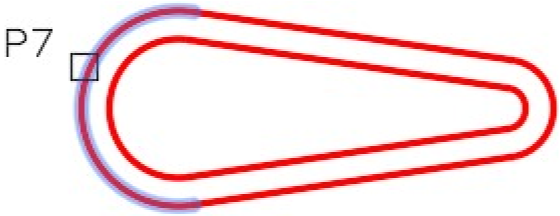


Figure Step 11A

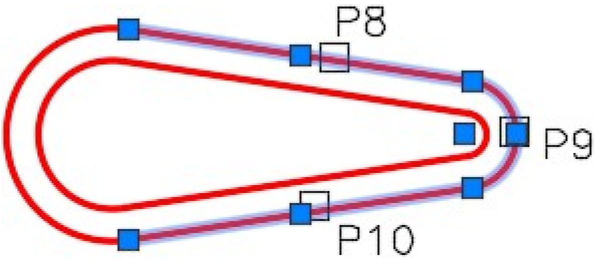
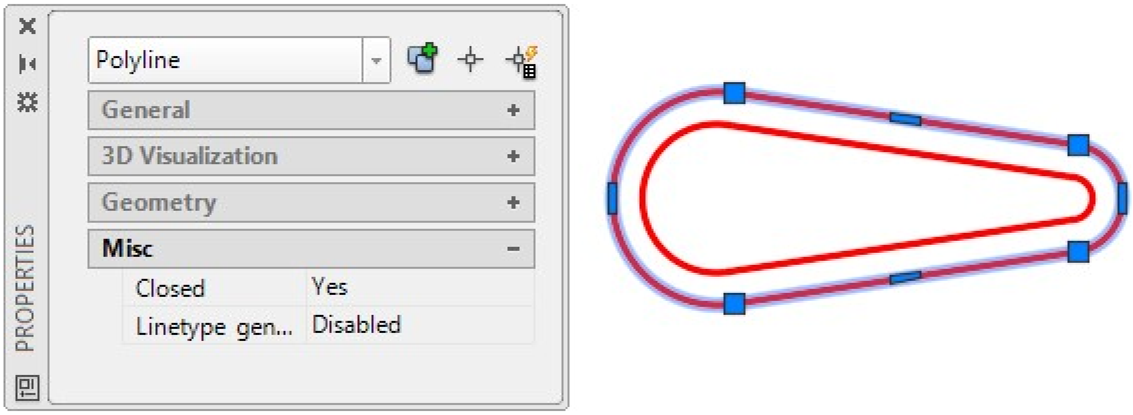


Figure Step 11B

Step 12

Without entering in a command, select the polyline that you created in Step 11. In the Properties window, check to ensure that it is a closed polyline. (Figure Step 12)


PROPERTIES
Polyline
General
3D Visualization
Geometry
Misc

Closed Yes

Linetype gen... Disabled

Figure Step 12

Step 13

Enter the PEDIT command as shown below. Using the ALL object selection method, create a polyline with the joining objects. (Figure Step 13)

Command: **PE**

Select polyline or [Multiple]: **P11**

Object selected is not a polyline

Do you want to turn it into one? <Y>

(Press enter to accept Yes.)

Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: **J**

Select objects: **ALL**

(Selecting all of the visible objects in the drawing.)

Command:

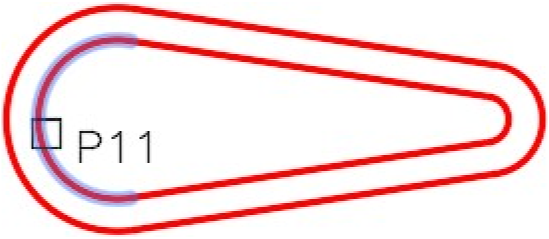
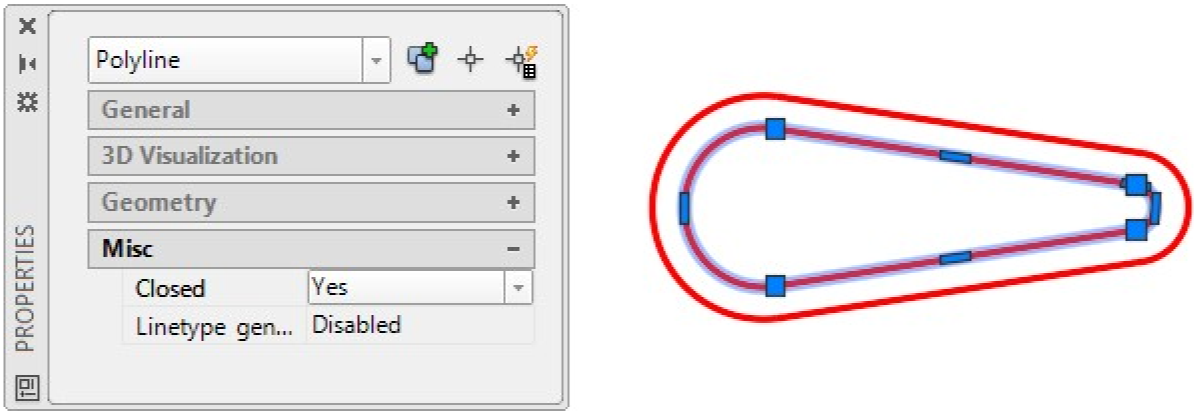


Figure Step 13

AUTHOR'S COMMENTS: The PEDIT command will find the objects that are connected exactly at their endpoints and join them to form the polyline.

Step 14

Using the Properties window, check to ensure that the polyline you created in Step 13 was created and is closed. Your drawing should appear as shown in the figure. (Figure Step 14)



Closed: Yes
Linetype gen...: Disabled

Figure Step 14

Step 15

Save and close the drawing.

Donuts

A *donut* is simply a circular closed polyline, with an assigned width. Donuts are created with the DONUT command. They can be drawn with a width or completely filled in so that they appear as a solid circle. Since a donut is a polyline, it can be edited with the PEDIT command and exploded with the EXPLODE command. See Figure 29-6 for three donut examples.

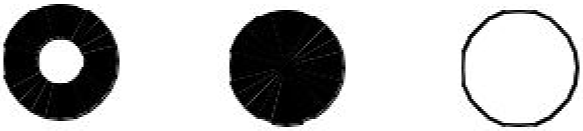
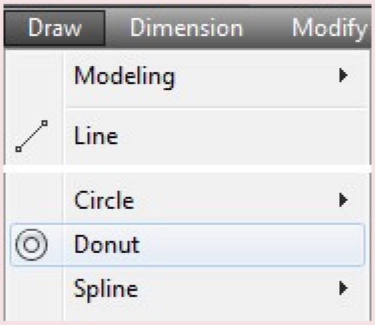


Figure 29-6
Donuts

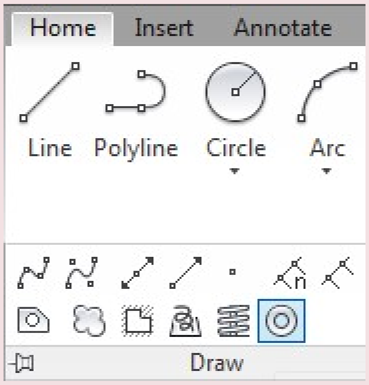
AutoCAD Command: **DONUT**

The DONUT command is used to create donuts.

Shortcut: **DO**


Draw
Dimension
Modify
Modeling
Line
Circle
Donut
Spline

Draw pull-down

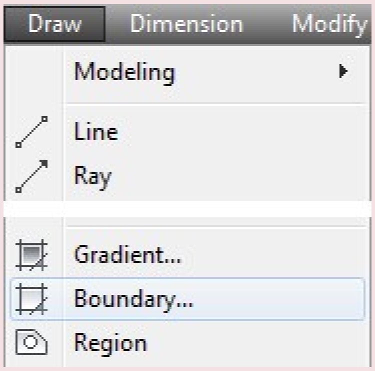

Home
Insert
Annotate
Line
Polyline
Circle
Arc
Draw

Home Ribbon

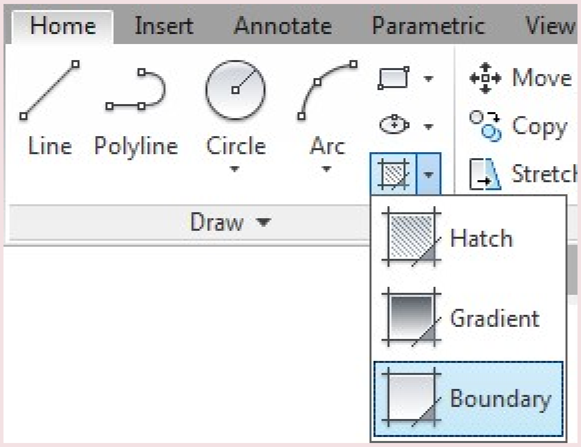
AutoCAD Command: **BOUNDARY**

The BOUNDARY command is used to create a closed polyline using existing objects that form an enclosed area on the drawing.

Shortcut: **BO**


Draw
Dimension
Modify
Modeling
Line
Ray
Gradient...
Boundary...
Region

Draw Pull-down


Home
Insert
Annotate
Parametric
View
Line
Polyline
Circle
Arc
Move
Copy
Stretch
Draw
Hatch
Gradient
Boundary

Home Ribbon

AutoCAD Command: **FILL**

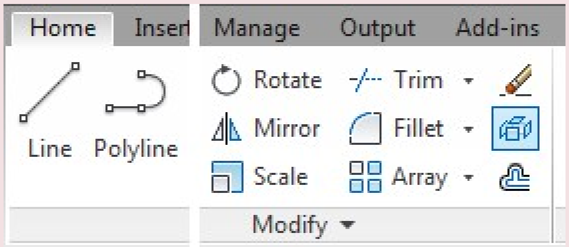
The FILL command is used to toggle the solid fill display inside polylines, solid, and hatches.

Shortcut: none

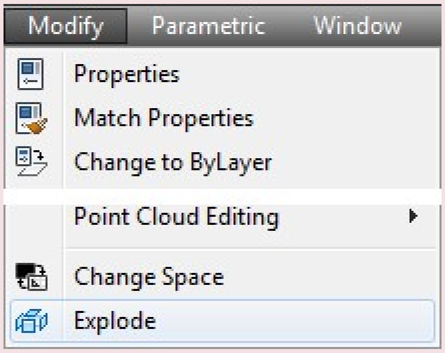
AutoCAD Command: **EXPLODE**

The EXPLODE command is used to convert polylines plus many other AutoCAD objects back into individual objects.

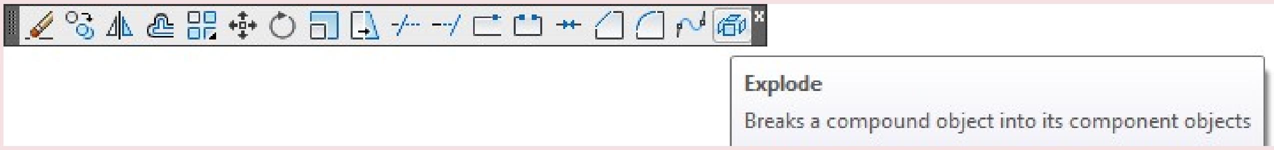
Shortcut: **X**


Home
Insert
Manage
Output
Add-ins
Line
Polyline
Rotate
Trim
Mirror
Fillet
Scale
Array
Modify

Home Ribbon


Modify
Parametric
Window
Properties
Match Properties
Change to ByLayer
Point Cloud Editing
Change Space
Explode

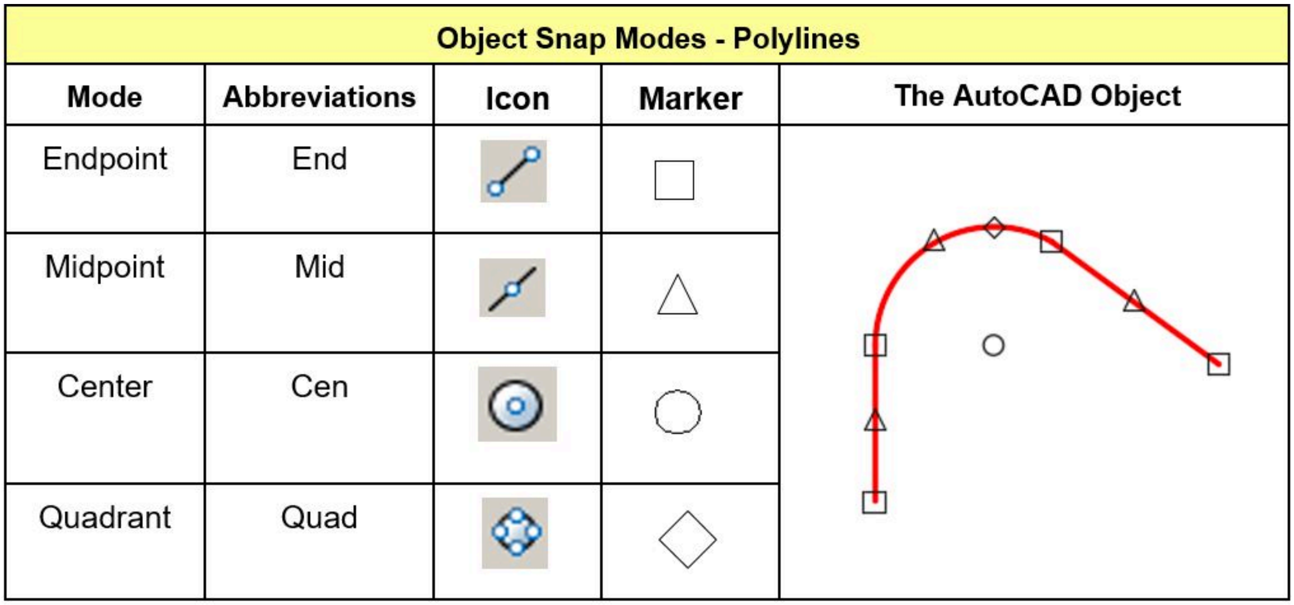
Modify Pull-down



Explode

Breaks a compound object into its component objects

Modify Toolbar



Object Snap Modes - Polylines				
Mode	**Abbreviations**	**Icon**	**Marker**	**The AutoCAD Object**
Endpoint	End		□	
Midpoint	Mid		△	
Center	Cen		○	
Quadrant	Quad		◇	

Figure 29-7
Object Snap Modes for Polylines

WORKALONG: Using the DONUT, FILL, EXPLODE and BOUNDARY Commands

Step 1

Open the drawing: AutoCAD 2D Workalong 29-1. Using the SAVEAS command, save the drawing with the name: AutoCAD 2D Workalong 29-2. (Figure Step 1)

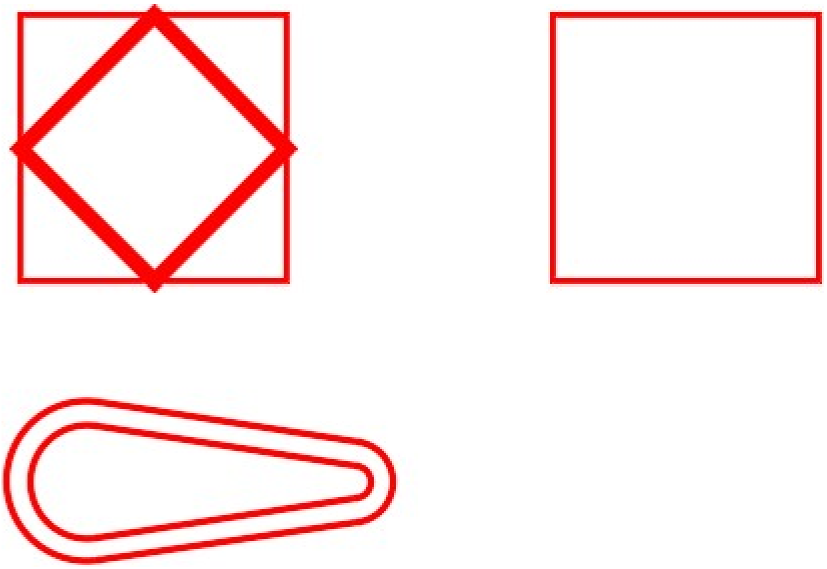


Figure Step 1

Step 2

On layer: Construction, draw a construction line diagonally from corner to corner in both squares as shown in the figure. (Figure Step 2)

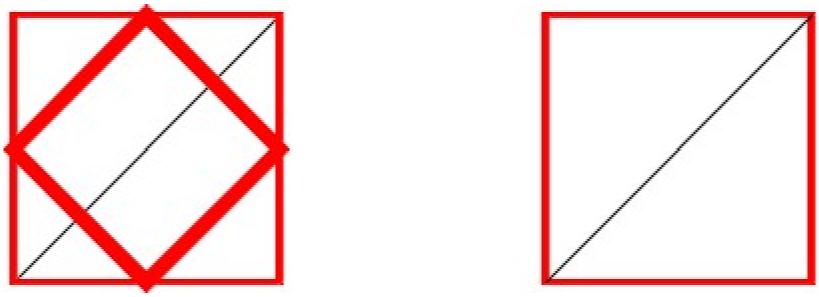


Figure Step 2

Step 3

Set layer: Object as the current layer. Enter the DONUT command, as shown below, to draw two donuts with their centers located in the center of the squares. Snap to the midpoint of the construction lines when locating the centers. (Figure step 3A and 3B)

Command: **DONUT**

Specify inside diameter of donut <0.5000>: **0**

(Setting the inside diameter to 0.)

Specify outside diameter of donut <0.8000>: **1**

(Setting the outside diameter to 1.)

Specify center of donut or <exit>: *(mid)* **P1**

Specify center of donut or <exit>:

Command: **DONUT**

Specify inside diameter of donut <0.0000>: **.5**

(Setting the inside diameter to 0.5)

Specify outside diameter of donut <1.0000>:

(Press enter to accept default.)

Specify center of donut or <exit>: *(mid)* **P2**

Specify center of donut or <exit>:

Command:

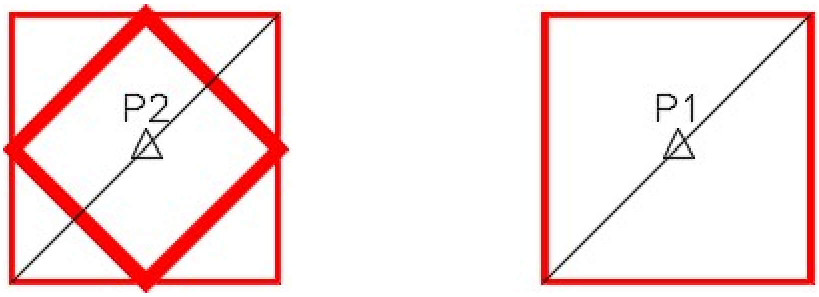


Figure Step 3A

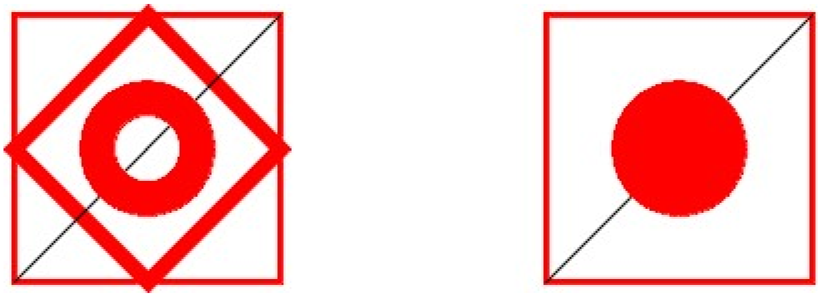


Figure Step 3B

Step 4

Turn layer: <u>Construction</u> off and enter the commands below to see the results of using the FILL command. (Figure Step 4A and 4B

```
Command: FILL

Enter mode [ON/OFF] <ON>: OFF

(Disabling fill mode.)

Command: REGEN

(Regenerating to display fill mode.)

REGEN Regenerating model. Command: FILL

Enter mode [ON/OFF] <OFF>: ON

(Enabling fill mode.)

Command: REGEN

(Regenerating to display fill mode.)

REGEN Regenerating model.

Command:
```

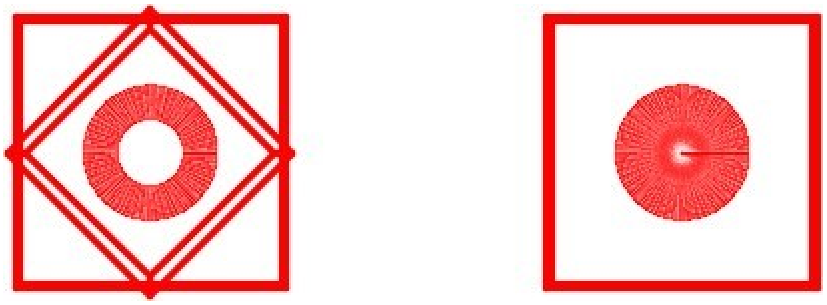


Figure Step 4A

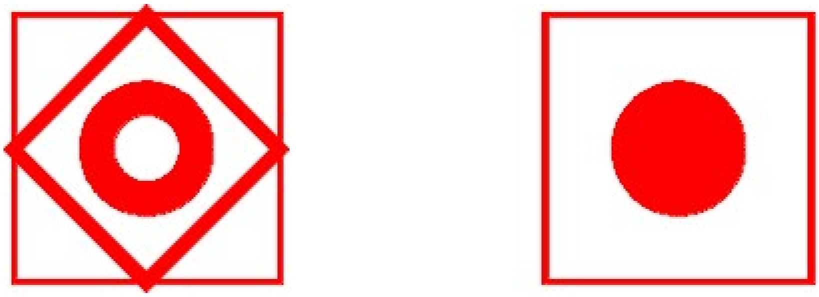


Figure Step 4B

AUTHOR’S COMMENTS: Enabling and disabling fill can help you become more productive. When fill is disabled, a drawing will regenerate faster. Fill can them be enabled after you complete the drawing.

Step 5

Using what you learned earlier in the book, mirror the lower object to match the dimensions shown in the figure. (Figure Step 5)

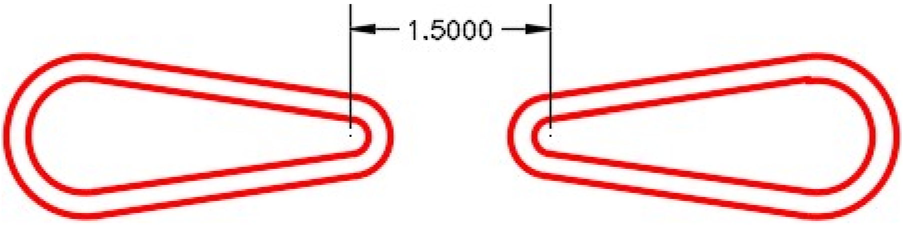


Figure Step 5

Step 6

Enter the EXPLODE command, as shown below, to explode the 5 plines back to their individual objects. (Figure Step 6A and 6B)

Command: **EXPLODE**

Select objects: **P3** 1 found

Select objects: **P4** 1 found, 2 total

Select objects: **P5** 1 found, 3 total

Select objects: **P6** 1 found, 4 total

Select objects: **P7** 1 found, 5 total

Select objects:

Command

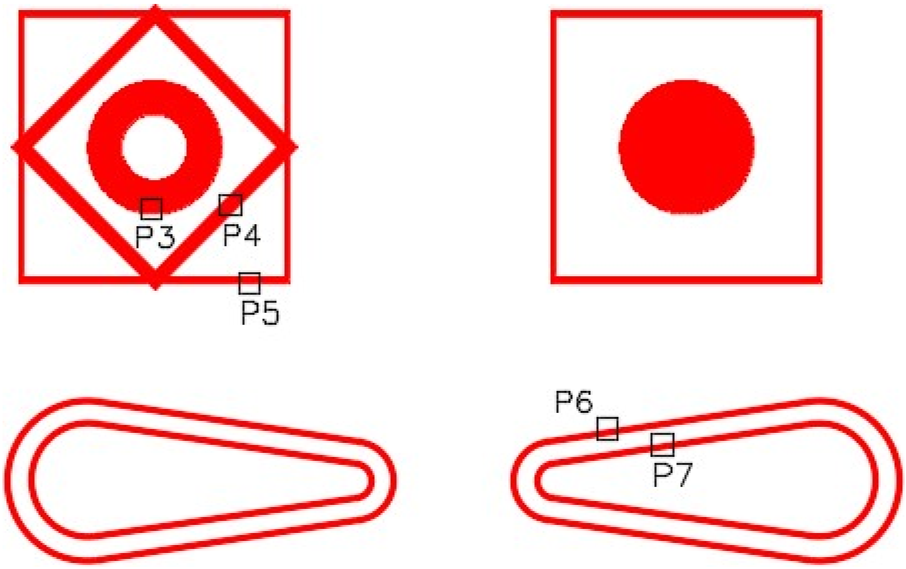


Figure Step 6A

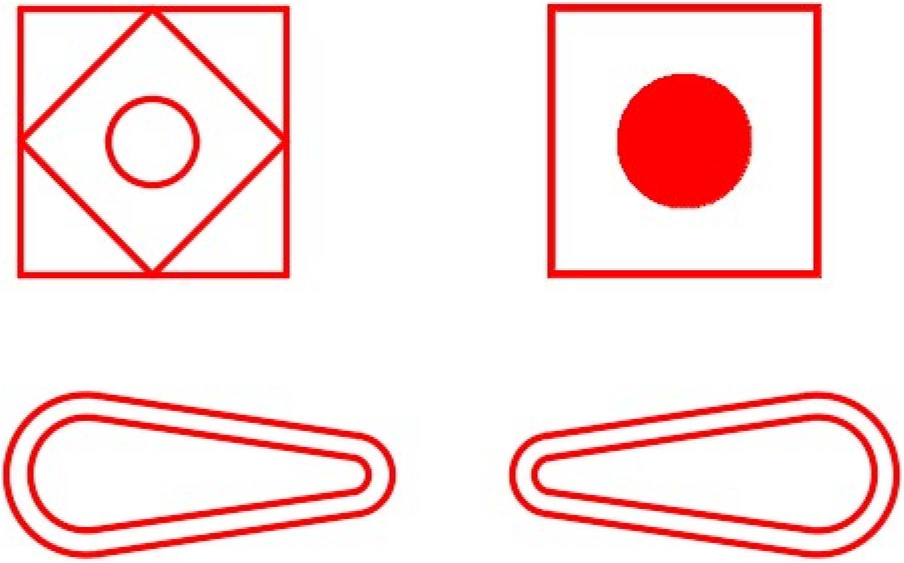


Figure Step 6B

AUTHOR’S COMMENTS: An exploded pline will lose its assigned width and all plines and donuts will be converted back to lines, circles and arcs.

Step 7

Erase the inside arcs and lines from the object on the right as shown in the figure. (Figure Step 7).

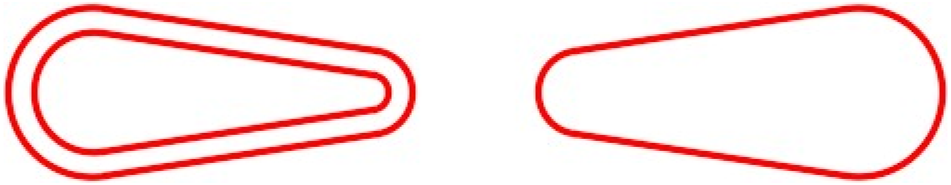


Figure Step 7

Step 8

Copy the object on the right locating the center of the large radius at X5,Y3.5 and then rotate it -45 degrees as shown in the figure. (Figure Step 8)

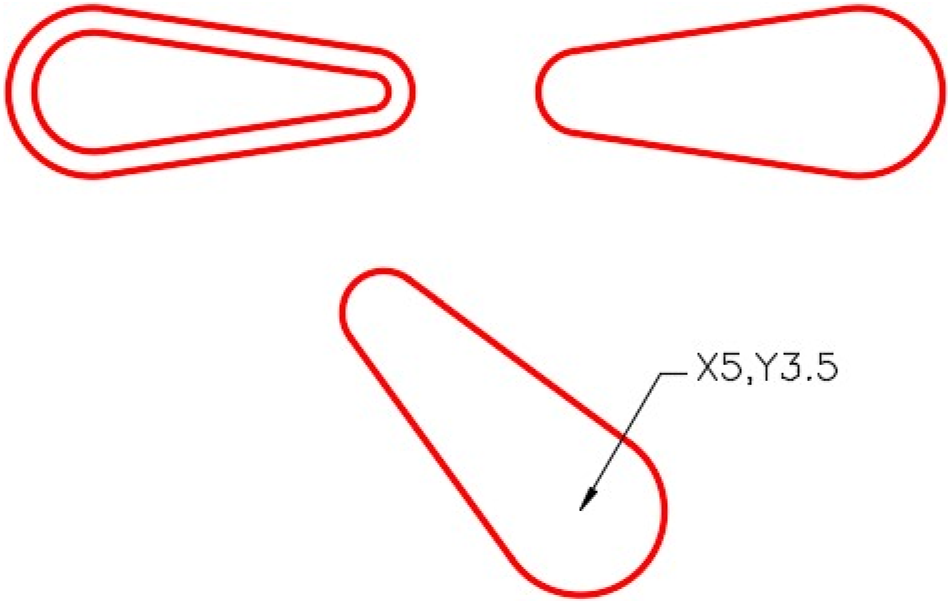


Figure Step 8

Step 9

Mirror and copy the object to match the dimensions as shown in the figure. (Figure Step 9A and 9B)

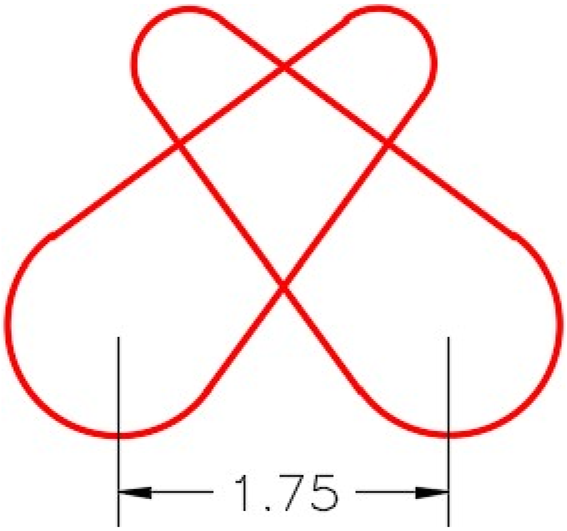


Figure Step 9A

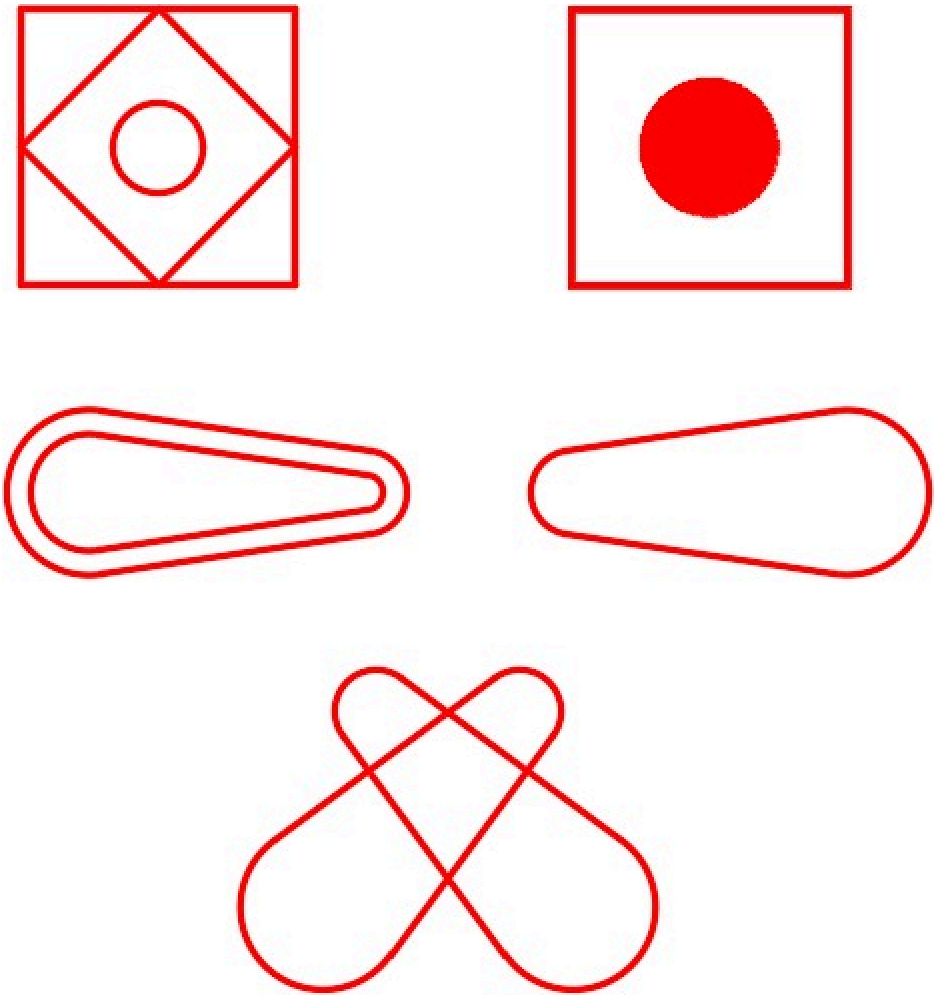
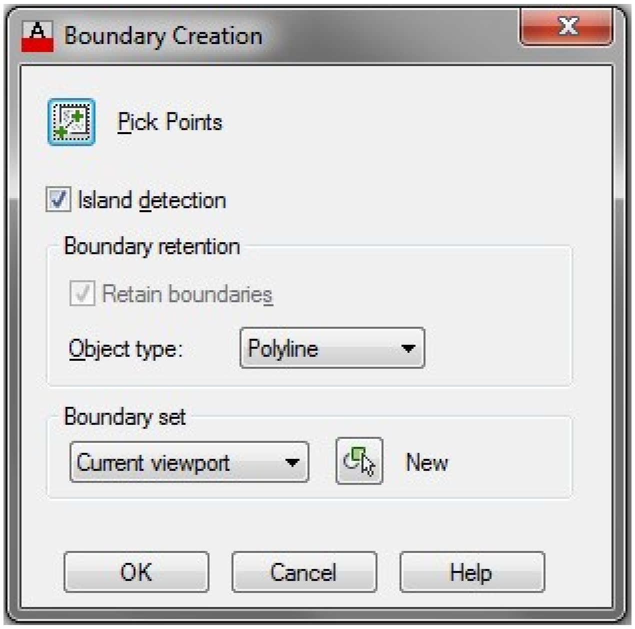


Figure Step 9B

Step 10

Set layer: Boundary as the current layer. Enter the BOUNDARY command. It will open the Boundary Creation dialogue box. Ensure that the settings in your dialogue box matches the figure. (Figure Step 10)


Boundary Creation
Pick Points
Boundary retention
Boundary set
New
OK
Cancel
Help

☑ Island detection

☑ Retain boundaries

Object type: Polyline

Current viewport

Figure Step 10

Step 11

Click the Pick Points icon and select the locations P7, P8 and P9 as shown in the figure. (Figure Step 11)

```
Command: BOUNDARY

Pick internal point: Selecting everything… Selecting everything visible…

Analyzing the selected data… Analyzing internal islands…

Pick internal point: P7

Analyzing internal islands… Pick internal point: P8

Analyzing internal islands… Pick internal point: P9

BOUNDARY created 3 polylines

Command:
```

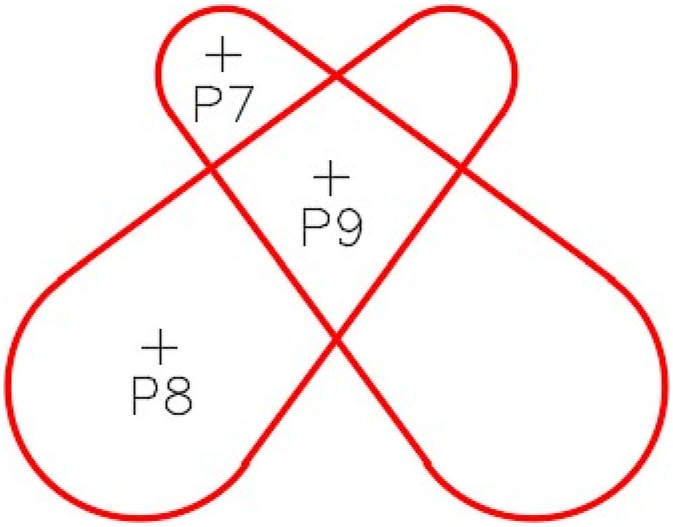


Figure Step 11

Step 12

Note how the BOUNDARY command creates closed polylines using the boundary of the selected areas. The plines are located on layer: Boundary, colored blue. (Figure Step 12)

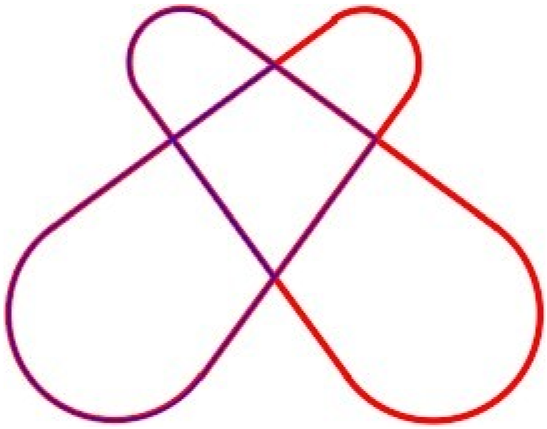


Figure Step 12

Step 13

To prove that you have created polylines, move them away from the object using the MOVE command as shown in the figure. The exact location where you move them is not important. Using the Properties window, confirm that they are closed polylines. (Figure Step 13)

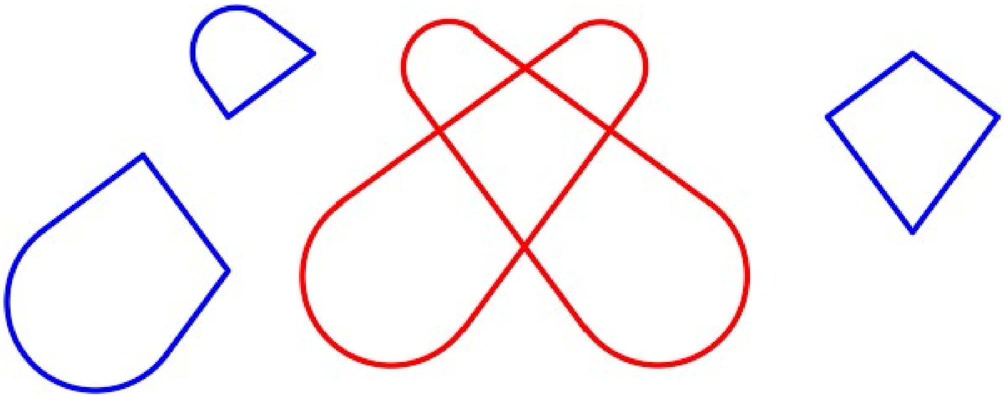


Figure Step 13

Step 14

Your drawing should appear similar to the figure. (Figure Step 14)

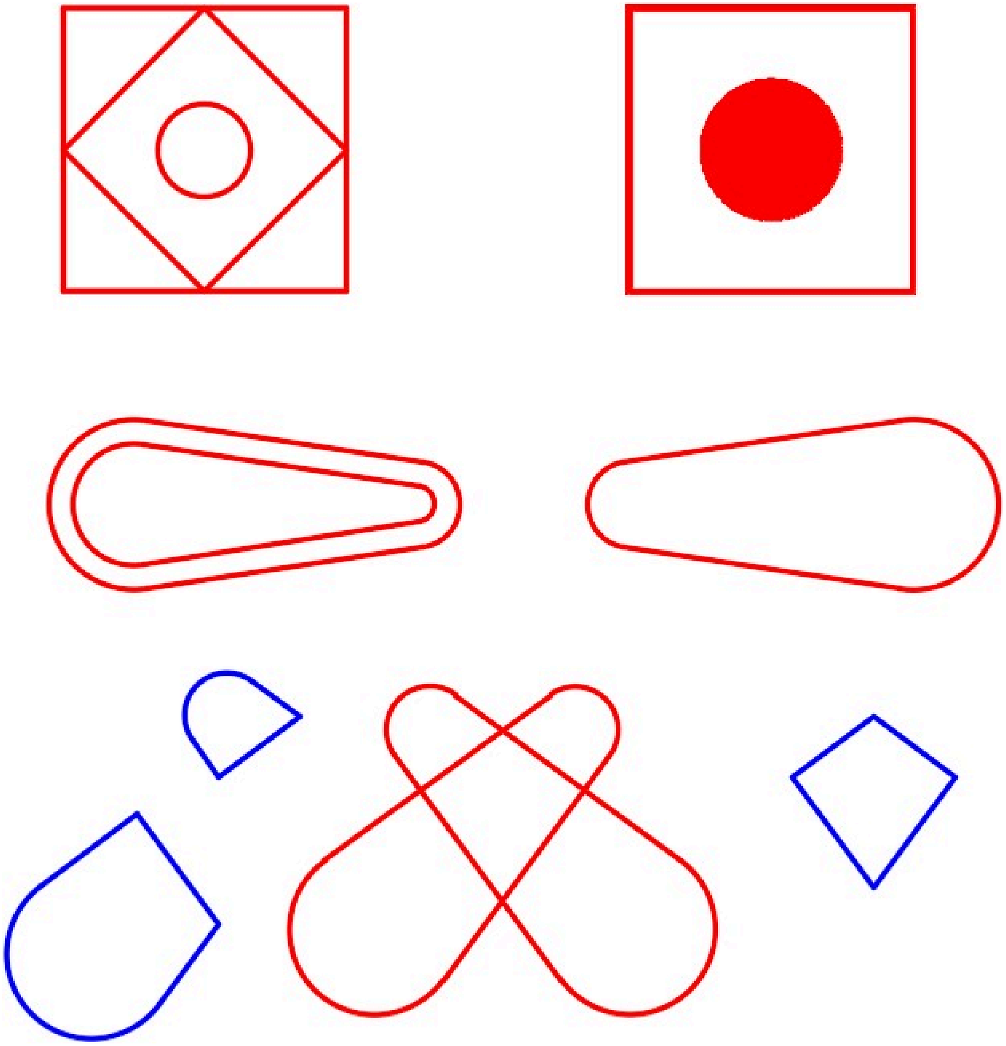


Figure Step 14

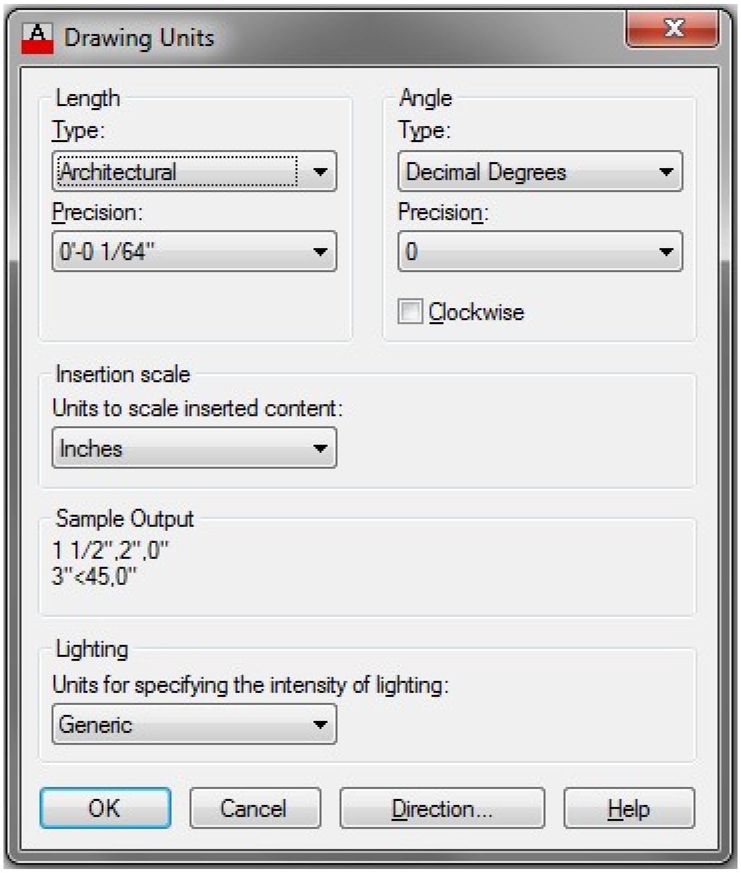
Step 15

Save and close the drawing.

Working in Feet and Inches

Up to this point in the book, all working units have been inches or millimeters. They both use the Length Type of Decimals. To work in feet and inches, the first thing you have to do is change the Length Type to Architectural. This is done using the UNITS command. In the Drawing Units dialogue box, pull-down the Length Type box and select Architectural as shown in Figure 29-8.

Pull-down the Length Precision box and select 0′-0 1/16″ as shown in Figure Step 29-9. The precision can be changed as required. Even though the length units have been set to Architectural, AutoCAD continues to work in inches. Therefore, when a number is entered and nothing else, AutoCAD will assume the number is inches. i.e.



Drawing Units

Length

Type: Architectural

Precision: 0'-0 1/64"

Angle

Type: Decimal Degrees

Precision: 0

☐ Clockwise

Insertion scale

Units to scale inserted content: Inches

Sample Output

1 1/2",2",0"

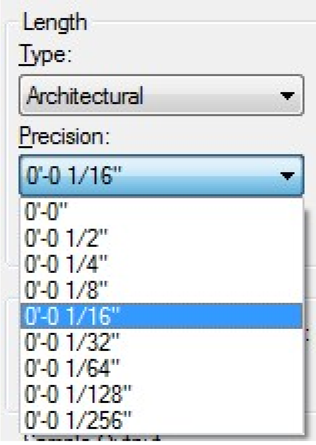
3"<45,0"

Lighting

Units for specifying the intensity of lighting: Generic

OK
Cancel
Direction...
Help

Figure 29-8
Drawing Units Dialogue Box



Length

Type: Architectural

Precision: 0'-0 1/16"

0'-0"
0'-0 1/2"
0'-0 1/4"
0'-0 1/8"
0'-0 1/16"
0'-0 1/32"
0'-0 1/64"
0'-0 1/128"
0'-0 1/256"

Figure 29-9
Setting Precision

Command: **LINE**

Specify first point: **2,2**

(AutoCAD interprets this number as 2",2")

Specify next point or [Undo]: **@4,0**

(AutoCAD interprets this as go 4″ in the X direction.)

Specify next point or [Undo]:

Command:

To enter feet and inches, enter a ‘ (single quote) after the number of feet and a ” (double quote) after the number of inches. For example:

Command: **LINE**

Specify first point: **2’3″,4’6″**

(AutoCAD interprets this number as 2 feet 3 inches.,4 feet 6 inches)

Specify next point or [Undo]: **@5’2″,0**

(AutoCAD interprets this as go 5 feet 2 inches in the X direction.)

Specify next point or [Undo]:

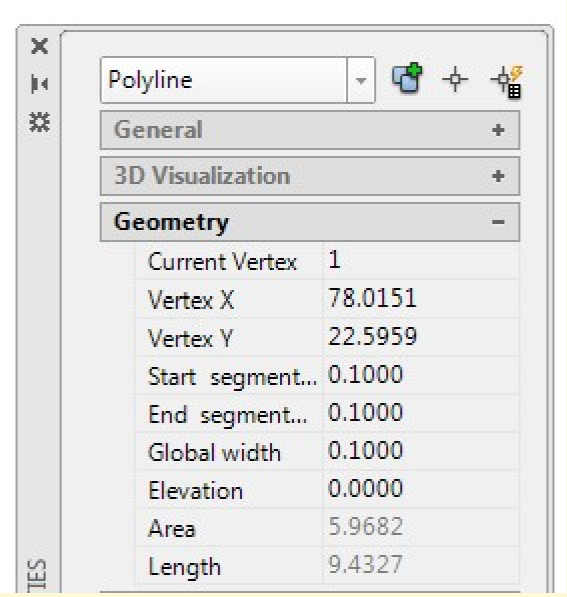
Command:

To work in fractions of an inch, enter the number as follows: **3’2-1/2″** or **3’2.5″**

MUST KNOW: When using the PEDIT command to convert existing objects into a polyline, the objects must be drawn perfectly with their endpoints meeting exactly. If they overlap, have a gap between them, or are stacked on top of a another, the PEDIT command will not create the polyline.

USER TIP: When editing polylines, it is sometimes easier to use the Properties window rather than the PEDIT command. The Properties window cannot do everything the PEDIT command can do. While a polyline cannot be created using the Properties window, it can do the following:

1. Change its width.
2. Change the vertex locations.
3. Open or close the pline.



Current Vertex 1
Vertex X 78.0151
Vertex Y 22.5959
Start segment... 0.1000
End segment... 0.1000
Global width 0.1000
Elevation 0.0000
Area 5.9682
Length 9.4327

USER TIP: After the text is entered to complete one titleblock, the text can easily be copied to the next drawing. It is much easier to copy the titleblock text and then edit it to describing the drawing then entering new text for each titleblock. Ensure that titleblocks are copied from English units to English units and metric units to metric units.

Key Principles

Key Principles in Module 29

1. A polyline is a series of line and/or arc segments connected to create a single object. Polylines can be created using the PLINE command drawing straight line segments, arc segments or a combination of the two. A polyline can be exploded back to its individual objects with the EXPLODE command.
2. A closed polyline creates a polygon. A polygon has many different uses in an AutoCAD drawing but will appear exactly the same as an open polyline. Therefore, it is important to be able to verify if a polyline is opened or closed. This is done using the Properties window.
3. Polylines can be drawn from scratch, using the PLINE command, but it is easier to create them using the PEDIT or BOUNDARY commands from existing objects.
4. AutoCAD always works in inches. If you want to enter numbers in feet and inches, you must change the Length Type settings to Architectural in the Drawing Units dialogue box.

Lab Exercise 29-1

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 29-1	2D Layout Metric	Millimeters

Step 1

On layer: Object, draw the object shown in the figures. (Figure Step 1A and 1B)

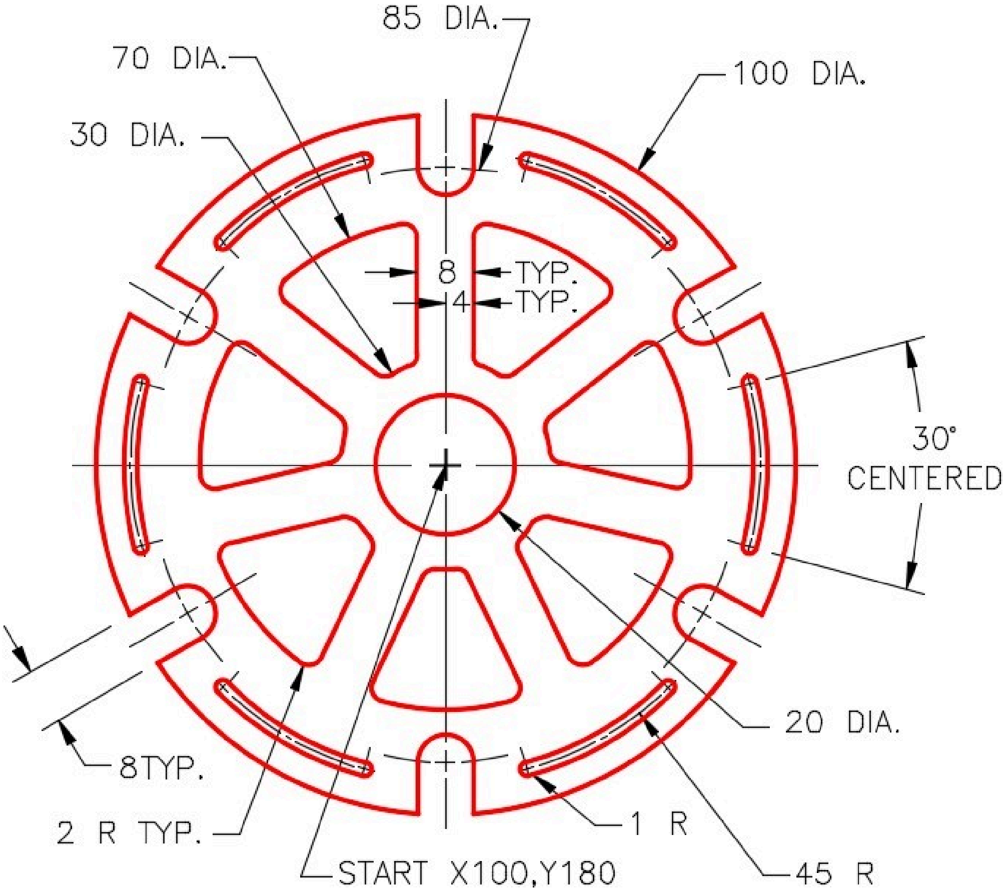

85 DIA.
70 DIA.
100 DIA.
30 DIA.
8 TYP.
4 TYP.
30°
CENTERED
20 DIA.
8TYP.
2 R TYP.
1 R
START X100,Y180
45 R

Figure Step 1A

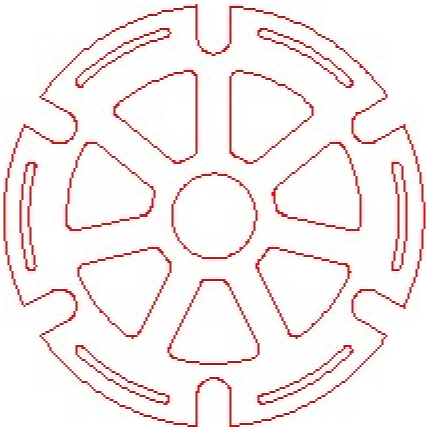


Figure Step 1B
Completed Drawing

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hint to help you.

Hint 1

Create the objects to be arrayed in 3 steps as shown in the following figures. Once they have all be created, you will only have to use the ARRAY command twice to complete the drawing. (Figure Hint 1A, 1B, and 1C)

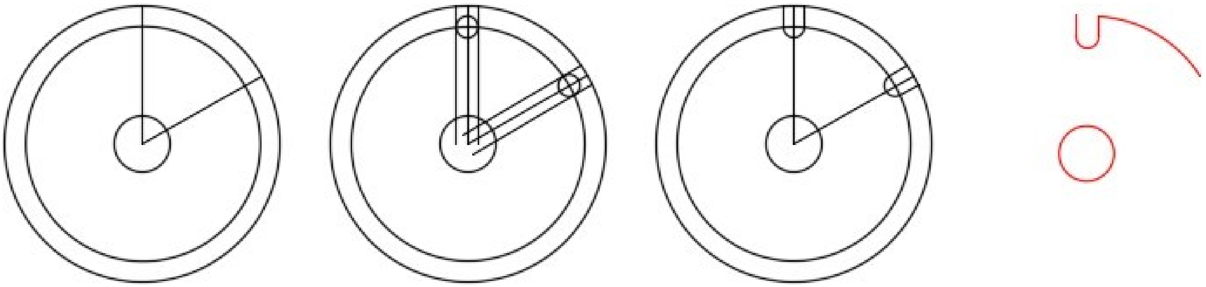


Figure Hint 1A

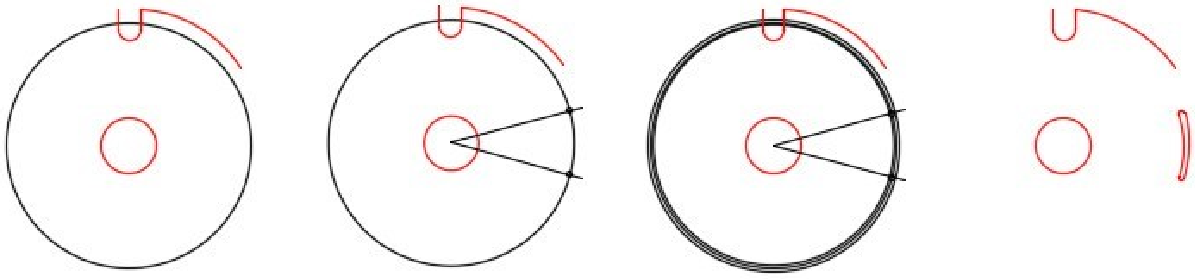


Figure Hint 1B

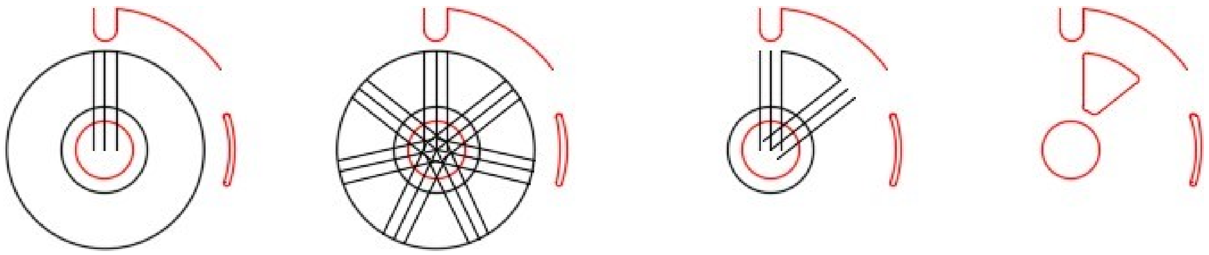


Figure Hint 1C

Lab Exercise 29-2

Time allowed: 40 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 29-2	N/A	Millimeters

Step 1

Open drawing: AutoCAD 2D Lab 29-1. Using the SAVEAS command, save and name the drawing: AutoCAD 2D Lab 29-2. (Figure Step 1)

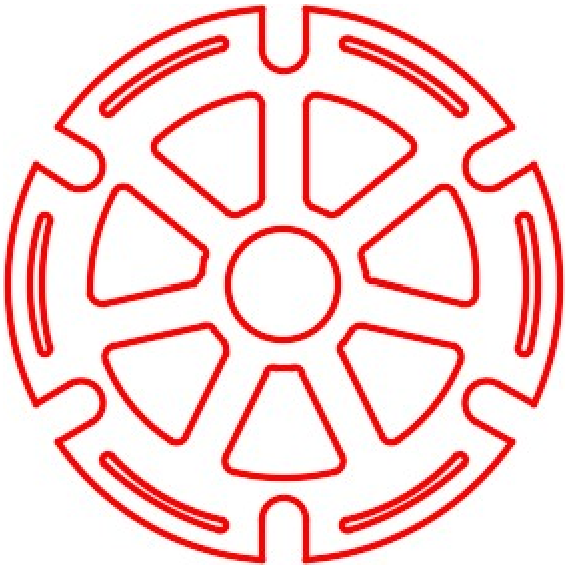


Figure Step 1

Step 2

Using the PEDIT command and the All method of selecting, create a closed pline from the objects that form the perimeter of the object. Change the pline to layer: Pline. Using the MOVE command, move the pline 125 millimeters in the positive X direction. (Figure Step 2)

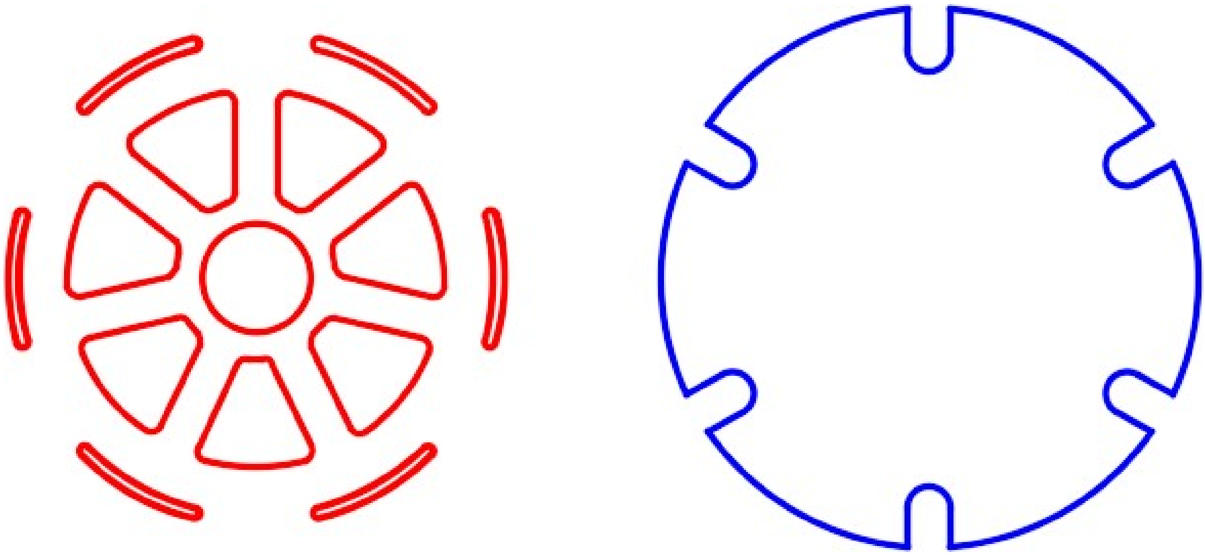


Figure Step 2

Step 3

Using the PEDIT command, 6 times, and the pick method of selecting the objects, create 6 closed plines of the slots. Copy the 6 plines 125 millimeters in the positive X direction. Change the layer of the copied plines to layer Pline. (Figure Step 3)

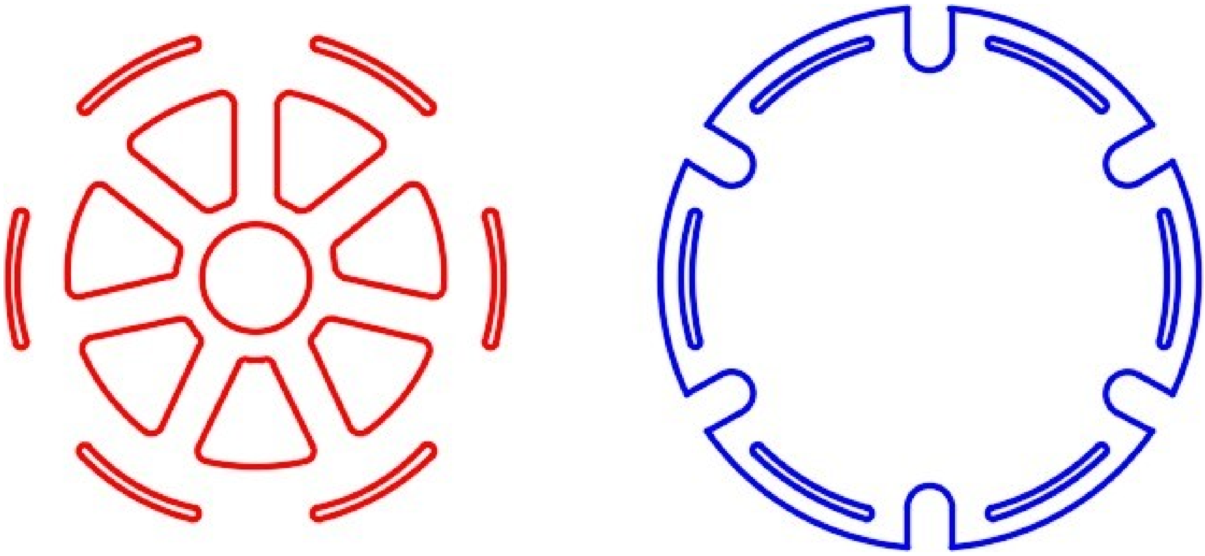


Figure Step 3

Step 4

Set layer: Pline as the current layer. Using the BOUNDARY command, 7 times, create a pline around and inner shapes to create 7 closed plines. Move the 7 plines 125 millimeters in the positive X direction. (Figure Step 4)

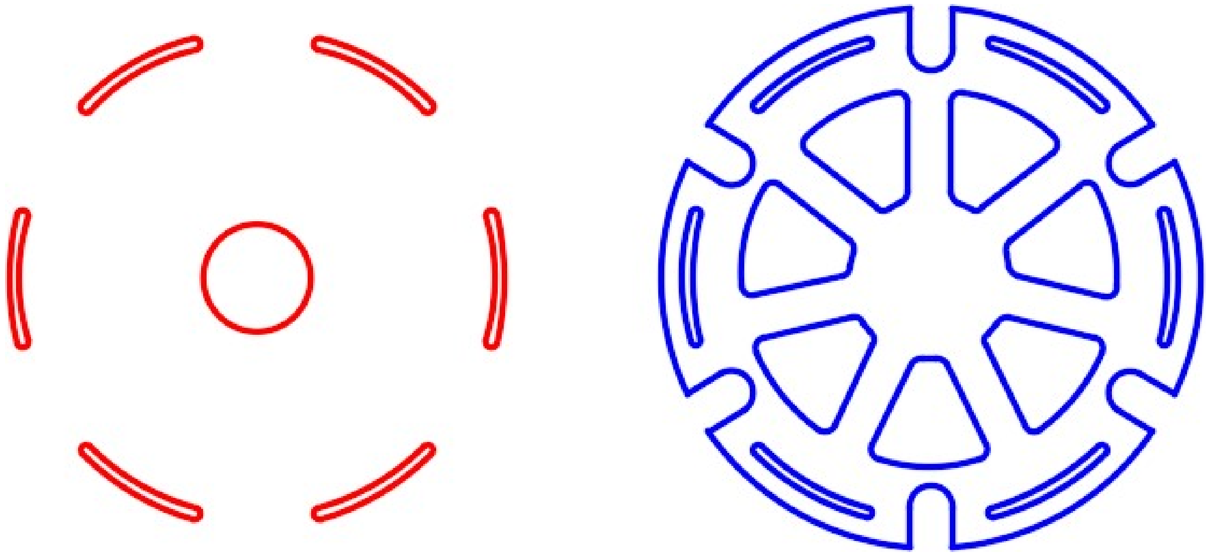


Figure Step 4

Step 5

Save and close the drawing.

Lab Exercise 29-3

Time allowed: 90 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 29-3	2D Layout English	Feet/Inches

Step 1

Start a new drawing using the template shown above.

Step 2

Using the UNITS command, change the linear units to Architectural with a precision of 1/16″.

Step 3

Using Steps 4 to 9, draw the floor plan of the house shown the figure. Use the LINE command to draw it. Use feet and inches as input units. (Figure Step 3)

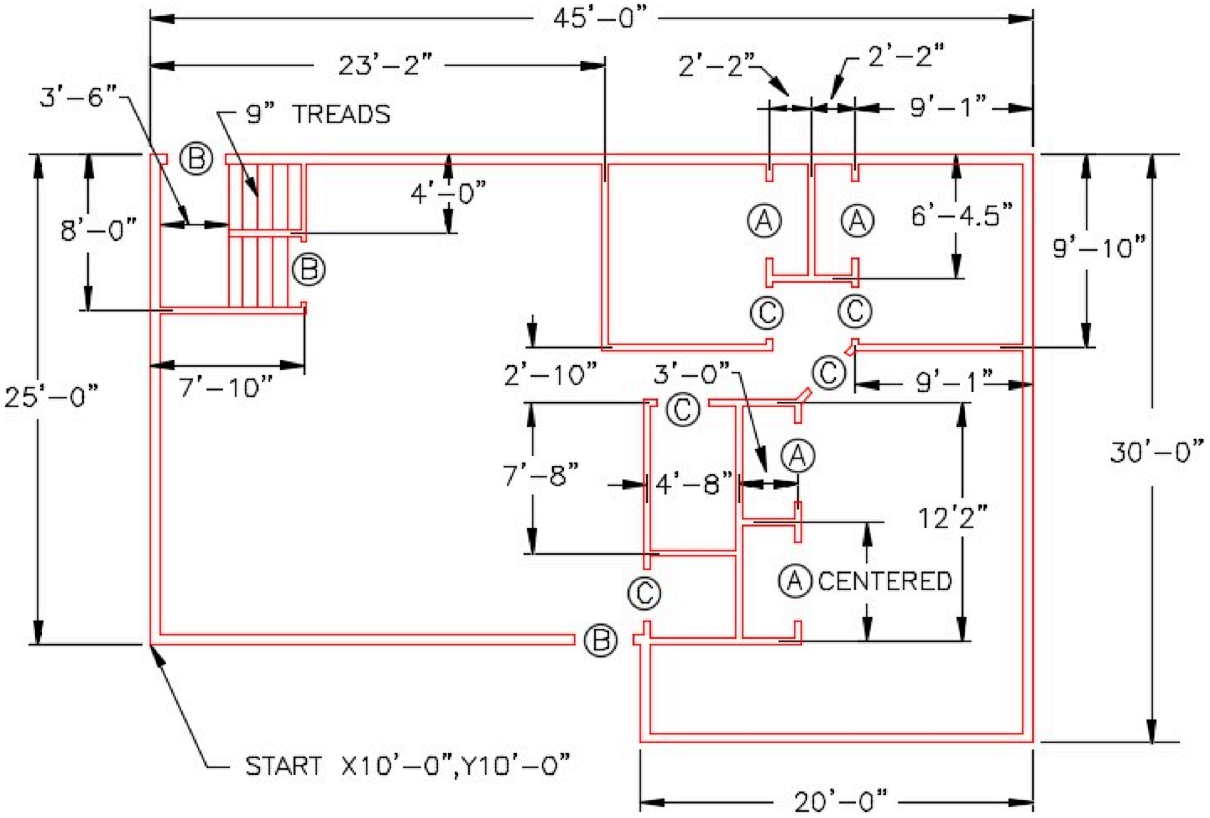

45'–0"
23'–2"
2'–2"
2'–2"
9'–1"
3'–6"
9" TREADS
B
4'–0"
8'–0"
A
A
6'–4.5"
9'–10"
B
C
C
25'–0"
7'–10"
2'–10"
3'–0"
C
9'–1"
C
7'–8"
4'–8"
A
12'2"
30'–0"
A CENTERED
C
B
START X10'–0",Y10'–0"
20'–0"

Figure Step 3

Step 4

On layer: Floor Plan, draw lines around the outside boundary of the house. (Figure Step 4)

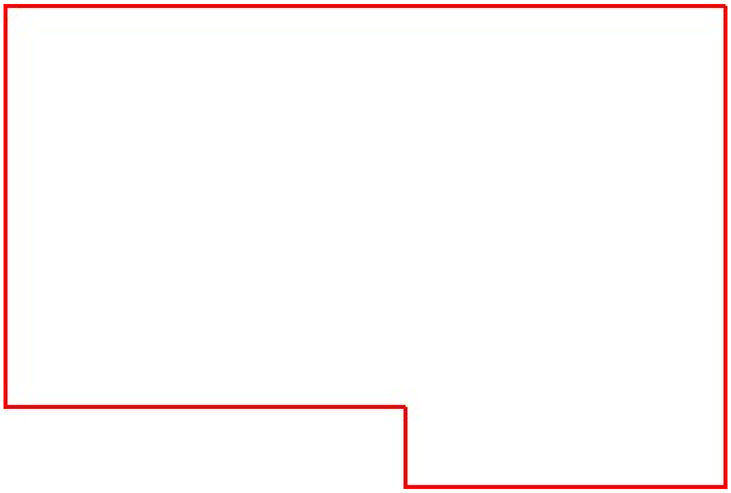


Figure Step 4

Step 5

Offset the outside boundary 6 inches toward the inside of the house. Trim or extend the lines to clean up the corners. (Figure Step 5)

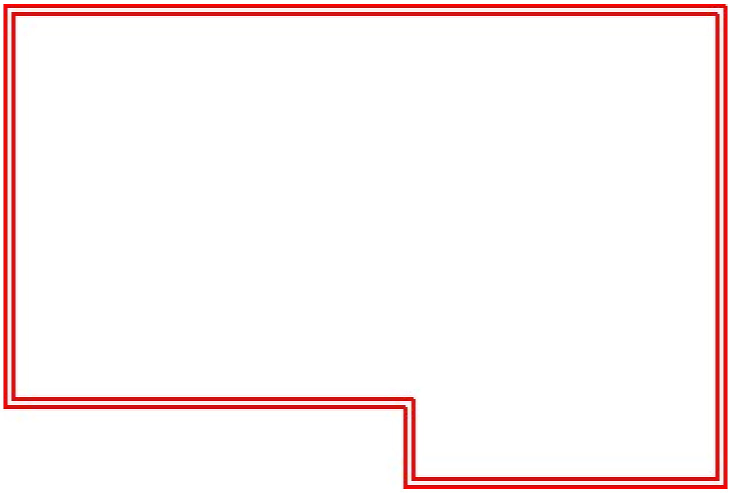


Figure Step 5

AUTHOR'S COMMENTS: The outside walls are 6″ wide.

Step 6

On layer: Construction, draw lines in the center location of the inside walls. Use Figure Step 6B to help you. (Figure Step 6A and 6B)

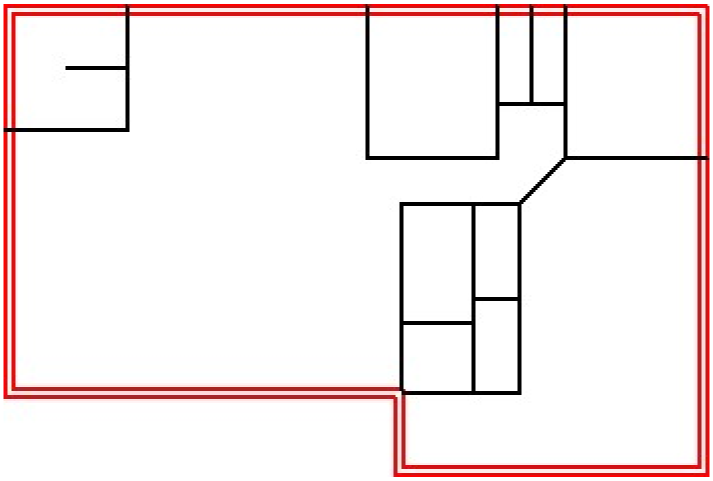


Figure Step 6A

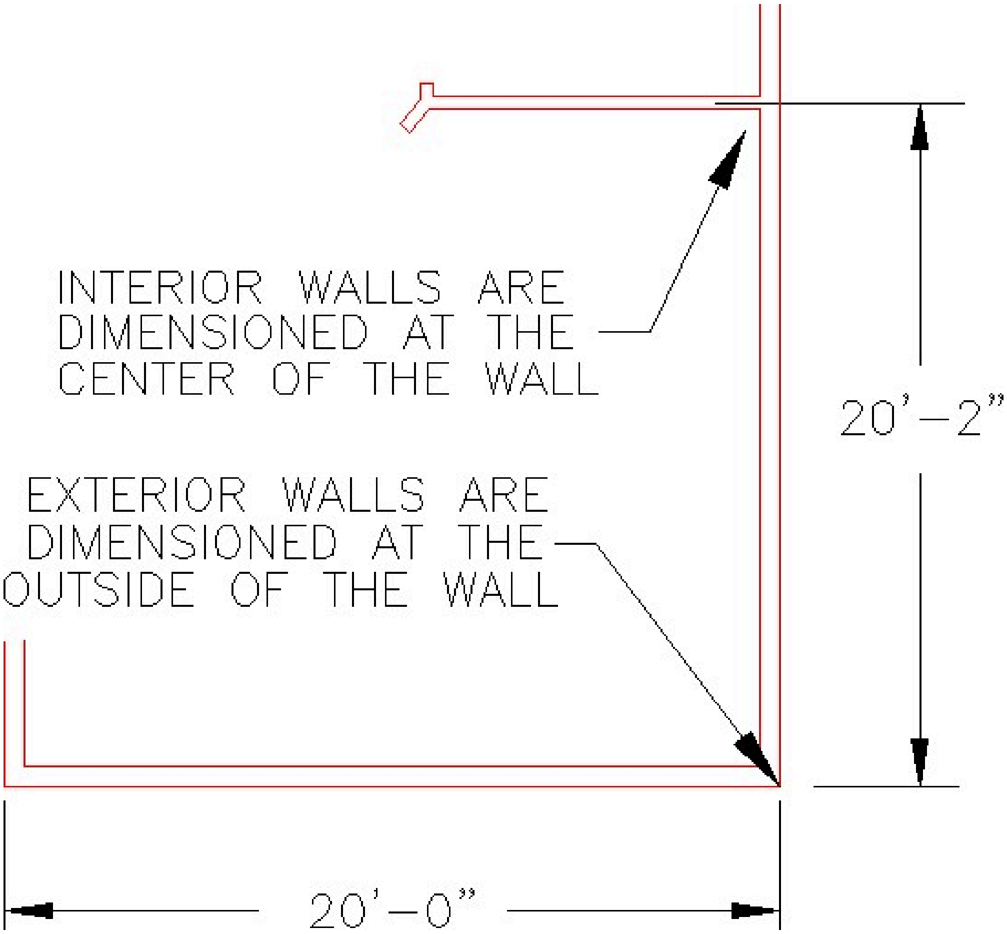

INTERIOR WALLS ARE DIMENSIONED AT THE CENTER OF THE WALL
EXTERIOR WALLS ARE DIMENSIONED AT THE OUTSIDE OF THE WALL
20'–2"
20'–0"

Figure Step 6B

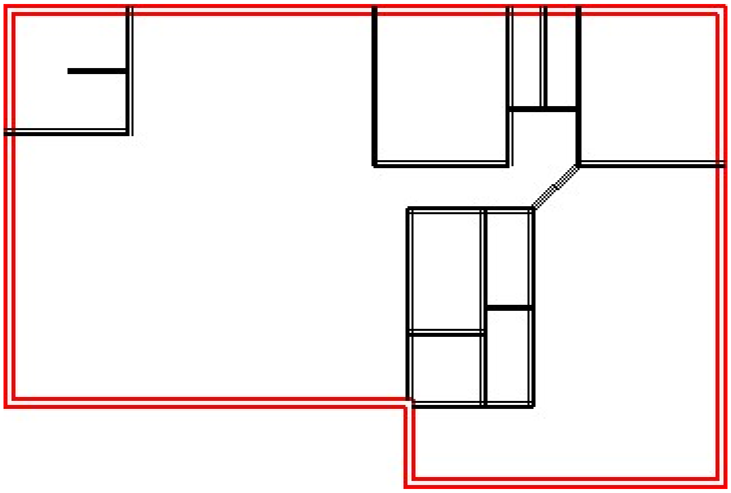


Figure Step 6C

Step 7

Offset the center lines 2 inches on each side the centerline to create 4 inch walls. (Figure step 7A and 7B)

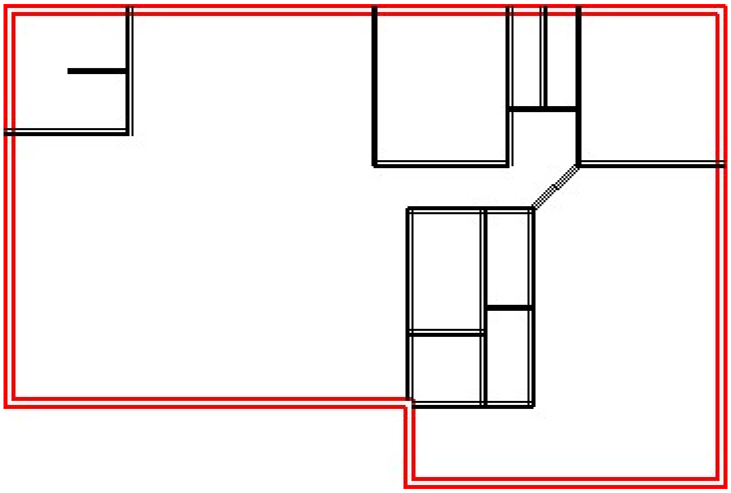


Figure Step 7A

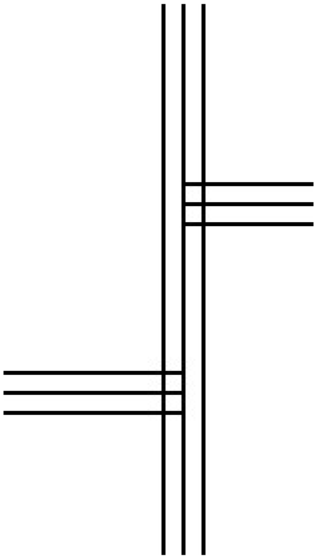


Figure Step 7B

AUTHOR’S COMMENTS: The inside walls are 4″ wide.

Step 8

Turn layer: Construction off. Using the TRIM and EXTEND commands, clean up the corners of all walls and change layer of all wall lines to layer: Floor Plan. Figure step 8A and 8B)

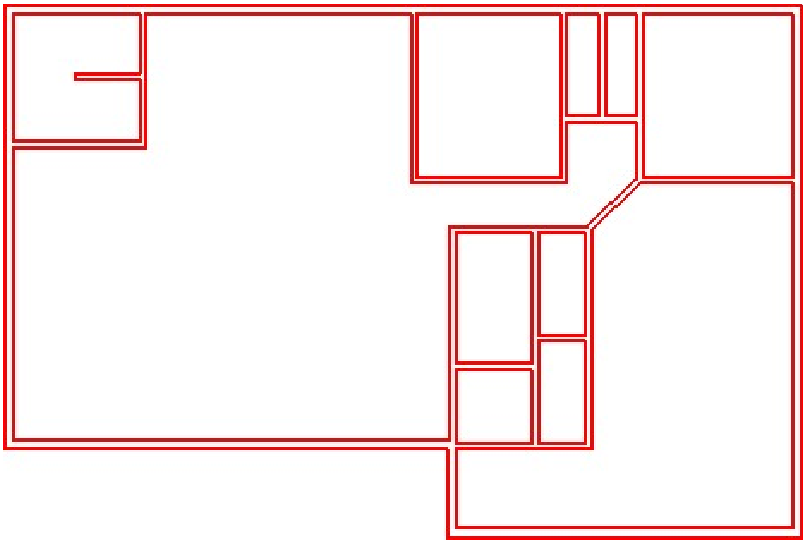


Figure Step 8A

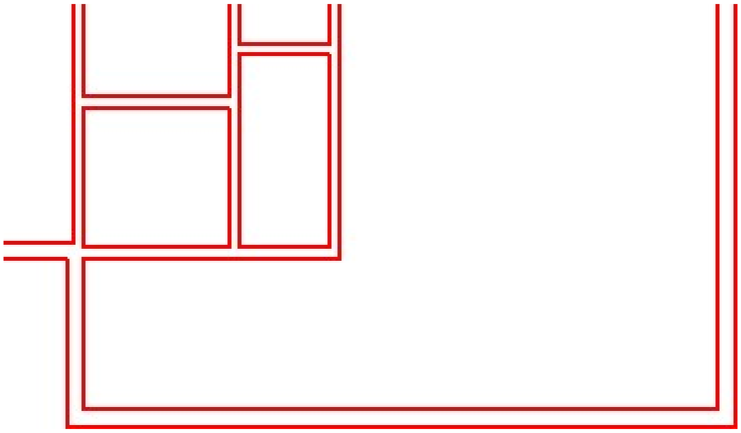


Figure Step 8B – Detail

Step 9

Using Figure 9A, 9B and 9C, cut the door openings in the walls. (Figure step 9A, 9B, 9C, and 9D)

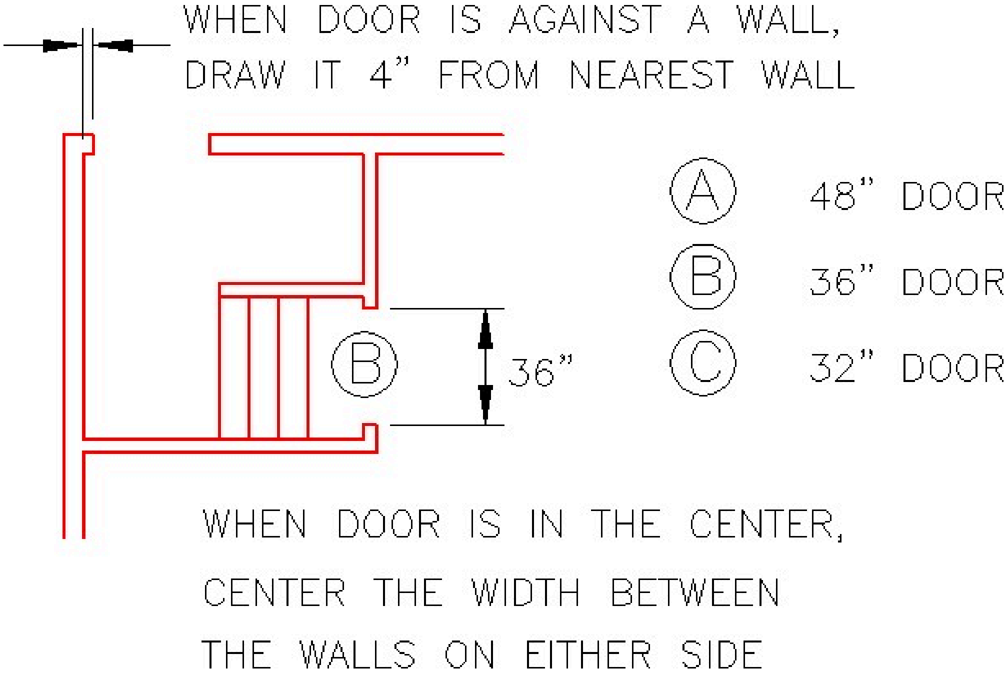

WHEN DOOR IS AGAINST A WALL,
DRAW IT 4" FROM NEAREST WALL
A
48" DOOR
B
36" DOOR
C
32" DOOR
B
36"
WHEN DOOR IS IN THE CENTER,
CENTER THE WIDTH BETWEEN
THE WALLS ON EITHER SIDE

Figure Step 9A

To draw a door, where the walls are horizontal or vertical , draw a line from the midpoint of the door opening with ortho mode enabled. Offset the line one-half the door width on each side and trim. (Figure Step 9B)

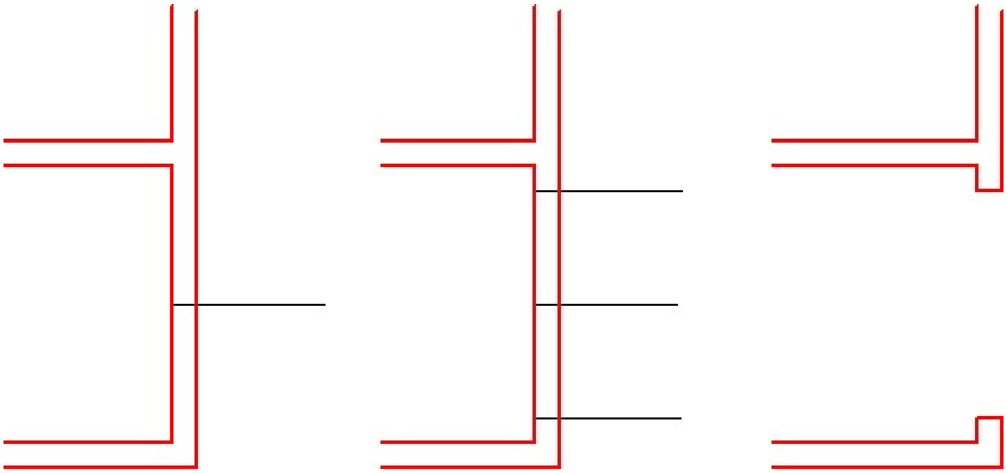


Figure Step 9B

When the walls are inclined, draw a line from a point in space, perpendicular to the line. Move the line by snapping its endpoint to the midpoint of the line. Offset the line by one-half of the door width on each side and trim. (Figure Step 9C and 9D)

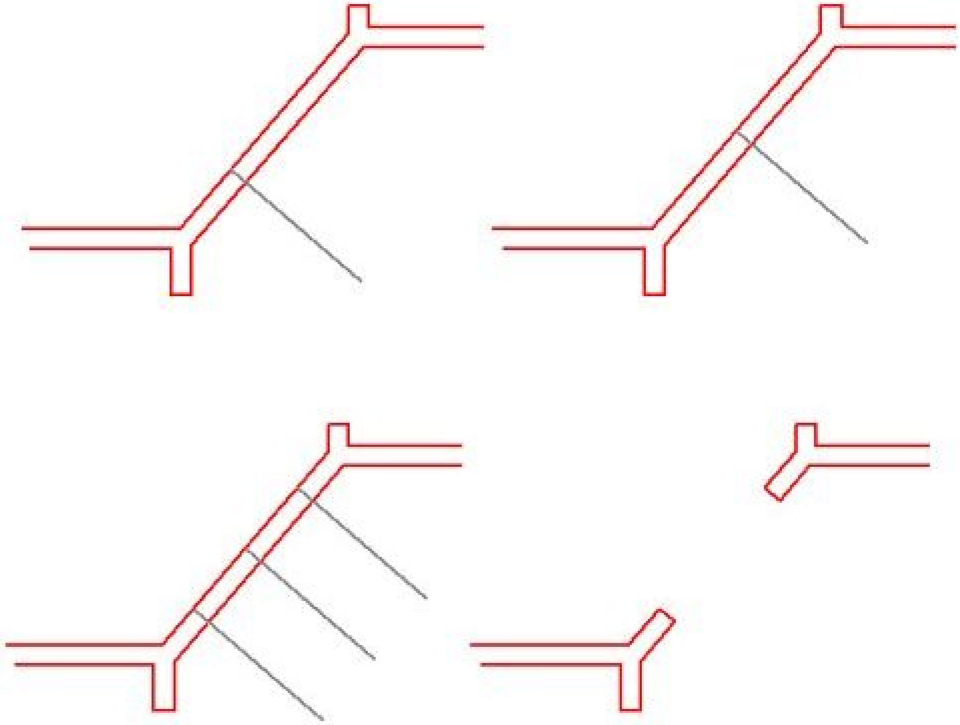


Figure Step 9C

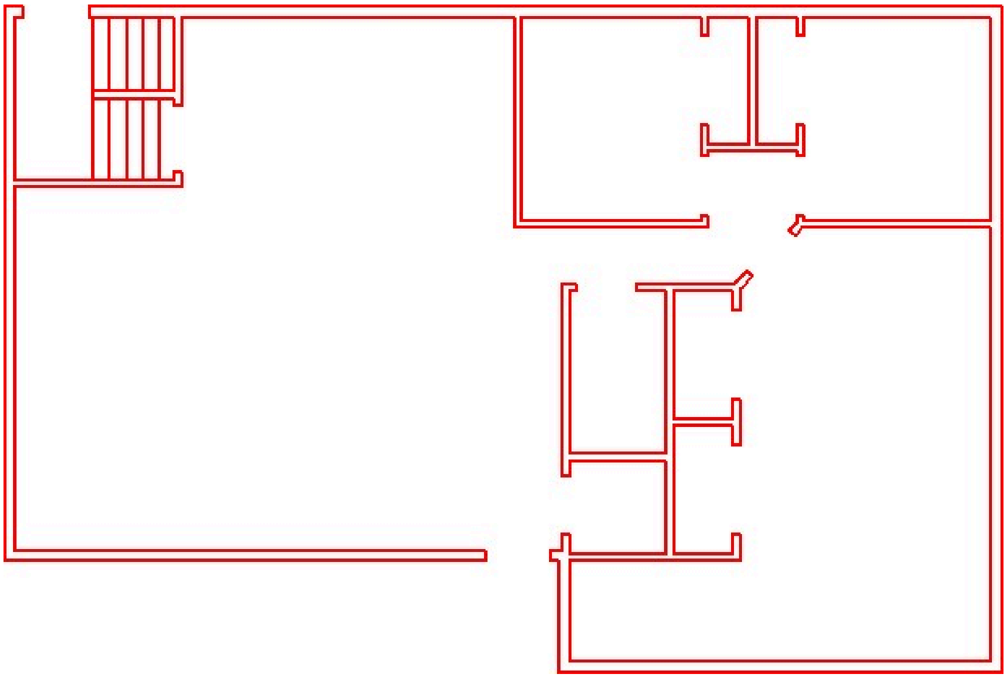


Figure Step 9D

Step 10

Enable layout: Module Layout C. On layer: Viewport, create a viewport and set the scale to 1/4″=1′-0″. Lock the display. (Figure Step 10)

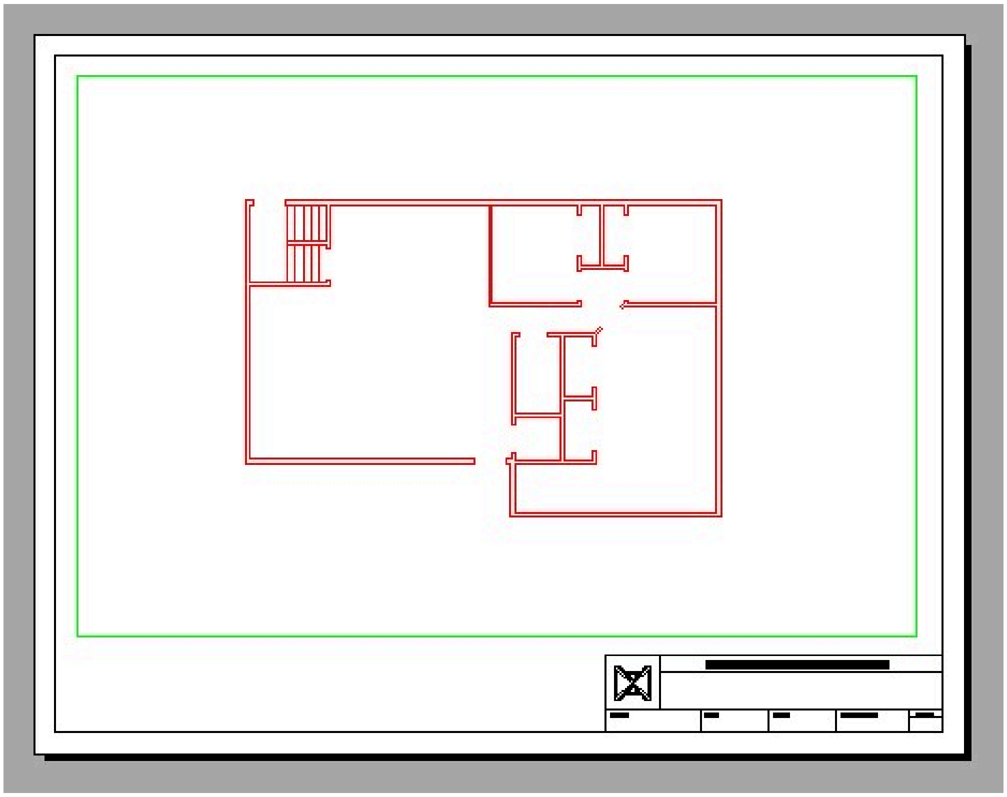


Figure Step 10

Step 11

Save and close the drawing.

Module 31: Competency Test 6 Open Book

Learning Outcomes

When you have completed this module, you will be able to:

1. Within a two hour time limit, complete a written exam and a lab exercise without the aid of a key.

The AutoCAD 2D book was written with competency based modules. What that means is that you have not completed each module until you have mastered it. The Competency Test module contains multiple choice questions and a comprehensive lab exercise to test your mastery of the set of modules that you completed. There are no answers or keys supplied in a Competency Test module since it is meant to be checked by your instructor. If there are any parts of this module that you have trouble completing, you should go back and reread the module or modules containing the information that you are having trouble with. If necessary, redo as many lab exercises required until you fully understand the material.

If you are Completing this book:

- Without the Aid of an Instructor, complete the written test and the lab exercise.
- In a Classroom with an Instructor, the instructor will give instructions on what to do after this module has been completed.

Multiple Choice Questions

Select the BEST answer.

1. Which one of the following commands can be used to find the perimeter of a closed polyline?
    A. PERIMETER
    B. AREA
    C. DIST
    D. INQUIRY
    E. LENGTH
2. Which one of the following commands can be used to control the size the grid displays on a drawing? Select the best answer.
    A. POINT

   - B. SNAP
   - C. UNITS
   - D. LIMITS
   - E. PEDIT

3. Which one of the following commands can be used to convert a polyline back to its original objects?
   - A. ERASE
   - B. EXPLODE
   - C. PEDIT
   - D. BOUNDARY
   - E. POLYGON
4. Which two of the following commands can be used to toggle a pline's width off and on?
   - A. FILL and EXPLODE
   - B. PEDIT and FILL
   - C. REGEN and PEDIT
   - D. EXPLODE and FILL
   - E. FILL and REGEN
5. What are the minimum and maximum number of sides that a polygon can be drawn using the POLYGON command?
   - A. 3 and 512
   - B. 3 and 1024
   - C. 3 and 256
   - D. 4 and 2048
   - E. 4 and 1024
6. Which one of the following function keys toggles snap mode?
   - A. F5
   - B. F6
   - C. F7
   - D. F8
   - E. F9
7. Which one of the following commands can be used to draw a square as a closed polygon?
   - A. INSERT
   - B. SQUARE
   - C. RECTANGLE

  D. BLOCK
  E. FILL
8. Which two commands can be used to toggle the display of a donut's solid fill off and on?
  A. FILL and EXPLODE
  B. PEDIT and FILL
  C. FILL and REGEN
  D. EXPLODE and FILL
  E. REGEN and PEDIT
9. Which one of the following function keys toggles the grid display?
  A. F5
  B. F6
  C. F7
  D. F8
  E. F9
10. When the PEDIT command is used to create a pline, what are the only two types of drawing objects that can be selected by the user?
  A. ellipses and arcs
  B. arcs and circles
  C. lines and arcs
  D. lines and circles
  E. lines and ellipses

Lab Exercise 31-1 Open Book

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 31-1	2D Layout Metric	Millimeters

Step 1

On layer: Object, draw the object shown in the figures. (Figure Step 1A and 1B)

Step 2

Change the width of the two regular polygons to a width of 1 and 2 as shown in Figure Step 1B)

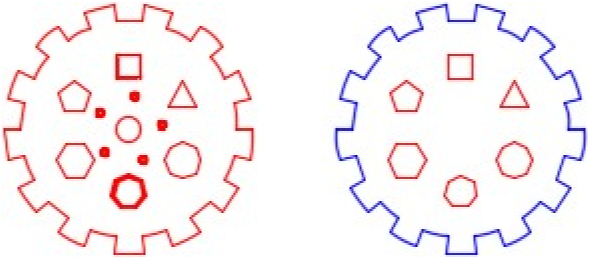


Figure Step 1A
Completed Drawing

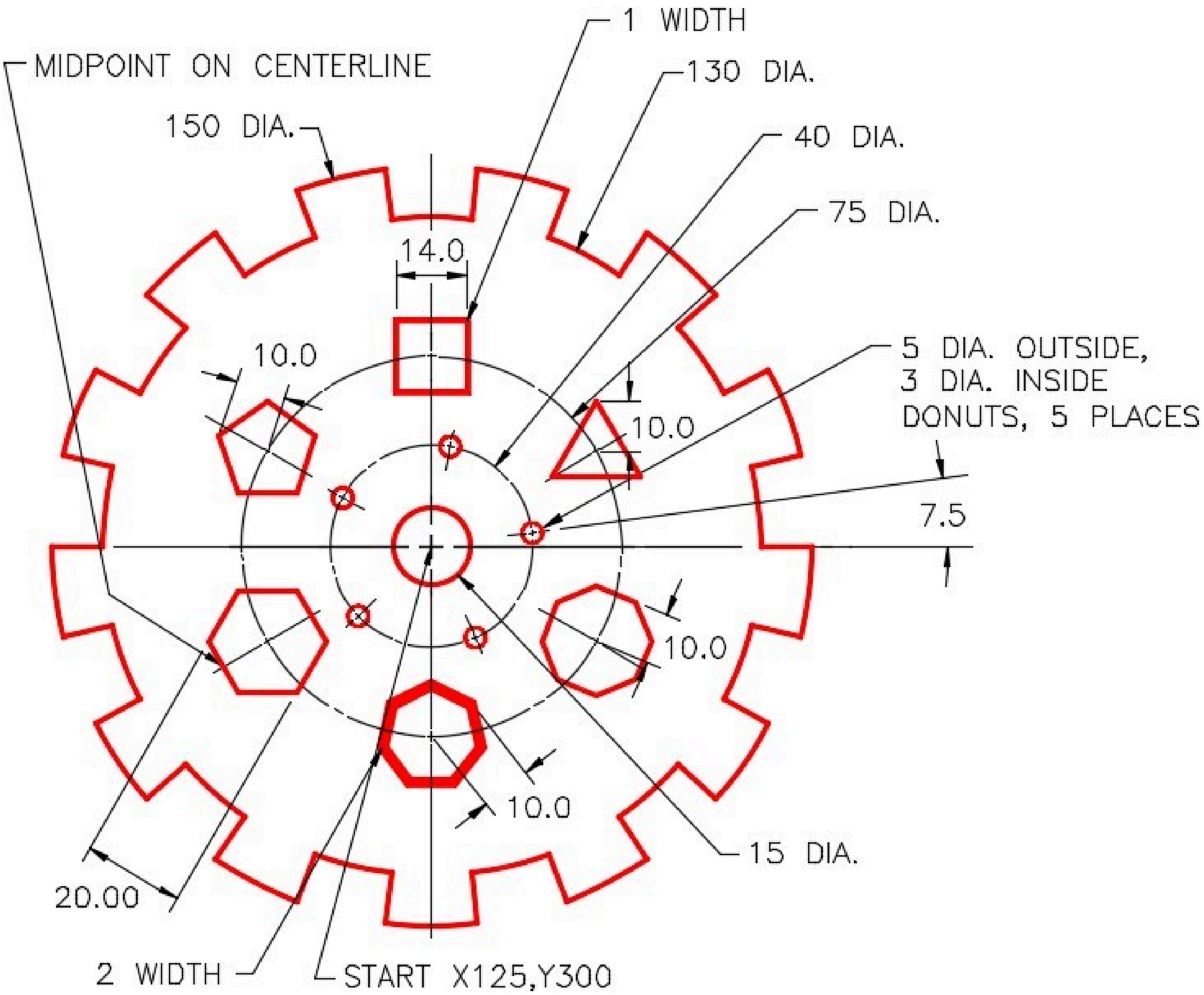

1 WIDTH
MIDPOINT ON CENTERLINE
130 DIA.
150 DIA.
40 DIA.
75 DIA.
14.0
10.0
5 DIA. OUTSIDE,
3 DIA. INSIDE
DONUTS, 5 PLACES
10.0
7.5
10.0
10.0
15 DIA.
20.00
2 WIDTH
START X125,Y300

Figure Step 1B

Step 3

Copy all objects, except the circles, 200 mm to the right. (Figure Step 3)

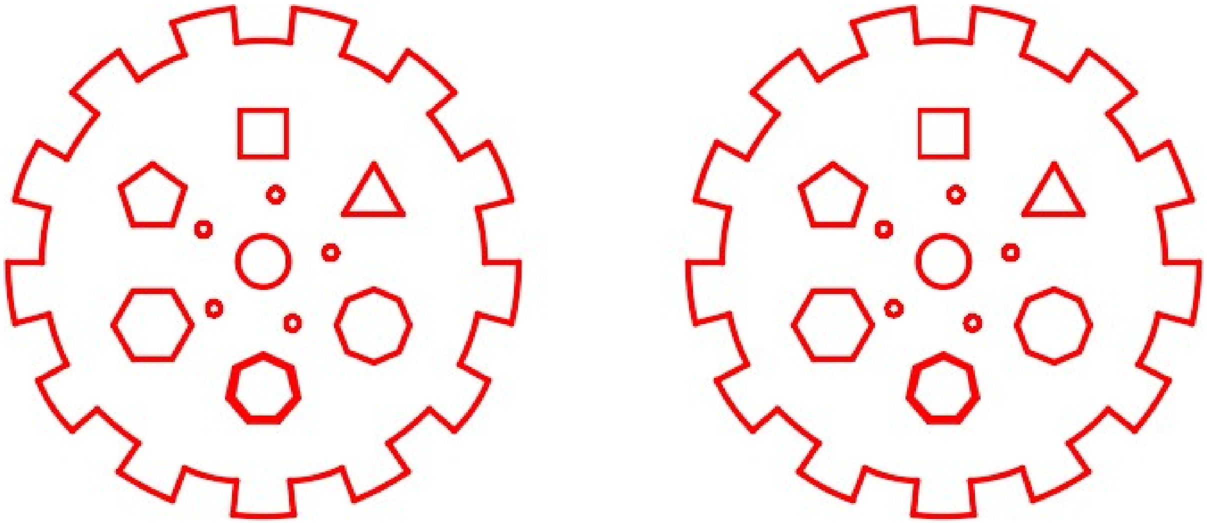


Figure Step 3

Step 4

Using the PEDIT command, change the widths of all plines, in the copied figure, to the width of 0. (Figure Step 4)

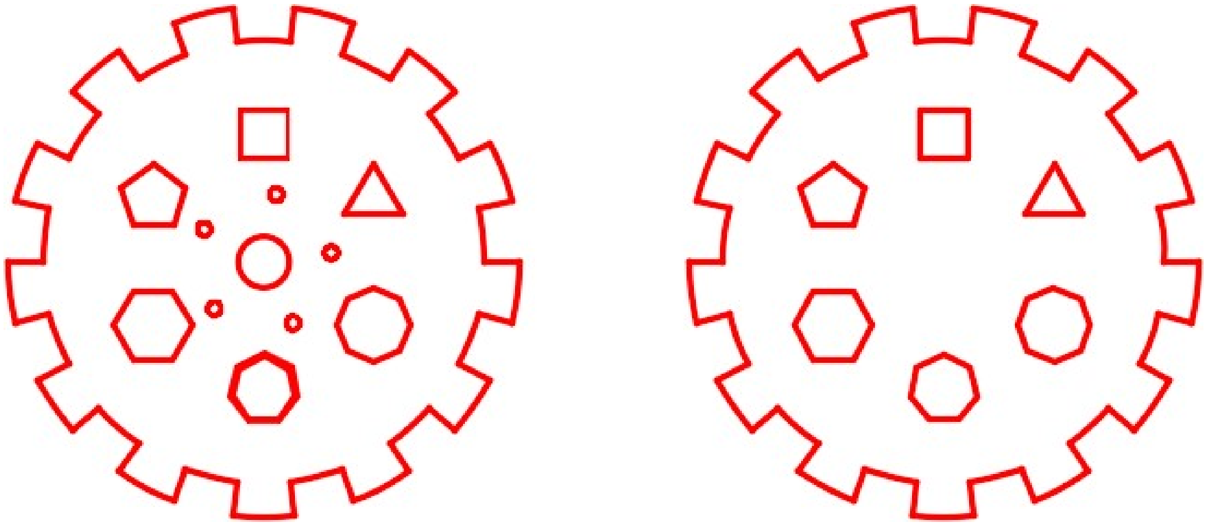


Figure Step 4

Step 5

On layer: Boundary, create a closed polyline of the outside perimeter of the copied object. (Figure Step 5)

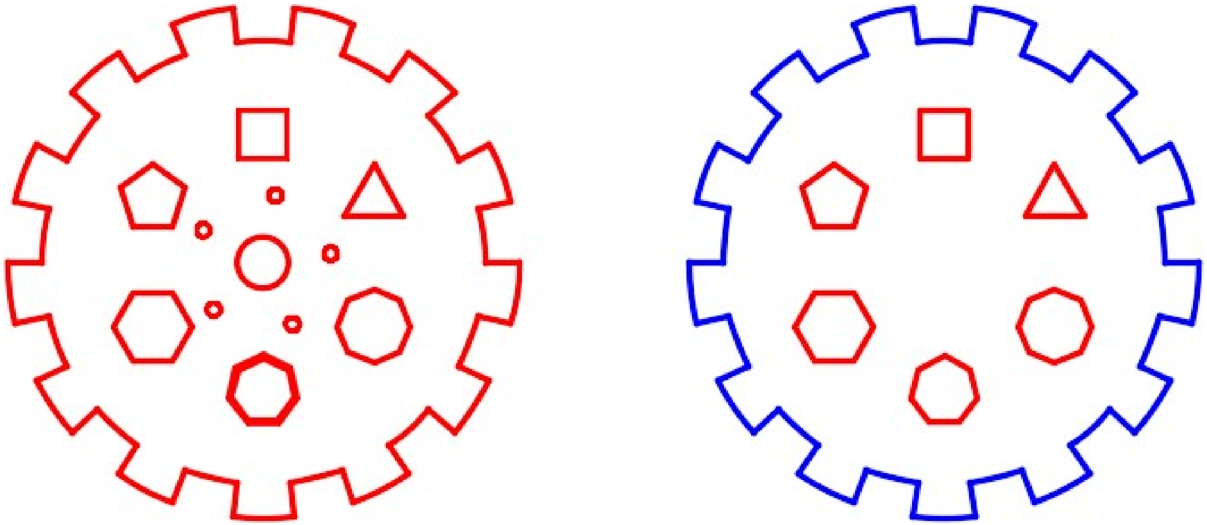


Figure Step 5

Step 6

Enable layout: Module Layout A2. On layer: Viewport, create three viewports. Set the scale of the viewports as shown in the figure and lock their display. (Figure Step 6)

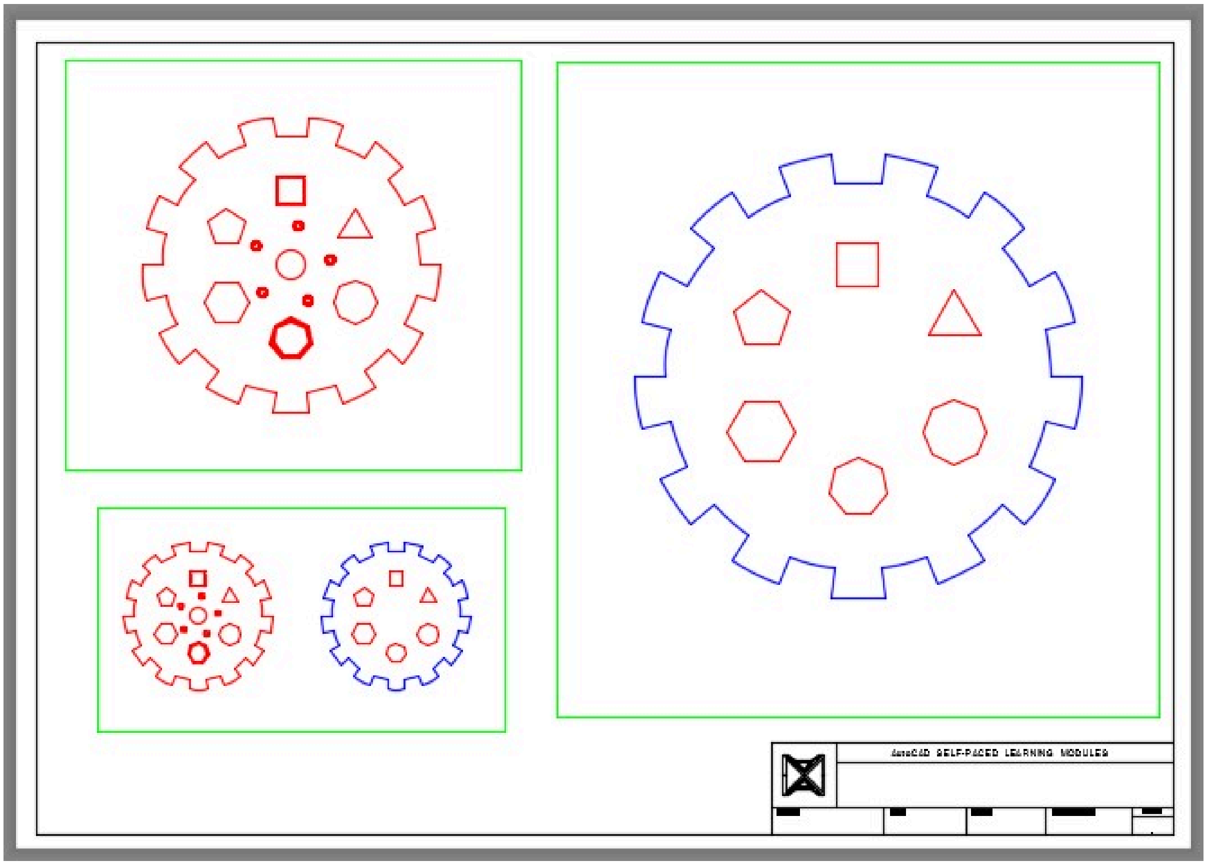


Figure Step 6

Step 7

Find the areas and perimeter shown in the figure. Set the current text style to: 2D Modules. In Paper space, on layer: Text, insert the text in the bottom left corner of the layout and include your answers. Set a text height that appears best on the drawing. (Figure Step 7)

AREA OF THE OUTSIDE OF THE OBJECT XX.XXXX SQ. MM

PERIMETER OF THE OUTSIDE OF OBJECT: XX.XXXX MM

AREA OF THE OUTSIDE OF THE OBEJCT MINUS THE SIX PLOYGONS: XX.XXXX SQ. MM

Figure Step 7

Step 8

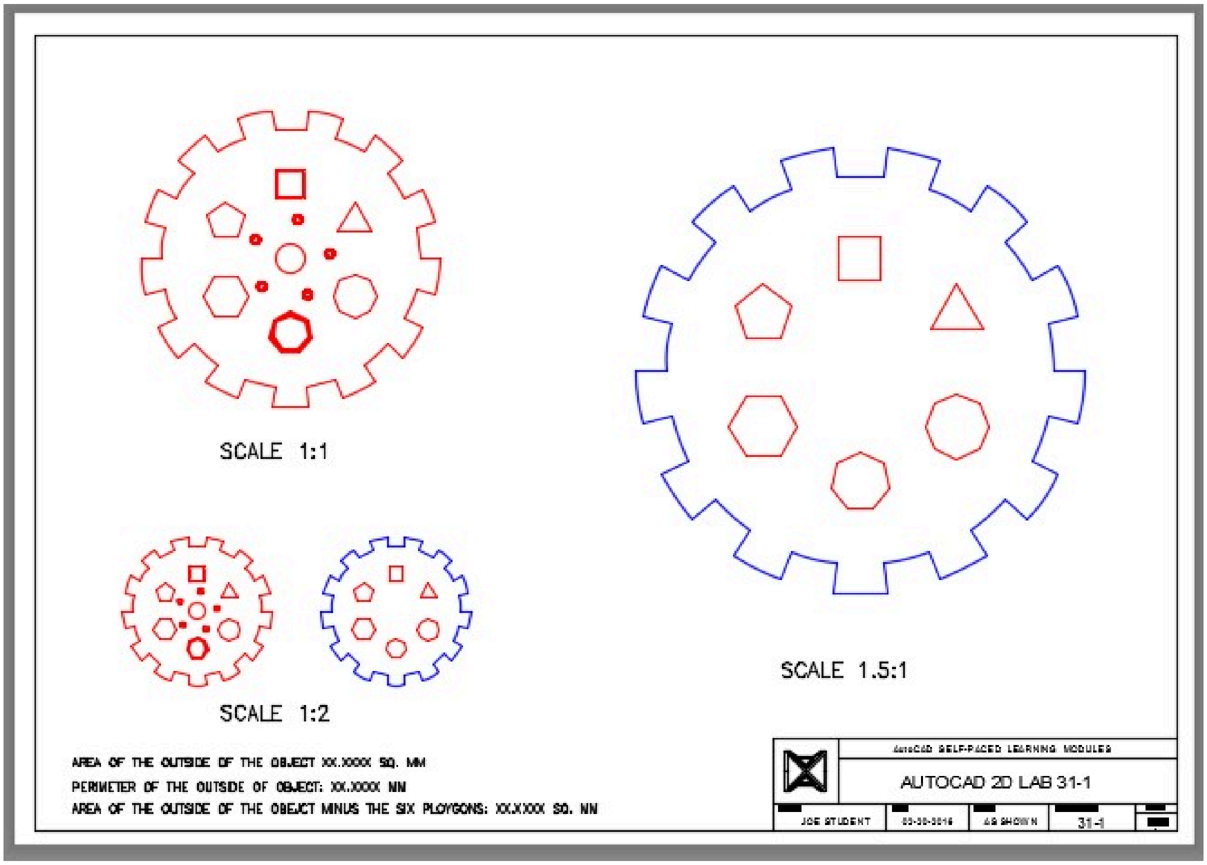
In Paper space, on layer: Text, insert the scales of each viewport as shown in the figure. Set a text height that appears best on the drawing.

Step 9

On layer: Titleblock Text, fill in the titleblock using the standard shown in Module 20.

Step 10

Turn layer: Viewport off. (Figure Step 10)


SCALE 1:1
SCALE 1.5:1
SCALE 1:2
AutoCAD SELF-PACED LEARNING MODULES
AUTOCAD 2D LAB 31-1
JOE STUDENT
03-30-2016
AS SHOWN
31-1

AREA OF THE OUTSIDE OF THE OBJECT XX.XXXX SQ. MM
PERIMETER OF THE OUTSIDE OF OBJECT: XX.XXXX MM
AREA OF THE OUTSIDE OF THE OBJECT MINUS THE SIX PLOYGONS: XX.XXXX SQ. MM

Figure Step 10

Part 7

Module 32: Hatching

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe crosshatching, hatching boundaries, hatching patterns, and associative and nonassociative hatching.
2. Explain sectional views, sections, and hatching in sectional views.
3. Apply the HATCH and HATCHEDIT commands to insert and edit crosshatching patterns.

Crosshatching

Many drawings require areas to be filled with a pattern. This is called *crosshatching* or just *hatching*. There are many uses for hatch patterns on a drawing including displaying an object's material type, the solid material that was cut through in the sectional view, difference materials used in one area to another, plus many other applications.

Hatching Boundaries

The HATCH command is used to insert hatch patterns. It allows you to select two different methods of selecting the boundary of the area to be filled in with the hatch pattern. The first method is picking an internal point and allow the HATCH command to find an enclosed boundary and the second method involves using an existing closed polyline to define the hatching boundary. The first method works well most of the time but for more complicated drawings, the second method must be used.

Hatching Patterns

There are many predefined hatch patterns including ANSI standards, ISO standards, and a solid fill pattern that are part of AutoCAD's software. All hatch patterns are constructed using lines but hatch patterns are inserted as a single drawing object. This helps them to be easily manipulated and also keeps the drawing's database small. A hatching pattern can be exploded, using the EXPLODE command, to convert it to individual lines.

Associative Verses Nonassociative Hatching

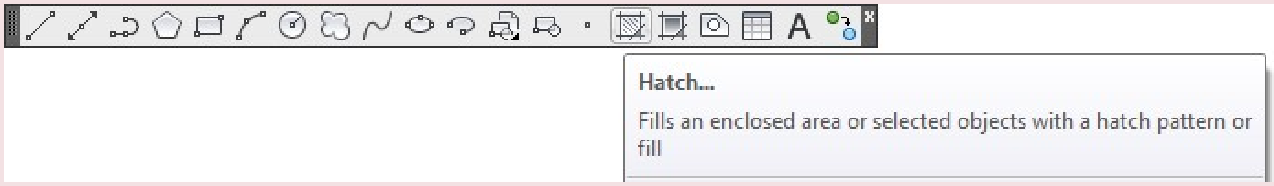
An *associative hatch* will automatically adjust when the boundary, that was used to create it, is moved or changed. A *nonassociative hatch* remains the same size and in the same location regardless of the changes made to the hatch's boundary. An associative hatch is the default in the HATCH command.

An associative hatch can be changed to a nonassociative hatch by changing the properties of the object with the Properties window or with the HATCHEDIT command. A nonassociative hatch can never be changed to a associative hatch. It would have to be deleted and reinserted.

AutoCAD Command: HATCH

The HATCH command is used to insert a hatch pattern.

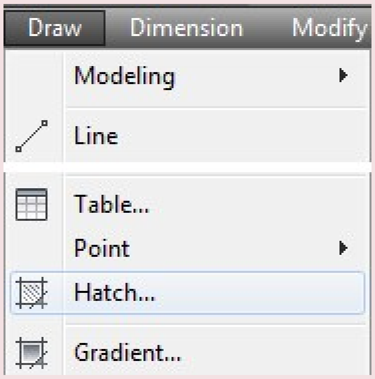
Shortcut: **H**



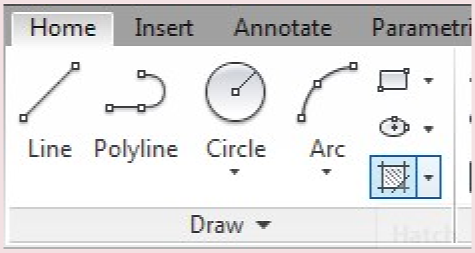
Hatch...

Fills an enclosed area or selected objects with a hatch pattern or fill

Draw Toolbar


Draw
Dimension
Modify
Modeling
Line
Table...
Point
Hatch...
Gradient...

Draw Pull-down

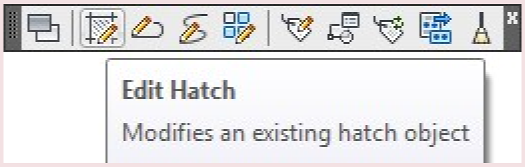

Home
Insert
Annotate
Parametri
Line
Polyline
Circle
Arc
Draw

Home Ribbon

AutoCAD Command: HATCHEDIT

The HATCHEDIT command is used to edit an existing hatch pattern.

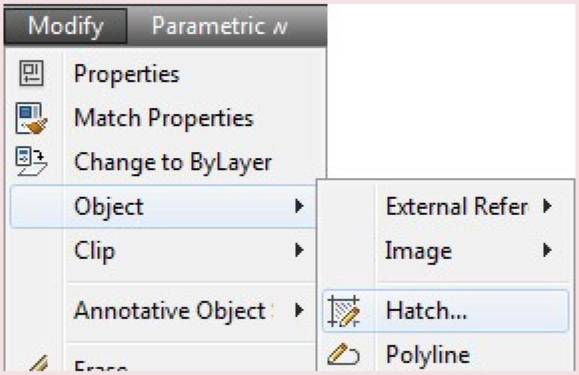
Shortcut: **none**



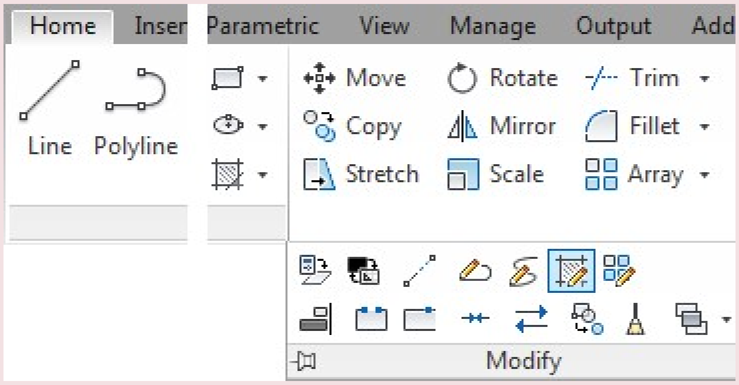
Edit Hatch

Modifies an existing hatch object

Modify II Toolbar


Modify
Parametric
Properties
Match Properties
Change to ByLayer
Object
Clip
Annotative Object
External Refer
Image
Hatch...
Polyline

Modify Pull-down


Home
Inser
Parametric
View
Manage
Output
Add
Line
Polyline
Move
Rotate
Trim
Copy
Mirror
Fillet
Stretch
Scale
Array
Modify

Home Ribbon

Drafting Lesson: Cross Sections – Part 1

When a view of an object requires a clearer description of its interior or it is hard to dimension because of the hidden lines, a cross section view can be drawn in place of the normal multiview. See Figure 32-1

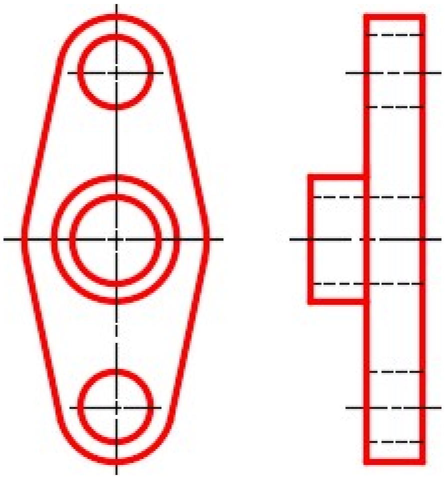


Figure 32-1
Normal Front and Right Side View of an Object

A *cross section* view, also called a *section*, is a view of the object as if it were cut along a cutting plane and the two pieces pulled apart exposing the inside of the object. See Figure 32-2 and 32-4.

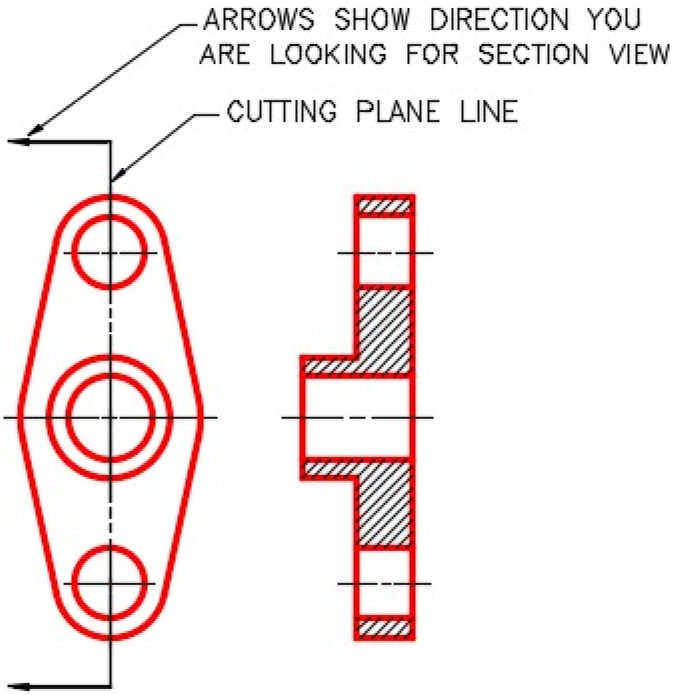

ARROWS SHOW DIRECTION YOU ARE LOOKING FOR SECTION VIEW
CUTTING PLANE LINE

Figure 32-2
A Front view and Right Side Section View

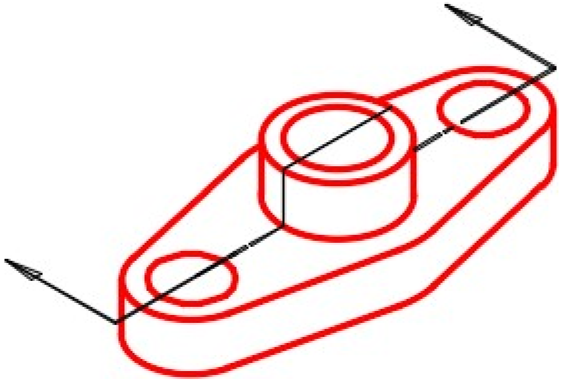


Figure 32-3
3D Model of the Object Showing the Cutting Plane Line

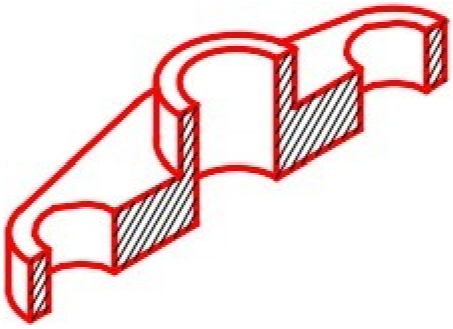


Figure 32-4
3D Model of the Object Showing it Cut on the Cutting Plane Line

A *cutting plane* line is the line along the object where the cut would have been made. See Figure 32-2 and 32-3. The arrows point in the direction that you are looking when drawing the section view. The surfaces of the object that are solid, when cut, are crosshatched.

WORKALONG: Inserting Hatch Patterns

Step 1

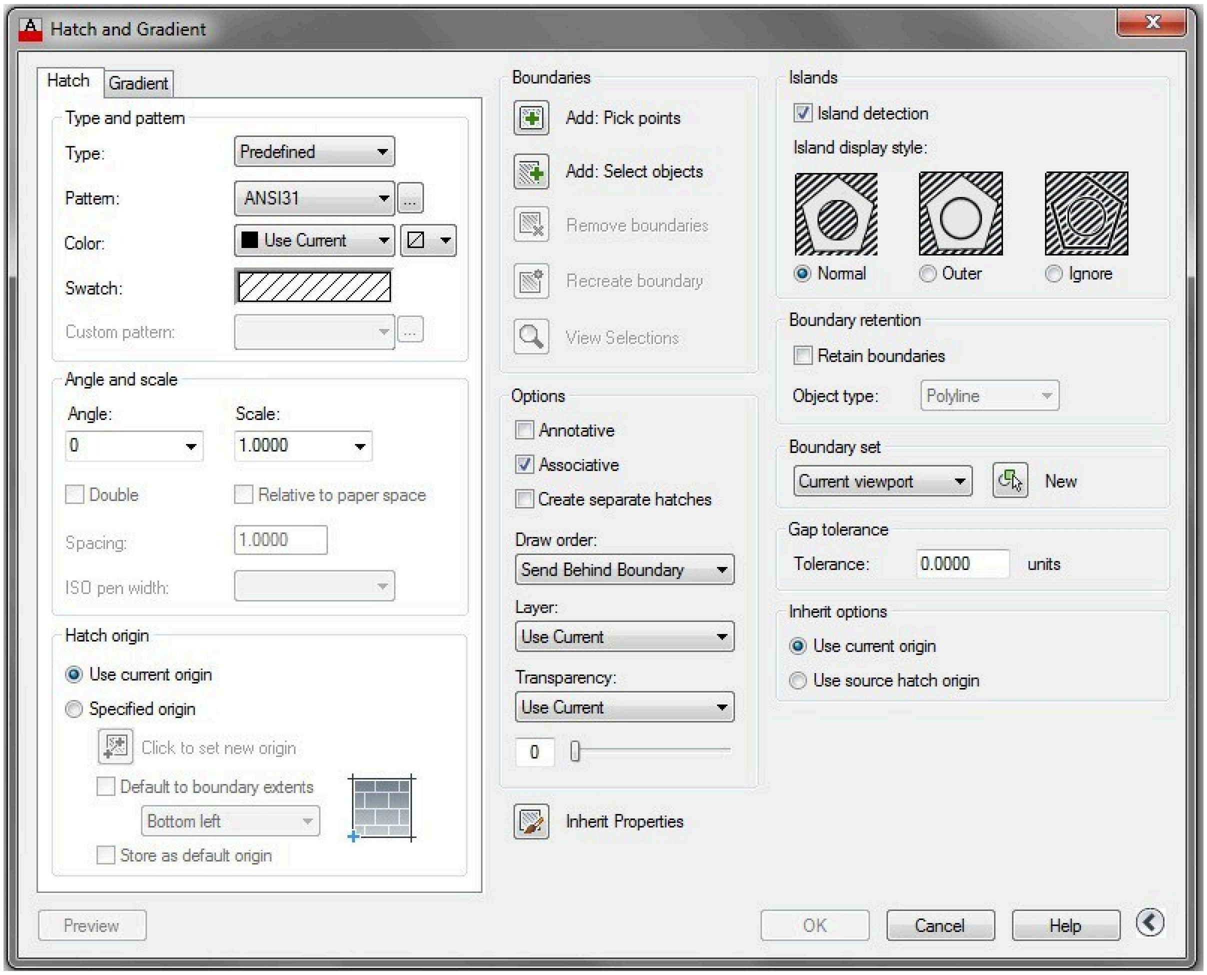
Start a new drawing using the template: 2D Layout English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 32-1.

Step 3

Enter the HATCH command. It will open the Hatch and Gradient dialogue box. The More/Less icon, in the bottom right corner of the dialogue box, is used to open or close the advanced portion of the dialogue box. (Figure Step 3)



Hatch and Gradient

Type and pattern
Type: Predefined
Pattern: ANSI31
Color: Use Current
Custom pattern:

Angle and scale
Angle: 0
Scale: 1.0000
☐ Double
☐ Relative to paper space
Spacing: 1.0000
ISO pen width:

Hatch origin
● Use current origin
○ Specified origin
☐ Default to boundary extents
Bottom left
☐ Store as default origin

Options
☐ Annotative
☑ Associative
☐ Create separate hatches
Draw order: Send Behind Boundary
Layer: Use Current
Transparency: Use Current
0

Islands
☑ Island detection
Island display style:
● Normal ○ Outer ○ Ignore

Boundary retention
☐ Retain boundaries
Object type: Polyline

Boundary set
Current viewport

Gap tolerance
Tolerance: 0.0000 units

Inherit options
● Use current origin
○ Use source hatch origin

Hatch
Gradient
Swatch:
Click to set new origin
Boundaries
Add: Pick points
Add: Select objects
Remove boundaries
Recreate boundary
View Selections
Inherit Properties
New
Preview
OK
Cancel
Help

Figure Step 3

MUST KNOW: A hatch object is constructed using lines but acts as a single drawing object. This helps you manipulate the hatch object and also keeps your drawing’s database small. A hatch object can be exploded into individual line objects with the EXPLODE command. If exploded, its associativity will be lost.

Step 4

Open drawing: AutoCAD 2D Workalong 30-1. Using the SAVEAS command, save and name the drawing: AutoCAD 2D Workalong 32-1. (Figure Step 4)

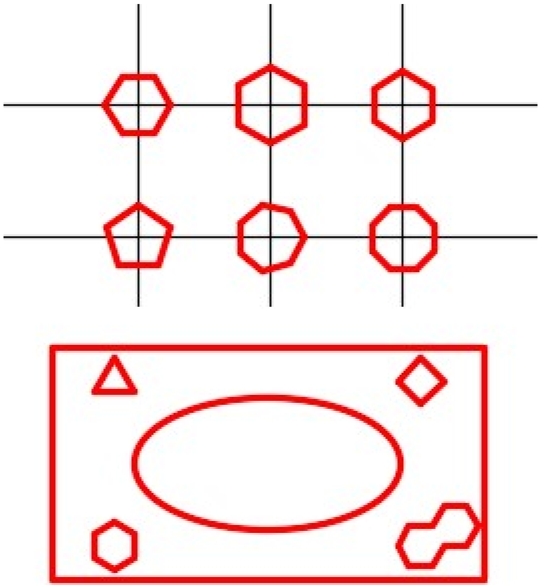


Figure Step 4

Step 5

Using the ROTATE command, rotate the diamond 45 degrees, using its center as the base point, to change it to a square. (Figure Step 5)

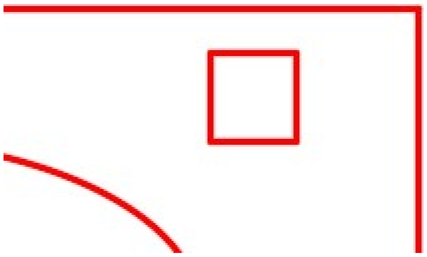


Figure Step 5

Step 6

Using the COPY command, copy the bottom object three times locating them by eye. Align the copies and center them using what you have learned to this point in the book. Their exact location is not important for this exercise but ensure that they are aligned.

Step 7

Erase the 6 regular polygons above the rectangles. On layer: Text, using the DTEXT command, add the text in each copy. Use the text style: 2D Modules at a height of 0.25. Your drawing should appear similar to the figure. (Figure Step 7)

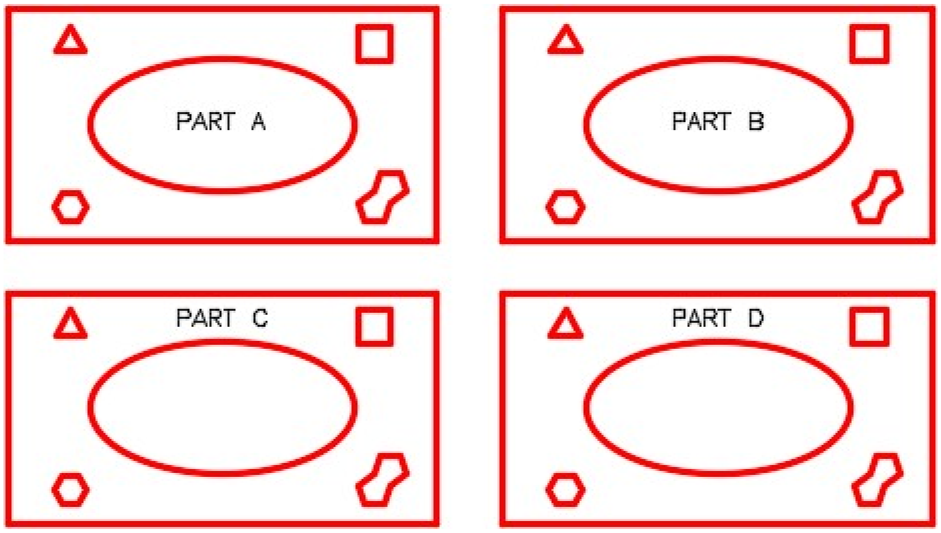
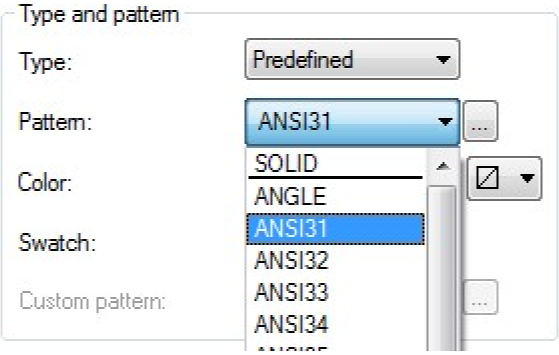

PART A
PART B
PART C
PART D

Figure Step 7

Step 8

Set the current layer to layer: Hatch. Enter the HATCH command. In the Hatch and Gradient dialogue box, set the Type and pattern to ANSI31. In the Angle and scale box, set the Angle to 0 and the Scale to 1. (Figure Step 8A and 8B)



Type and pattern

Type: Predefined

Pattern: ANSI31

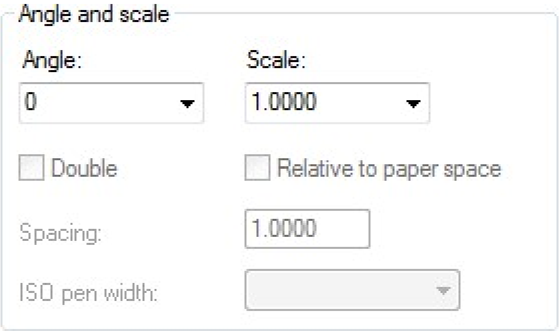
Color:

Swatch:

Custom pattern:

SOLID
ANGLE
ANSI31
ANSI32
ANSI33
ANSI34

Figure Step 8A



Angle and scale

Angle: 0

Scale: 1.0000

☐ Double

☐ Relative to paper space

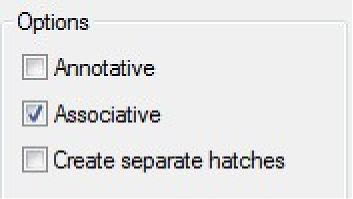
Spacing: 1.0000

ISO pen width:

Figure Step 8B

Step 9

In the Options box, ensure that Associative is enabled. (Figure Step 9)



Options

☐ Annotative

☑ Associative

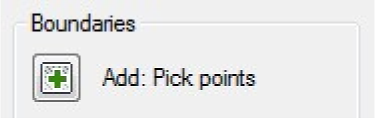
☐ Create separate hatches

Figure Step 9

AUTHOR'S COMMENTS: When a hatch pattern is created as an associative, it will automatically adjust when the boundary, that was used when it was created, is moved or changed. A nonassociative hatch remains the same size and in the same location regardless of the changes made to the hatch's boundary.

Step 10

In the Boundaries box, click the Add Pick points icon and then pick the point P1 in Part A as shown in the figure. The hatch pattern will hatch the ellipse, without hatching over the text. (Figure Step 10A, 10B, and 10C)



Boundaries

Add: Pick points

Figure Step 10 A

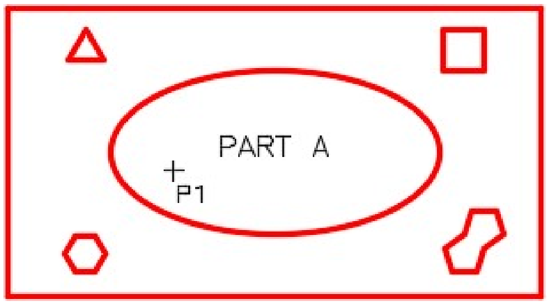


Figure Step 10B

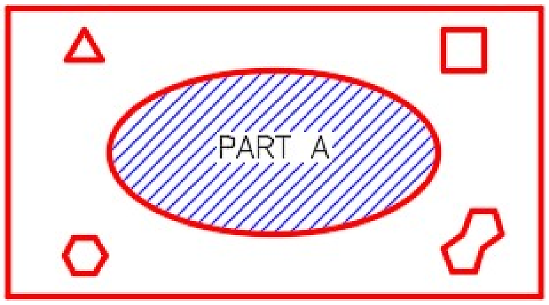


Figure Step 10C

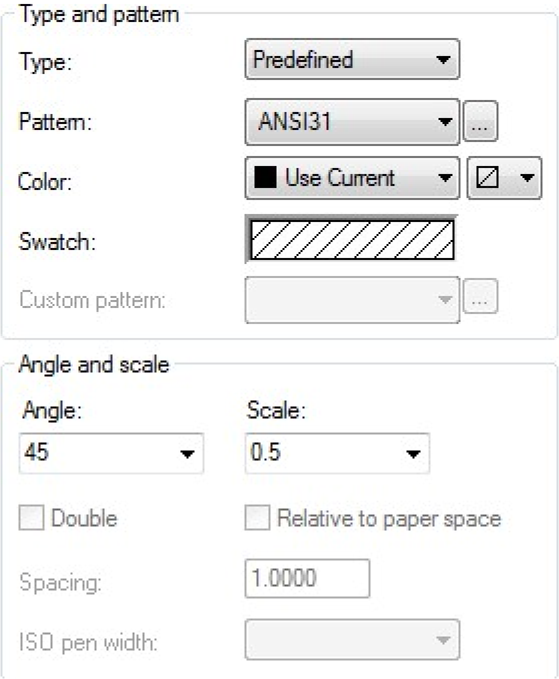
AUTHOR’S COMMENTS: The area to be hatched can be hatched using either the Pick Point method or the Select Object method. They will produce very different results and are used for different purposes as you will see as you work your way through this module.

Step 11

Click OK to complete the command.

Step 12

Enter the HATCH command again. In the Hatch dialogue box, set the Pattern to ANSI31, the Angle to 45 degrees and the Scale to 0.5. (Figure step 12)



Type and pattern

Type: Predefined

Pattern: ANSI31

Color: Use Current

Swatch:

Custom pattern:

Angle and scale

Angle: 45

Scale: 0.5

☐ Double

☐ Relative to paper space

Spacing: 1.0000

ISO pen width:

Figure Step 12

Step 13

Click the Add Select objects icon and when prompted, select the ellipse in Part B as shown in the figure. The hatch pattern will ignore everything inside the object as you can see in the figure. (Figure Step 13A, 13B, and 13C)

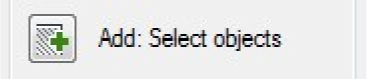


Figure Step 13A

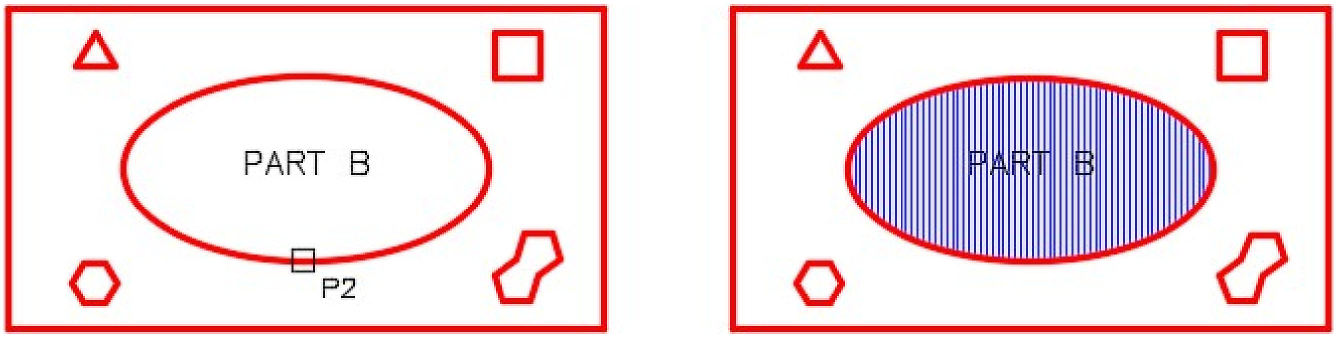
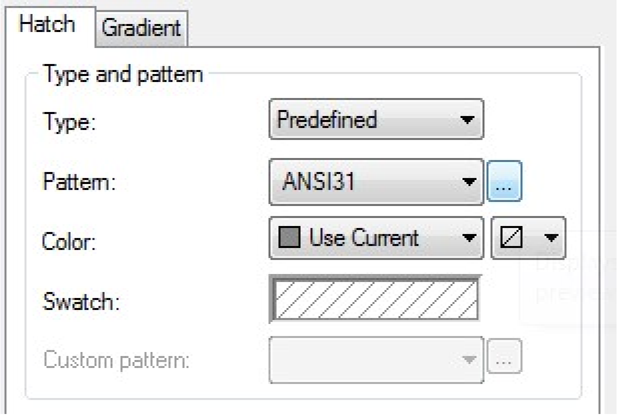


Figure Step 13B *Figure Step 13C*

Step 14

Enter the HATCH command. Click the … icon at the end of the Pattern box. This will open the Hatch Pattern Palette dialogue box. Select the AR-HBONE pattern. (Figure Step 14A and 14B)



Type and pattern

Type: Predefined

Pattern: ANSI31 ...

Color: Use Current

Swatch:

Custom pattern: ...

Figure Step 14A

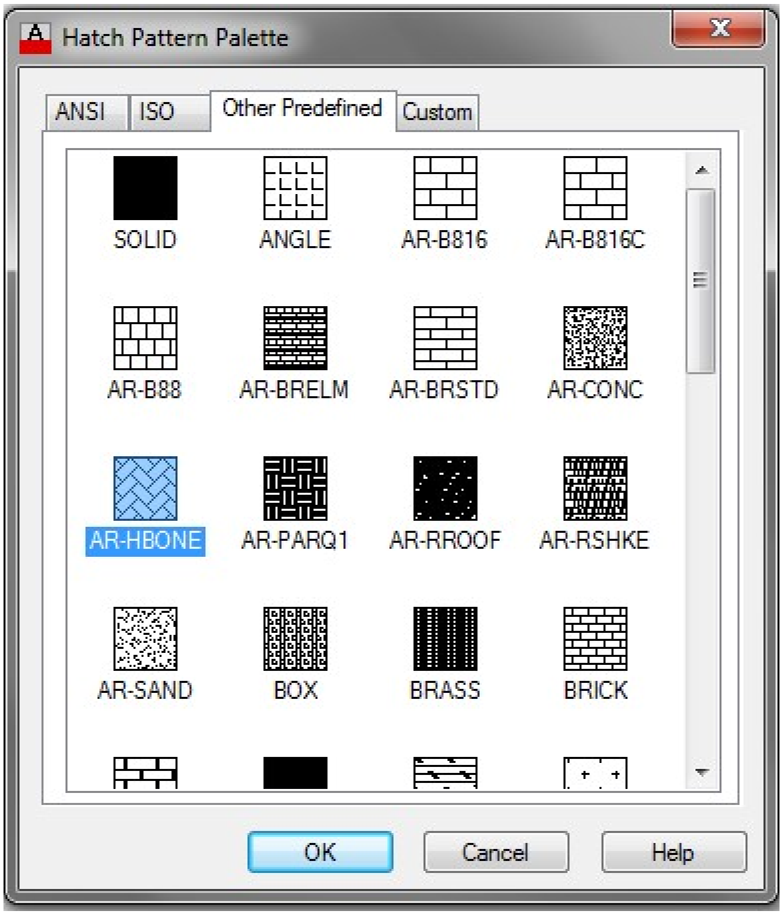
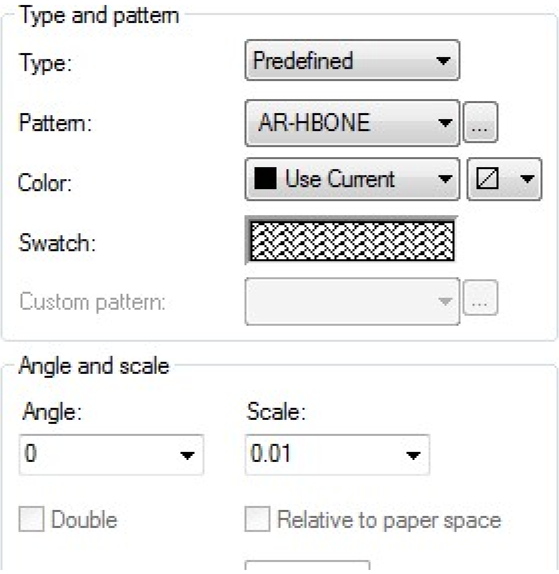

Hatch Pattern Palette
ANSI
ISO
Other Predefined
Custom
SOLID
ANGLE
AR-B816
AR-B816C
AR-B88
AR-BRELM
AR-BRSTD
AR-CONC
AR-HBONE
AR-PARQ1
AR-RROOF
AR-RSHKE
AR-SAND
BOX
BRASS
BRICK
OK
Cancel
Help

Figure Step 14B

Step 15

With the AR-HBONE pattern selected, set the Angle to 0 and the Scale to 0.01. (Figure Step 15)



Type and pattern

Type: Predefined

Pattern: AR-HBONE

Color: Use Current

Swatch:

Custom pattern:

Angle and scale

Angle: 0

Scale: 0.01

☐ Double

☐ Relative to paper space

Figure Step 15

Step 16

Using what you just learned, insert the hatching patterns in Part A using the Select Object method. Use the following specifications: (Figure Step 16)

Triangle – Pattern Hr-HBone

Angle – 0 Scale – 0.1

Square – Pattern Honey

Angle – 0 Scale – 0.5

Hexagon – Pattern Gravel

Angle – 0 Scale – 0.25

8 Sided Polygon – Pattern Solid, Angle N/A – Scale – N/A

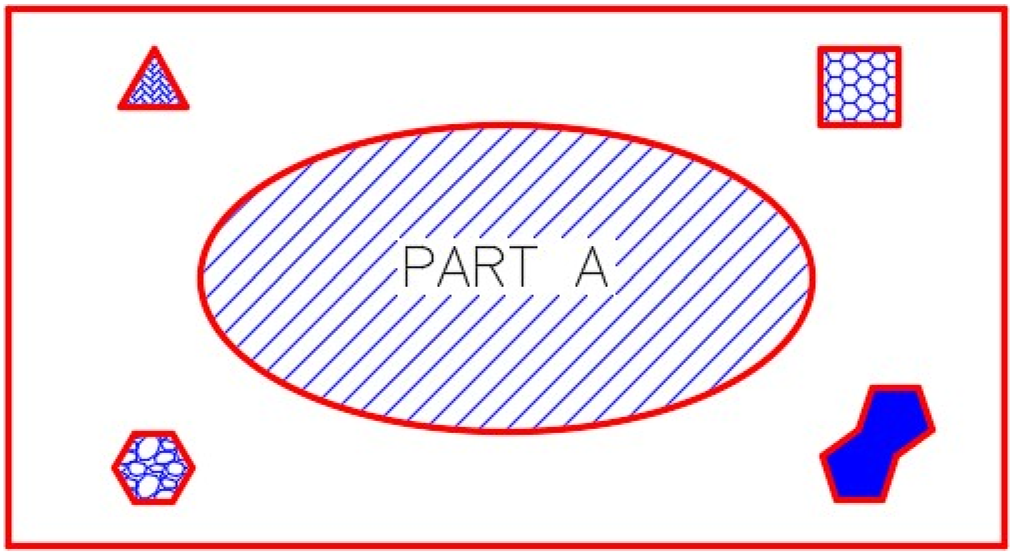


Figure Step 16

Step 17

On layer: Object, draw a 1 inch diameter circle in the center of the ellipses in Part C and Part D. (Figure Step 17)

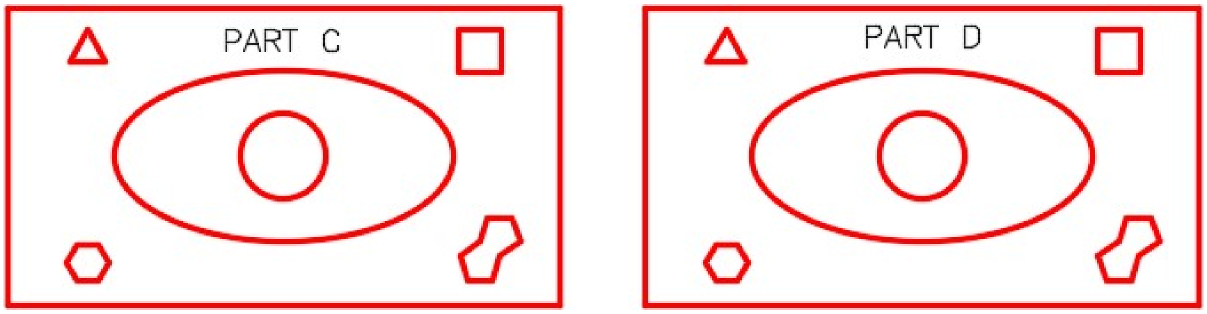
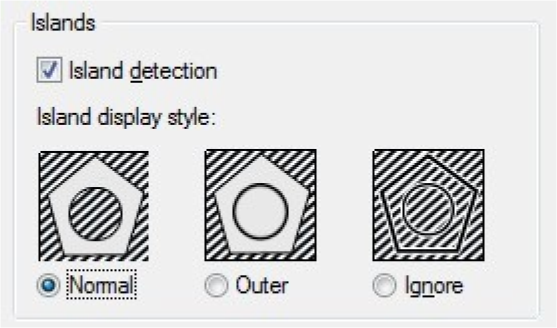


Figure Step 17

Step 18

Enter the HATCH command. Ensure that the Islands box is set to Normal as shown in the figure. Set the Types and pattern to Stars, the Angle to 0 and Scale to 0.5. (Figure Step 18A and 18B)



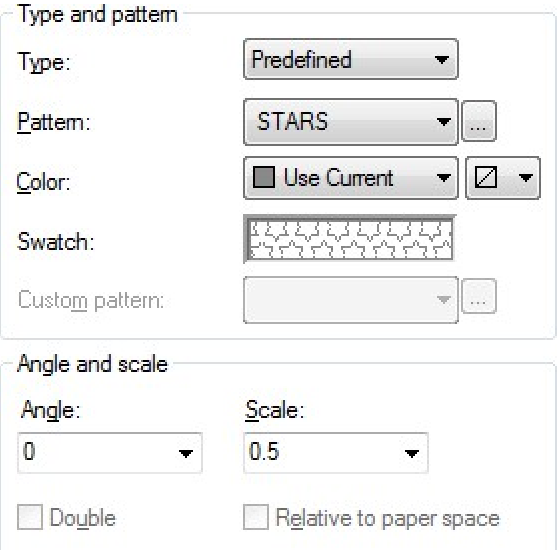
Islands

☑ Island detection

Island display style:

● Normal ○ Outer ○ Ignore

Figure Step 18A



Type and pattern

Type: Predefined

Pattern: STARS

Color: Use Current

Swatch:

Custom pattern:

Angle and scale

Angle: 0

Scale: 0.5

☐ Double ☐ Relative to paper space

Figure Step 18B

Step 19

Click the Add Pick points icon. In Part C, select inside the rectangle as shown by P4. Part C should hatch as shown in the figure. (Figure Step 19A and 19B)

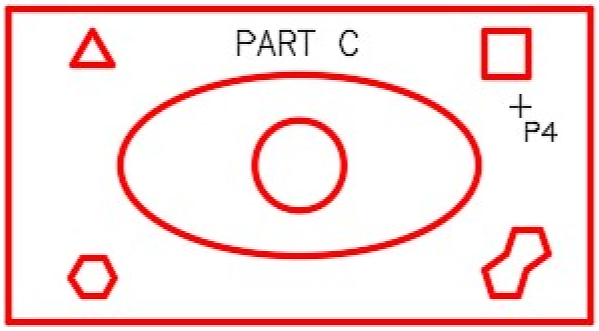


Figure Step 19A

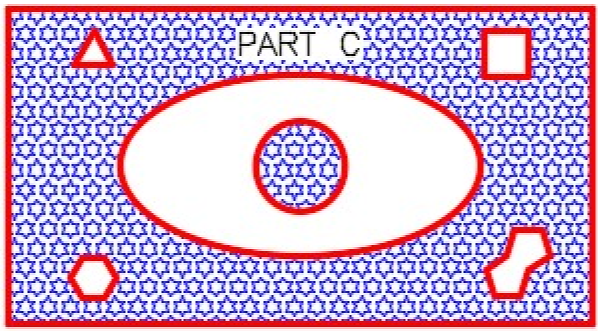
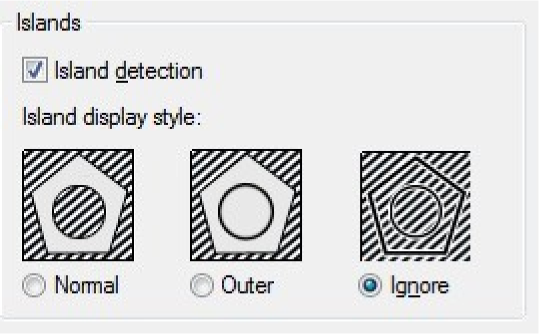


Figure Step 19B

AUTHOR'S COMMENTS: When the Islands display style is set to Normal, the hatching pattern will hatch every other island.

Step 20

Enter the HATCH command and set the Islands display style to Ignore. (Figure Step 20)



Islands

☑ Island detection

Island display style:

○ Normal ○ Outer ● Ignore

Figure Step 20

Step 21

Click the Add Pick points icon and select the location P5 in Part D. (Figure Step 21A and 21B)

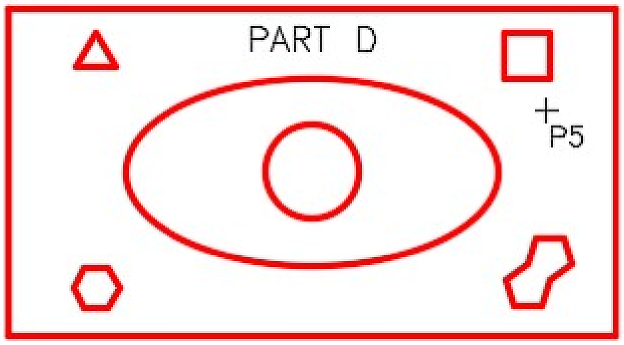


Figure Step 21A

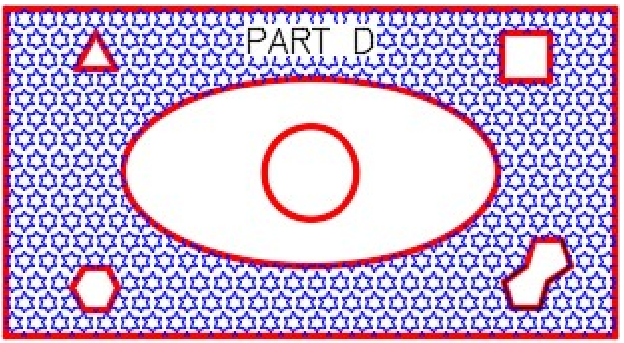


Figure Step 21B

AUTHOR'S COMMENTS: Note how all islands are ignored.

Step 22

Your completed drawing should match the figure. (Figure Step 22)

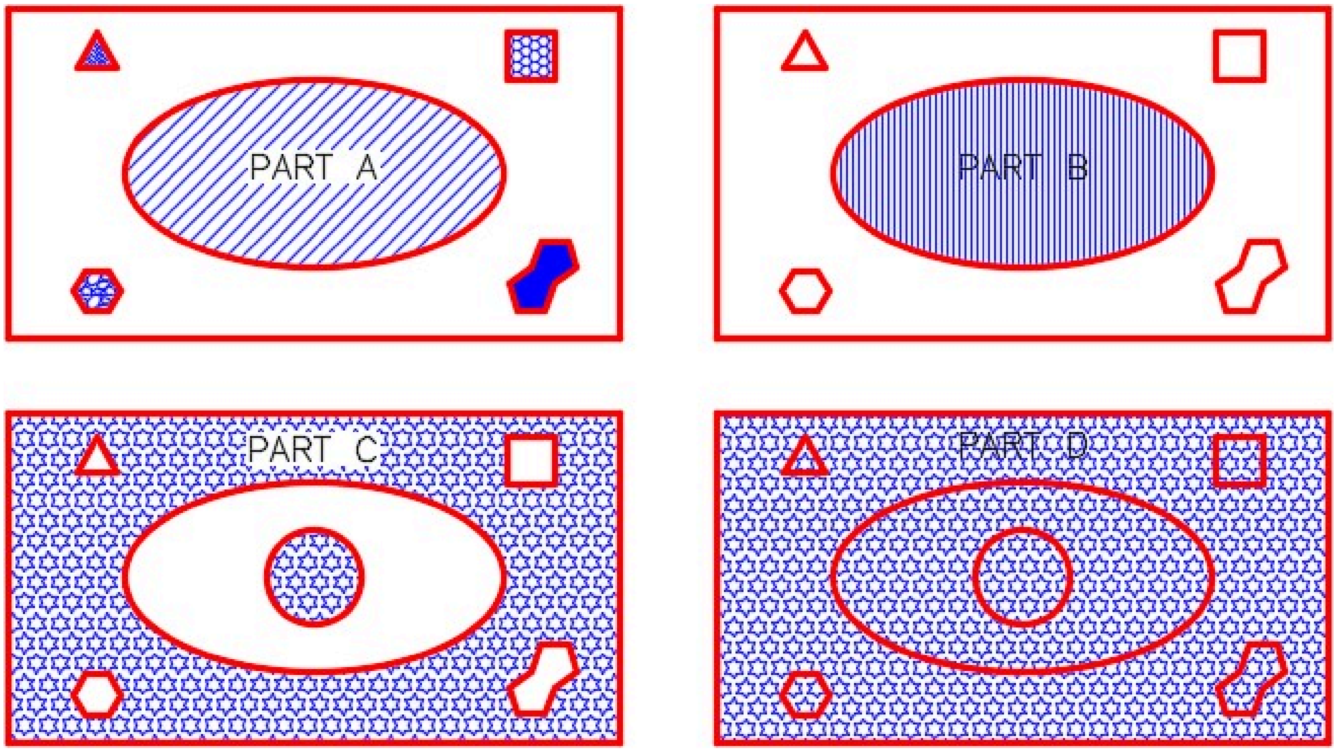
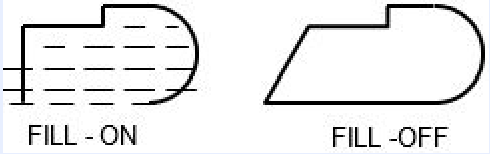

PART A
PART B
PART C
PART D

Figure Step 22

Step 23

Save and close the drawing.

MUST KNOW: The FILL command controls the fill of a hatching pattern plus other feature in AutoCAD. If toggled off, hatch patterns will not appear on the drawing. This can help you work faster since the drawing regenerates faster. Do not forget to turn the FILL on when finished. The REGEN command must be executed to display the current fillmode after the FILL command has been executed.



WORKALONG: Using the HATCHEDIT Command

Step 1

Open the drawing: AutoCAD 2D Workalong 32-1. Using the SAVEAS command, save and name the drawing: AutoCAD 2D Workalong 32-2. (Figure Step 1)

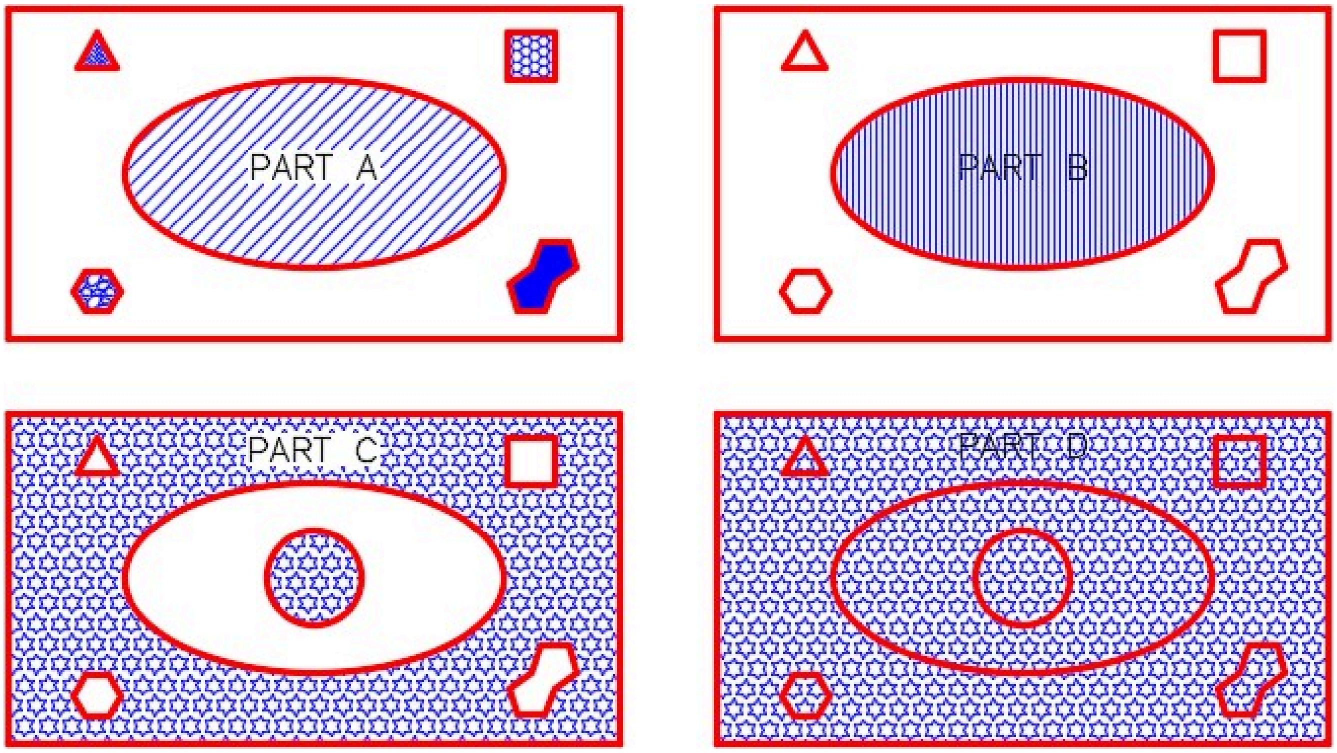


Figure Step 1

Step 2

Enter the HATCHEDIT command, as shown below. (Figure Step 2)

Command: **HATCHEDIT**

Select hatch object: **P1**

(Select the hatching pattern inside the ellipse of PART A. This will open the Hatch Edit dialogue box.)

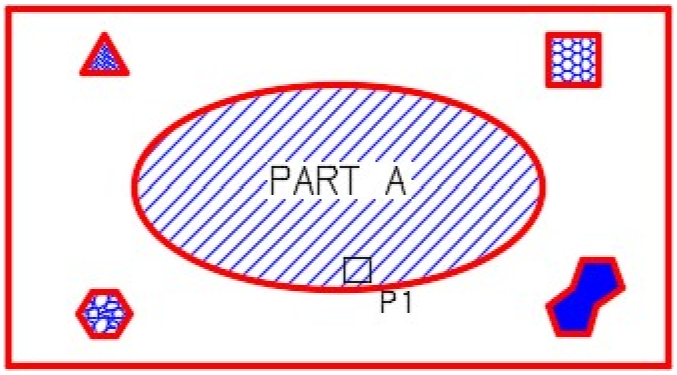
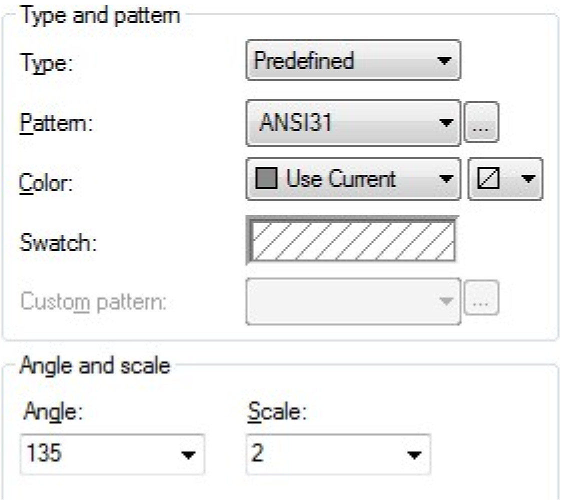


Figure Step 2

Step 3

In the Angle and scale box change the Angle to 135 and the Scale to 2 as shown in Figure Step 3A. Click OK. Notice the change to the hatching pattern. (Figure Step 3A and 3B)


Type and pattern

Type: Predefined

Pattern: ANSI31

Color: Use Current

Swatch:

Custom pattern:

Angle and scale

Angle: 135

Scale: 2

Figure Step 3A

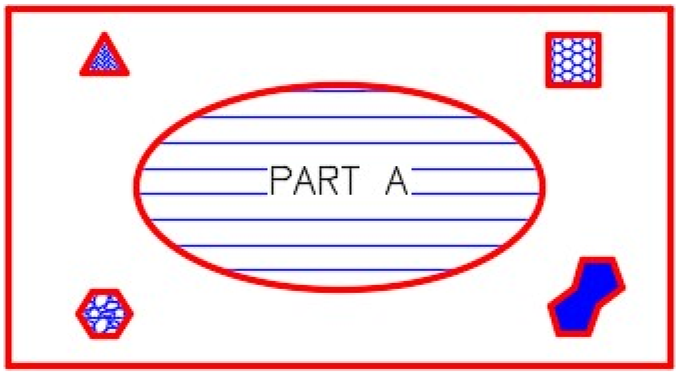
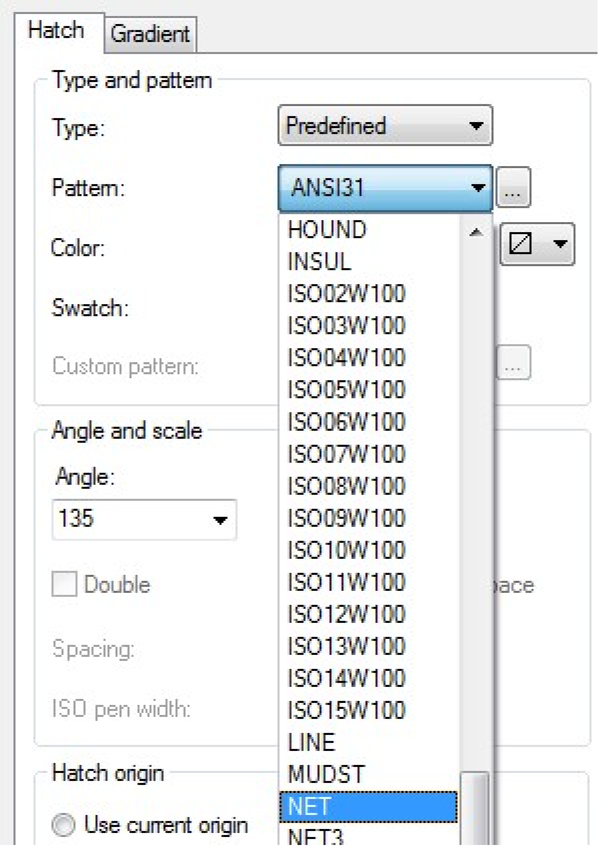


Figure Step 3B

Step 4

To edit the hatching pattern for Part B, double click the pattern to open the Hatchedit dialogue box. On the pull-down list for Pattern, select Net. (Figure Step 4A and 4B)



Type and pattern

Type: Predefined

Pattern: ANSI31

Angle and scale

Angle: 135

☐ Double

Hatch
Gradient
Color:
Swatch:
Custom pattern:
Spacing:
ISO pen width:
Hatch origin
○ Use current origin
HOUND
INSUL
ISO02W100
ISO03W100
ISO04W100
ISO05W100
ISO06W100
ISO07W100
ISO08W100
ISO09W100
ISO10W100
ISO11W100
ISO12W100
ISO13W100
ISO14W100
ISO15W100
LINE
MUDST
NET
NET3
...
ace

Figure Step 4A

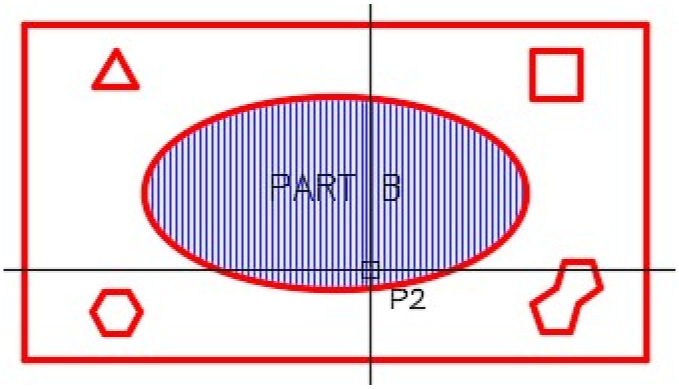
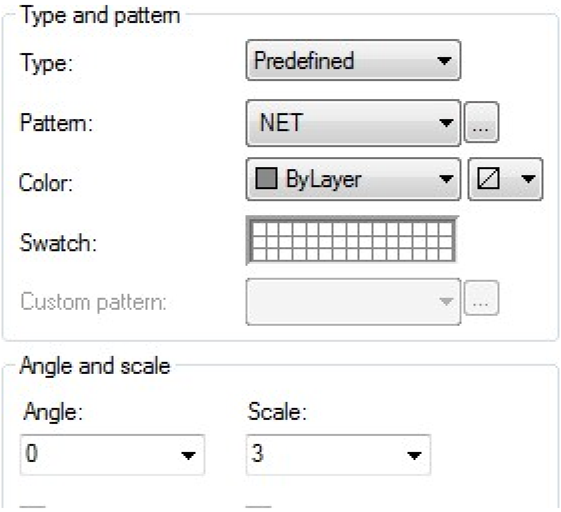


Figure Step 4B

Step 5

In the Angle and scale box, change the Angle to 0 and the Scale to 3. Click OK. Note the change to the hatching pattern. (Figure Step 5A and 5B)



Type and pattern

Type: Predefined

Pattern: NET

Color: ByLayer

Swatch:

Custom pattern:

Angle and scale

Angle: 0

Scale: 3

Figure Step 5A

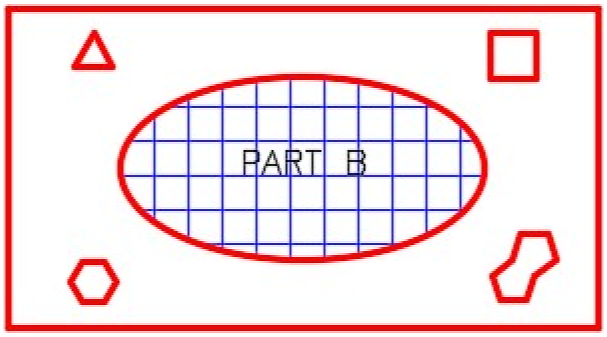


Figure Step 5B

Step 6

Delete the hatch in Part D. (Figure Step 6)

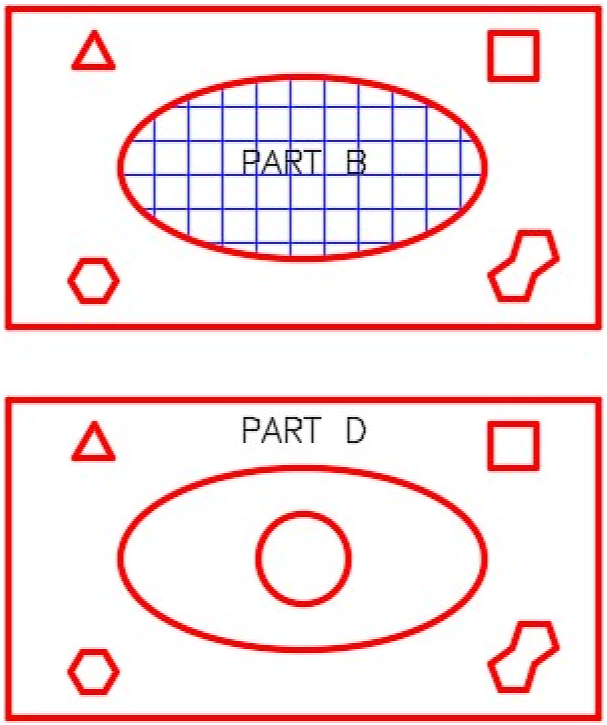


Figure Step 6

Step 7

Enter the HATCH command and click the Inherit Properties icon. Select the Net hatch pattern in Part B. With the paint brush graphic cursor, select inside the ellipse in Part D. (Figure Step 7A, 7B, 7C, and 7D)

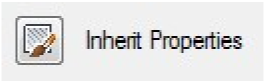


Figure Step 7A

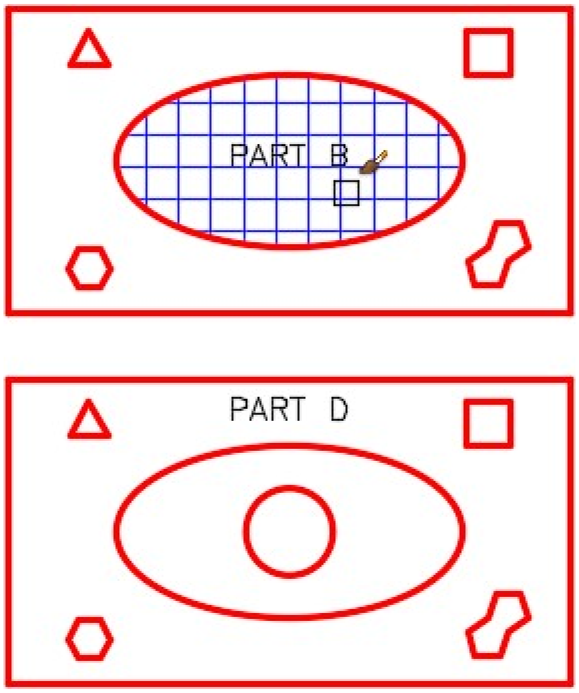

PART B
PART D

Figure Step 7B

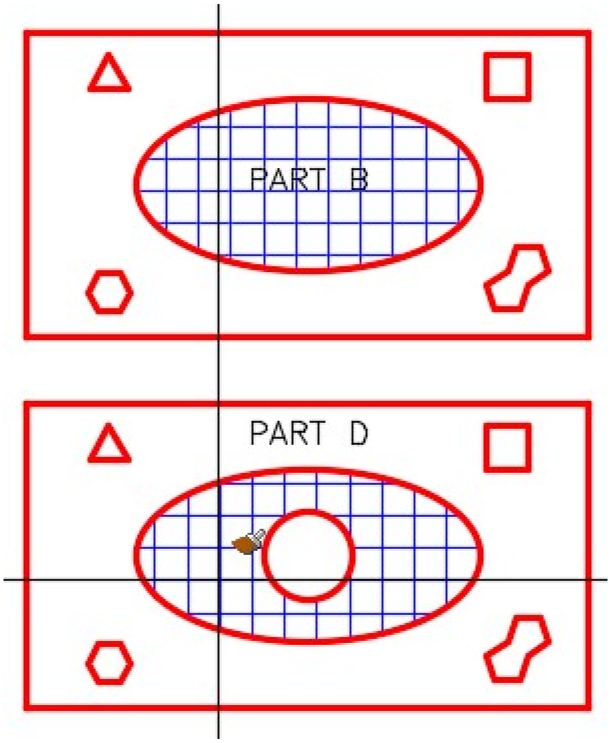

PART B
PART D

Figure Step 7C

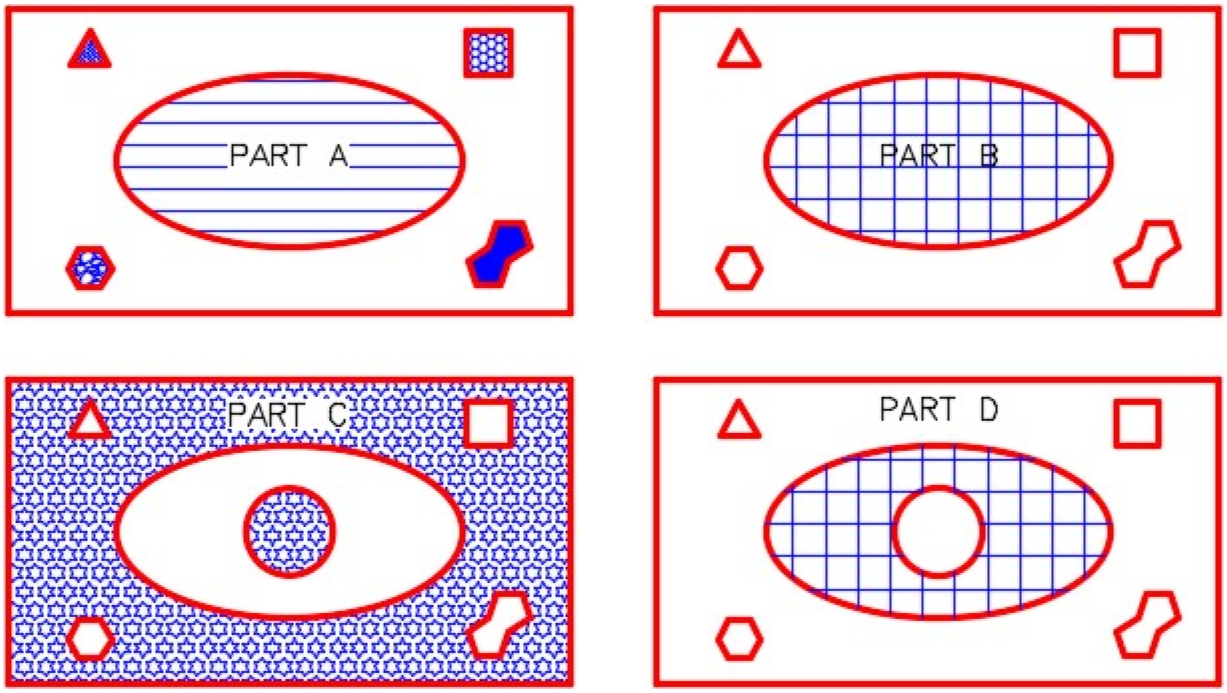


Figure Step 7D

Step 8

Enter the commands shown below. Note how the hatching patterns will not display when Fill is Off and display when it is On. (Figure Step 8A and 8B)

```
Command: FILL

Enter mode [ON/OFF] <ON>: OFF

Command: REGEN

Regenerating model.

Command: FILL

Enter mode [ON/OFF] <OFF>: ON

Command: REGEN

Regenerating model. Command
```

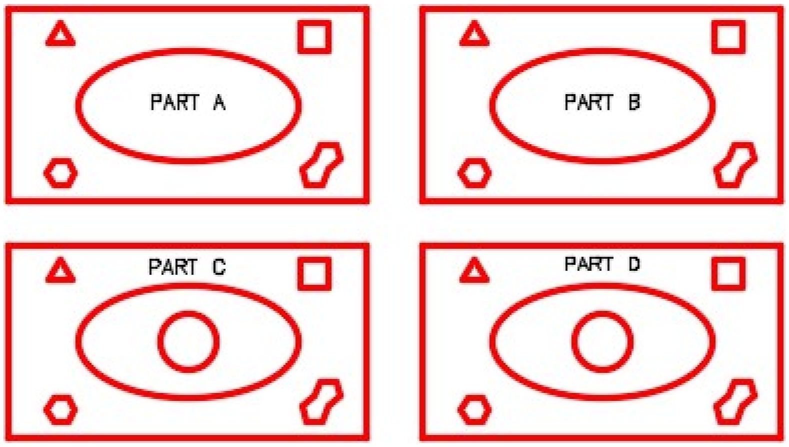


Figure Step 8A

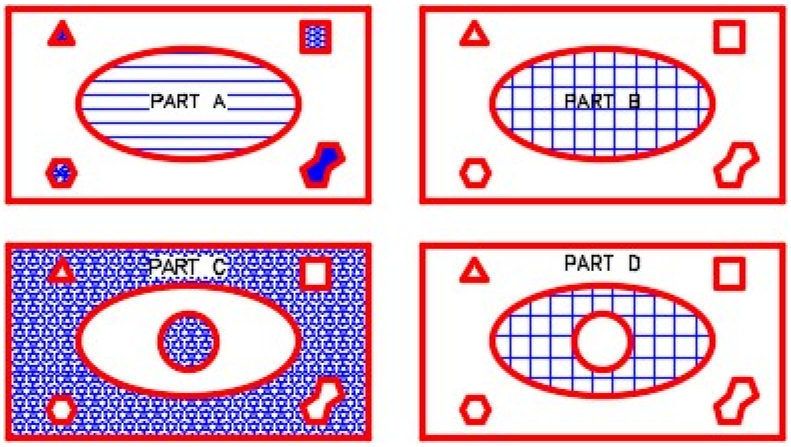


Figure Step 8B

AUTHOR’S COMMENTS: To speed drawing regeneration, turn Fill off and on while you are working on the drawing. Turn it on when you complete the drawing and are ready to plot.

Key Principles

Key Principles in Module 32

1. When using the Select object method to hatch, a closed polyline must be created before executing the HATCH command. Since a circle, rectangle, or an ellipse are closed objects, they can be used as a closed polyline.
2. All hatch patterns are constructed using lines but the hatch pattern is created as a single object in the drawing. This helps to manipulate the hatch patterns and keep the drawing’s database smaller. A hatching pattern can be exploded back into its individual line objects using the EXPLODE command.
3. An associative hatch will automatically adjust when the boundary, that was used when it was

created, is moved or changed. A nonassociative hatch remains the same size and in the same location regardless of the changes made to the hatch boundary.

Lab Exercise 32-1

Time allowed: 40 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 32-1	2D Layout English	Inches

Step 1

On layer: Object, draw the top and front views of the object shown in figures. (Figure Step 1)

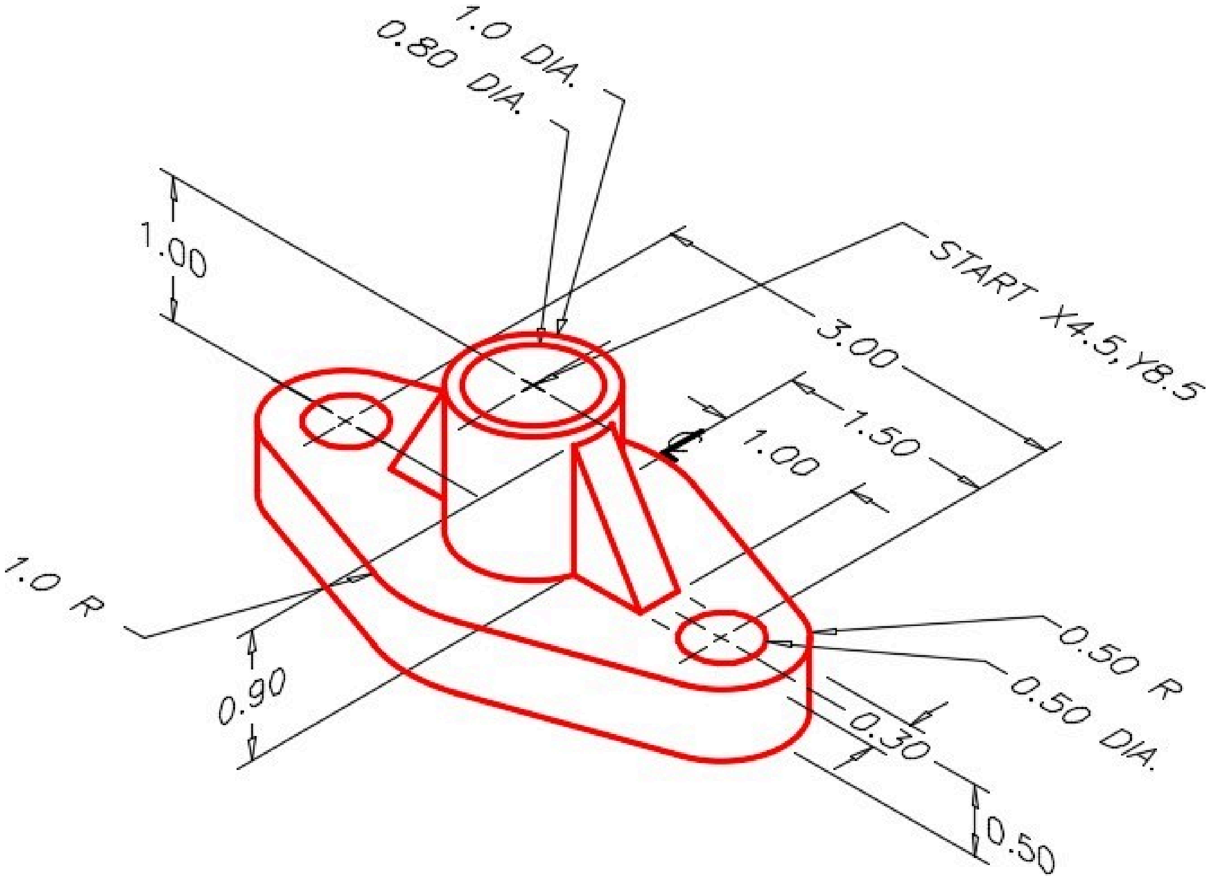

1.0 DIA.
0.80 DIA.
1.00
START X4.5,Y8.5
3.00
1.50
1.00
1.0 R
0.90
0.50 R
0.50 DIA.
0.30
0.50

Figure Step 1

Step 2

Use the spacing between views as shown in the figure. (Figure Step 2)

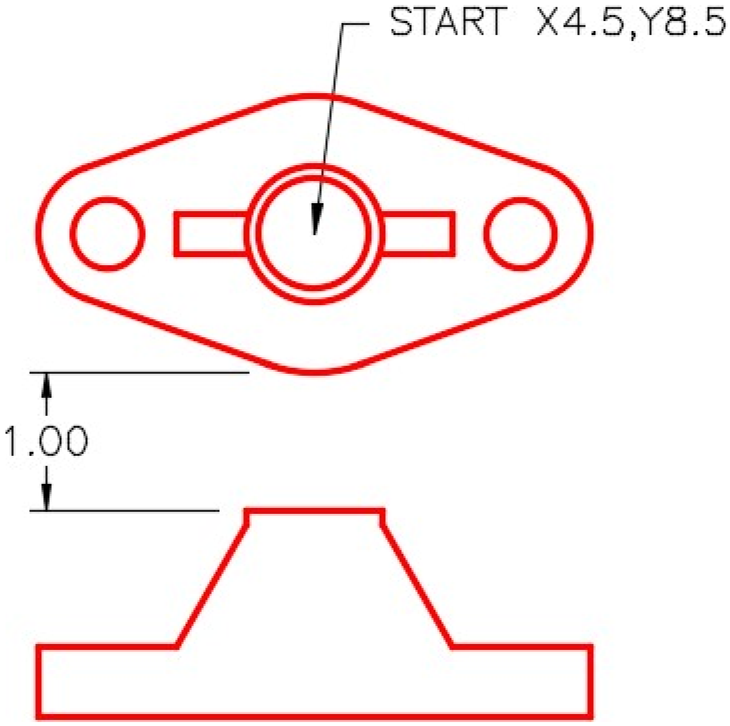


Figure Step 2
Top and Front View Locations

Step 3

On the Front view, add the full cross section. The cutting plane line is shown in Figure Step 3A. On layer: Hatch, insert the cross hatching pattern using the pattern ANSI31. Select an appropriate scale. (Figure Step 3A and 3B)

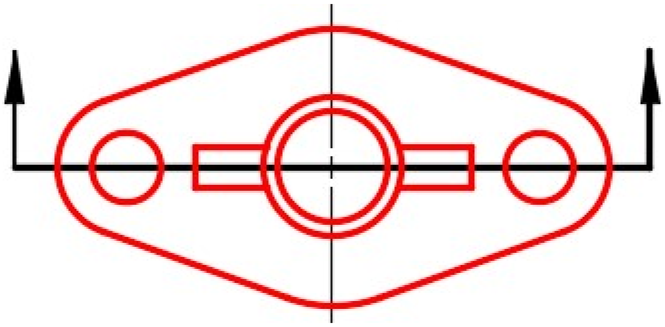


Figure Step 3A

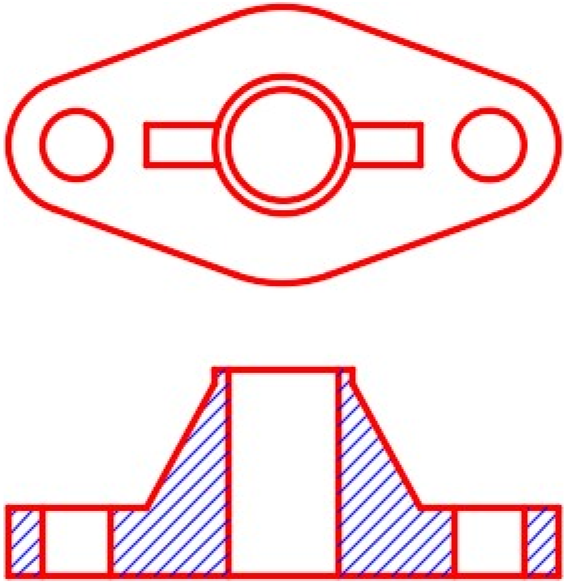
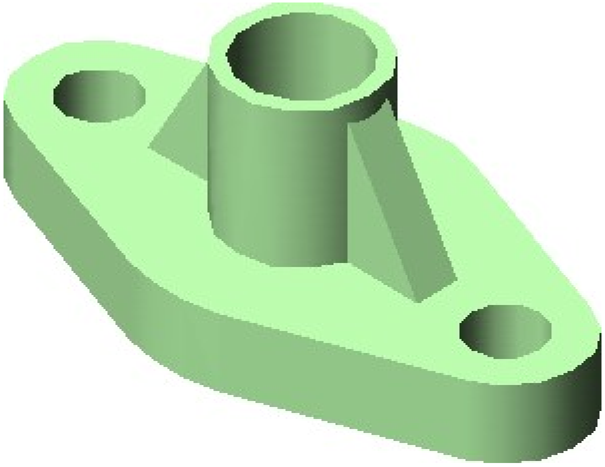


Figure Step 3B
Completed Drawing

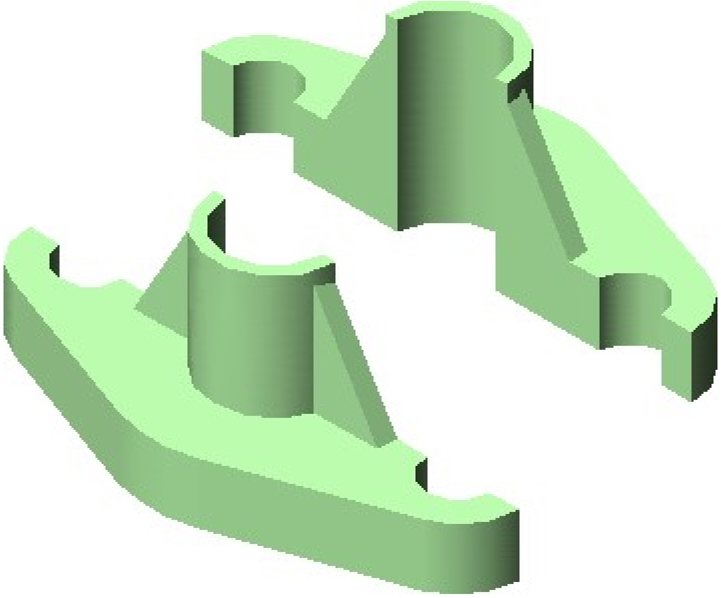
AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 1

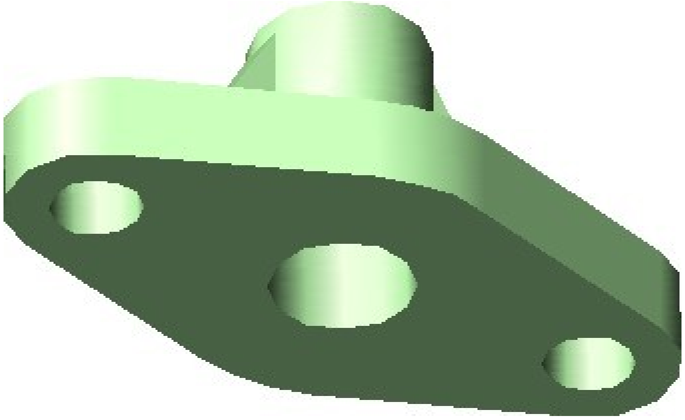
If you have trouble visualizing the object and the section view you have to draw, study the solid models of the object shown below. The solid modeling views were done using AutoCAD. Solid Modeling is taught in the AutoCAD 3D book.



Solid Model Isometric View



Solid Model Sliced on Cutting Plane Line



Solid Model View from Bottom

Lab Exercise 32-2

Time allowed: 50 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 32-2	2D Layout English	Inches

Step 1

On layer: Object, draw the Front and Right Side views of the figure. Draw the Right Side view as a full section. (Figure Step 1A, 1B, and 1C)

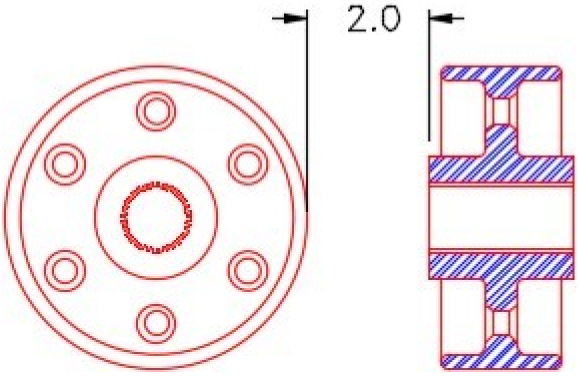


Figure Step 1A
Location of the Views

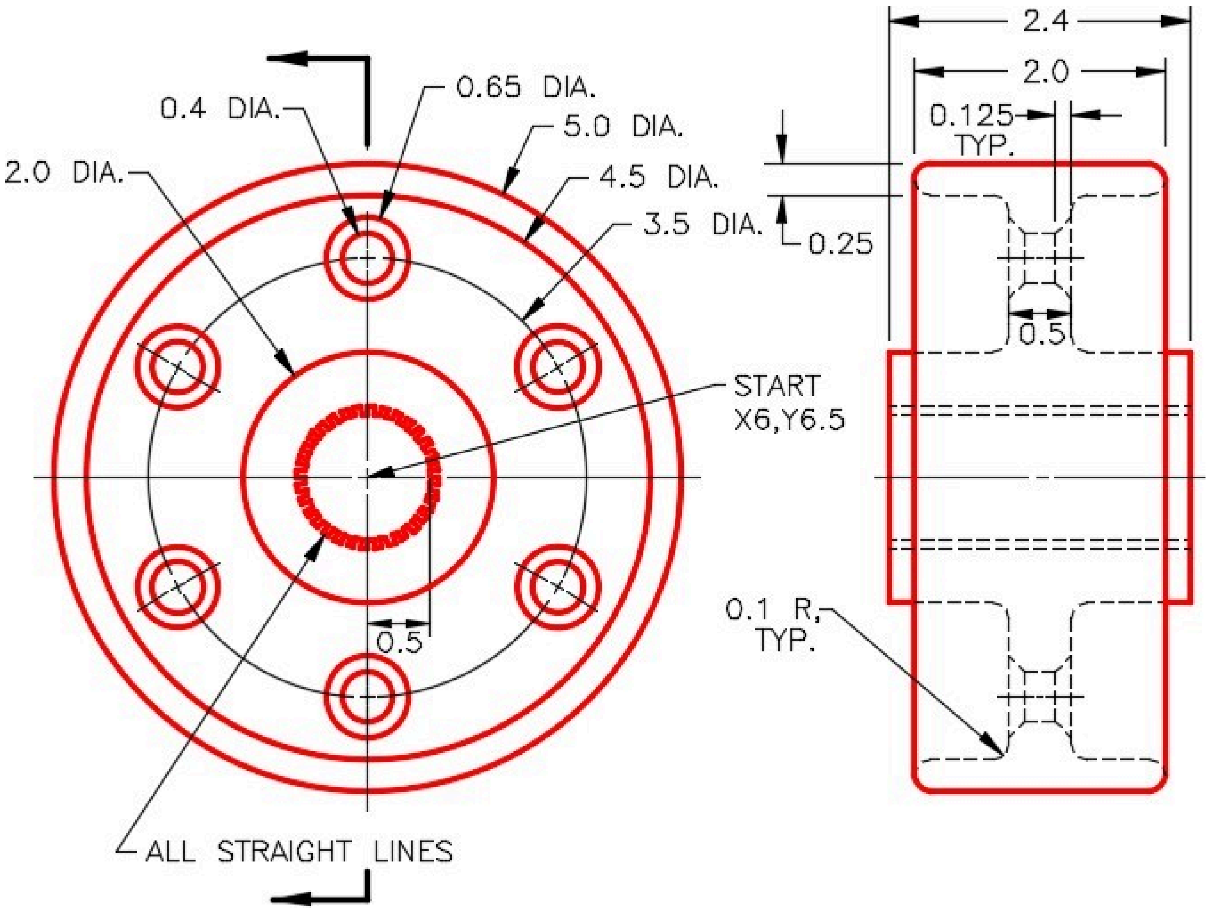

0.4 DIA.
0.65 DIA.
5.0 DIA.
2.0 DIA.
4.5 DIA.
3.5 DIA.
START
X6,Y6.5
0.5
ALL STRAIGHT LINES
2.4
2.0
0.125
TYP.
0.25
0.5
0.1 R,
TYP.

Figure Step 1B

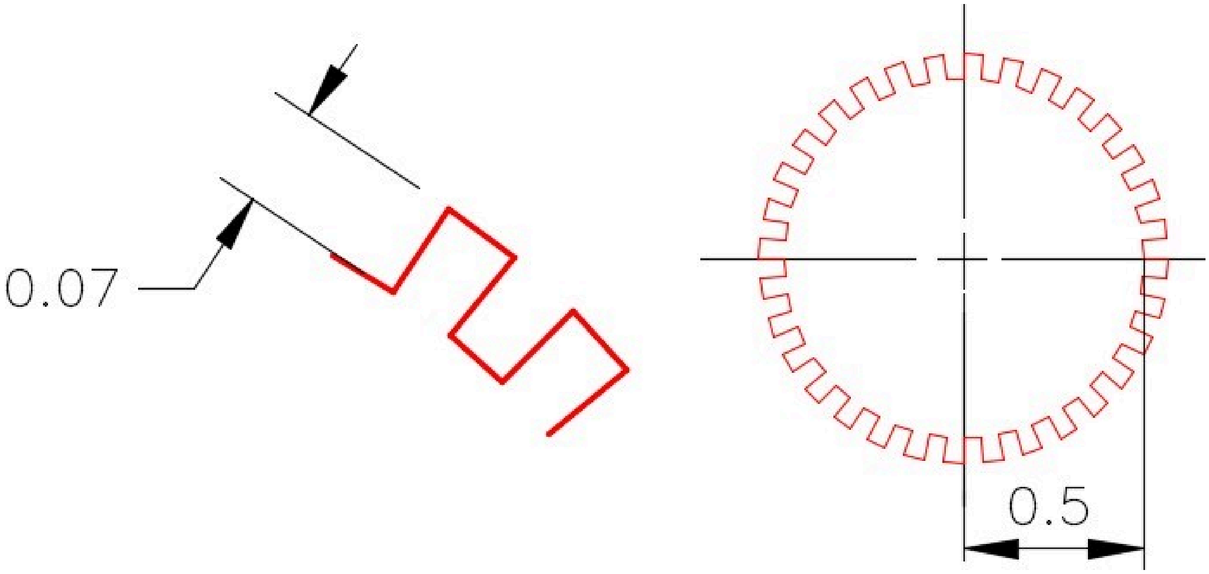


Figure Step 1C
Detail of Gear Teeth

Step 2

On layer: Hatch, insert the cross hatching pattern using the pattern ANSI32 at a scale of 0.5. (Figure Step 2)

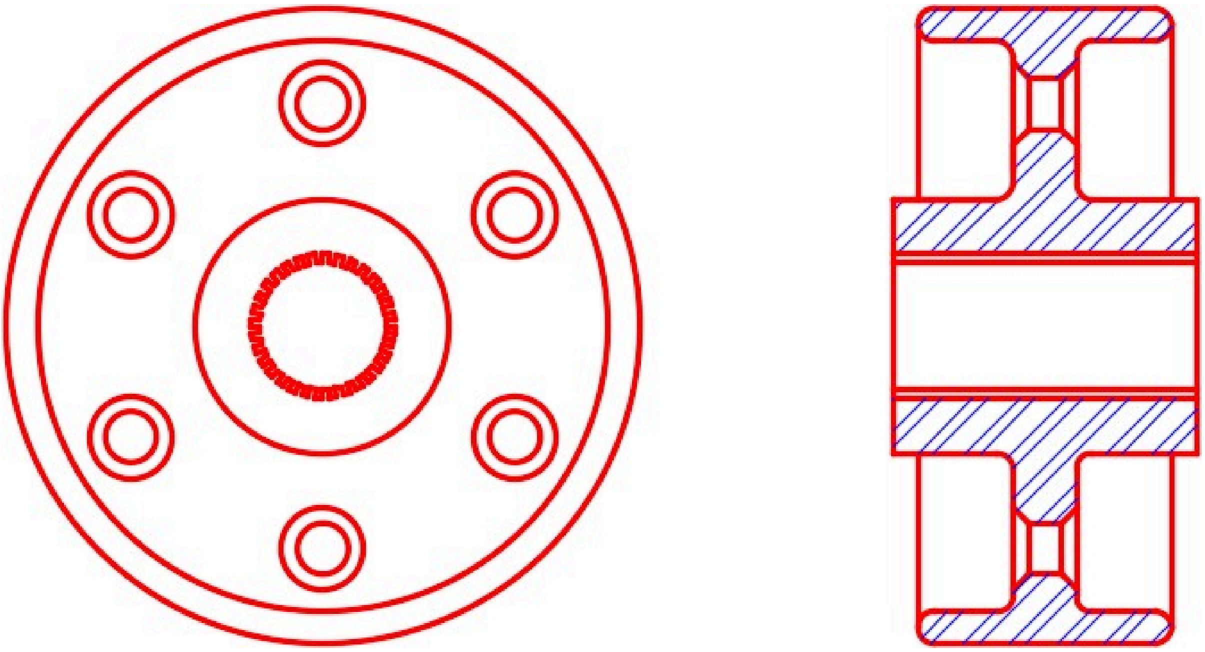


Figure Step 2

AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hint. If you get stuck and cannot complete it on your own, use the following hint to help you.

Hint 1

To draw the gear teeth, see the five steps in the figure. Step 1 – Draw one line of the teeth. Step 2 – Array the line 64 times. Step 3 and 4 – Add lines to create one tooth and the repeat pattern. Erase the unwanted lines. Step 5 – Array the tooth 32 times. (Figure Hint 1)

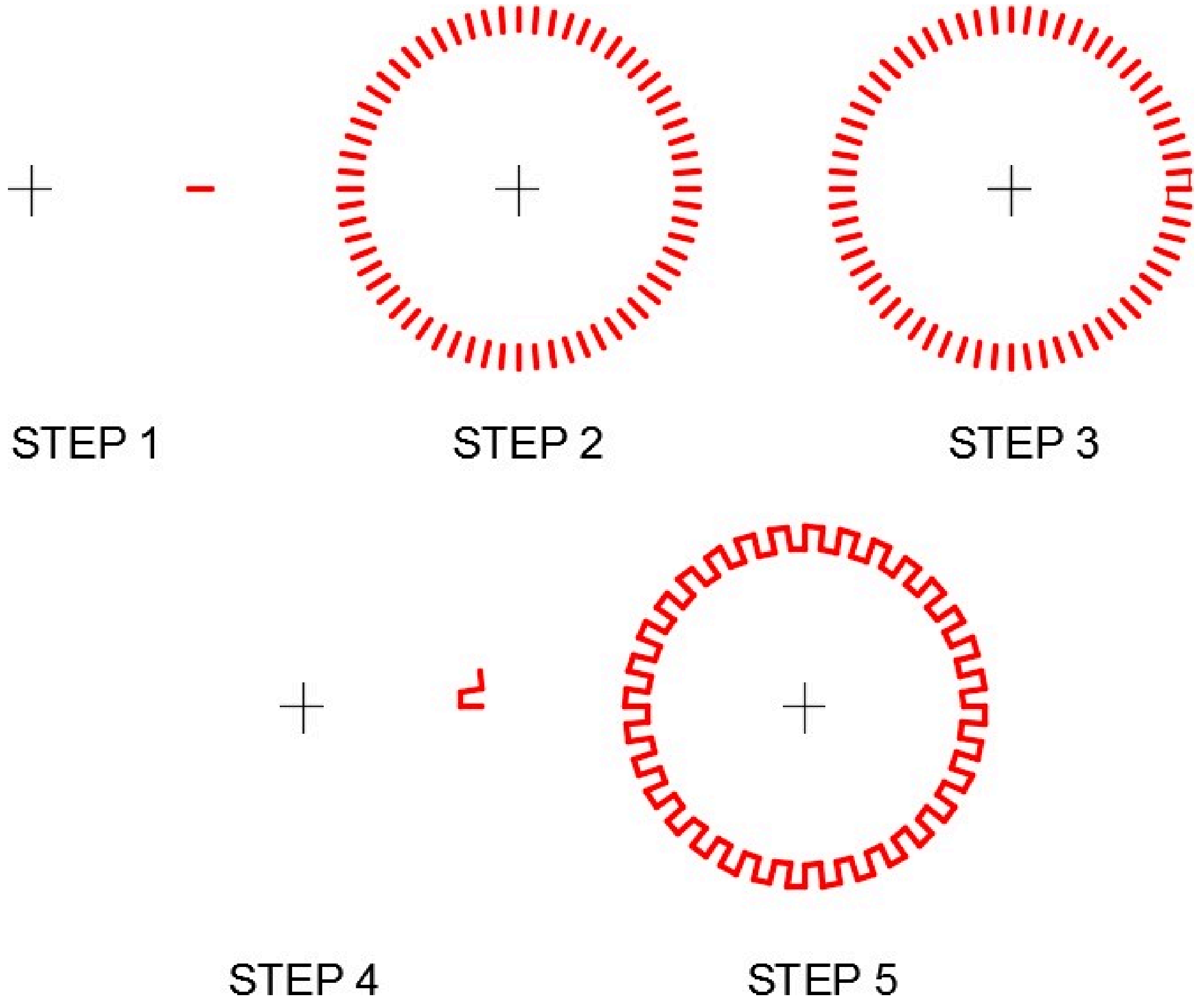
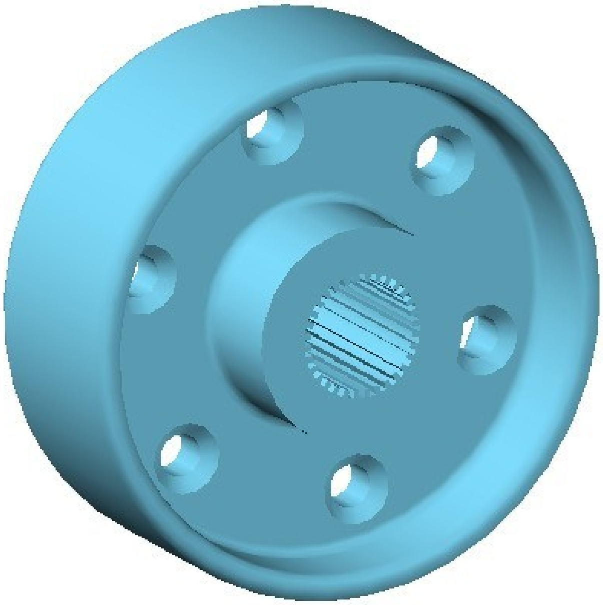


Figure Hint 1

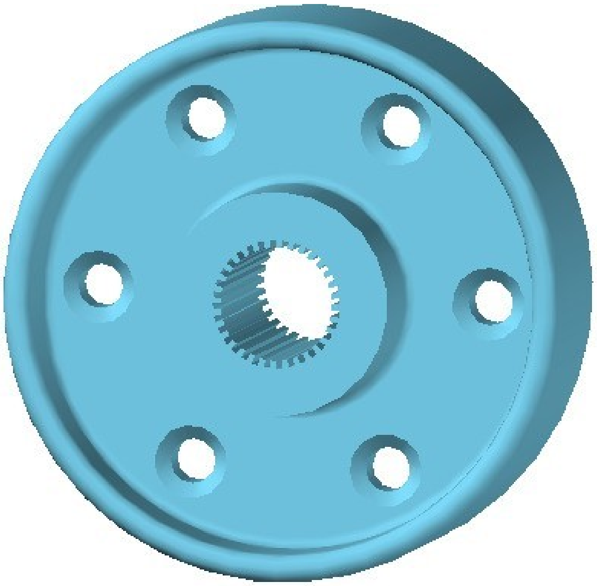
AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hints. If you get stuck and cannot complete it on your own, use the following hints to help you.

Hint 2

If you have trouble visualizing the object, study the solid models of the object shown below. The solid modeling views were done using AutoCAD. Solid Modeling is taught in the AutoCAD 3D book.



Solid Model of Object



Solid Model of Object

Lab Exercise 32-3

Time allowed: 80 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 32-3	2D Layout Metric	Millimeters

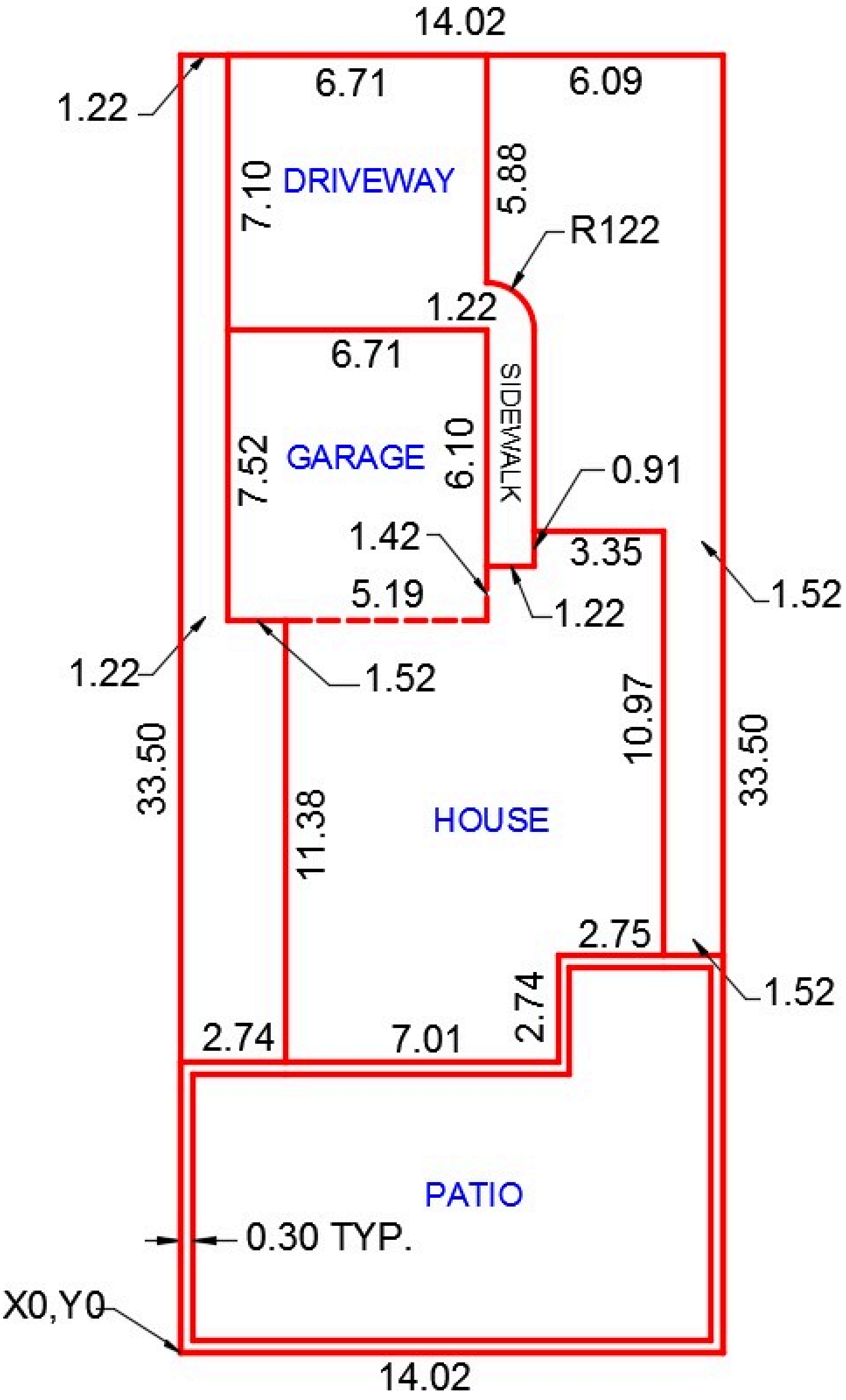

14.02
1.22
6.71
6.09
7.10
DRIVEWAY
5.88
R122
1.22
6.71
SIDEWALK
7.52
GARAGE
6.10
0.91
1.42
3.35
5.19
1.22
1.52
1.22
1.52
10.97
33.50
33.50
11.38
HOUSE
2.75
1.52
2.74
2.74
7.01
PATIO
0.30 TYP.
X0,Y0
14.02

Figure Step 1A

Step 1

Using the dimensioned drawing as a reference, on layer: Lot Lines, draw only the four outside lot lines or the boundary of the lot. The dimensions on the drawing are in meters but the input is in millimeters so you must be careful. 1 meter = 1000 millimeters (Figure Step 1A and 1B)

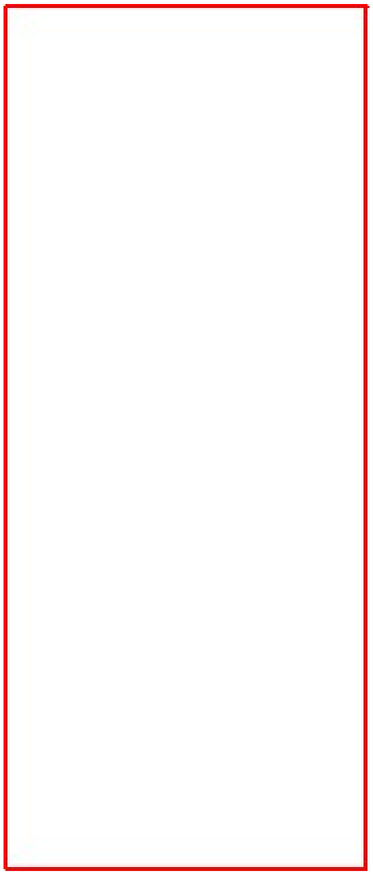


Figure Step 1B

AUTHOR'S COMMENTS: When the dimension of a line is 14.02, draw the line 14020 mm.

Step 2

Using the dimensioned drawing as a reference, on layer: Object, draw the lines and arcs of the driveway, garage, house and patio. (Figure Step 2)

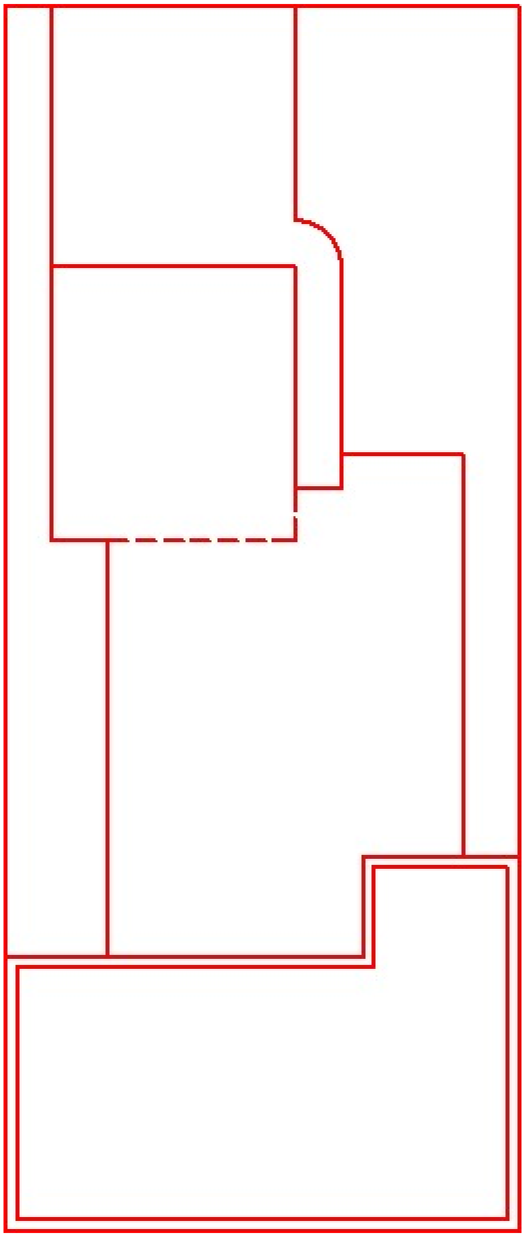


Figure Step 2

Step 3

On layer: Boundary, use the BOUNDARY command to create a closed pline around each of the seven areas shown in the figure. (Figure Step 3)

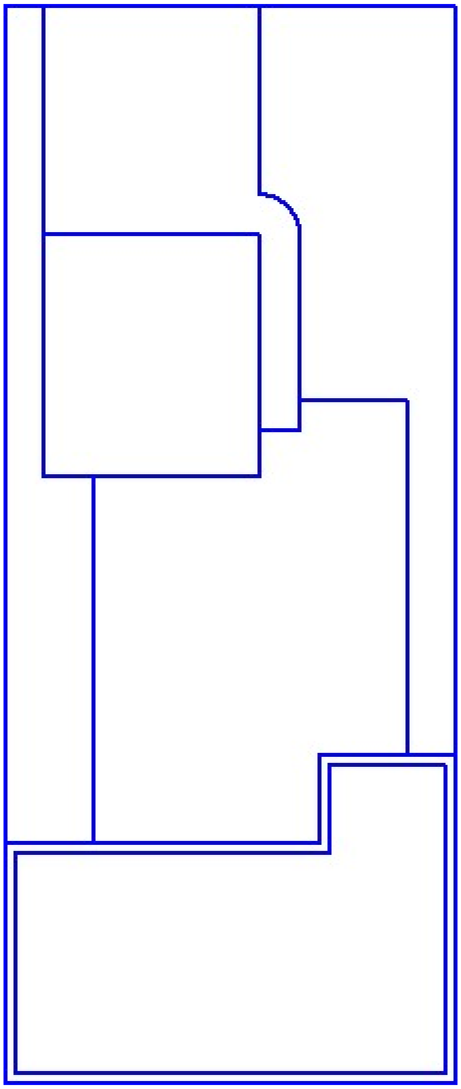


Figure Step 3

Step 4

Turn layer: Object off.

Step 5

On layer: Lot Hatch, insert the hatch patterns in the seven areas as follows:

A Grass

B AR-CONC

C Net

D Net – at an angle of 45 degrees

E Gravel

F AR-HBONE

G Solid

(Figure Step 5A and 5B)

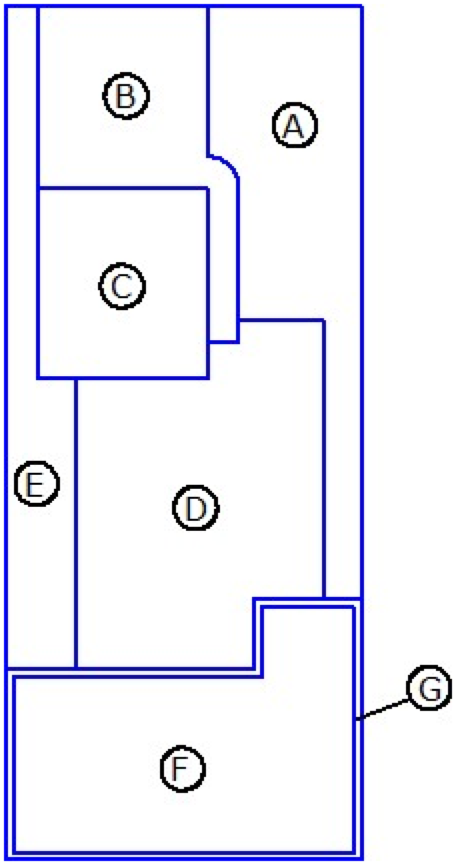


Figure Step 5A

AUTHOR'S COMMENTS: You will have to adjust the scale of each hatch pattern. Insert them and then edit them until they appear correct on your drawing. If HATCH will not insert them at the scale you selected, change the scale and try again.

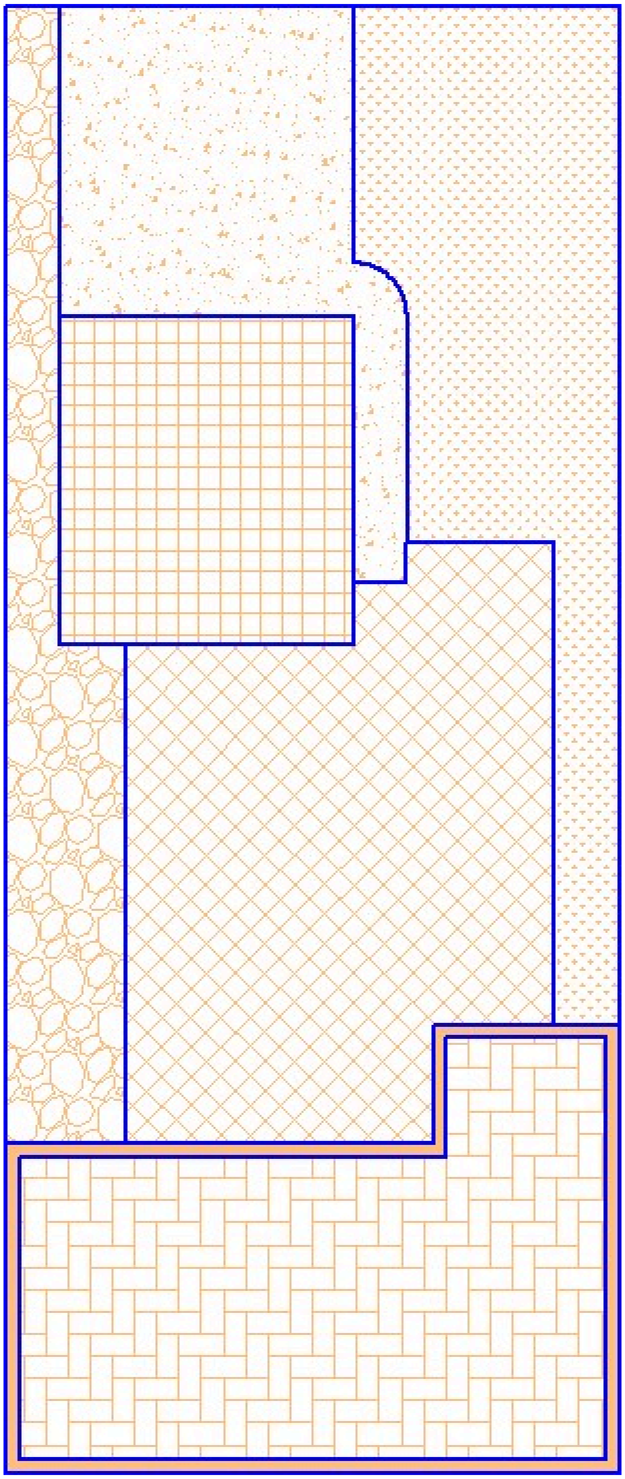


Figure Step 5B

Step 6

Enable layout: Module Layout A2. On layer: Viewport, create a viewport and set the scale to 1:100 (0.01). (Figure Step 6)

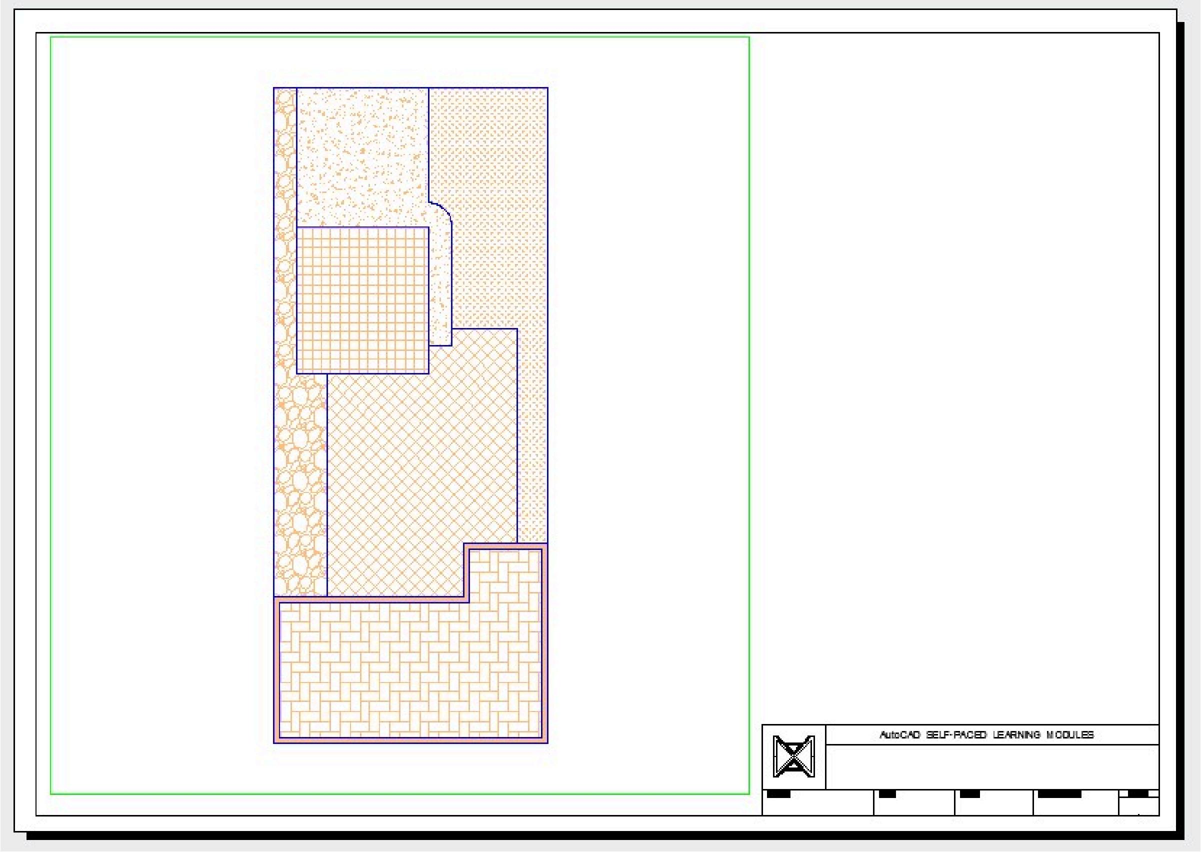
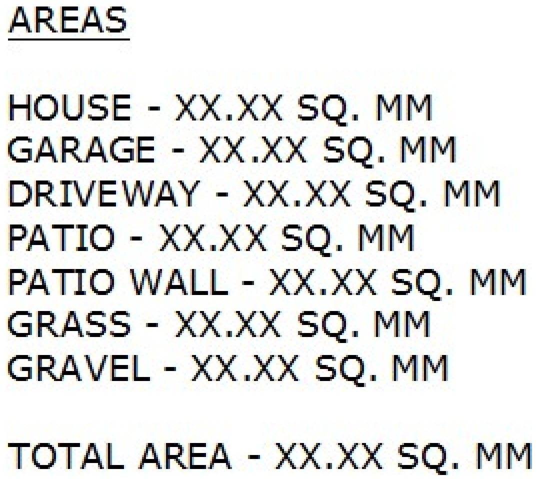

AutoCAD SELF-PACED LEARNING MODULES

Figure Step 6

Step 7

Find the areas and of the seven areas. Set the text style to:2D Modules. In Paper space, on layer: Text, insert the text as shown in the figure and include your answers. Set an appropriate text height. (Figure Step 7)



AREAS

HOUSE - XX.XX SQ. MM
GARAGE - XX.XX SQ. MM
DRIVEWAY - XX.XX SQ. MM
PATIO - XX.XX SQ. MM
PATIO WALL - XX.XX SQ. MM
GRASS - XX.XX SQ. MM
GRAVEL - XX.XX SQ. MM

TOTAL AREA - XX.XX SQ. MM

Figure Step 7

Step 8

On layer: Titleblock Text, fill in the titleblock using the standard shown in Module 20.

Step 9

Turn layer: Viewport off. (Figure Step 9)

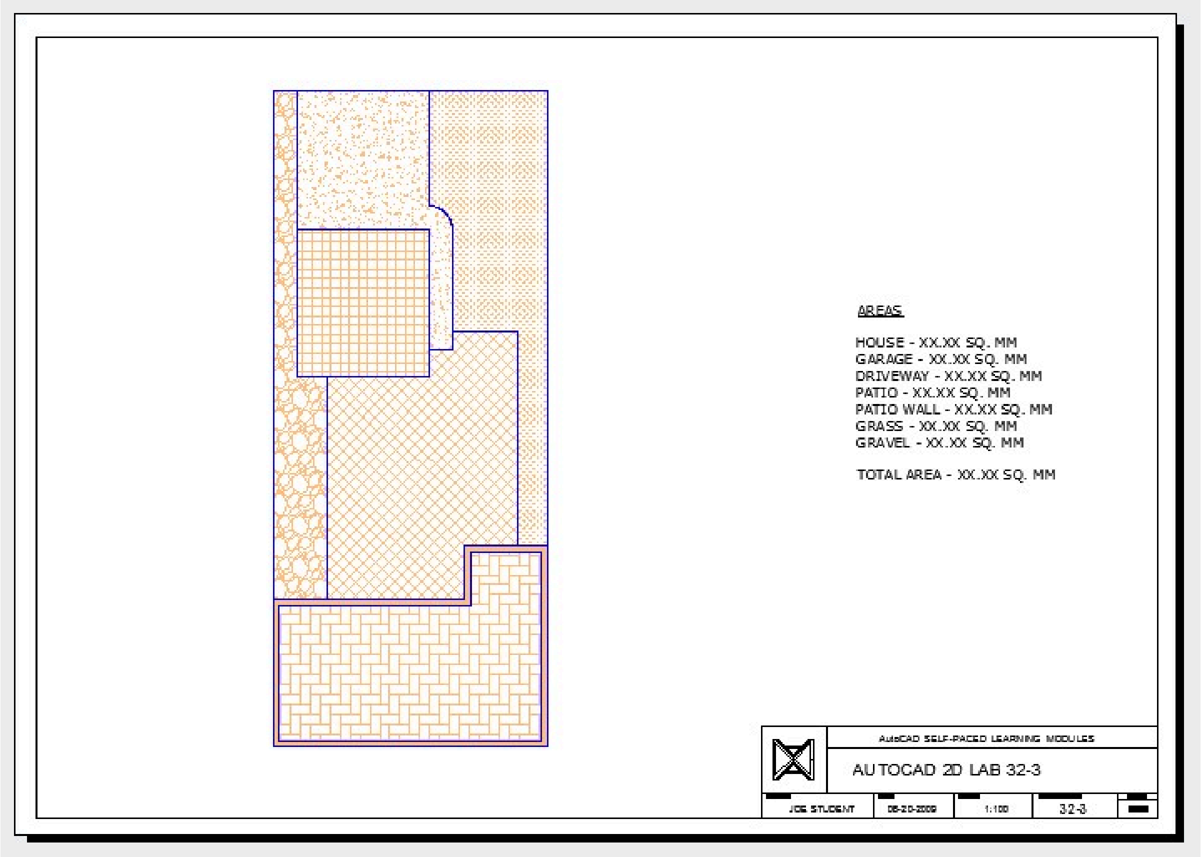

AREAS
HOUSE - XX.XX SQ. MM
GARAGE - XX.XX SQ. MM
DRIVEWAY - XX.XX SQ. MM
PATIO - XX.XX SQ. MM
PATIO WALL - XX.XX SQ. MM
GRASS - XX.XX SQ. MM
GRAVEL - XX.XX SQ. MM
TOTAL AREA - XX.XX SQ. MM
AutoCAD SELF-PACED LEARNING MODULES
AUTOCAD 2D LAB 32-3
JOE STUDENT
08-20-2009
1:100
32-3

Figure Step 9

Step 10

Save and close the drawing.

Module 33: Blocks

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe a block and explain how blocks are stored and inserted in a drawing file.
2. Describe how blocks work with layers.
3. Apply the BLOCK and INSERT commands.
4. Explain the importance of layer 0 when blocks are created.

Blocks

A *block* is a collection of drawing objects that acts as one object. After defining a block, it can be inserted into the current drawing as many times as required. It can also be inserted into other drawings or shared with other users. Regardless of the number of times a block has been inserted into the drawing, it will not significantly change the size of the drawing's database. The EXPLODE command can be used to explode a block back into its individual drawing objects.

Every drawing file has an invisible data area called the block definition table. The *block definition table* stores all block definitions in that drawing and contains all of the data associated with each block. It is these block definitions that the INSERT command refers to when it is used to insert a block into the drawing. The only way to remove a block from the definition table is to purge it using the PURGE command. All inserted blocks must be deleted before the block definition can be purged from the drawing. Purging is taught in Module 34.

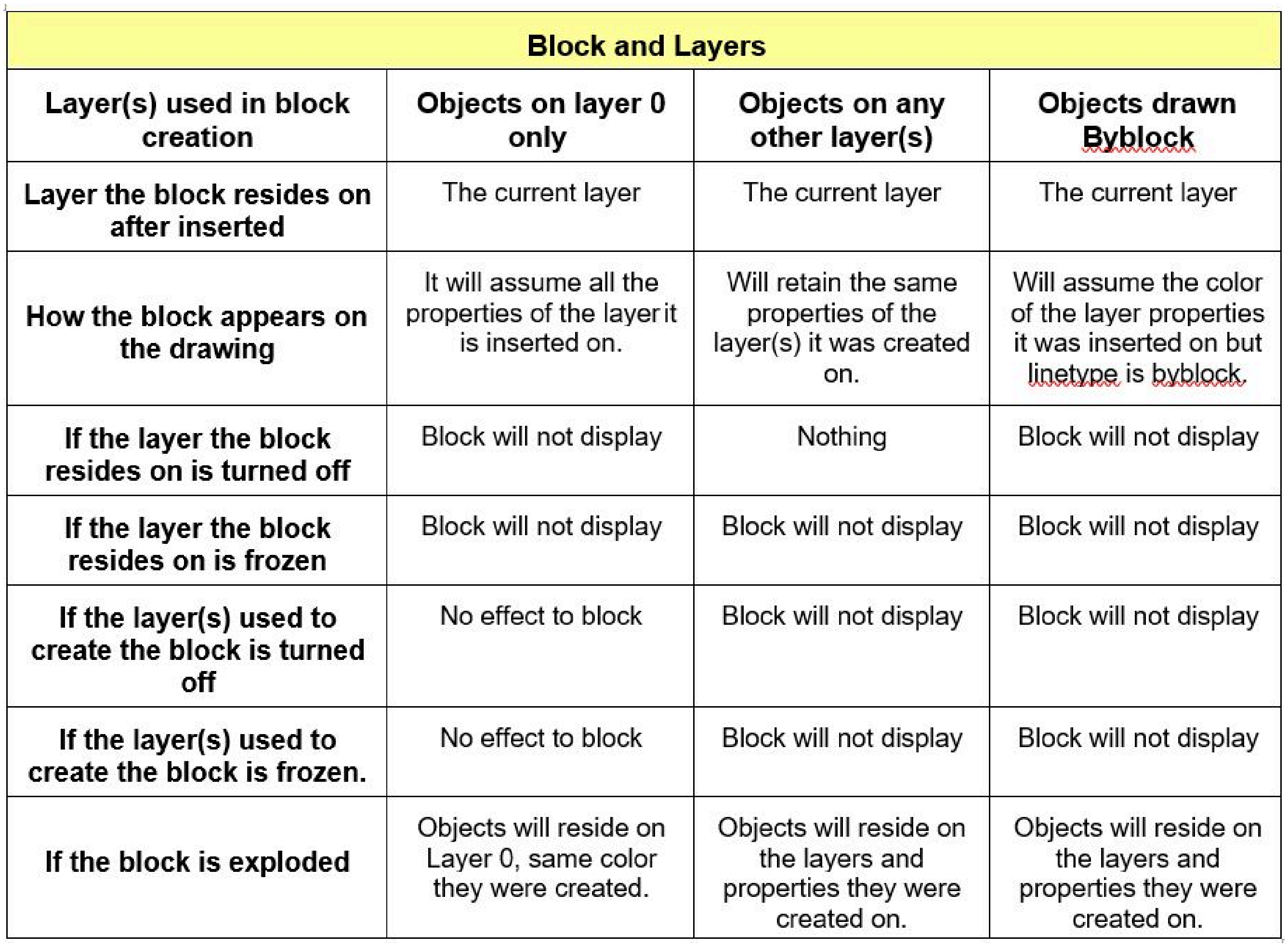
The BLOCK command is used to create a block in the definition table. The INSERT command is used to insert a block, from the definition table, into the drawing. A block can be scaled and/or rotated when it is inserted or it can be scaled and/or rotated after it has been inserted by changing the block's properties or by using the SCALE and ROTATE commands.

There are two methods available to edit an existing block. The first is to insert the block into the drawing, explode it, and make the necessary changes. After the changes are made, redefine the block and assign the original name. When a block is edited and redefined, it will overwrite the block definition in the block definition table and all insertions of that block will change to appear as the revised block. The second method is to use the BEDIT command. BEDIT is taught in the AutoCAD 2D Advanced book.

Blocks can also have intelligence, called *attributes*, attached to them. After blocks with assigned attributes are defined and inserted into a drawing, the data attached to them can be extracted.

Attributes and data extraction are taught in the AutoCAD 2D Advanced book.

It is essential to know how blocks and layers work in a drawing. When a block is inserted in a drawing, it will reside on the current layer but it is more important to know what layer the drawing objects, that were used to create the block, were drawn on. See Figure 33-1.



Block and Layers

Layer(s) used in block creation	**Objects on layer 0 only**	**Objects on any other layer(s)**	**Objects drawn Byblock**
Layer the block resides on after inserted	The current layer	The current layer	The current layer
How the block appears on the drawing	It will assume all the properties of the layer it is inserted on.	Will retain the same properties of the layer(s) it was created on.	Will assume the color of the layer properties it was inserted on but linetype is byblock.
If the layer the block resides on is turned off	Block will not display	Nothing	Block will not display
If the layer the block resides on is frozen	Block will not display	Block will not display	Block will not display
If the layer(s) used to create the block is turned off	No effect to block	Block will not display	Block will not display
If the layer(s) used to create the block is frozen.	No effect to block	Block will not display	Block will not display
If the block is exploded	Objects will reside on Layer 0, same color they were created.	Objects will reside on the layers and properties they were created on.	Objects will reside on the layers and properties they were created on.

Figure 33-1
Blocks and Layers

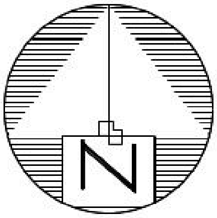
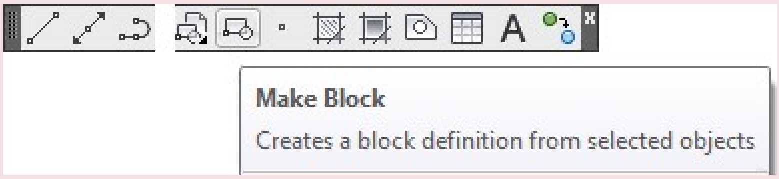
Object Snap Modes - Blocks				
Mode	**Abbreviation**	**Icon**	**Marker**	**The AutoCAD Object**
Insert	Ins	⧉	⧉	

Figure 33-2
Object Snap Mode – Blocks

AutoCAD Command: BLOCK

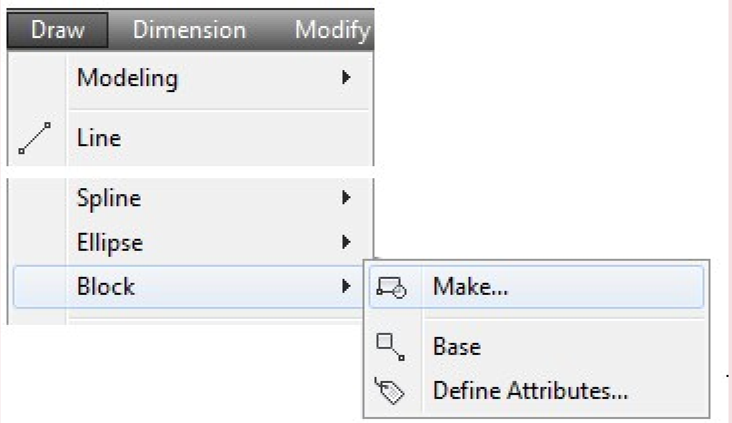
The BLOCK command is used to create a block in the block definition table in the current drawing.

Shortcut: **B**

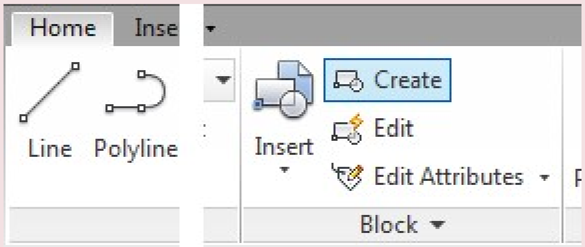


Make Block
Creates a block definition from selected objects

Block Toolbar


Draw
Dimension
Modify
Modeling
Line
Spline
Ellipse
Block
Make...
Base
Define Attributes...

Draw Pull-down

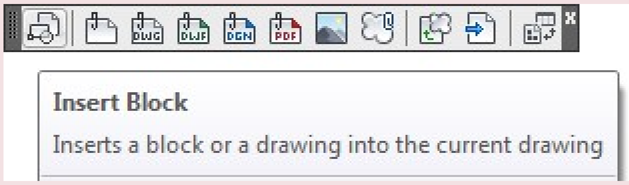

Home
Inse
Line
Polyline
Insert
Create
Edit
Edit Attributes
Block

Home Ribbon

AutoCAD Command: INSERT

The INSERT command is used to insert a block into the current drawing from the current drawings block definition table.

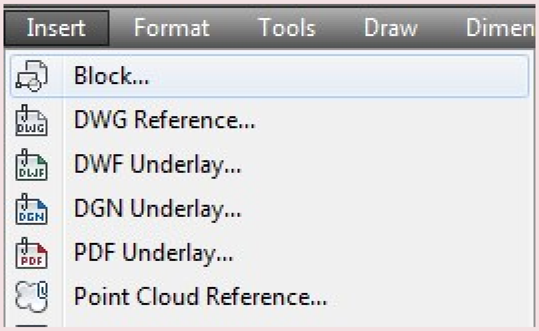
Shortcut: none



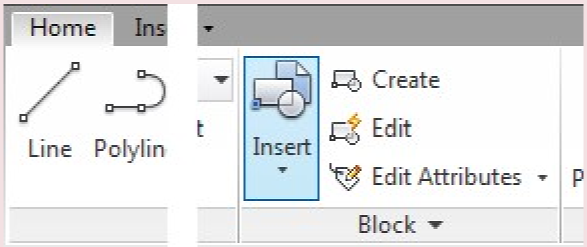
Insert Block

Inserts a block or a drawing into the current drawing

Insert Toolbar


Insert
Format
Tools
Draw
Dimen
Block...
DWG Reference...
DWF Underlay...
DGN Underlay...
PDF Underlay...
Point Cloud Reference...

Insert Pull-down


Home
Ins
Line
Polylin
Insert
Create
Edit
Edit Attributes
Block

Home Ribbon

AUTHOR’S COMMENTS: Layer 0 (zero), colored white/black, is a special layer and color for blocks.

When inserted, a block defined using drawing objects drawn on layer 0 (colored white/black), will take the properties of the layer it is inserted on. If the block is exploded, the individual drawing objects will reside on layer 0 (colored white/black). If a block is defined, using objects drawn on any other layer, and inserted it will display the color that it was constructed on and not the color assigned to the layer it resides on.

WORK ALONG: Creating and Inserting Blocks

Step 1

Start a new drawing using the template: 2D Layout English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 33-1.

Step 3

On layer: Construction, draw the three horizontal and 3 vertical lines shown in the dimensioned drawing. Start the first line anywhere in model space. (Figure Step 3)

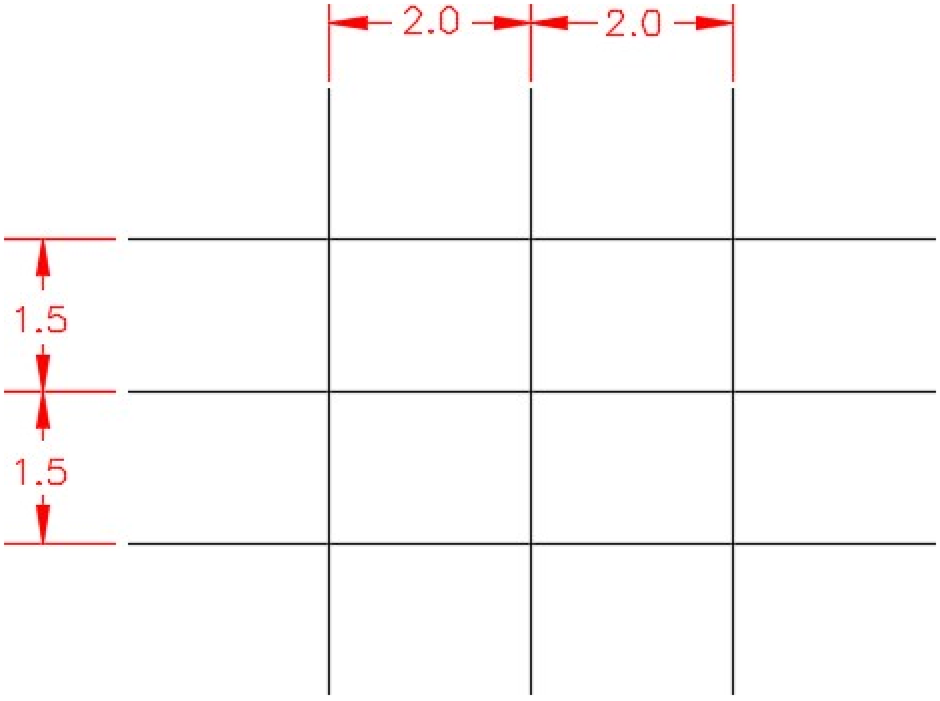


Figure Step 3

Step 4

On layer: 0, draw the three polygons as shown in the figure. Draw them anywhere on the drawing. (Figure Step 4A and 4B)

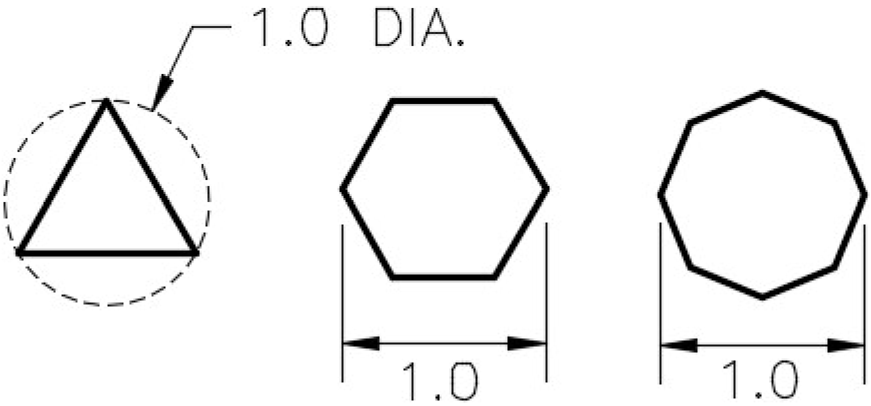


Figure Step 4A

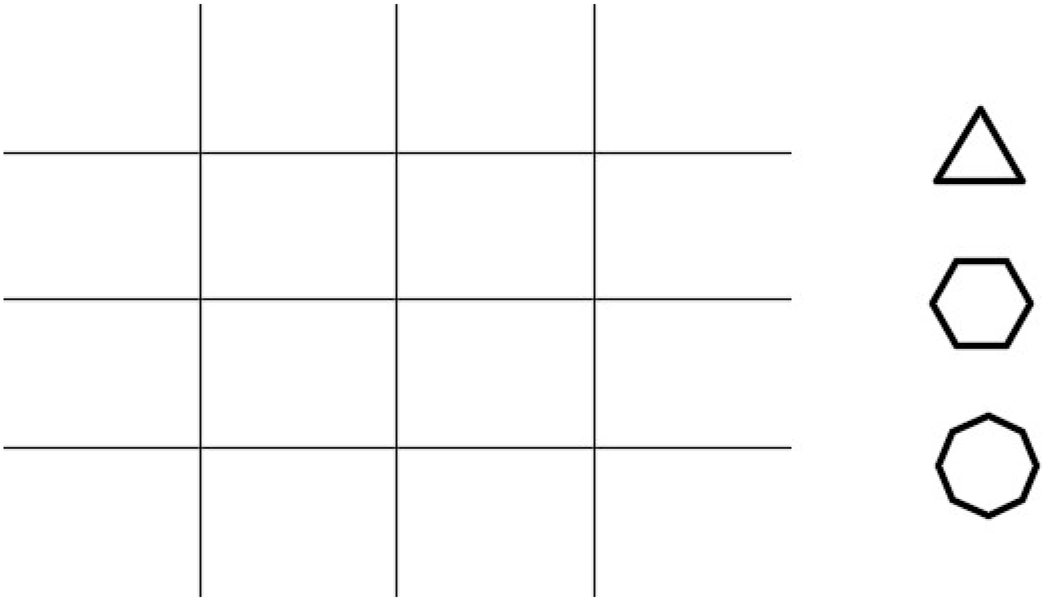
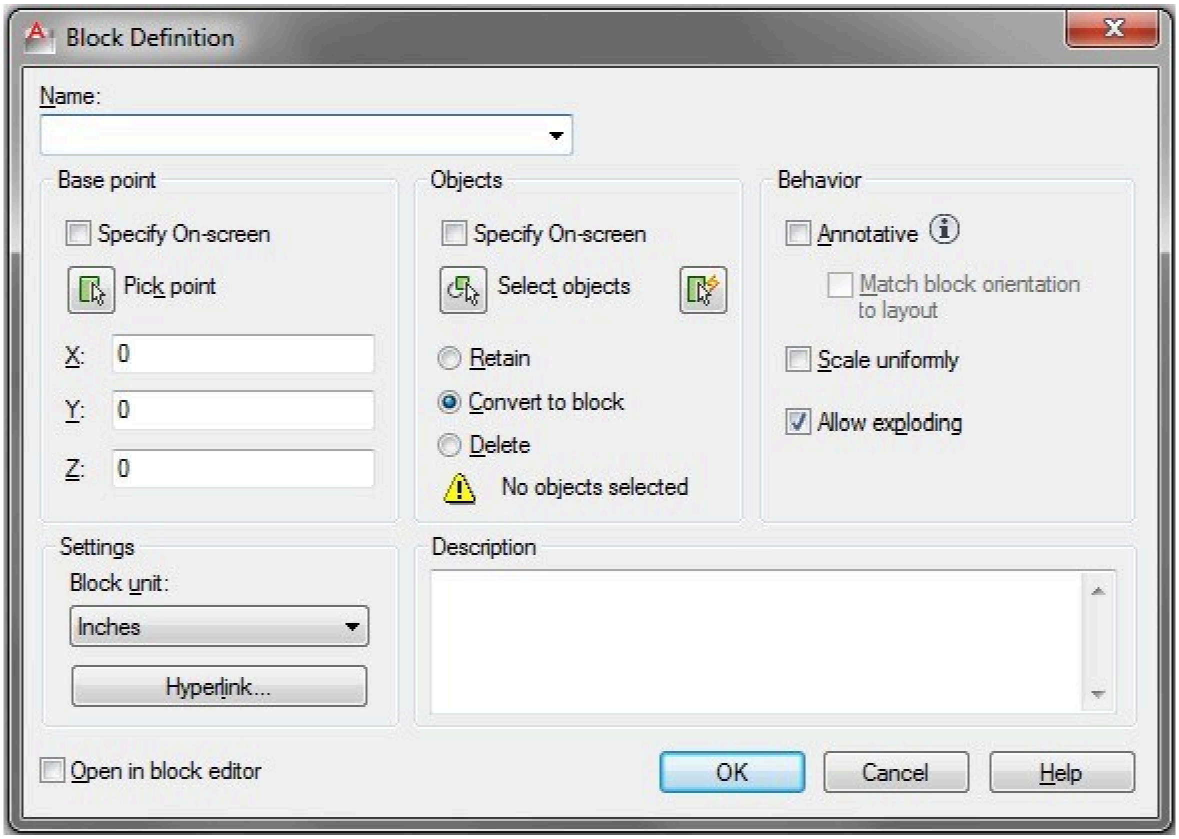


Figure Step 4B

Step 5

Enter the BLOCK command. It will open the Block Definition dialogue box. (Figure Step 5)


Block Definition
Base point
Pick point
Objects
Select objects
Behavior
Settings
Hyperlink...
Description
OK
Cancel
Help

Name:

☐ Specify On-screen

X: 0

Y: 0

Z: 0

☐ Specify On-screen

○ Retain

● Convert to block

○ Delete

No objects selected

☐ Annotative

☐ Match block orientation to layout

☐ Scale uniformly

☑ Allow exploding

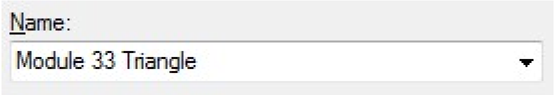
Block unit: Inches

☐ Open in block editor

Figure Step 5

Step 6

In the Name box, enter the name: Module 33 Triangle for the name of the block. (Figure Step 6)

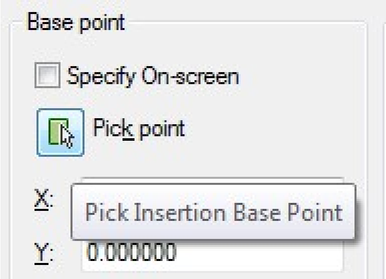


Name: Module 33 Triangle

Figure Step 6

Step 7

Click the Pick point icon to select the Base point for the block. When prompted for the base point, snap to the top corner of the triangle. (Figure Step 7A and 7B)


Base point
Pick point
Pick Insertion Base Point

☐ Specify On-screen

X:

Y: 0.000000

Figure Step 7A

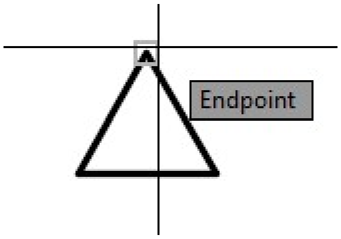
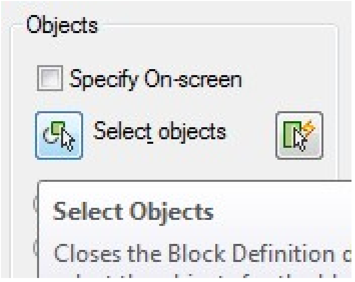


Figure Step 7B

AUTHOR'S COMMENTS: The base point of a block is important since it is the same point as the insert point you will be using when you insert the block.

Step 8

In the Objects area, click the Select objects icon. When prompted to select objects, use a window to select the triangle. (Figure Step 8A and 8B)


Objects
Specify On-screen
Select objects

Select Objects

Closes the Block Definition c

Figure Step 8A

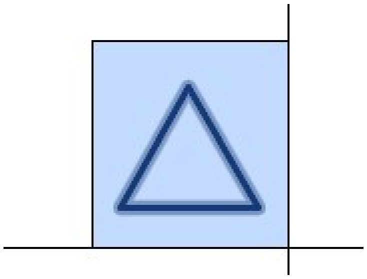
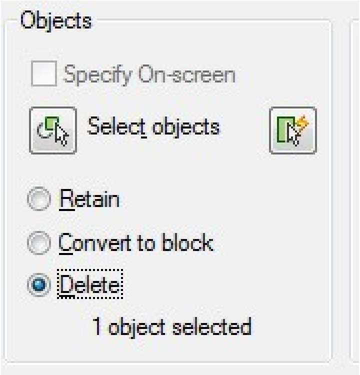


Figure Step 8B

Step 9

In the Objects area, enable the Delete button. (Figure Step 9)


Objects
Specify On-screen
Select objects

☐ Specify On-screen

○ Retain

○ Convert to block

● Delete

1 object selected

Figure Step 9

AUTHOR'S COMMENTS:

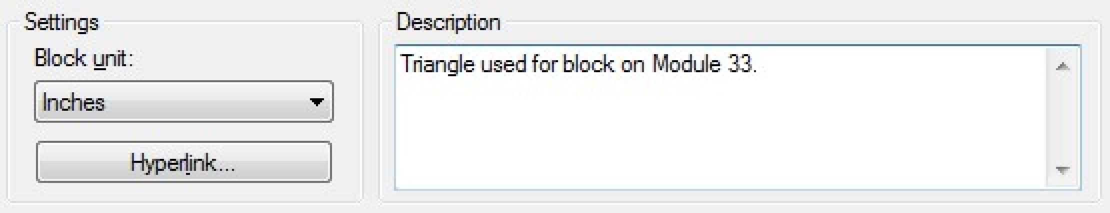
Delete – the objects used to create the block will be deleted.

Retain – the objects used to create the block will remain in as individual objects.

Convert to block – the objects that were used to create the block are converted to a block.

Step 10

In the Settings area, set the Block unit to Inches and in the Description area, enter the description as shown in the figure. (Figure Step 10)



Block unit: Inches

Description: Triangle used for block on Module 33.

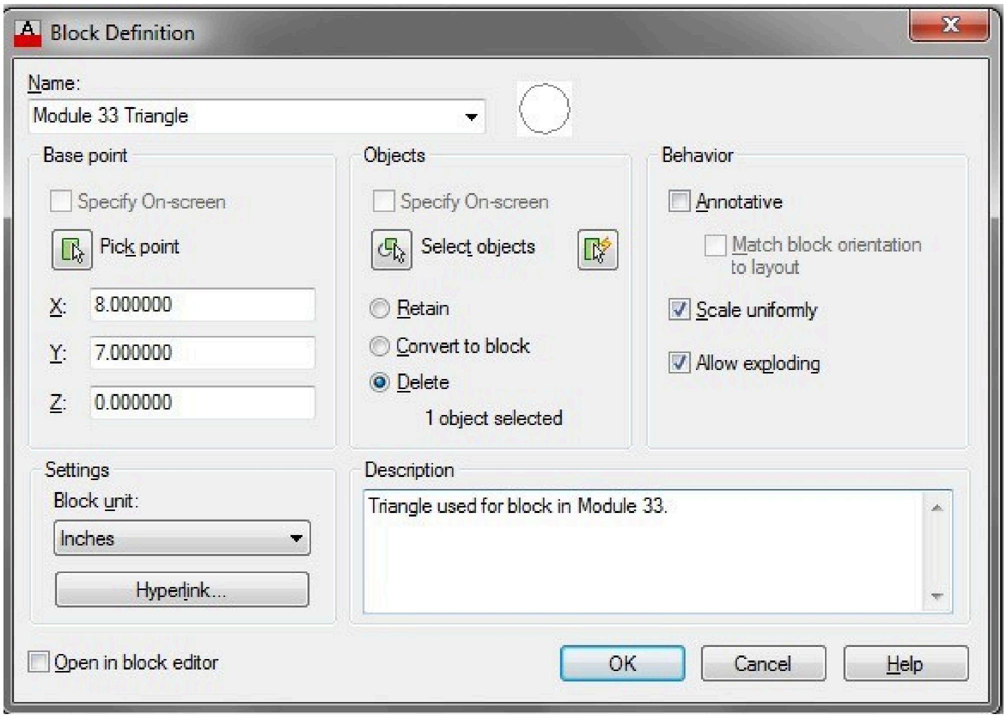
Figure Step 10

AUTHOR’S COMMENTS: The Block unit refers to the units that the block is drawn in. If they match the Insertion scale, when the block is inserted, the block will be inserted at full size or a scale factor of 1.

MUST KNOW: A block is a collection of drawing objects that acts as one object. Every drawing file has an invisible data area called the block definition table. The block definition table stores all the block definitions in that drawing. When a block is inserted into the drawing, it will reside on the current layer. A block can be exploded with theEXPLODE command. When a block is exploded, it loses all of its block properties and it is converted back to the original drawing objects and properties that it had when it was created.

Step 11

Except for the Base point, the Block Definition dialogue box should now match the figure. (Figure Step 11)


Block Definition

Name: Module 33 Triangle

Base point
☐ Specify On-screen
Pick point
X: 8.000000
Y: 7.000000
Z: 0.000000

Objects
☐ Specify On-screen
Select objects
○ Retain
○ Convert to block
● Delete
1 object selected

Behavior
☐ Annotative
☐ Match block orientation to layout
☑ Scale uniformly
☑ Allow exploding

Settings
Block unit: Inches
Hyperlink...

Description
Triangle used for block in Module 33.

☐ Open in block editor

Figure Step 11

Step 12

Click OK and your drawing should now appear similar to the figure. (Figure Step 12)

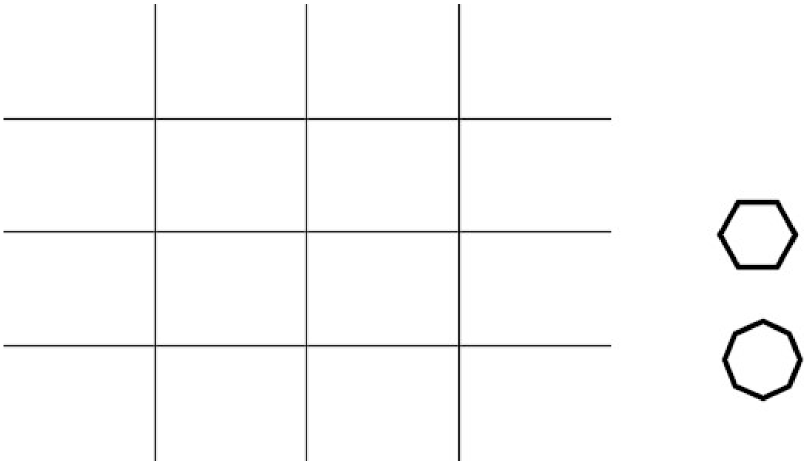


Figure Step 12

Step 13

Using what you just learned, create a block from the hexagon and the octagon. For both, use the center of the object as the base point. To do that, you must first draw a construction line so that you can snap to its midpoint when specifying the base point. Ensure that you do not select the construction line as one of the block objects. Name the blocks: Module 33 Hexagon and Module 33 Octagon. (Figure Step 13A and 13B)

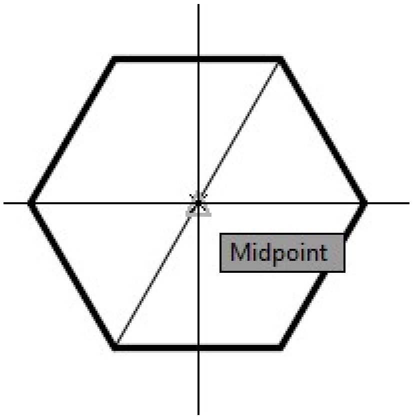


Figure Step 13A

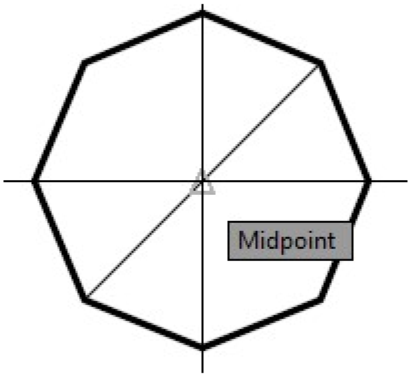


Figure Step 13B

Step 14

Your drawing should now appear as shown in the figure. (Figure Step 14)

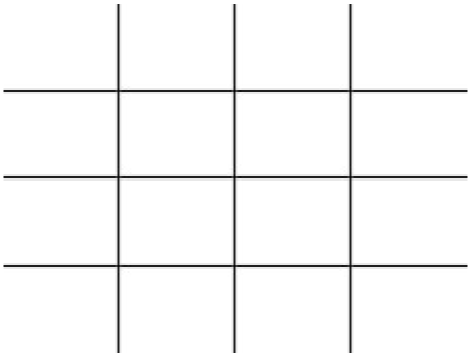


Figure Step 14

Step 15

Enter the UNITS command. In the Drawing Units dialogue box, set the Insertion scale to Inches. (Figure Step 15)

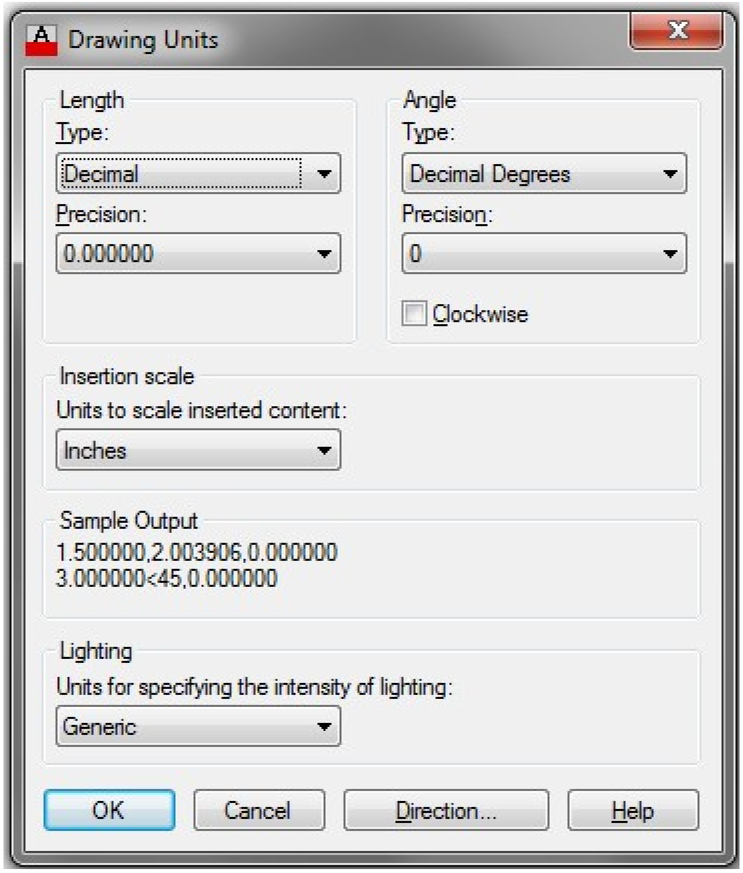
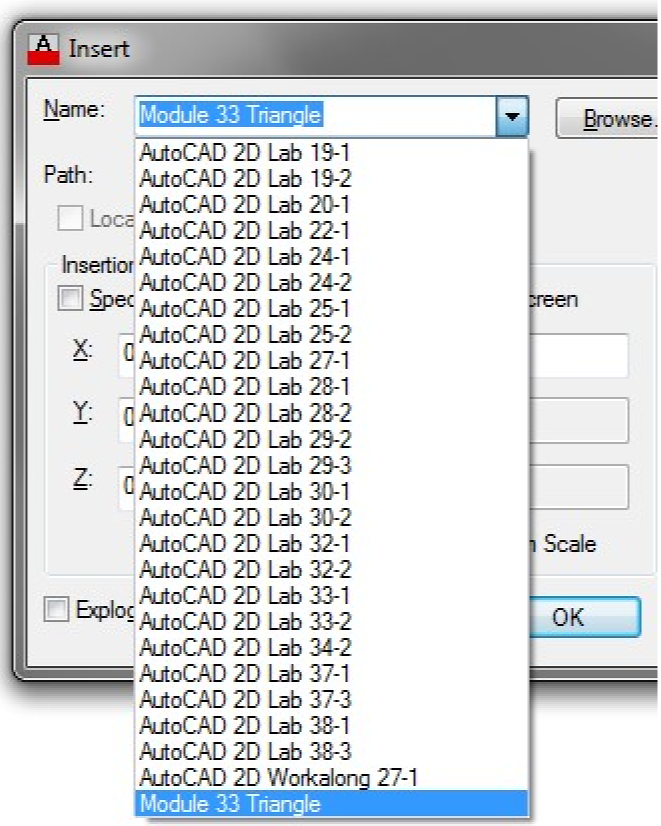

Drawing Units
Length
Type:
Decimal
Precision:
0.000000
Angle
Type:
Decimal Degrees
Precision:
0
☐ Clockwise
Insertion scale
Units to scale inserted content:
Inches
Sample Output
1.500000,2.003906,0.000000
3.000000<45,0.000000
Lighting
Units for specifying the intensity of lighting:
Generic
OK
Cancel
Direction...
Help

Figure Step 15

AUTHOR'S COMMENTS: The Insertion scale units are the units that the blocks are inserted in the current drawing. If they match the Block unit of the block being inserted, the block will be inserted at full size or a scale factor of 1. If not, a scale factor will be applied when the block is inserted.

Step 16

Set layer: 0 as the current layer. Enter the INSERT command to open the Insert dialogue box. In the Name: box, pull down the list and select the block: Module 33 Triangle. (Figure Step 16)



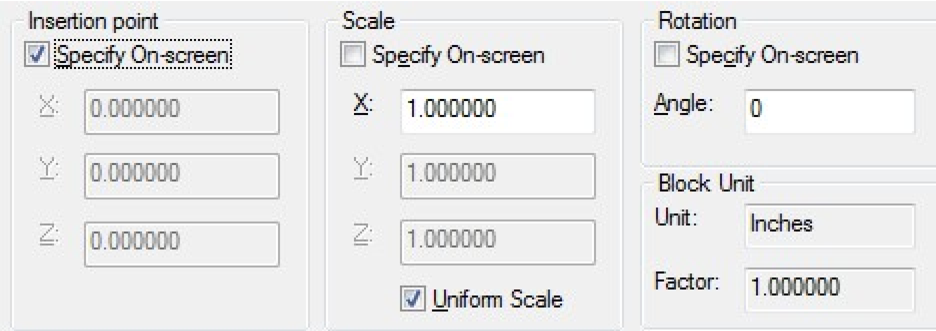
Name: Module 33 Triangle

Insert
Browse.
Path:
Loca
Insertior
Spec
X: 0
Y: 0
Z: 0
reen
Scale
Explo
OK
AutoCAD 2D Lab 19-1
AutoCAD 2D Lab 19-2
AutoCAD 2D Lab 20-1
AutoCAD 2D Lab 22-1
AutoCAD 2D Lab 24-1
AutoCAD 2D Lab 24-2
AutoCAD 2D Lab 25-1
AutoCAD 2D Lab 25-2
AutoCAD 2D Lab 27-1
AutoCAD 2D Lab 28-1
AutoCAD 2D Lab 28-2
AutoCAD 2D Lab 29-2
AutoCAD 2D Lab 29-3
AutoCAD 2D Lab 30-1
AutoCAD 2D Lab 30-2
AutoCAD 2D Lab 32-1
AutoCAD 2D Lab 32-2
AutoCAD 2D Lab 33-1
AutoCAD 2D Lab 33-2
AutoCAD 2D Lab 34-2
AutoCAD 2D Lab 37-1
AutoCAD 2D Lab 37-3
AutoCAD 2D Lab 38-1
AutoCAD 2D Lab 38-3
AutoCAD 2D Workalong 27-1
Module 33 Triangle

Figure Step 16

Step 17

Set the Insert point, Scale and Rotation boxes as shown in the figure. (Figure Step 17)



Insertion point
☑ Specify On-screen
X: 0.000000
Y: 0.000000
Z: 0.000000

Scale
☐ Specify On-screen
X: 1.000000
Y: 1.000000
Z: 1.000000
☑ Uniform Scale

Rotation
☐ Specify On-screen
Angle: 0

Block Unit
Unit: Inches
Factor: 1.000000

Figure Step 17

Step 18

Click OK and when prompted for the insertion point, snap to the intersection of the upper left construction lines. (Figure Step 18)

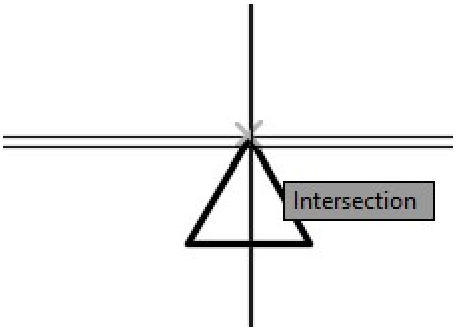
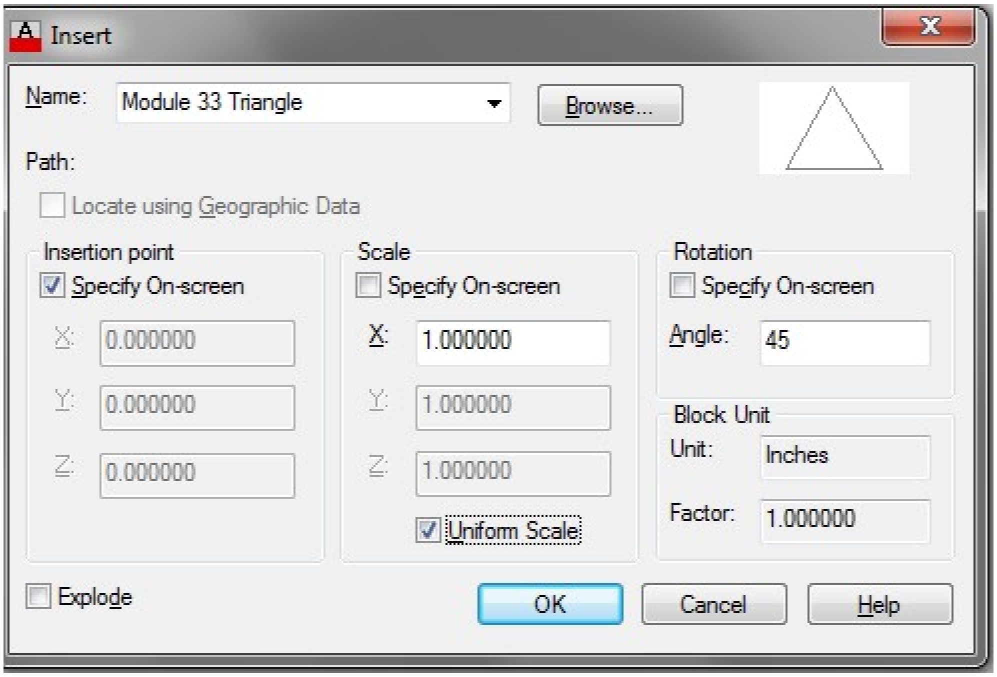


Figure Step 18

AUTHOR'S COMMENTS: The base point you selected when creating the block will be located at the insertion point.

Step 19

Enter the INSERT command and select the block: Module 33 Triangle. Set the Angle to 45. Insert the block locating the insertion point as shown in the figure. (Figure Step 19A and 19B)



Name: Module 33 Triangle

Path:

☐ Locate using Geographic Data

Insertion point
☑ Specify On-screen
X: 0.000000
Y: 0.000000
Z: 0.000000

Scale
☐ Specify On-screen
X: 1.000000
Y: 1.000000
Z: 1.000000
☑ Uniform Scale

Rotation
☐ Specify On-screen
Angle: 45

Block Unit
Unit: Inches
Factor: 1.000000

☐ Explode

Figure Step 19A

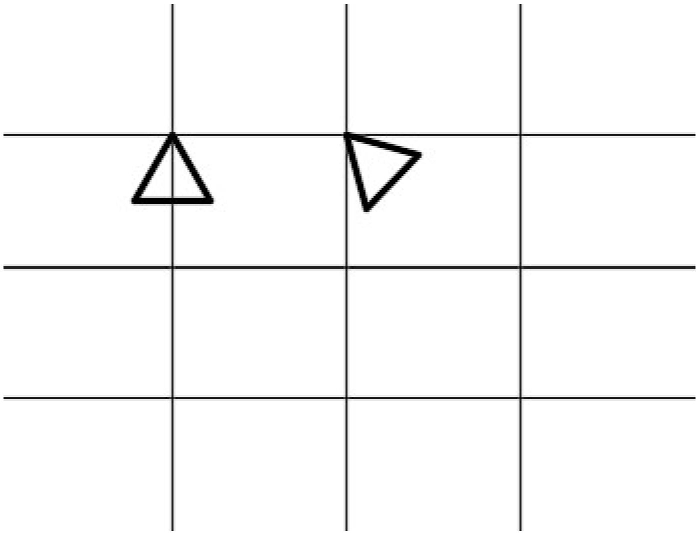
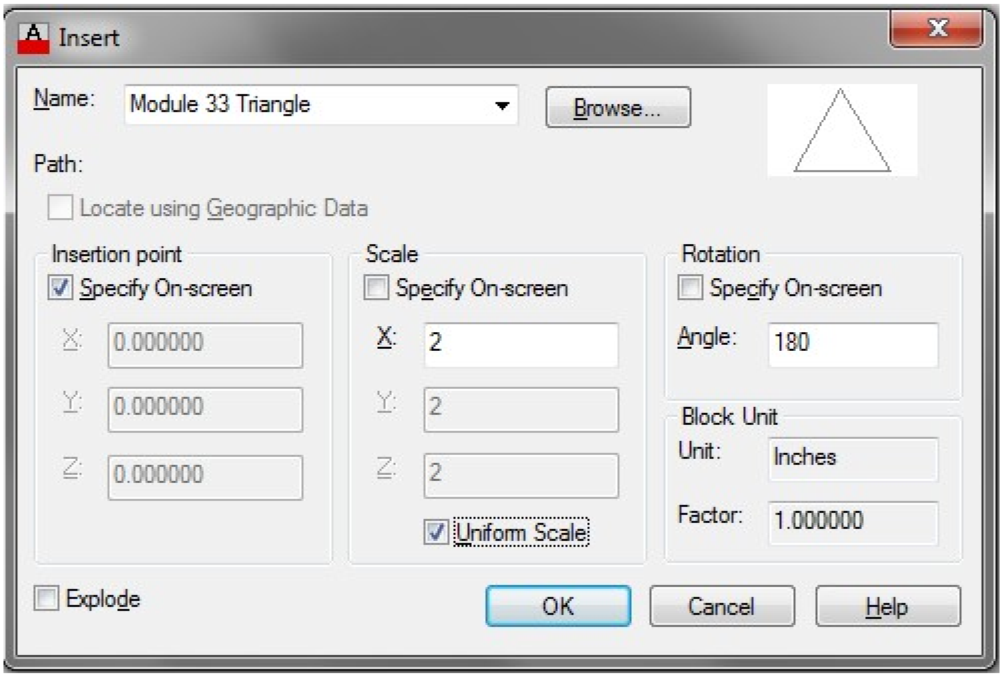


Figure Step 19B

MUST KNOW: Layer 0 (zero), assigned the color white/black, is a special layer and color for blocks. An inserted block that was created using objects drawn on layer 0 will reside on and take the properties of the layer that it is inserted on. If the block is exploded, the individual objects belonging to that block, will reside on layer 0 (colored white/black). When a block is created using objects drawn on any other layer, it will display the properties of that layer even though it is inserted on the layer with a different color assigned to it. If it is exploded, the individual objects will reside on the layer that they were originally drawn on.

Step 20

Enter the INSERT command and select the block: Module 33 Triangle. Set the Angle to 180, the X Scale to 2 and enable Uniform Scale. Insert the block as shown in the figure. (Figure Step 20A and 20B)



Name: Module 33 Triangle

Path:

☐ Locate using Geographic Data

Insertion point
☑ Specify On-screen
X: 0.000000
Y: 0.000000
Z: 0.000000

Scale
☐ Specify On-screen
X: 2
Y: 2
Z: 2
☑ Uniform Scale

Rotation
☐ Specify On-screen
Angle: 180

Block Unit
Unit: Inches
Factor: 1.000000

☐ Explode

Figure Step 20A

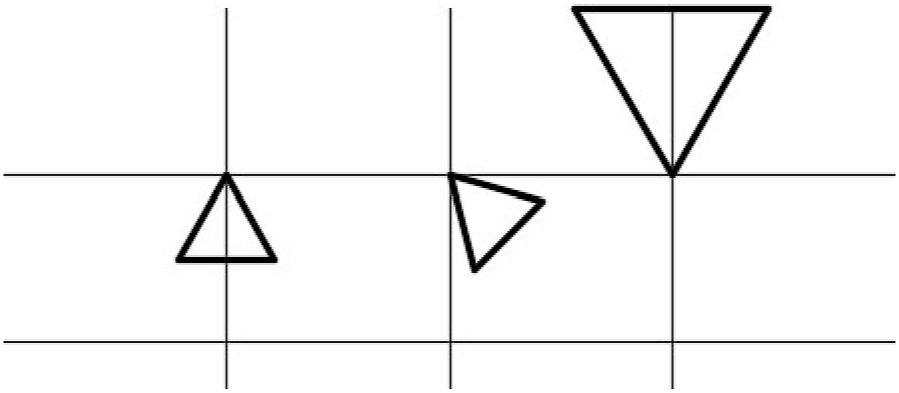


Figure Step 20B

Step 21

Set layer: Blocks as the current layer. Using what you just learned, insert three: Module 33 Hexagon blocks locating the insert point at the intersections of the construction lines as shown in the figure. The left and center blocks are inserted at scale factor of 1 and the one on the right at a scale factor of 1.5. Set the appropriate angles to match the figure. (Figure Step 21)

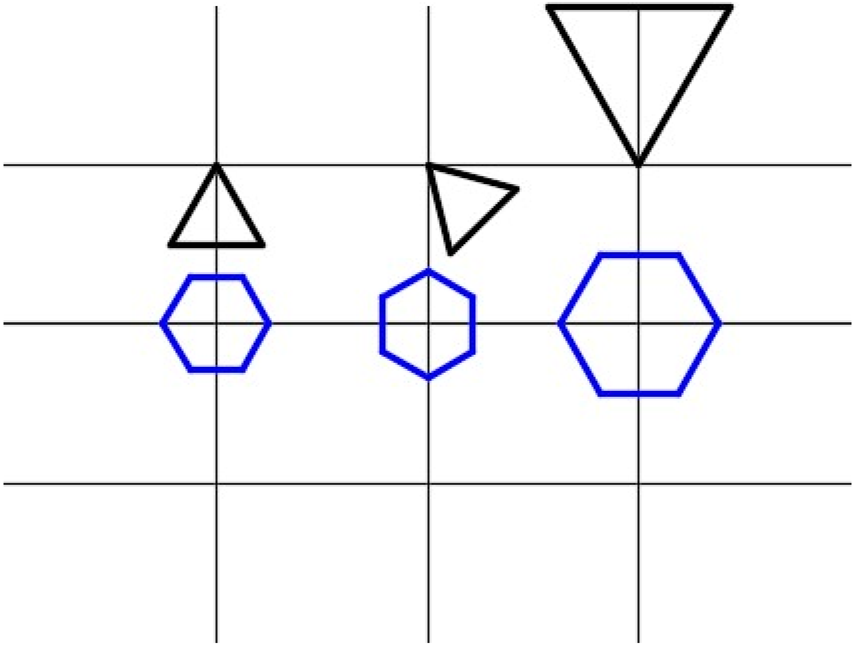


Figure Step 21

AUTHOR’S COMMENTS: Note that when you insert a block on layer Blocks, it takes the color of that layer.

Step 22

Set layer: Object as the current layer. Using what you just learned, insert three: Module 33 Octagon blocks placing the insert point as shown in the figure. The left and center blocks are inserted at a scale factor of 1 and the one on the right at a scale factor of 1.25. Set the appropriate angle to match the figure. (Figure Step 22)

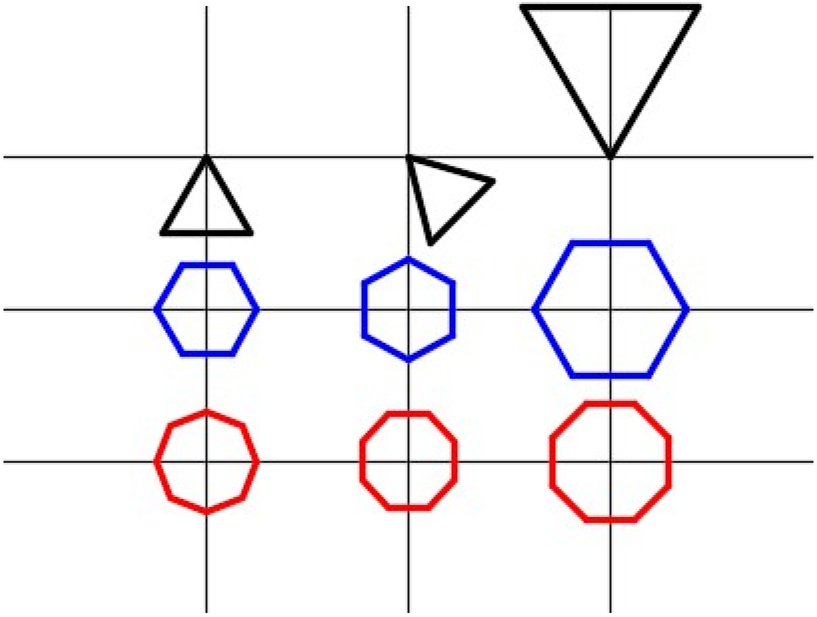


Figure Step 22

Step 23

Explode the left block: Module 33 Octagon as shown in the figure. (Figure Step 23)

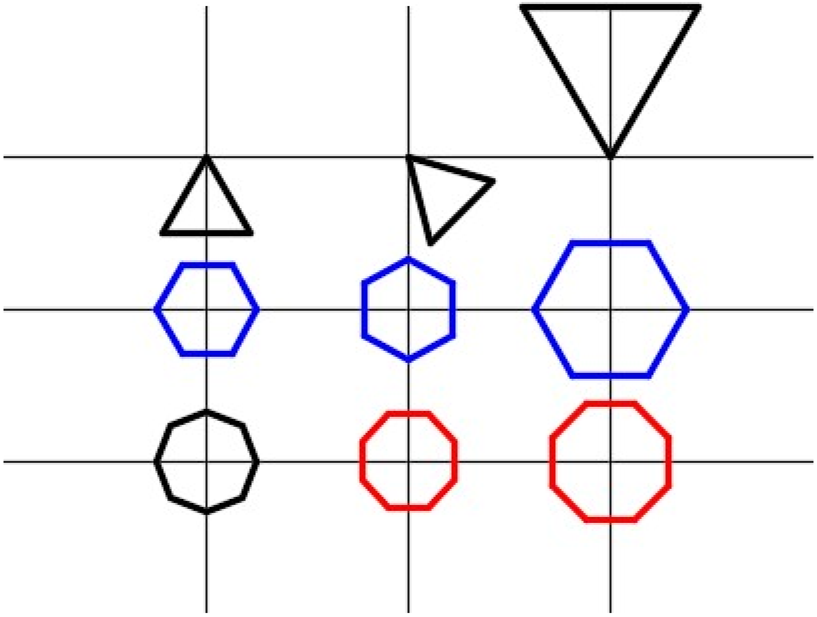


Figure Step 23

AUTHOR'S COMMENTS: Note that when the block is exploded, the drawing objects reside on layer 0 and display the color black/white.

Step 24

Turn layer: Construction off. (Figure Step 24)

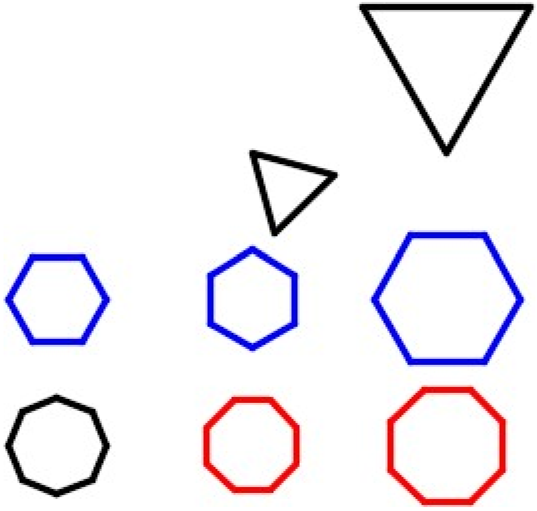


Figure Step 24

Step 25

Insert a block: Module 33 Triangle anywhere in model space. Explode the block and change the objects to layer: Object. On layer: Object, draw lines from each corner to the midpoint of the opposite line. (Figure Step 25).

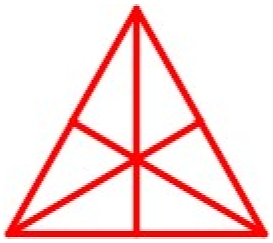
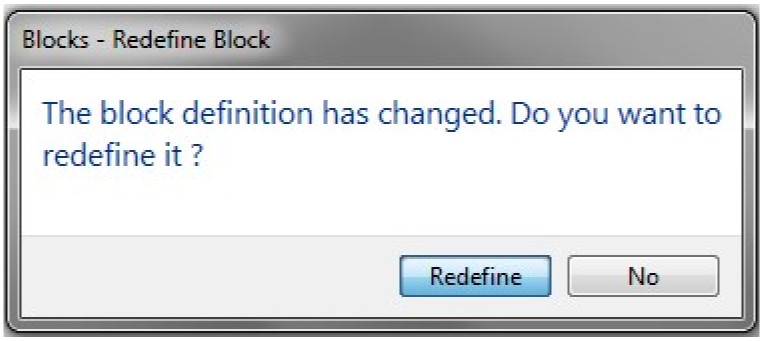


Figure Step 25

Step 26

Using the BLOCK command, create a block of the triangle that you created in Step 25. Assign it the name: Module 33 Triangle. Keep the same base point (top corner of triangle).

When you execute the command, the alert box will open warning you that a block with the same name is already defined in the drawing. Click Yes to accept. Take note how all of the blocks, named: Module 33 Triangle, will now display the new block definition. (Figure Step 26A and 26B)



Blocks - Redefine Block

The block definition has changed. Do you want to redefine it ?

Figure Step 26A

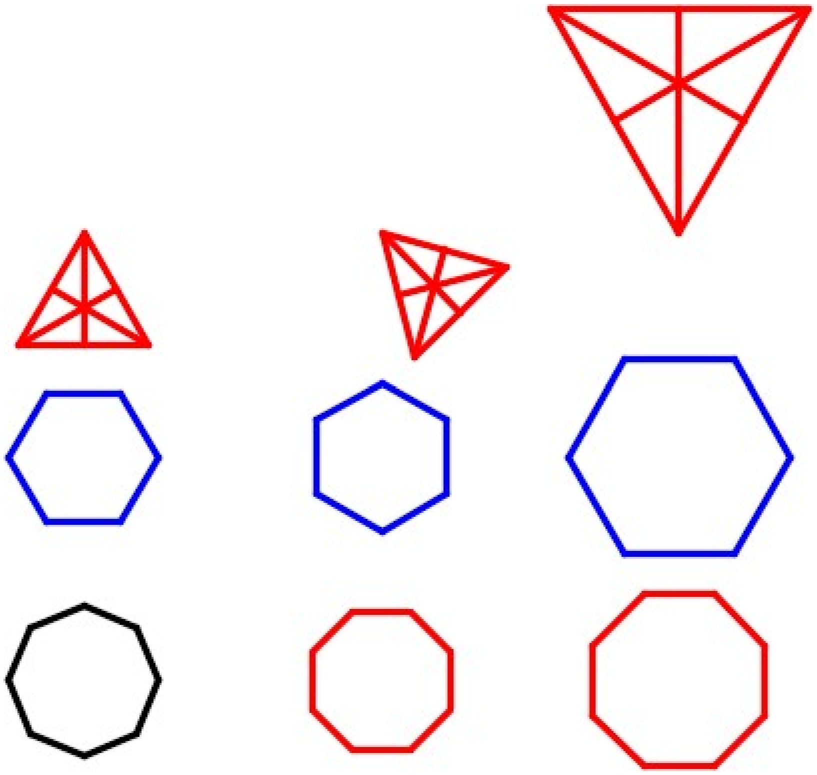
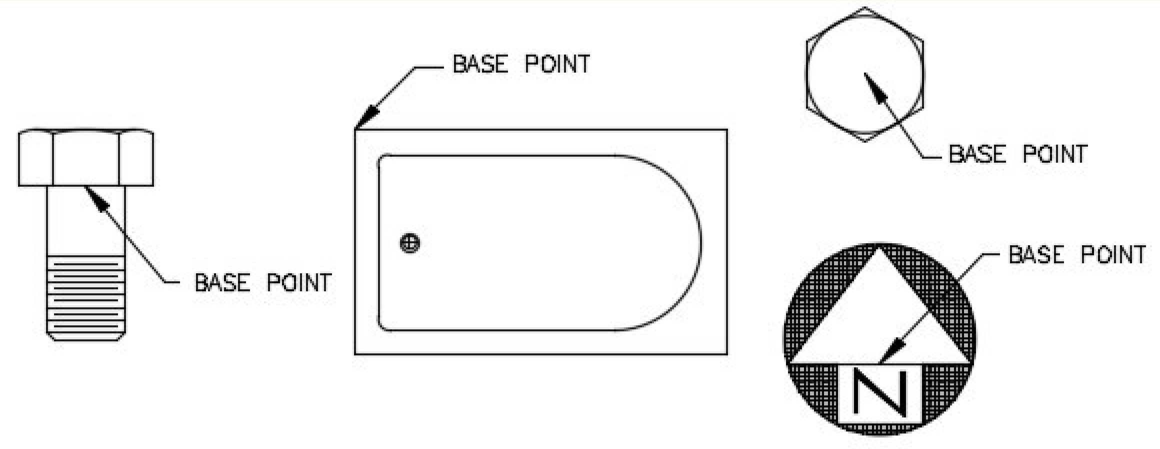


Figure Step 26B

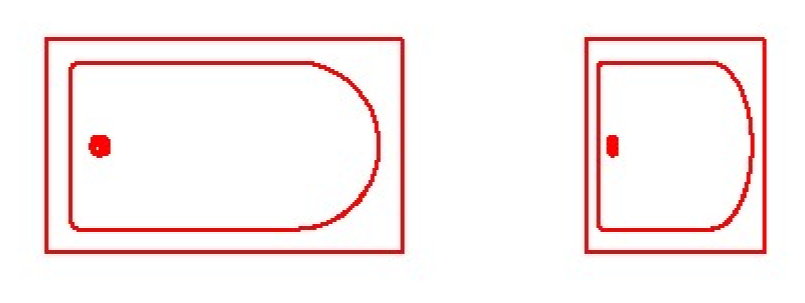
Step 27

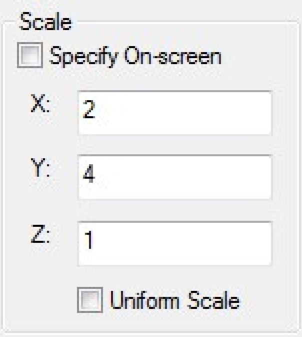
Save and close the drawing.

USER TIP: The location of the base point that is selected when creating a block is important. Each block is unique and has a best location for its base point. When selecting the location for a base point, imagine what location would be best when the block is inserted into a drawing. Study the blocks below and the base point locations that were selected for each of them.



MUST KNOW: To insert a block with different X and Y scales, disable the Uniform scale box and enter the scales and as shown in the figures below.





Scale

☐ Specify On-screen

X: 2

Y: 4

Z: 1

☐ Uniform Scale

Key Principles

Key Principles in Module 33

1. A block is a collection of objects that acts as one object. Regardless of the number of times the block has been inserted into the drawing, it will not significantly change the size of the drawing's database.
2. The only way to remove a block from the block definition table is to purge it with the PURGE command.
3. Layer 0 (zero), assigned the color black/white, is a special layer and color for blocks. A block that was created using objects drawn on layer 0 will reside on and take the properties of the layer it is inserted on.

Lab Exercise 33-1

Time allowed: 90 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 33-1	N/A	Feet and /Inches

Step 1

Open the drawing: AutoCAD 2D Lab 29-3. (Figure Step 1)

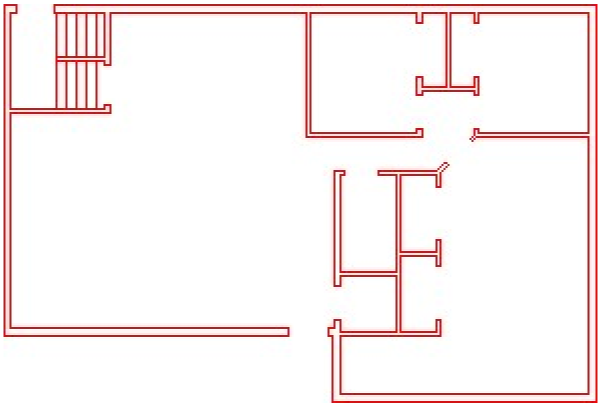


Figure Step 1

Step 2

Using the SAVEAS command, save and name the drawing: AutoCAD 2D Lab 33-1.

Step 3

Make the changes to the floor plan as shown in the figure. Note the size changes and some walls were removed. Use the STRETCH command to make all of the size changes. (Figure Step 3)

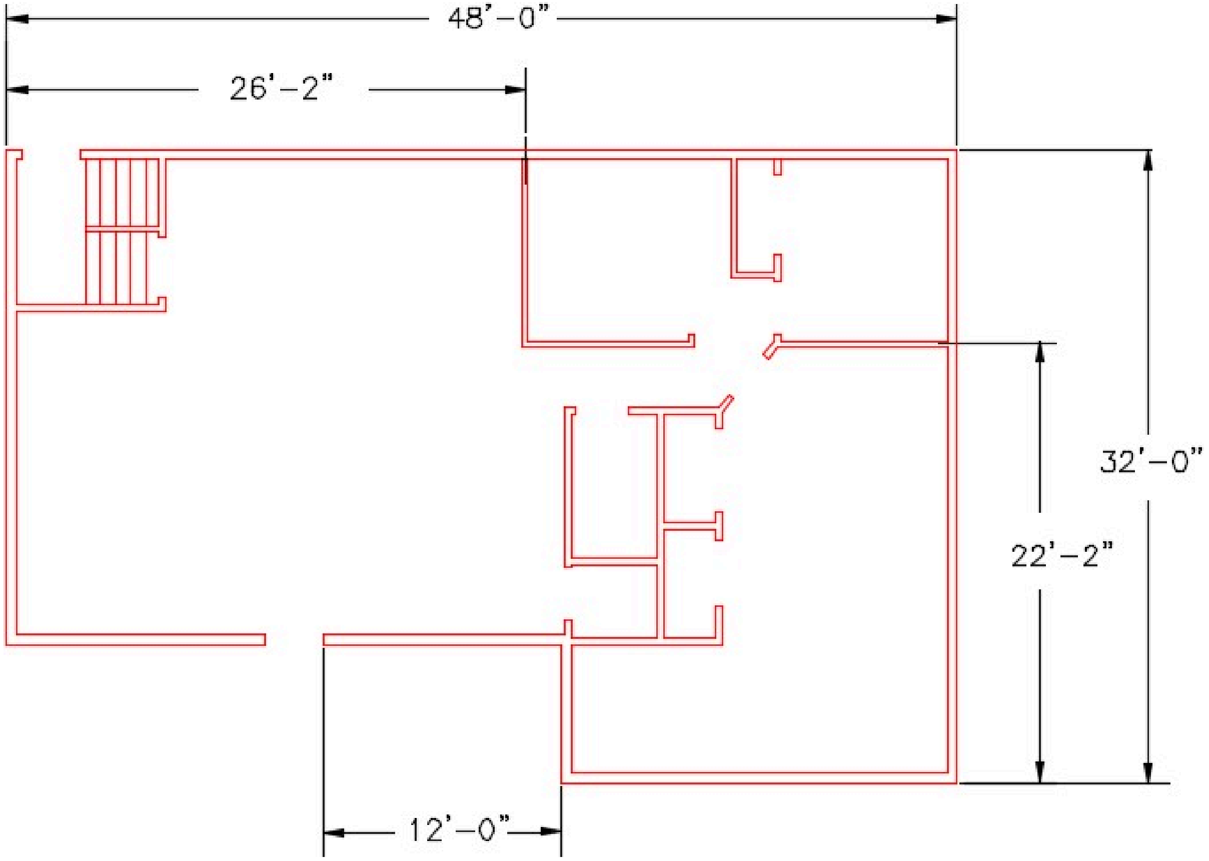
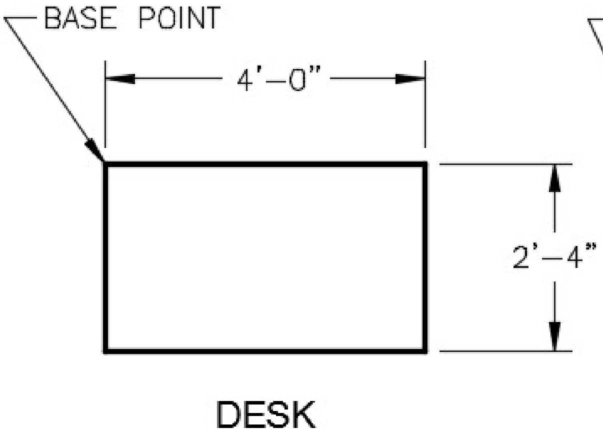
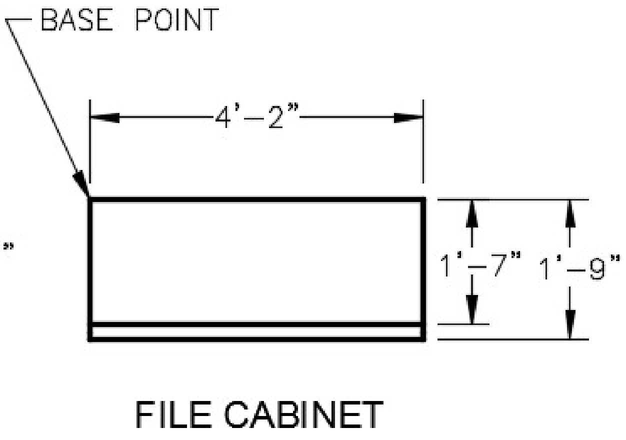

48'–0"
26'–2"
32'–0"
22'–2"
12'–0"

Figure Step 3

Step 4

Draw the 10 objects shown in the figures anywhere in model space. Draw them on layer: 0. Ensure it is assigned the color Black/White. (Figure Step 4A and 4B)


BASE POINT
4'–0"
2'–4"
DESK


BASE POINT
4'–2"
1'–7"
1'–9"
FILE CABINET

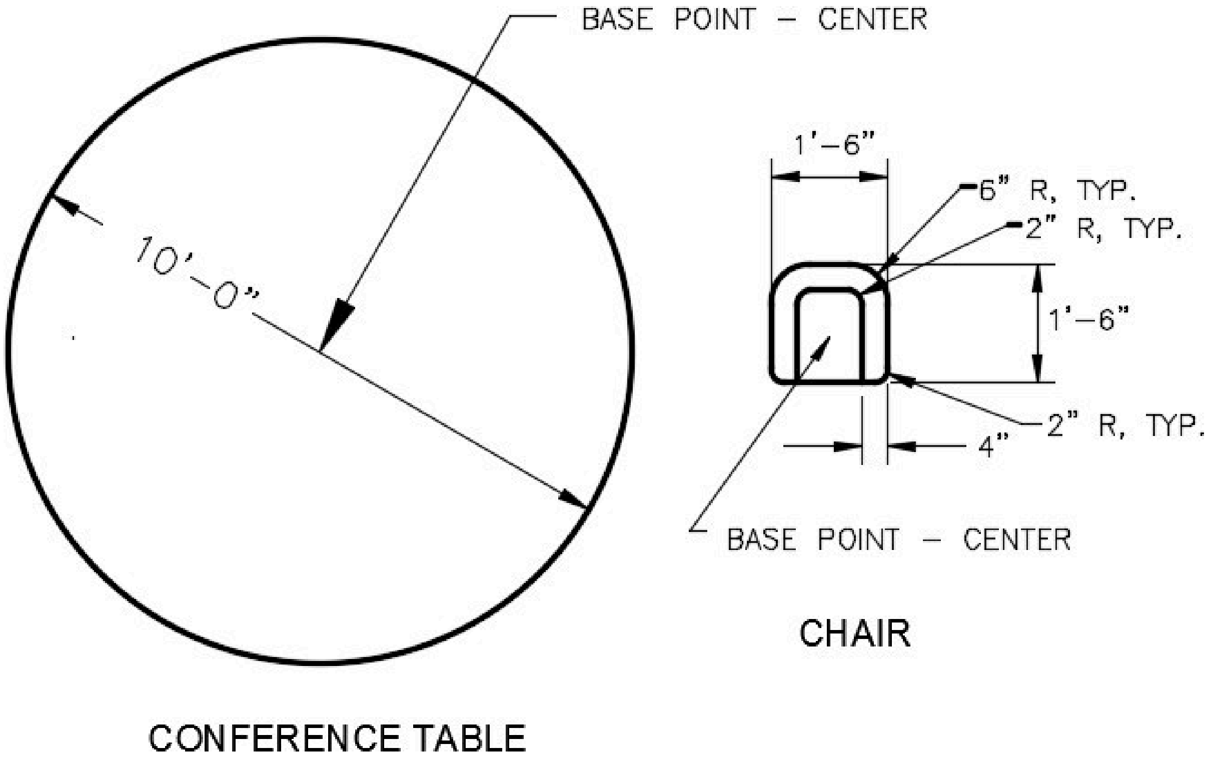
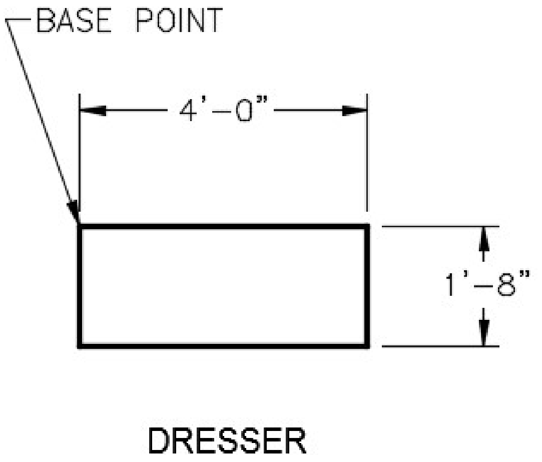
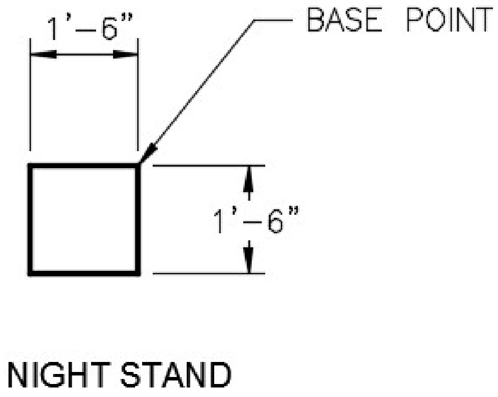

BASE POINT – CENTER
10'-0"
CONFERENCE TABLE
1'-6"
6" R, TYP.
2" R, TYP.
1'-6"
2" R, TYP.
4"
BASE POINT – CENTER
CHAIR

Figure Step 4A


BASE POINT
4'-0"
1'-8"
DRESSER


1'-6"
BASE POINT
1'-6"
NIGHT STAND

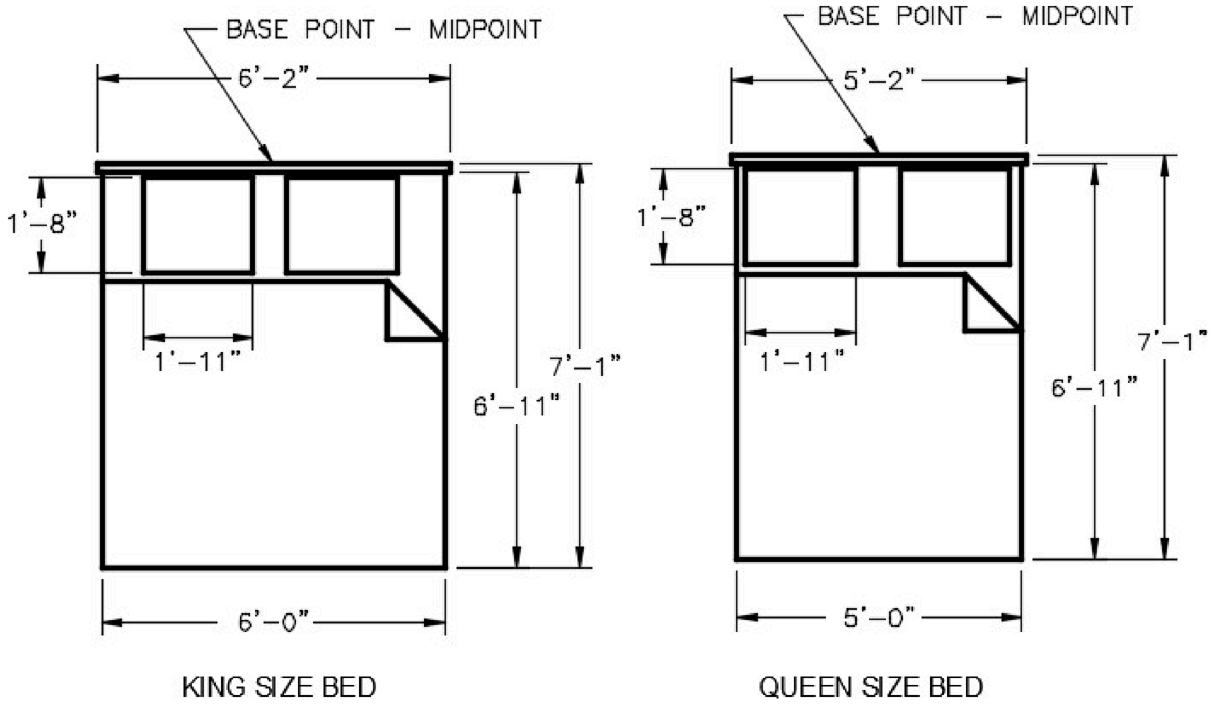

BASE POINT – MIDPOINT
6'–2"
1'–8"
1'–11"
6'–11"
7'–1"
6'–0"
KING SIZE BED
BASE POINT – MIDPOINT
5'–2"
1'–8"
1'–11"
6'–11"
7'–1"
5'–0"
QUEEN SIZE BED

Figure Step 4B

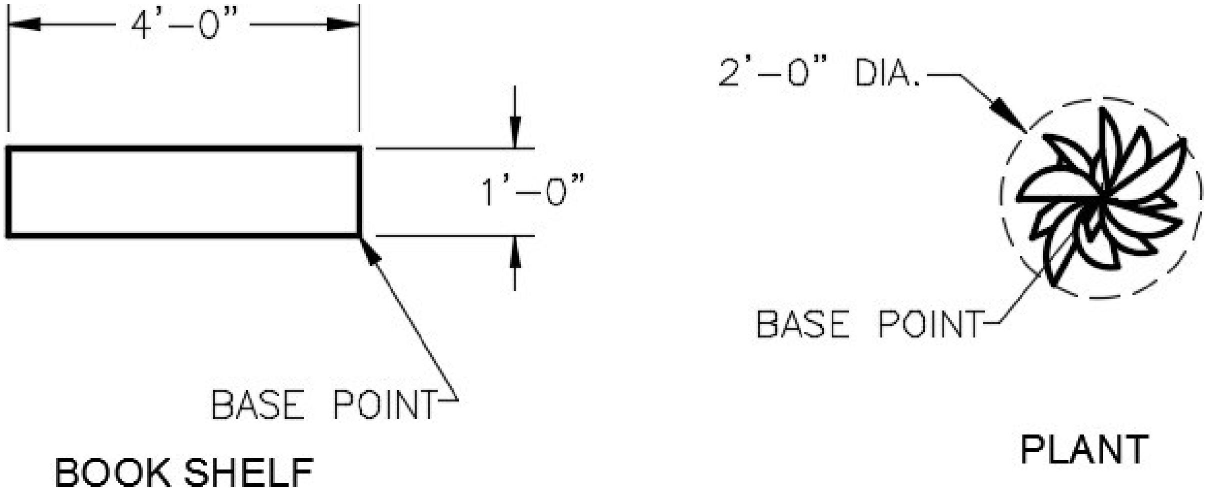
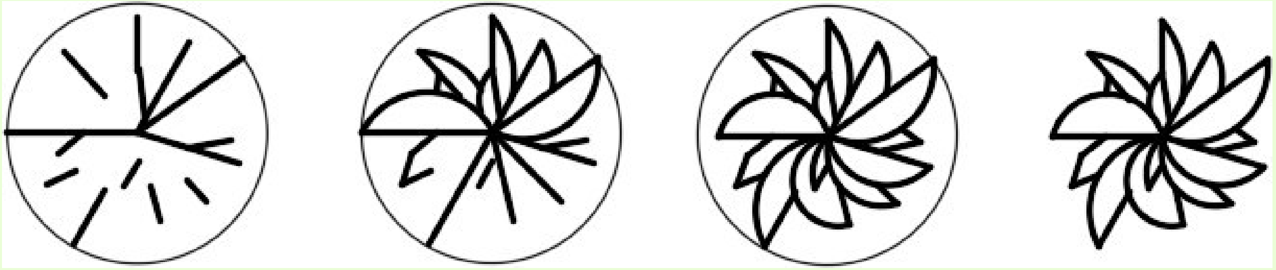

4'–0"
1'–0"
BASE POINT
BOOK SHELF
2'–0" DIA.
BASE POINT
PLANT

Figure Step 4C

AUTHOR'S COMMENTS: The plant block can be drawn by eye. Draw a construction circle and draw random lines. Use a 3 point arcs to complete it. (Figure Hint 1)



Step 5

Create a block from each object. Use the following:

A Name the block as shown under the

B Set the BLOCK command to delete the blocks after

C Set the base point as shown on the

D Give the block a description and set the Block unit to

Step 6

On layer: Furniture, insert the blocks as shown in the figure. Match the drawing the best that you can by eye. (Figure Step 6)

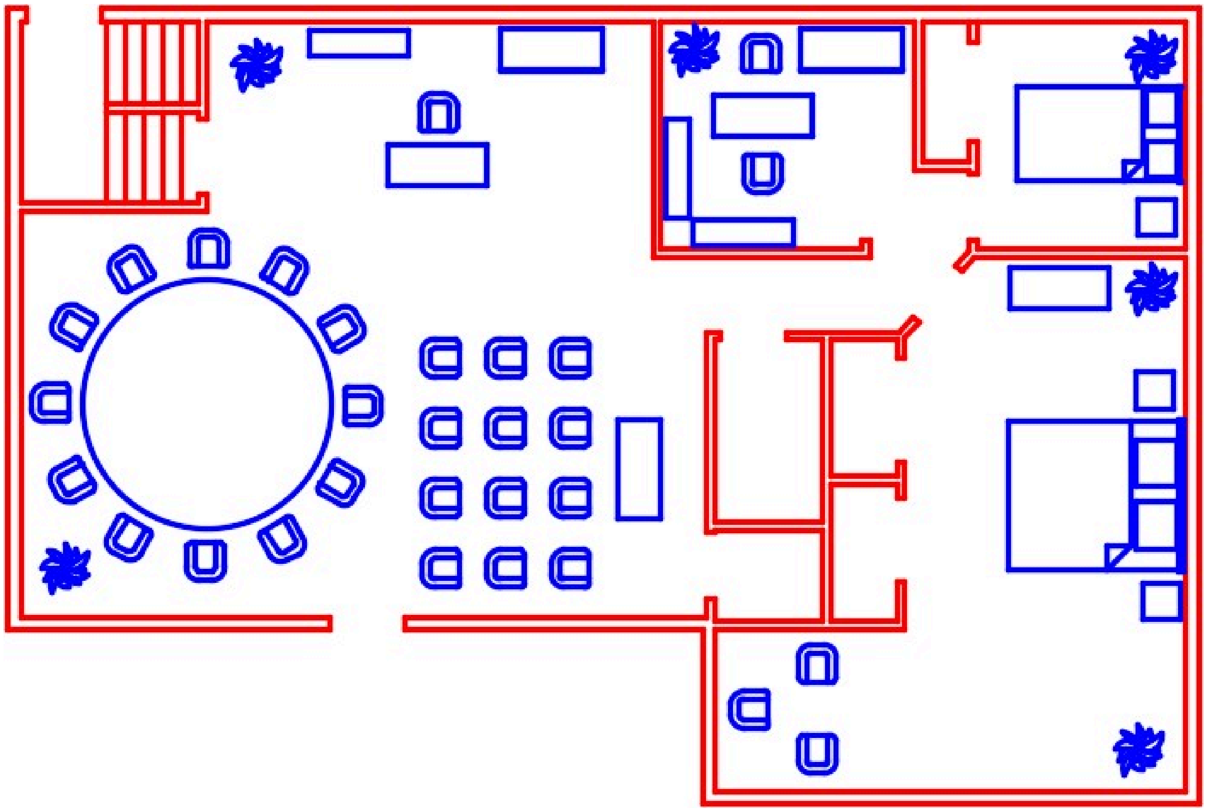


Figure Step 6

AUTHOR'S COMMENTS: Insert one block for each required object and then use the ARRAY or COPY commands to speed the insert process.

Step 7

Enable layout: Module Layout D. On layer: Viewport, create a viewport setting its scale to 1/2″=1′-0″ and lock the display. In Paper space, on layer: Text, label the rooms as shown. Pick your own text size and font. Fill in the titleblock. (Figure Step 7)

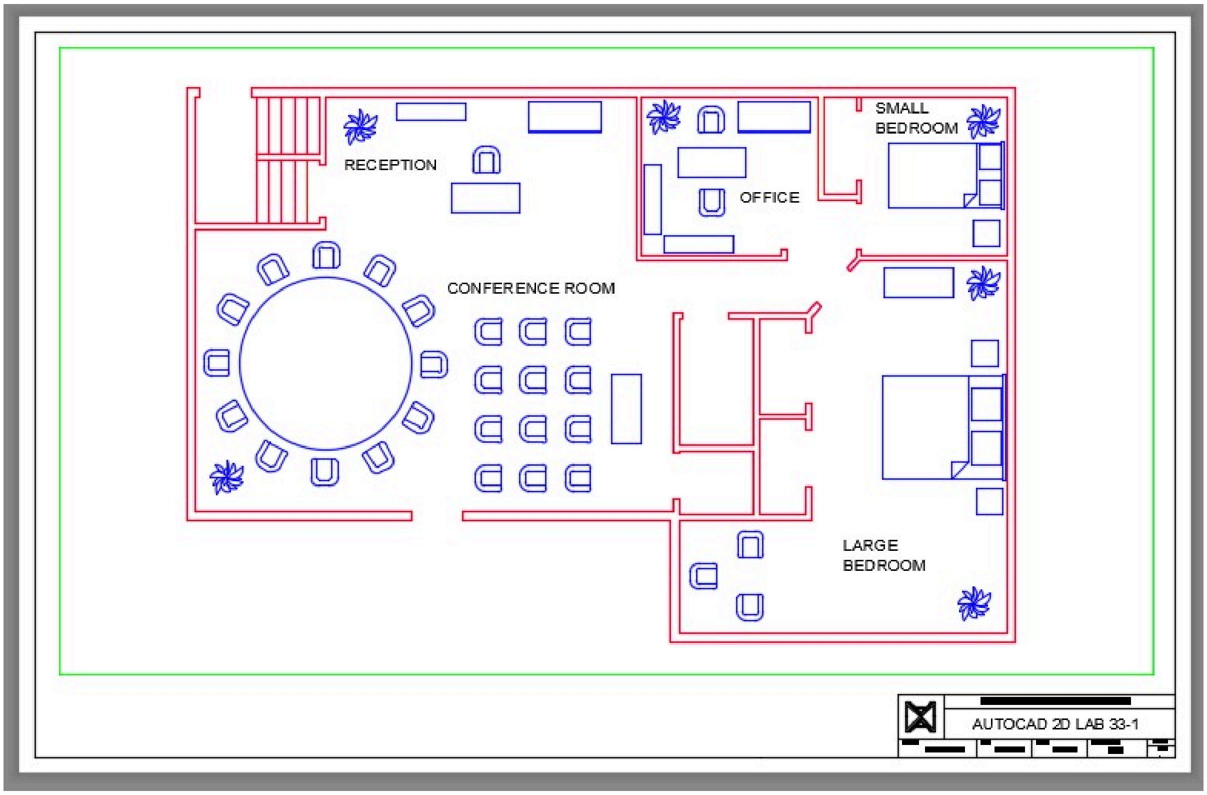

SMALL
BEDROOM
RECEPTION
OFFICE
CONFERENCE ROOM
LARGE
BEDROOM
AUTOCAD 2D LAB 33-1

Figure Step 7

Step 8

Turn layer: Viewport off to complete the drawing. (Figure Step 8)

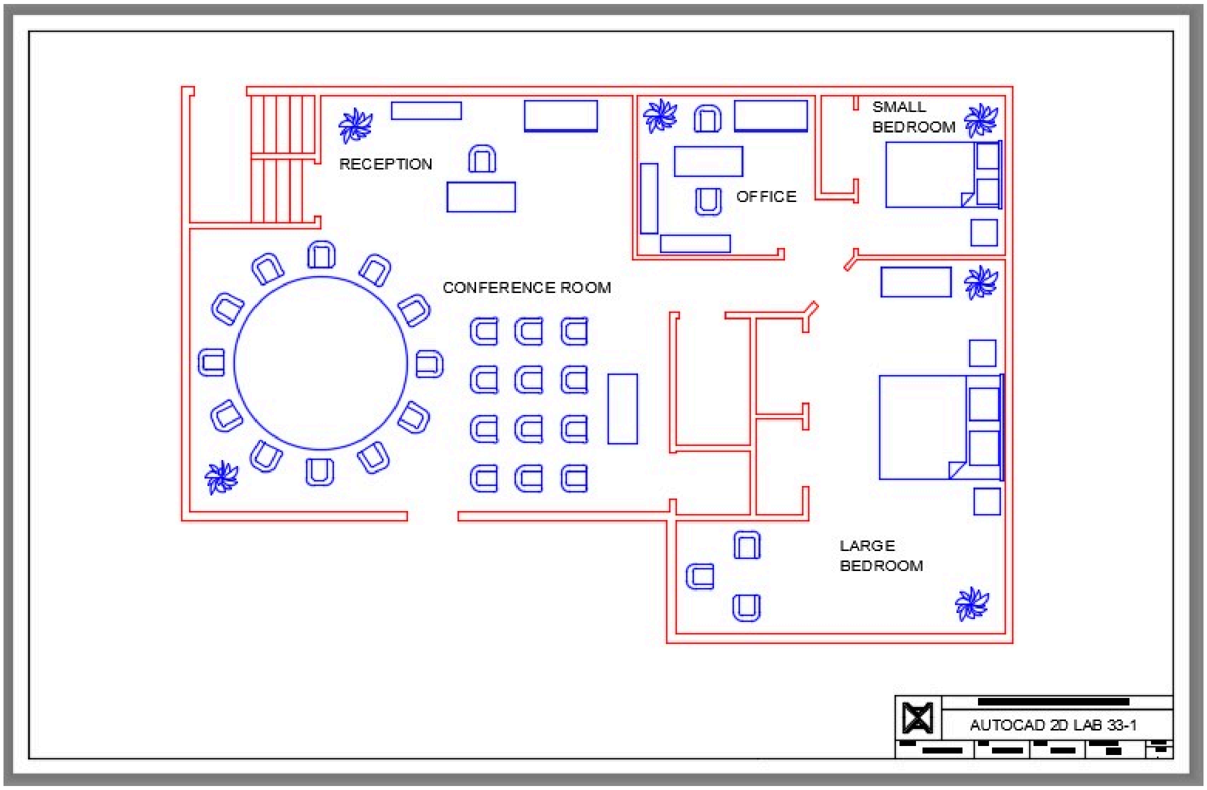

RECEPTION
OFFICE
SMALL BEDROOM
CONFERENCE ROOM
LARGE BEDROOM
AUTOCAD 2D LAB 33-1

Figure Step 8

Step 9

Save and close the drawing.

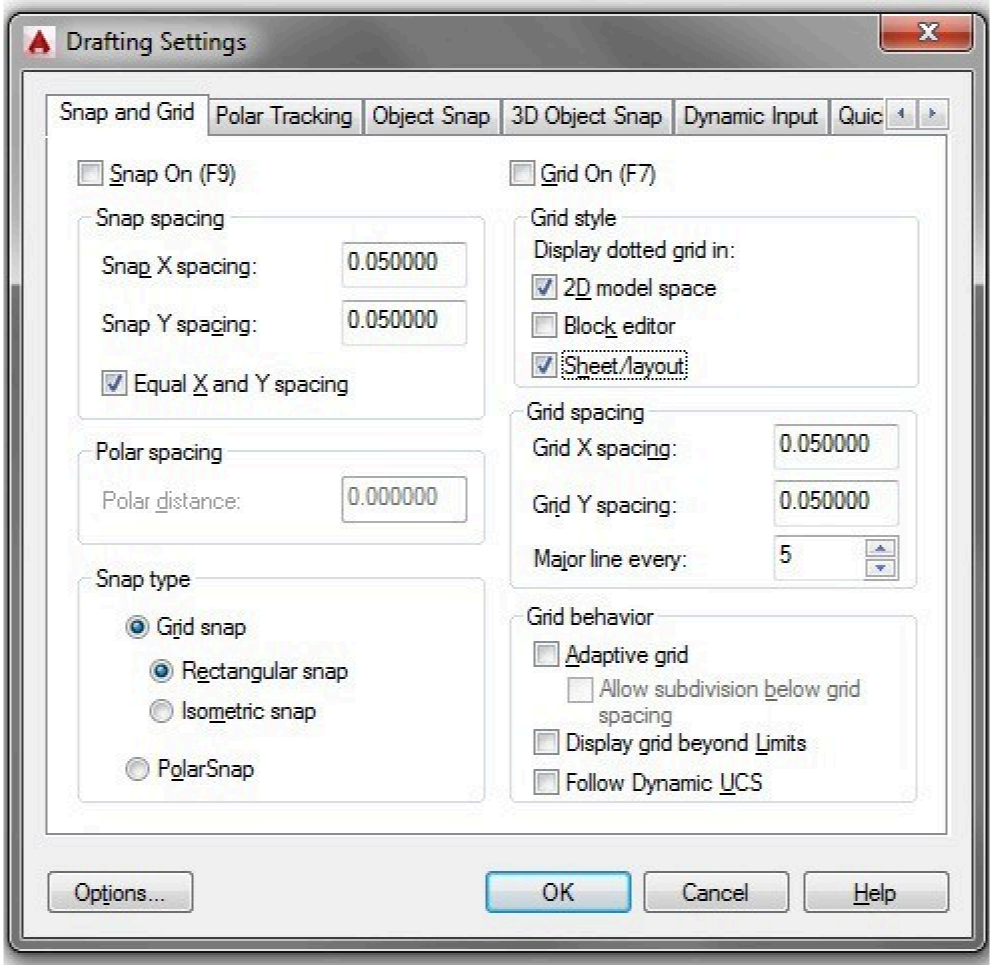
Lab Exercise 33-2

Time allowed: 30 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 33-2	2D Layout English	Inches

Step 1

Open the Drafting Setting dialogue box. Set the grid and snap spacing to 0.05 units and any other settings to match the figure. Ensure that you disable the Adaptive grid and the Display grid beyond Limits box. (Figure step 1)



Drafting Settings
Snap and Grid
Polar Tracking
Object Snap
3D Object Snap
Dynamic Input
Quic
Options...
OK
Cancel
Help

☐ Snap On (F9)

Snap spacing

Snap X spacing: 0.050000

Snap Y spacing: 0.050000

☑ Equal X and Y spacing

Polar spacing

Polar distance: 0.000000

Snap type

● Grid snap

● Rectangular snap

○ Isometric snap

○ PolarSnap

☐ Grid On (F7)

Grid style

Display dotted grid in:

☑ 2D model space

☐ Block editor

☑ Sheet/layout

Grid spacing

Grid X spacing: 0.050000

Grid Y spacing: 0.050000

Major line every: 5

Grid behavior

☐ Adaptive grid

☐ Allow subdivision below grid spacing

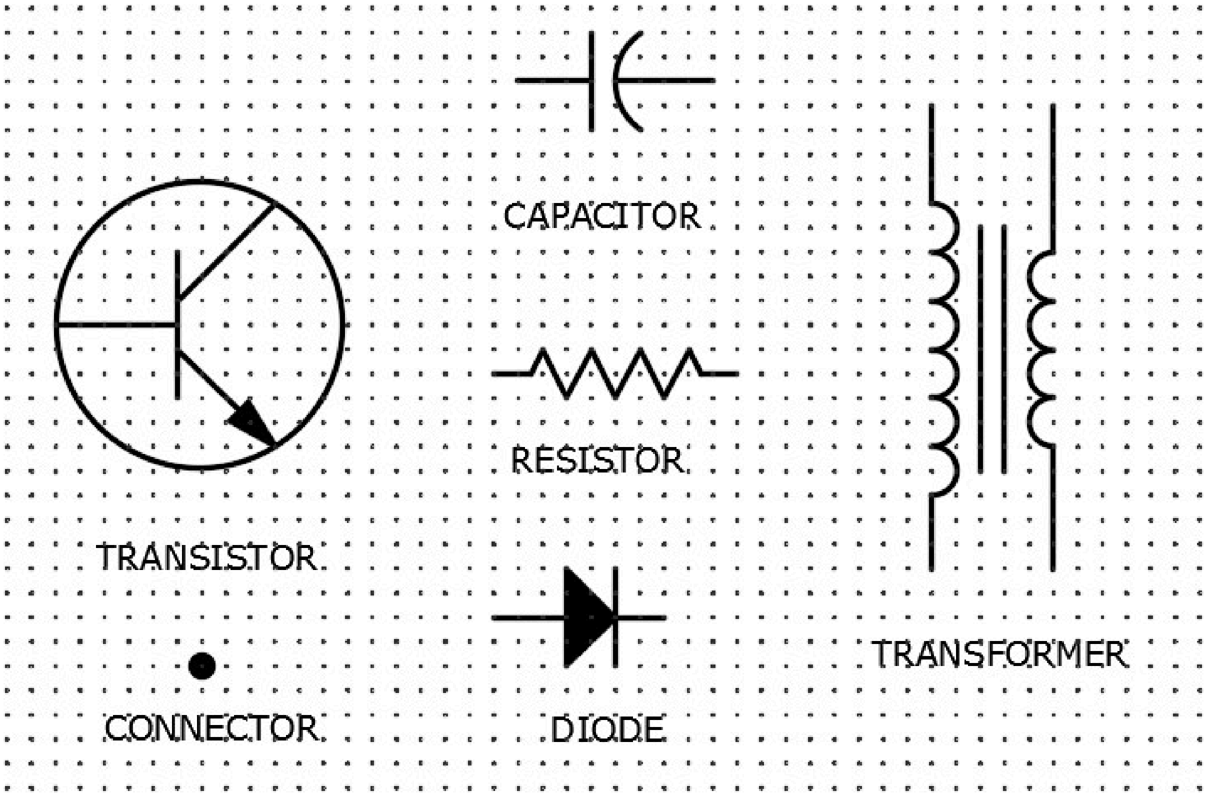
☐ Display grid beyond Limits

☐ Follow Dynamic UCS

Figure Step 1

Step 2

With the grid and snap enabled, draw the five symbols shown below anywhere in model pace. Draw them on layer 0. You will have zoom in close to draw the symbols. (Figure Step 2)


CAPACITOR
TRANSISTOR
RESISTOR
TRANSFORMER
CONNECTOR
DIODE

Step 2

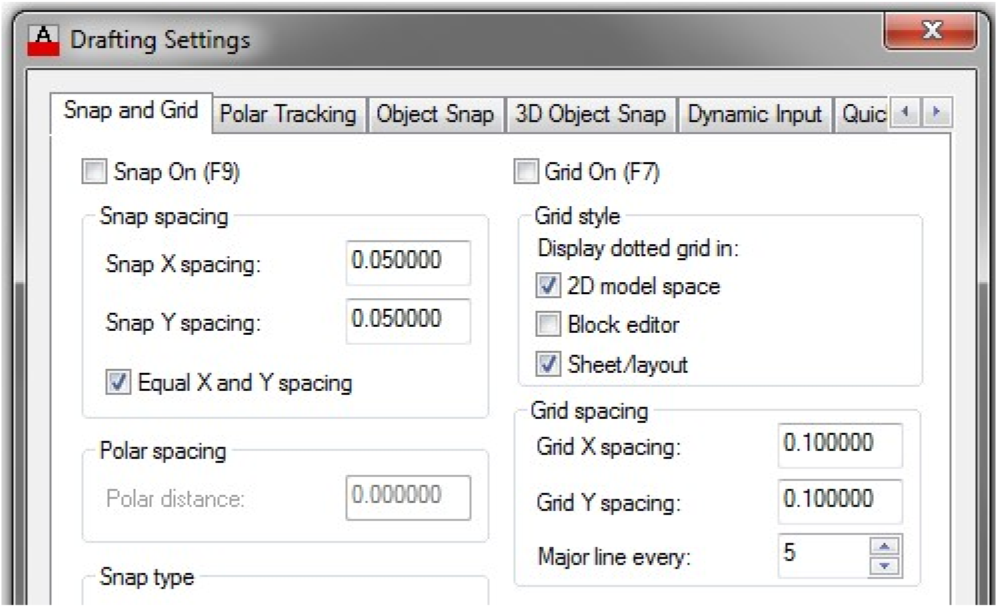
AUTHOR'S COMMENTS: Ensure that you draw each symbol by snapping to grid points as shown below. Notice how each of the connecting points of the symbols are located on a grid point.

Step 3

Make a block of each symbol. Name them with the name shown below the drawing. Set the Block units to Inches. Delete the objects when you the block is created.

Step 4

Set the grid to 0.1 and the snap to 0.05. (Figure Step 4)



Drafting Settings
Snap and Grid
Polar Tracking
Object Snap
3D Object Snap
Dynamic Input
Quic

☐ Snap On (F9)

Snap spacing

Snap X spacing: 0.050000

Snap Y spacing: 0.050000

☑ Equal X and Y spacing

Polar spacing

Polar distance: 0.000000

Snap type

☐ Grid On (F7)

Grid style

Display dotted grid in:

☑ 2D model space

☐ Block editor

☑ Sheet/layout

Grid spacing

Grid X spacing: 0.100000

Grid Y spacing: 0.100000

Major line every: 5

Figure Step 4

AUTHOR'S COMMENTS: To draw the solid areas of the symbols, first draw the outside shape with polylines or circles and then hatch them with a solid pattern..

Step 5

On layer: Object, draw the lines and the terminating circles for the schematic diagram. Draw it by eye but do your best to match the figure. Ensure that you snap to grid point for the endpoints of all lines. (Figure Step 5)

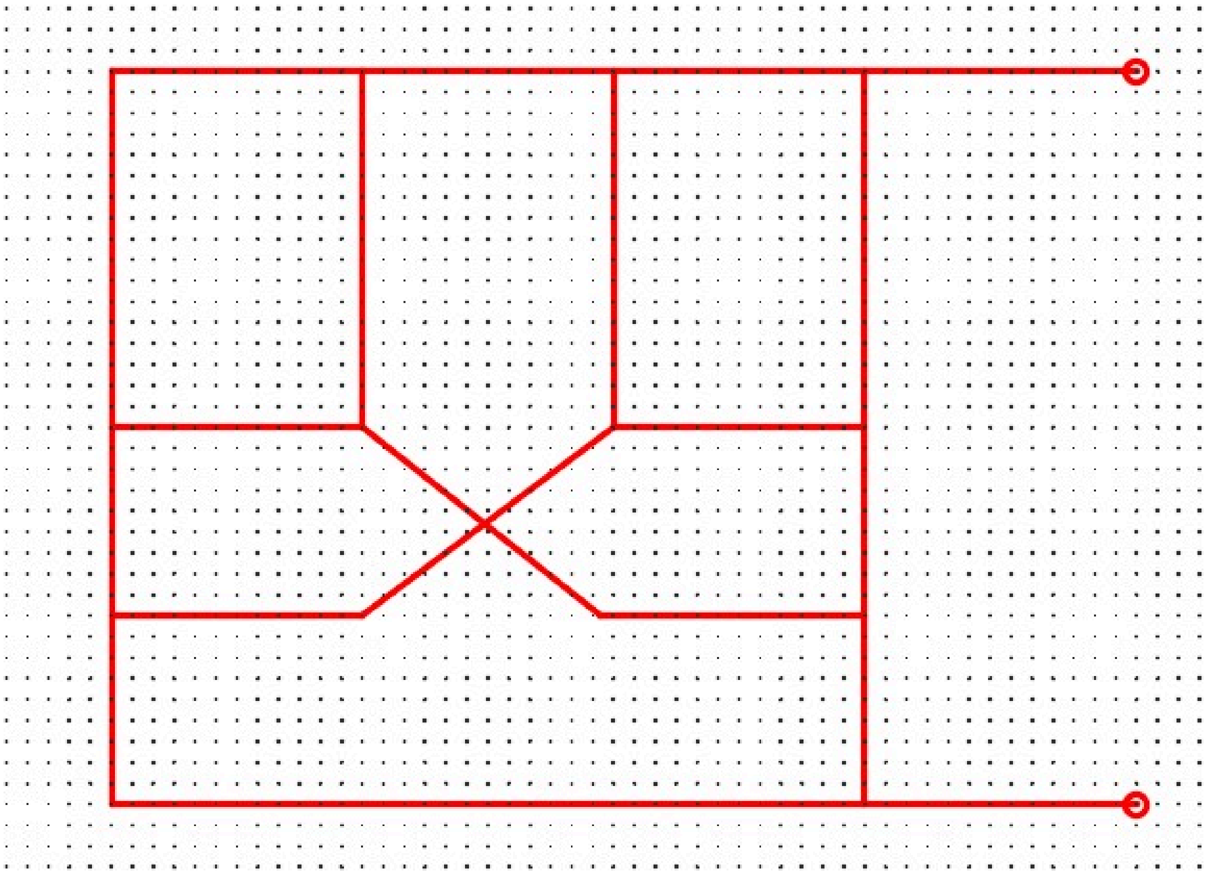


Figure Step 5

Step 6

On layer: Symbols, insert the blocks and trim the lines. (Figure Step 6)

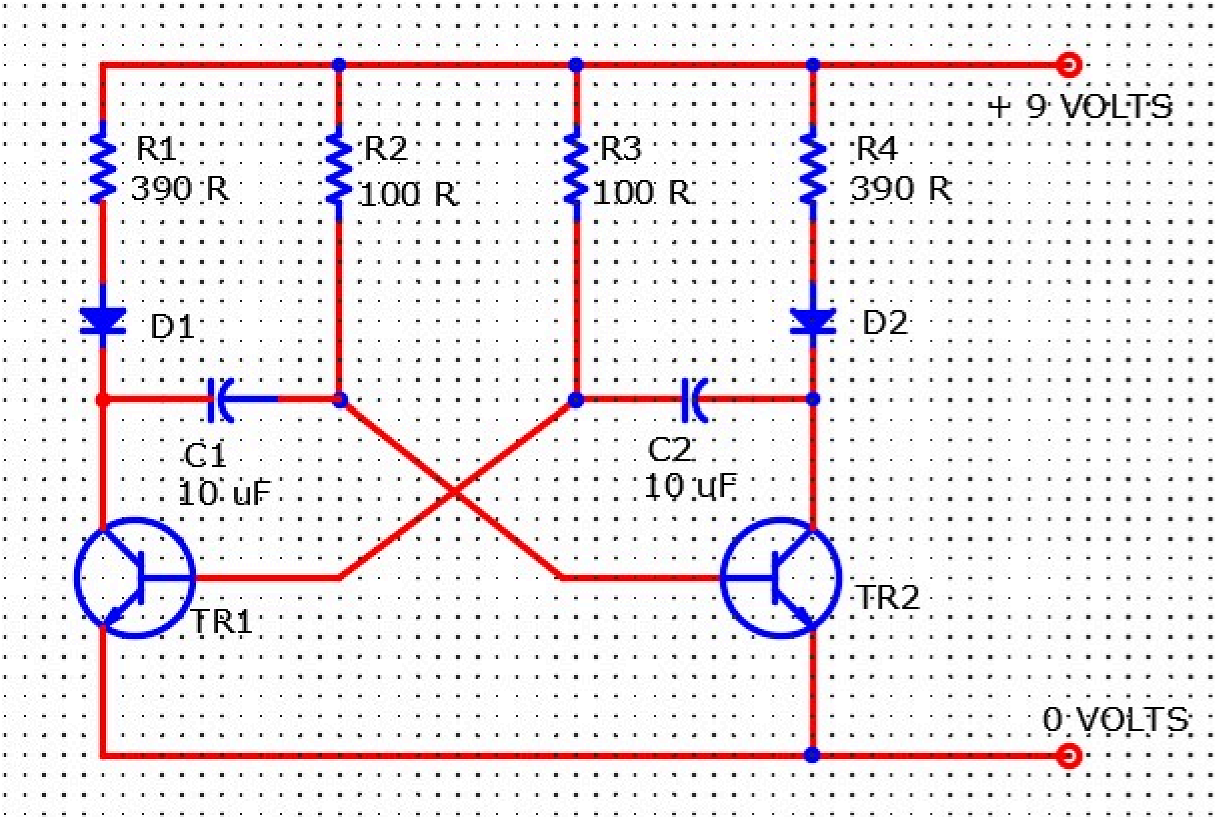

R1
390 R
R2
100 R
R3
100 R
R4
390 R
+ 9 VOLTS
D1
D2
C1
10 uF
C2
10 uF
TR1
TR2
0 VOLTS

Figure Step 6

Step 7

On layer: Viewport, create viewport setting the scale to 1.25:1 and lock the display. (Figure Step 7)

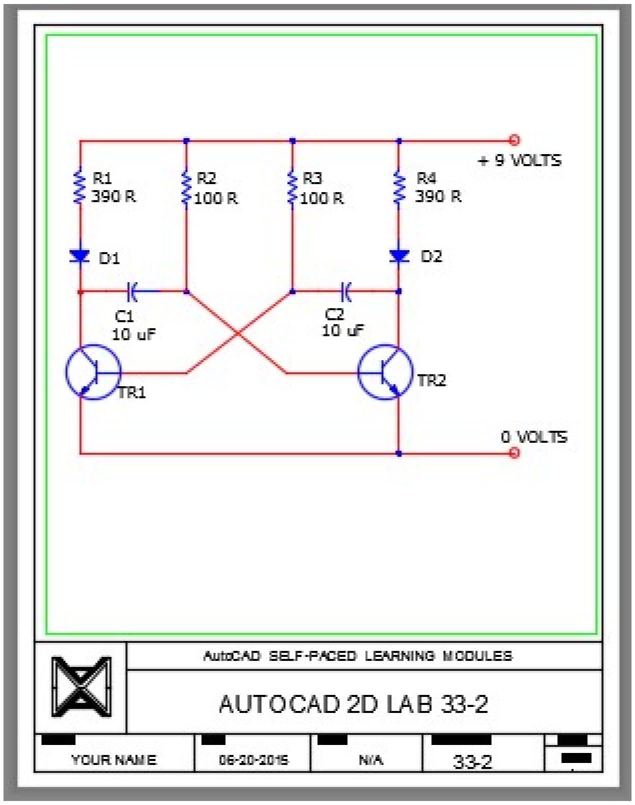
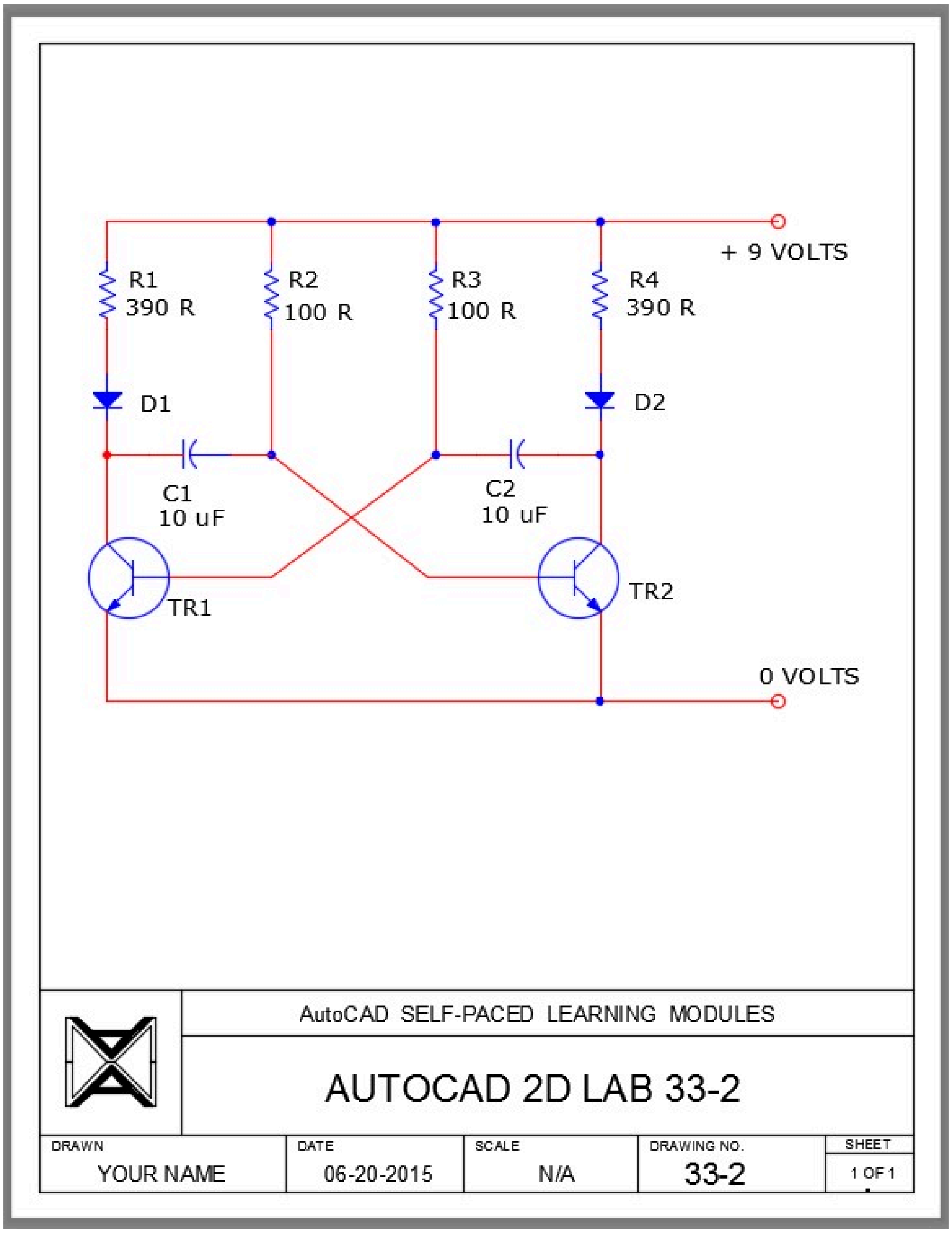

R1
390 R
R2
100 R
R3
100 R
R4
390 R
+ 9 VOLTS
D1
D2
C1
10 uF
C2
10 uF
TR1
TR2
0 VOLTS
AutoCAD SELF-PACED LEARNING MODULES
AUTOCAD 2D LAB 33-2
YOUR NAME
06-20-2015
N/A
33-2

Figure Step 7

Step 8

In Paper space, on layer: Text, add the text as shown in the figure. Pick your own font and text height. Fill in the titleblock and turn layer: Viewport off to complete the drawing. (Figure step 8)


R1
390 R
R2
100 R
R3
100 R
R4
390 R
+ 9 VOLTS
D1
D2
C1
10 uF
C2
10 uF
TR1
TR2
0 VOLTS

AutoCAD SELF-PACED LEARNING MODULES

AUTOCAD 2D LAB 33-2

DRAWN	DATE	SCALE	DRAWING NO.	SHEET
YOUR NAME	06-20-2015	N/A	33-2	1 OF 1

Figure Step 8

Module 34: Drawing Utilities

Learning Outcomes

When you have completed this module, you will be able to:

1. Explain drawing and file maintenance to help manage and control AutoCAD drawing files.
2. Describe a nested object.
3. Apply the TIME, PURGE, and RENAME commands.

Drawing Utilities

It is very important for you as the AutoCAD Operator to maintain your drawings and drawing files on a regular basis. It is very easy, while working day-to-day producing drawings, to let things slide and not correct small items in the drawings or the drawing files. If left unchecked, this habit can lead to a large problem in the future, sometimes almost impossible to correct.

Take a few minutes, while working on each drawing, to maintain objects, layers, blocks, xref, names, and the different styles. For the drawing files, create an organized logical folder structure making more folders rather than less and dividing drawing files into smaller organized groups. Create a good naming system for the folders and drawings. Give them descriptive names that can be sorted alphabetically. Remove unwanted files, and keep your .bak files under control by deleted ones that are no longer required.

AutoCAD has several utilities that are helpful to organize and maintain drawings. File Explorer can be used to maintain the drawing files. In this module, the PURGE and RENAME commands are covered to help you in your day-to-day drawing maintenance.

Nested Objects

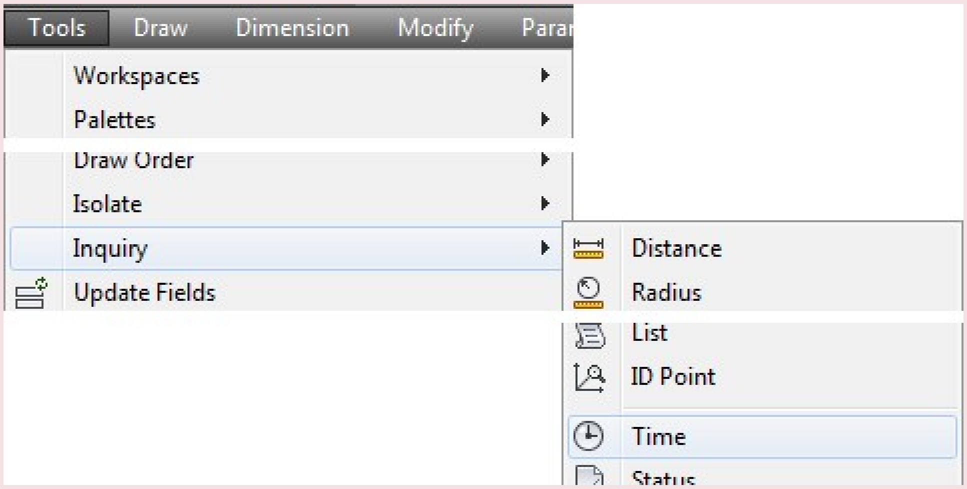
A *nested object* is a named object referencing another named object. One example of a nested object would be an object in a block located on layer Object but the block is on layer Block. The Object layer would appear not to have any objects on it, but in fact, an object in the block is on that layer and that makes it a nested object. Sometimes it is desirable to nest items to help draw and design more efficiently. An example of this is as follows: A drawing has several components. Draw each component separately and create a block of each. Assemble all the blocks and then make a block of the assembly. The blocks inserted in the assembly block are nested blocks. Nested objects are not necessarily a bad thing, but they can cause problem if they are not used correctly or carelessly inserted.

AutoCAD does not allow a block to be created using a block that references itself. An example of this would be a block created using an object that contains a block with the same name. One the most common errors in block creation is creating a block selecting a block as one of the selected objects. Ensure that all drawing objects are exploded before creating a block using them.

AutoCAD Command: TIME

The TIME command is used to display and track the amount of time that a drawing file is opened in AutoCAD.

Shortcut: none

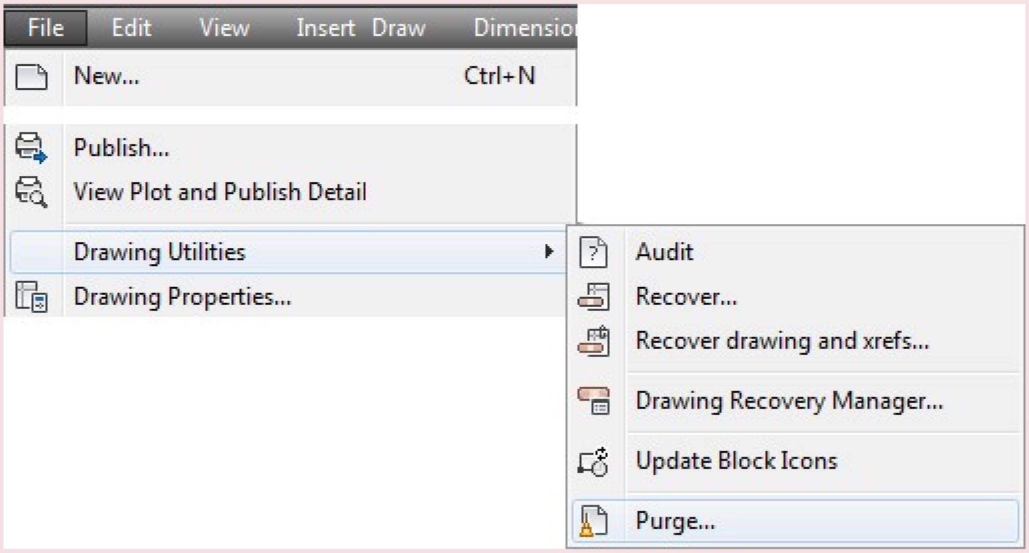

Tools
Draw
Dimension
Modify
Paran
Workspaces
Palettes
Draw Order
Isolate
Inquiry
Update Fields
Distance
Radius
List
ID Point
Time
Status

Tools Pull-down

AutoCAD Command: PURGE

The PURGE command is used to remove block definitions, dimension styles, groups, layers, linetypes, text styles, zero-length geometry, and empty text objects.

Shortcut: **PU**

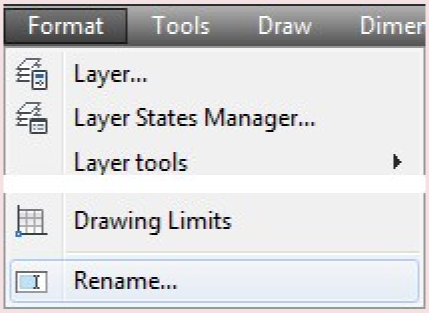

File
Edit
View
Insert
Draw
Dimensio
New...
Ctrl+N
Publish...
View Plot and Publish Detail
Drawing Utilities
Drawing Properties...
Audit
Recover...
Recover drawing and xrefs...
Drawing Recovery Manager...
Update Block Icons
Purge...

File Pull-down

AutoCAD Command: RENAME

The RENAME command is used to rename such things as block definitions, dimension styles, layers, linetypes, text styles, UCS's, viewports, and views.

Shortcut: **REN**


Format
Tools
Draw
Dimen
Layer...
Layer States Manager...
Layer tools
Drawing Limits
Rename...

Format Pull-down

WORKALONG: Using the TIME, PURGE and RENAME Commands

Step 1

Start a new drawing using the template: 2D Layout English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 34-1.

Step 3

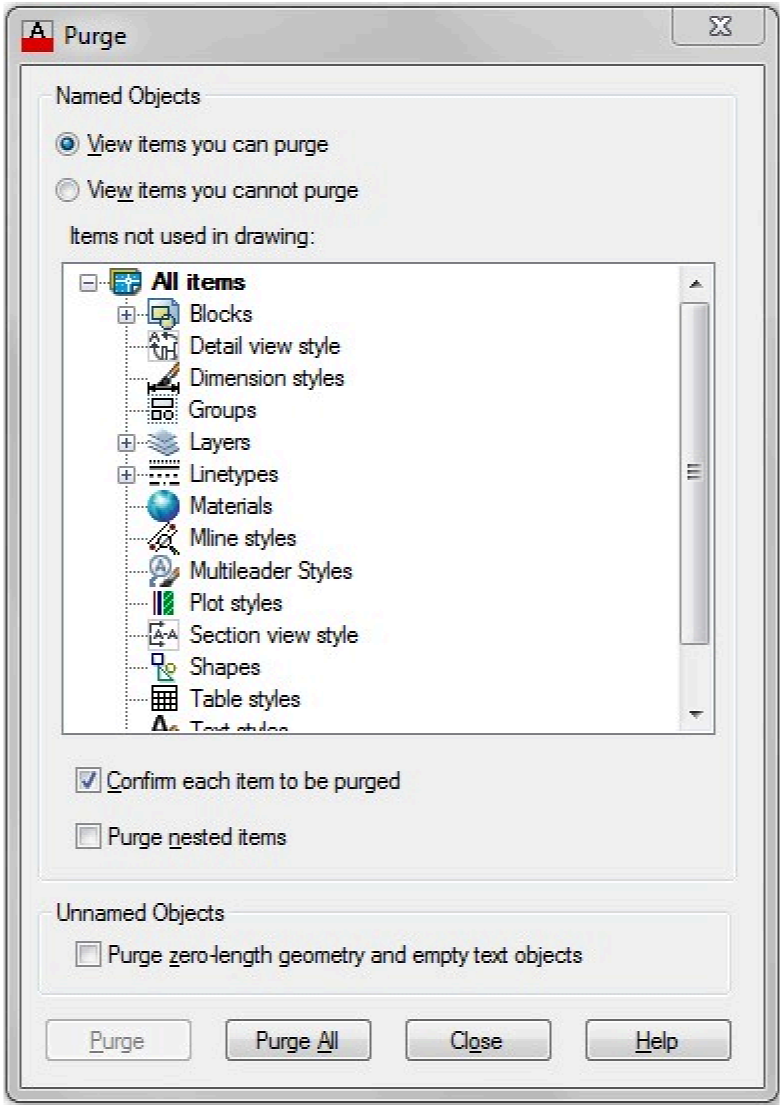
Enter the TIME command, as shown below.

```
Command: TIME

Current time: Wednesday, March 01, 2016 11:49:30:937 AM

Times for this drawing:

Created: Wednesday, March 01, 2016 11:48:42:453 AM

Last updated: Wednesday, March 01, 2016 11:48:42:453 AM

Total editing time: 0 days 00:12:48:672

Elapsed timer (on): 0 days 00:12:48:484

Next automatic save in: 0 days 00:09:13:172

Enter option [Display/ON/OFF/Reset]:

Command:
```

AUTHOR’S COMMENTS: Your answers will differ from this example.

Step 4

Enter the PURGE command. It will open the Purge dialogue box. (Figure Step 4)



Purge

Named Objects

● View items you can purge

○ View items you cannot purge

Items not used in drawing:

☑ Confirm each item to be purged

☐ Purge nested items

Unnamed Objects

☐ Purge zero-length geometry and empty text objects

All items
Blocks
Detail view style
Dimension styles
Groups
Layers
Linetypes
Materials
Mline styles
Multileader Styles
Plot styles
Section view style
Shapes
Table styles
Text styles
Purge
Purge All
Close
Help

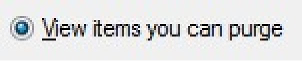
Figure Step 4

AUTHOR'S COMMENTS: You can only purge items from a drawing if they are NOT being used at the time. For example, an item that CAN be purged is a block that is defined but is not inserted in the drawing or a layer that was created but does not contain any objects. The items that can be purged will display with a + (plus sign) preceding them.

Step 5

Ensure that the View items you can purge is enabled. (Figure Step 5)

You can only purge items from a drawing if they are NOT being used at the time. For example, an item that CAN be purged is a block that is defined but is not inserted in the drawing or a layer that was created but does not contain any objects. The items that can be purged will display with a + (plus sign) preceding them.

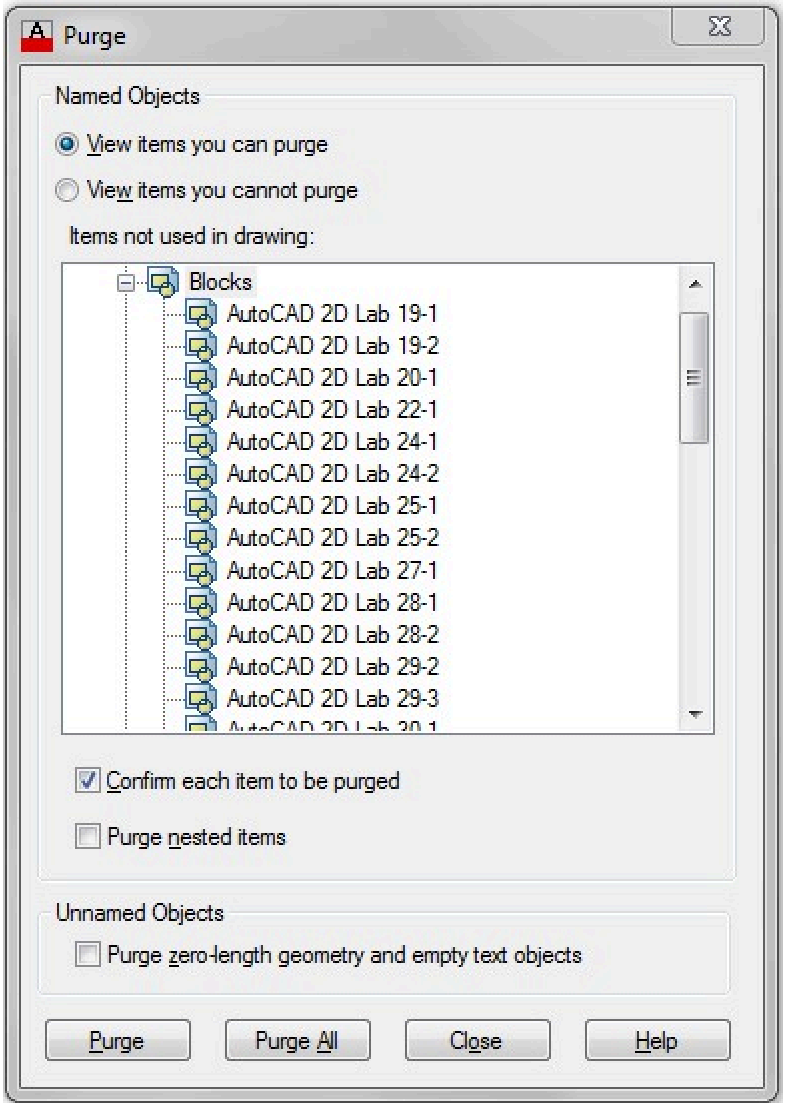


● View items you can purge

Figure Step 5

Step 6

Expand All items and then Blocks by clicking the + (plus sign) preceding each. When you see the – (minus sign), the item is expanded and will display all of the items that can be purged. All of the blocks in the Block Definition table that are NOT inserted in the drawing will display. (Figure Step 6)



Named Objects

● View items you can purge

○ View items you cannot purge

Items not used in drawing:

- Blocks
  - AutoCAD 2D Lab 19-1
  - AutoCAD 2D Lab 19-2
  - AutoCAD 2D Lab 20-1
  - AutoCAD 2D Lab 22-1
  - AutoCAD 2D Lab 24-1
  - AutoCAD 2D Lab 24-2
  - AutoCAD 2D Lab 25-1
  - AutoCAD 2D Lab 25-2
  - AutoCAD 2D Lab 27-1
  - AutoCAD 2D Lab 28-1
  - AutoCAD 2D Lab 28-2
  - AutoCAD 2D Lab 29-2
  - AutoCAD 2D Lab 29-3

☑ Confirm each item to be purged

☐ Purge nested items

Unnamed Objects

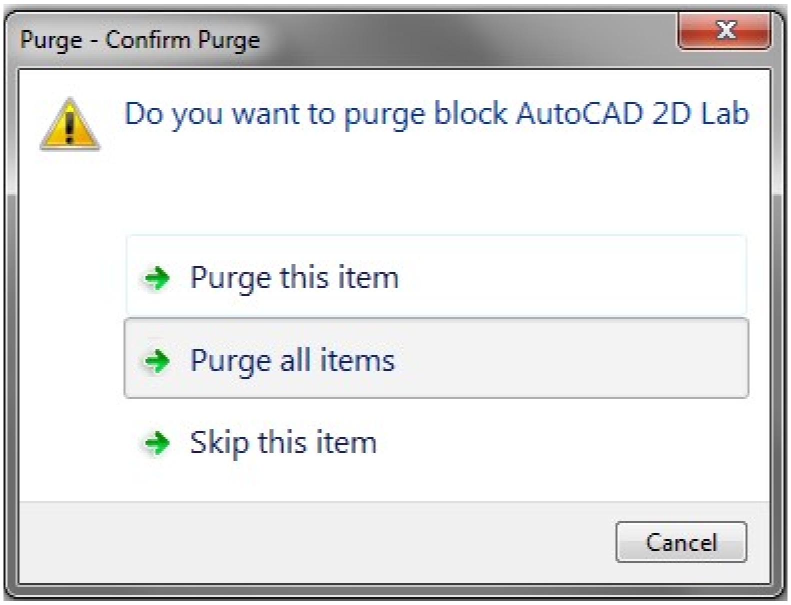
☐ Purge zero-length geometry and empty text objects

Figure Step 6

AUTHOR'S COMMENTS: Your list may differ from this example.

Step 7

Select the item Blocks. Click the Purge All button. When asked to confirm the purging, click Purge all items. (Figure Step 7)



Purge - Confirm Purge

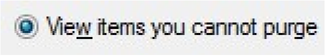
Do you want to purge block AutoCAD 2D Lab

Figure Step 7

AUTHOR'S COMMENTS: After you complete the purge, note how the item Blocks no longer has a + (plus sign) proceeding it.

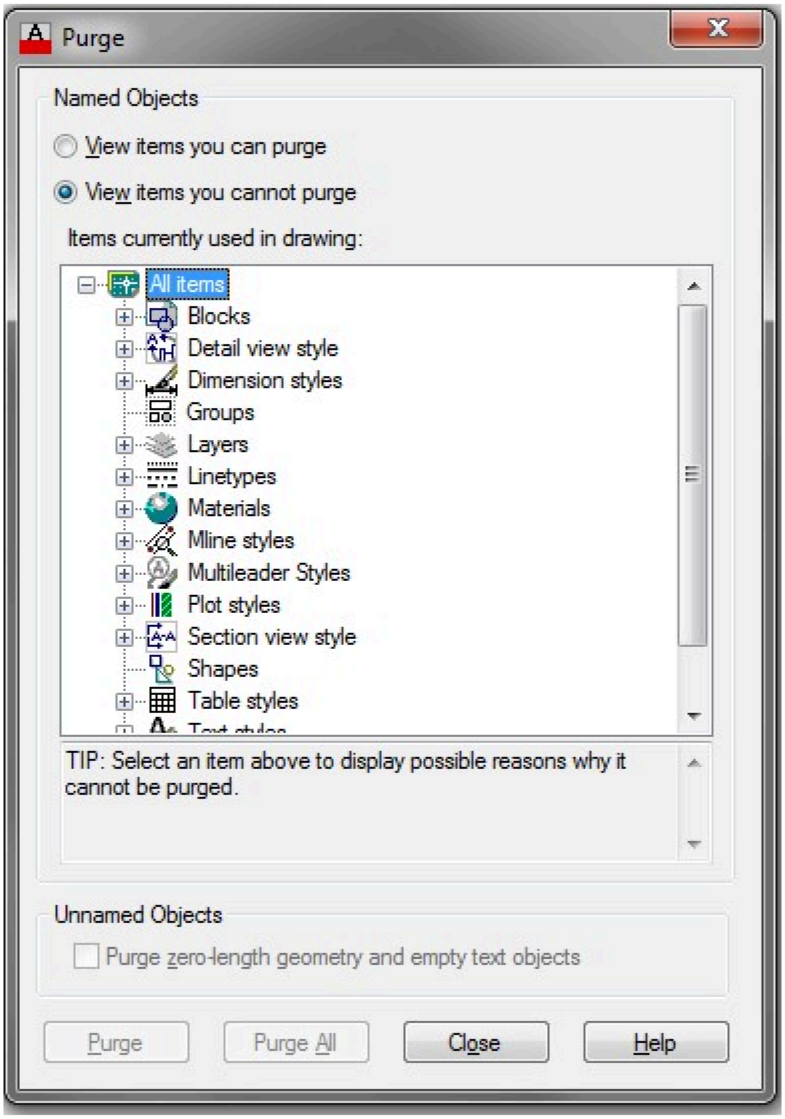
Step 8

Enable the View items you cannot purge button. The items that you CANNOT purge will display in the Purge dialogue box. (Figure Step 8A and 8B)



● View items you cannot purge

Figure Step 8A


Purge
Named Objects
Items currently used in drawing:
All items
Blocks
Detail view style
Dimension styles
Groups
Layers
Linetypes
Materials
Mline styles
Multileader Styles
Plot styles
Section view style
Shapes
Table styles
Text styles
Unnamed Objects
Purge
Purge All
Close
Help

○ View items you can purge

● View items you cannot purge

TIP: Select an item above to display possible reasons why it cannot be purged.

☐ Purge zero-length geometry and empty text objects

Figure Step 8B

Step 9

Close the dialogue box. Enter the TIME command as shown below.

Command: **TIME**

Current time: Wednesday, March 01, 2016 2:04:49:703 PM

Times for this drawing:

Created: Wednesday, March 01, 2016 1:21:07:562 PM

Last updated: Wednesday, March 01, 2016 1:21:07:562 PM

Total editing time: 0 days 15:43:42:157

Elapsed timer (on): 0 days 15:43:42:141

Next automatic save in: 0 days 00:07:33:123

Enter option [Display/ON/OFF/Reset]: **R**

(Reset the timer to zero. This is an elapsed timer which allows you to time a special event while the total time the drawing is active is also being timed.)

Enter option [Display/ON/OFF/Reset]: **D**

(The Display option simply forces the command line to display the time values.)

Current time: Wednesday, March 01, 2016 2:05:08:312 PM

Times for this drawing:

Created: Wednesday, March 01, 2016 1:21:07:562 PM

Last updated: Wednesday, March 01, 2016 1:21:07:562 PM

Total editing time: 0 days 00:44:00:766

Elapsed timer (on): 0 days 00:00:08:437 Next automatic save in: 0 days 00:07:33:123 Enter option [Display/ON/OFF/Reset]:

Command:

AUTHOR'S COMMENTS: Your answers will differ.

Step 10

On layer: 0, draw the three objects shown in the figure. Draw the circle 1 inch in diameter and the length of each side of the square and the diamond 1 inch. The location where you draw them is not important. (Figure Step 10)

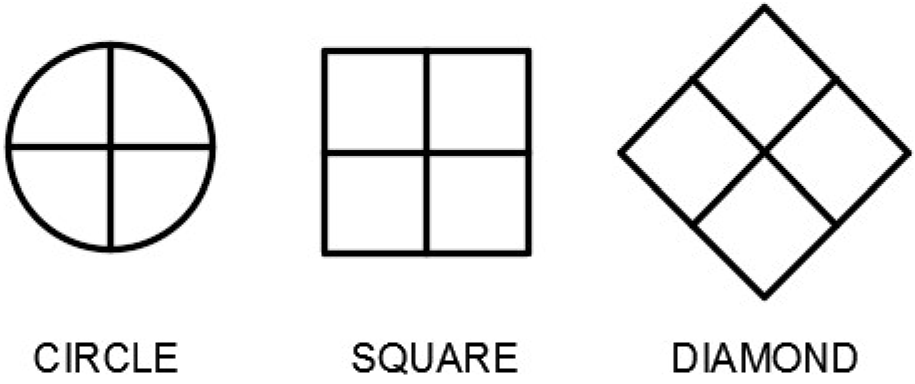


Figure Step 10

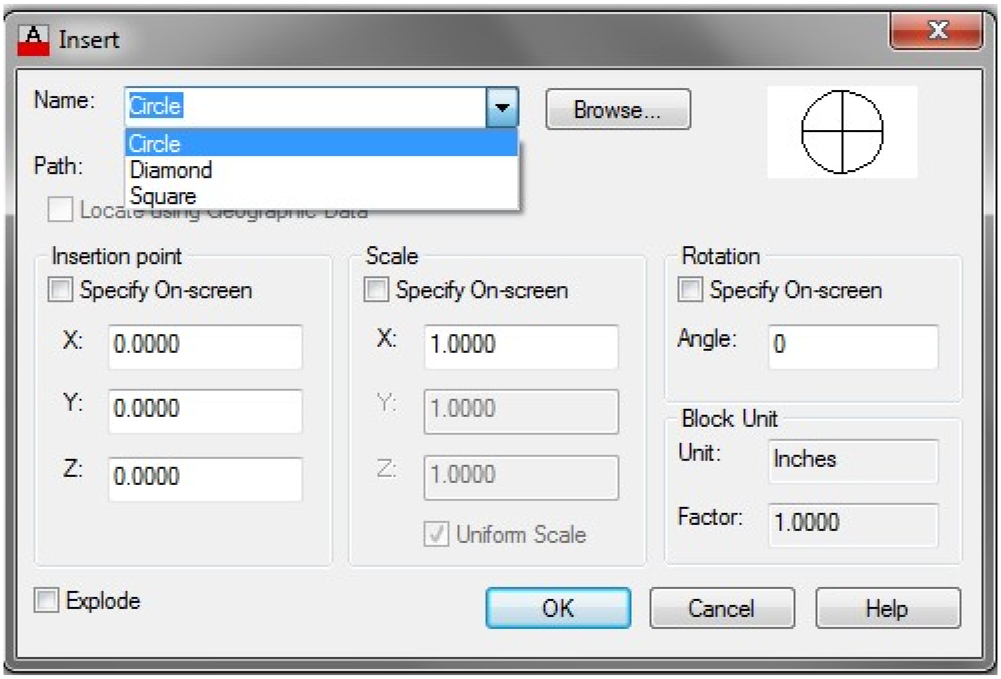
Step 11

Create three blocks, one from each object, and name them: Circle, Square, and Diamond respectfully.

Delete the drawing objects when you create the block for each. After you have created all three blocks, your drawing should appear blank.

Step 12

Check to ensure that the blocks were created by entering the INSERT command. In the list of names, you should only see the three blocks. (Figure Step 12)



Name: Circle

Path:

☐ Locate using Geographic Data

Insertion point
☐ Specify On-screen
X: 0.0000
Y: 0.0000
Z: 0.0000

Scale
☐ Specify On-screen
X: 1.0000
Y: 1.0000
Z: 1.0000
☑ Uniform Scale

Rotation
☐ Specify On-screen
Angle: 0

Block Unit
Unit: Inches
Factor: 1.0000

☐ Explode

Figure Step 12

Step 13

Enter the RENAME command. It will open the Rename dialogue box. Select Blocks in the Named Objects column. The three blocks you created in Step 11 will display in the Items column. (Figure Step 13)

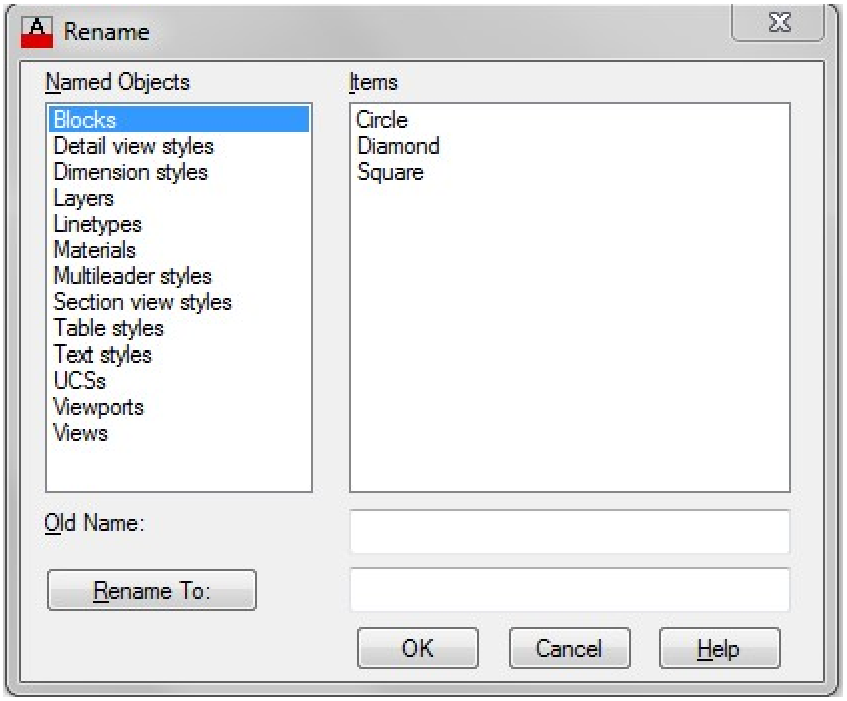
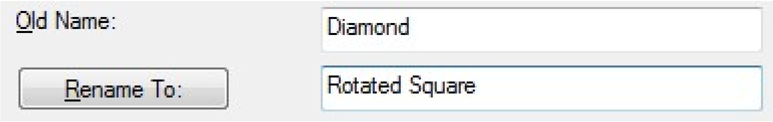

Rename
Named Objects
Blocks
Detail view styles
Dimension styles
Layers
Linetypes
Materials
Multileader styles
Section view styles
Table styles
Text styles
UCSs
Viewports
Views
Items
Circle
Diamond
Square
Old Name:
Rename To:
OK
Cancel
Help

Figure Step 13

Step 14

Click the block: Diamond in the Items column. Note how it now appears in the Old Name: box. Enter the new block name: Rotated Square in the Rename To: box. Click the Rename To: button to execute the name change. The renamed block will now appear in the Items box. (Figure Step 14A and 14B)



Old Name: Diamond

Rename To: Rotated Square

Figure Step 14A

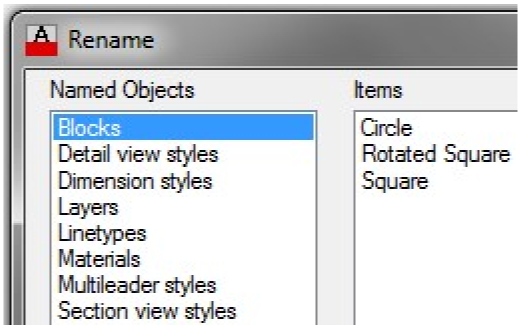

Rename
Named Objects
Blocks
Detail view styles
Dimension styles
Layers
Linetypes
Materials
Multileader styles
Section view styles
Items
Circle
Rotated Square
Square

Figure Step 14B

Step 15

Using the INSERT command, insert the blocks: Square and Rotated Square. The location where you insert them is not important. Your drawing should appear as shown in the figure. (Figure Step 15)

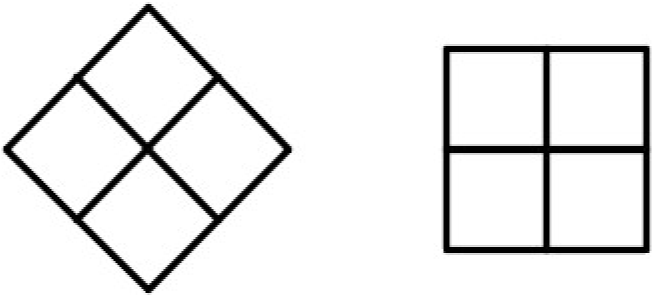
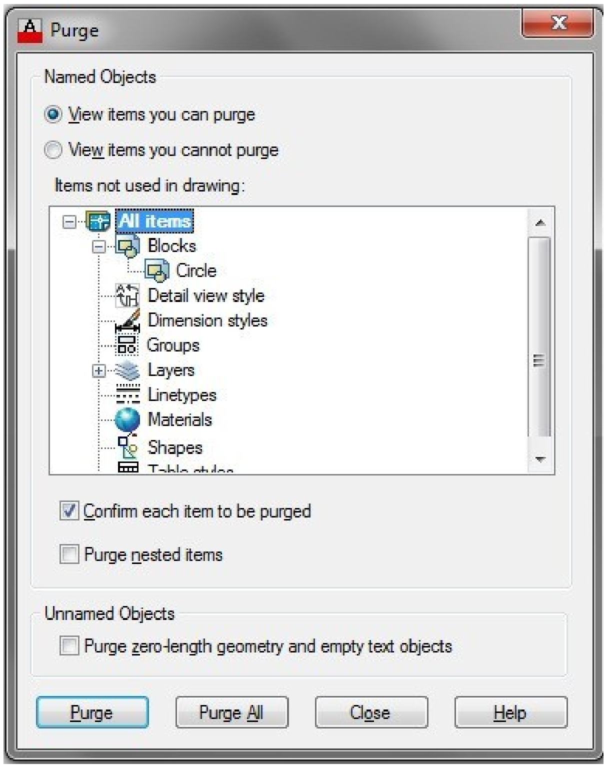


Figure Step 15

> **MUST KNOW:** The PURGE command removes unwanted items from the drawing. In some cases, it is the only way to remove them. If an item is being used in the drawing, it cannot be purged. For example, a layer with an object on it cannot be purged, nor can a block that has been inserted onto the drawing.

Step 16

Enter the PURGE command. In the Purge dialogue box, enable View items you can purge. Expand the item Blocks and the Circle block should display as shown in the figure. (Figure Step 16)



Purge

Named Objects

● View items you can purge

○ View items you cannot purge

Items not used in drawing:

- All items
  - Blocks
    - Circle
  - Detail view style
  - Dimension styles
  - Groups
  - Layers
  - Linetypes
  - Materials
  - Shapes

☑ Confirm each item to be purged

☐ Purge nested items

Unnamed Objects

☐ Purge zero-length geometry and empty text objects

Figure Step 16

Step 17

Using what you learned earlier in this workalong, purge the block: Circle from the drawing.

Step 18

Enter the TIME command as shown below. Command: **TIME**

```
Currenttime: Wednesday, March 01, 2016 2:30:21:015PM

Times for this drawing:

Created: Wednesday, March 01, 2016 1:21:07:562 PM

Last updated: Wednesday, March 01, 2016 1:21:07:562 PM

Total editing time: 0 days 01:09:13:485

Elapsed timer (on): 0 days 00:25:21:156

Next automatic save in: 0 days 00:02:58:672

Command:
```

AUTHOR'S COMMENTS: Your answers will differ from this example.

Step 19

Find the record the following answers in your drawing:

A Total editing time in the drawing.

B Time in Elapsed timer.

Step 20

Save and close the drawing.

Drafting Lesson: Cross Sections – Part 2

Types of Section Views

There are many types of section views that can be drawn. The full, half, aligned and revolved section, as shown in figures below, are the most common ones.

Full Section

A *full section* is the most commonly used cross section. It cuts completely through the object along the cutting plane. See Figure 35-1.

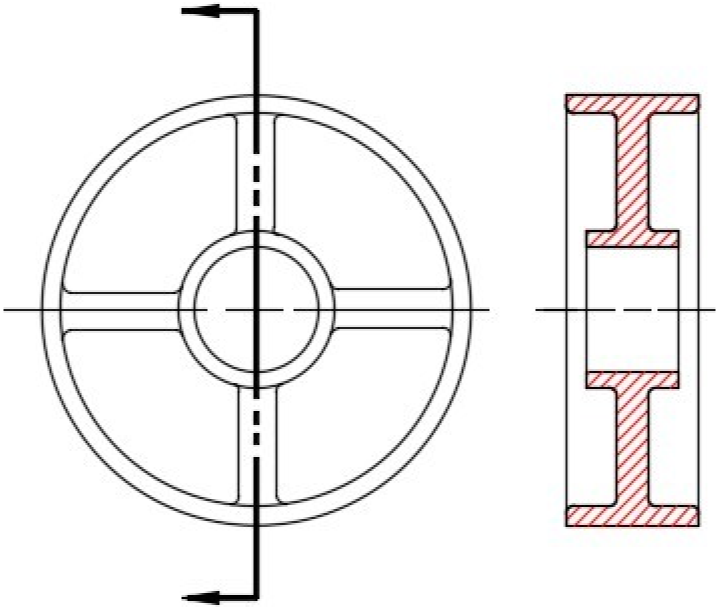


Figure 35-1
Full Section

Half Section

The *half section* only cuts through half of the object. It is used to section a symmetrical object where both halves are identical. See Figure 35-2.

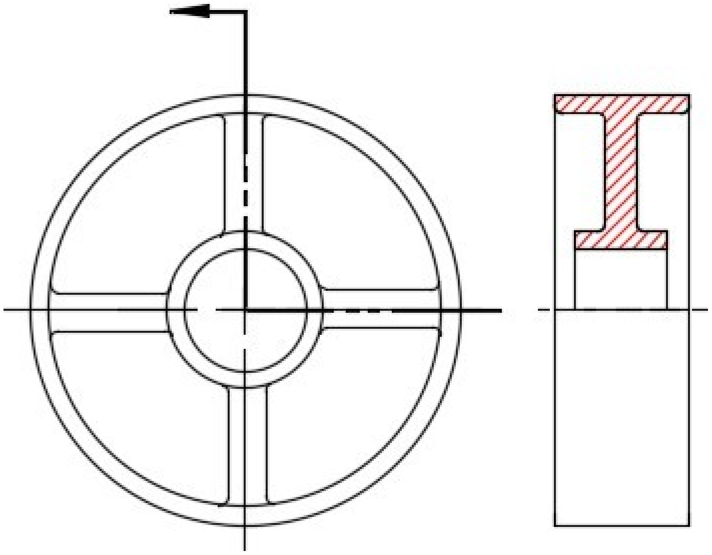


Figure 35-2
Half Section

Aligned Section

An *aligned section* is used to align spokes along the cutting plane on a symmetrical object. Take note how the spokes are not on the cutting plane but are aligned in the section view. See Figure 35-3.

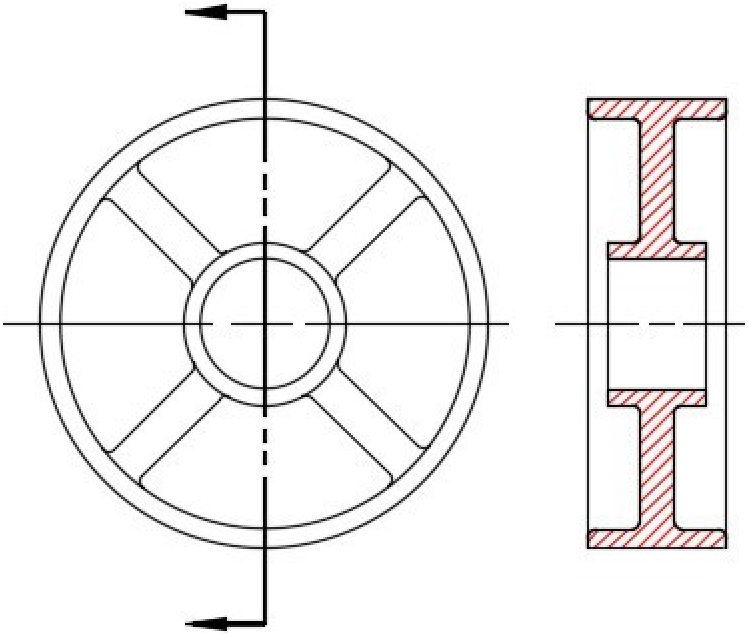


Figure 35-3
Aligned Section

Revolved Section

A *revolved section* is a simple way to show the shape of an object i.e. A spoke. Revolve the section on the object as shown in Figure 35-4.

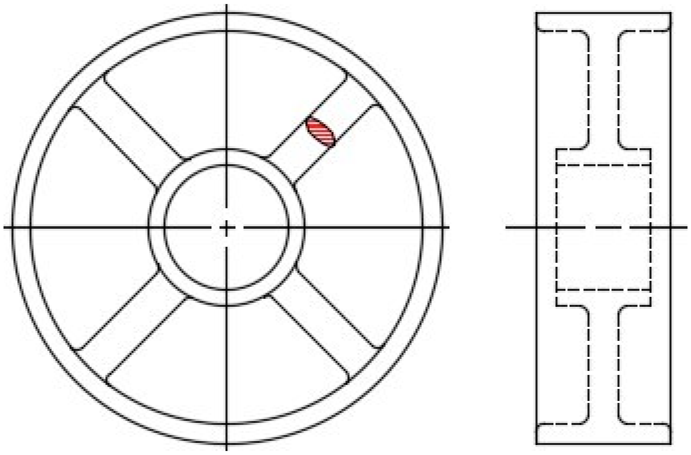
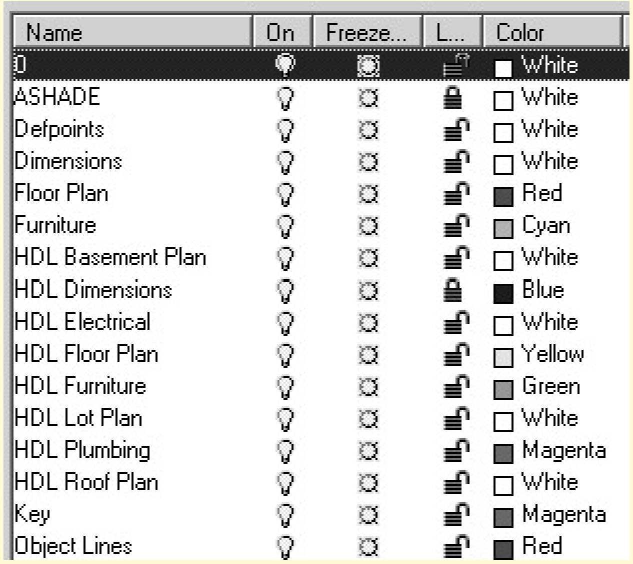
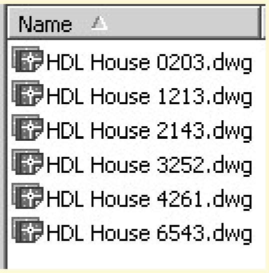


Figure 35-4
Revolved Section

USER TIP: When entering names for drawings, blocks, layers, and other items it is important to name them so they are descriptive and can be found easily. Naming them so they group together or are easy to find in a search is important. They should also be unique to your company and can be listed in alphabetical order. Let’s assume that you are a house designer and the name of your company is House Designer Ltd. For a naming scheme on layers in your drawings you might want to name all your layers starting with the letters HDL. Such as HDL Floor Plan, HDL Furniture, etc. Notice how they appear on the layer list, shown on the right, and how they are grouped together, easy to find and unique to your company. To name the drawings you create for your company. Use a naming scheme such as HDL House 0203, HDL House 3252, HDL House 2143 etc. On the left, is a partial list of how your drawing files would appear in Windows Explorer.

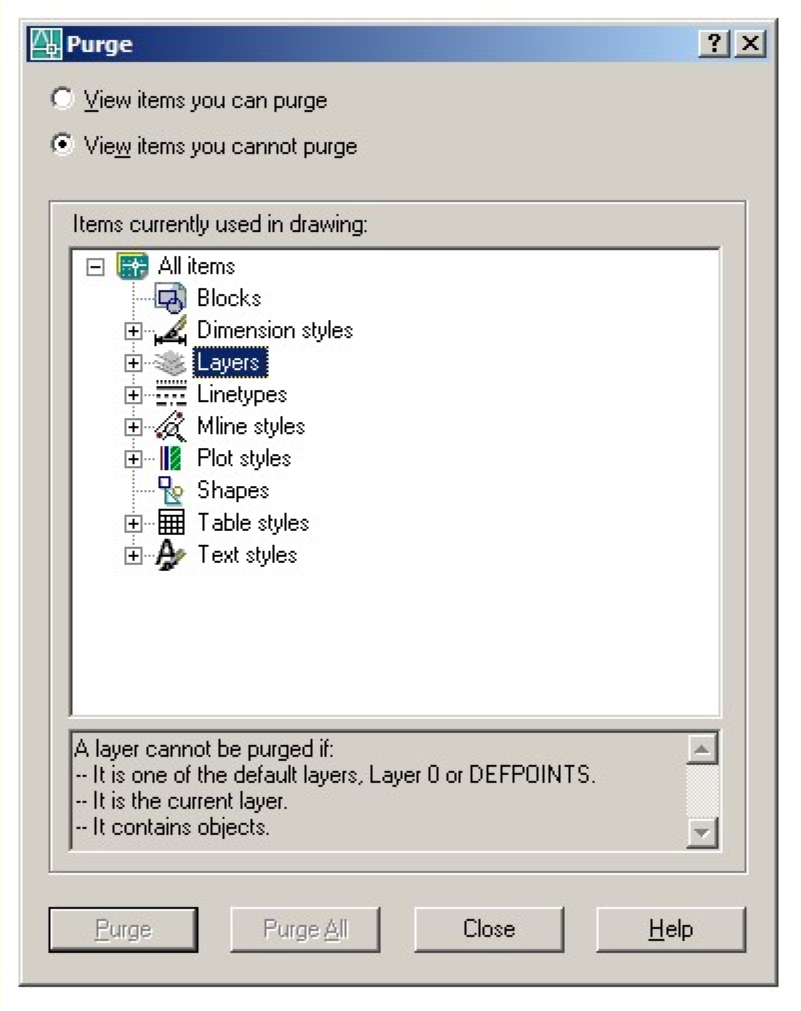


Name	On	Freeze...	L...	Color
0	💡	☼	🔓	☐ White
ASHADE	💡	☼	🔒	☐ White
Defpoints	💡	☼	🔓	☐ White
Dimensions	💡	☼	🔓	☐ White
Floor Plan	💡	☼	🔓	■ Red
Furniture	💡	☼	🔓	■ Cyan
HDL Basement Plan	💡	☼	🔓	☐ White
HDL Dimensions	💡	☼	🔒	■ Blue
HDL Electrical	💡	☼	🔓	☐ White
HDL Floor Plan	💡	☼	🔓	■ Yellow
HDL Furniture	💡	☼	🔓	■ Green
HDL Lot Plan	💡	☼	🔓	☐ White
HDL Plumbing	💡	☼	🔓	■ Magenta
HDL Roof Plan	💡	☼	🔓	☐ White
Key	💡	☼	🔓	■ Magenta
Object Lines	💡	☼	🔓	■ Red



Name ▲
HDL House 0203.dwg
HDL House 1213.dwg
HDL House 2143.dwg
HDL House 3252.dwg
HDL House 4261.dwg
HDL House 6543.dwg

USER TIP: When an item you wish to purge from a drawing is listed in the items you cannot purge, click the item and Purge dialogue box will display tips why the item cannot be purged as you can see below. Using these tips, you can then figure out what changes would have to be made to the drawing before the items can be purged.



Purge

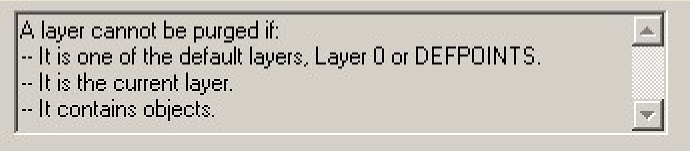
○ View items you can purge

● View items you cannot purge

Items currently used in drawing:

All items
Blocks
Dimension styles
Layers
Linetypes
Mline styles
Plot styles
Shapes
Table styles
Text styles
Purge
Purge All
Close
Help

A layer cannot be purged if:
-- It is one of the default layers, Layer 0 or DEFPOINTS.
-- It is the current layer.
-- It contains objects.



A layer cannot be purged if:
-- It is one of the default layers, Layer 0 or DEFPOINTS.
-- It is the current layer.
-- It contains objects.

Key Principles

Key Principles in Module 34

1. The PURGE command removes unwanted items from the drawing. In some cases, it is the only way to remove them. If an item is being used, it cannot be purged.

Lab Exercise 34-1

Time allowed: 60 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 34-1	2D Layout Metric	Millimeters

Step 1

On layers: Object and Hidden, draw the Front and Right Side view as shown in the figure. (Figure Step 1A and 1B)

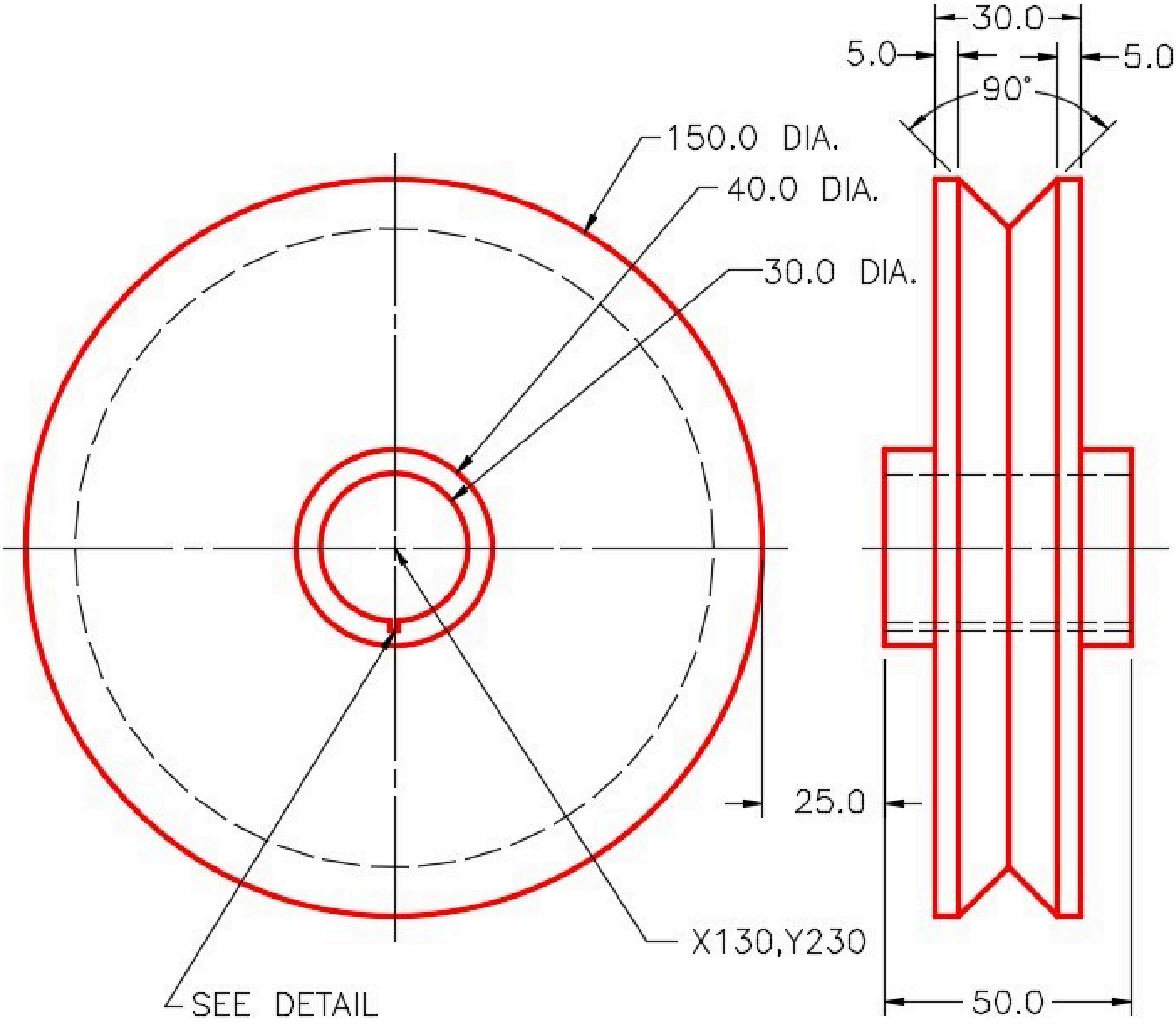

30.0
5.0
5.0
90°
150.0 DIA.
40.0 DIA.
30.0 DIA.
25.0
X130,Y230
SEE DETAIL
50.0

Figure Step 1A

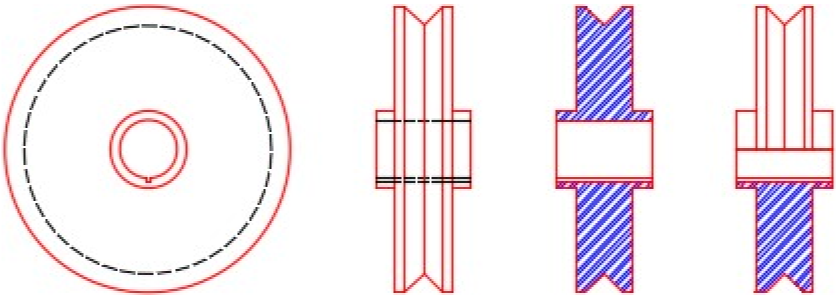


Figure Step 1B
Completed Drawing

Step 2

On layers: Object and Hatch, draw a full section and a half section. Use hatching pattern: ANSI32. Select an appropriate scale to match the figure the best that you can. (Figure Step 2A and 2B)

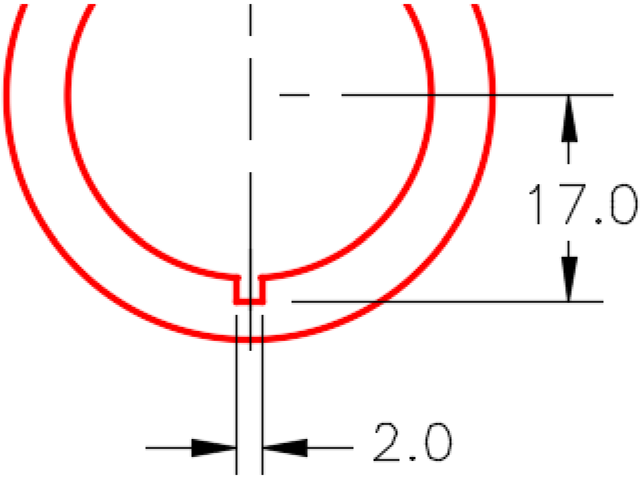


Figure Step 2A
Detail of Key

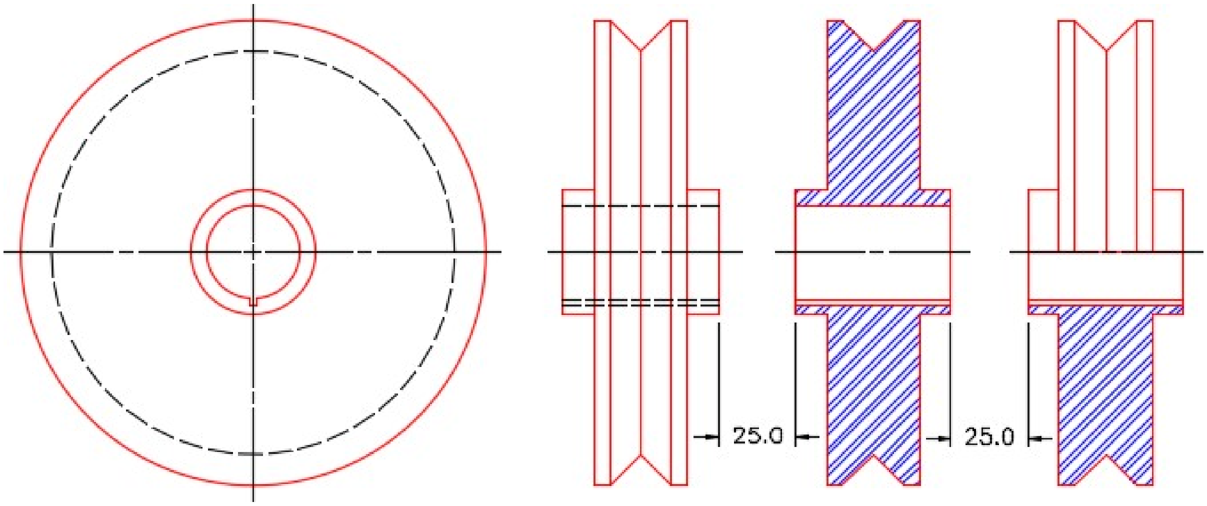


Figure Step 2B

Hint 1

To help you visualize the object that you are drawing. (Figure Step 1A and 1B)

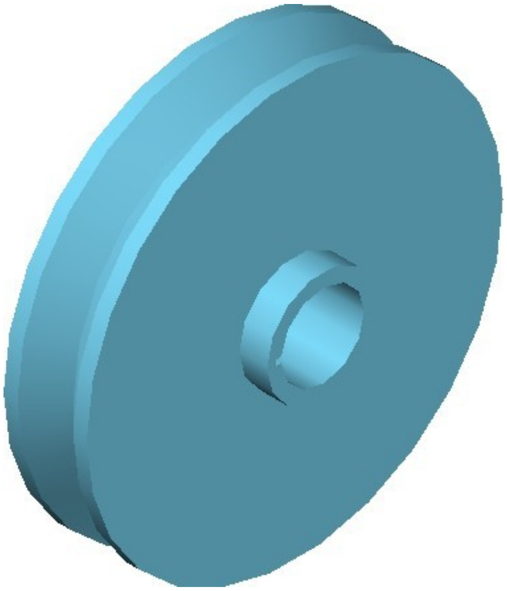


Figure Hint 1A
Solid Model

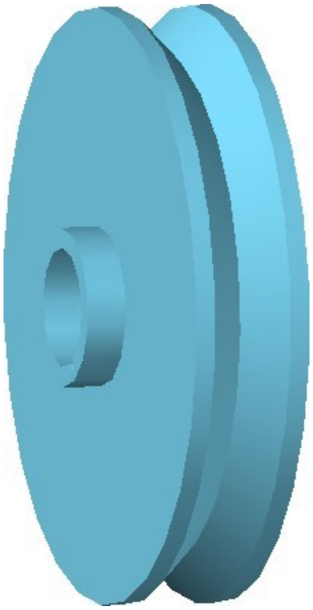


Figure Hint 1B
Solid Model

Lab Exercise 34-2

Time allowed: 30 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 34-2	N/A	Feet/Inches

Step 1

Open the drawing: AutoCAD 2D Lab 33-1. (Figure Step 1)

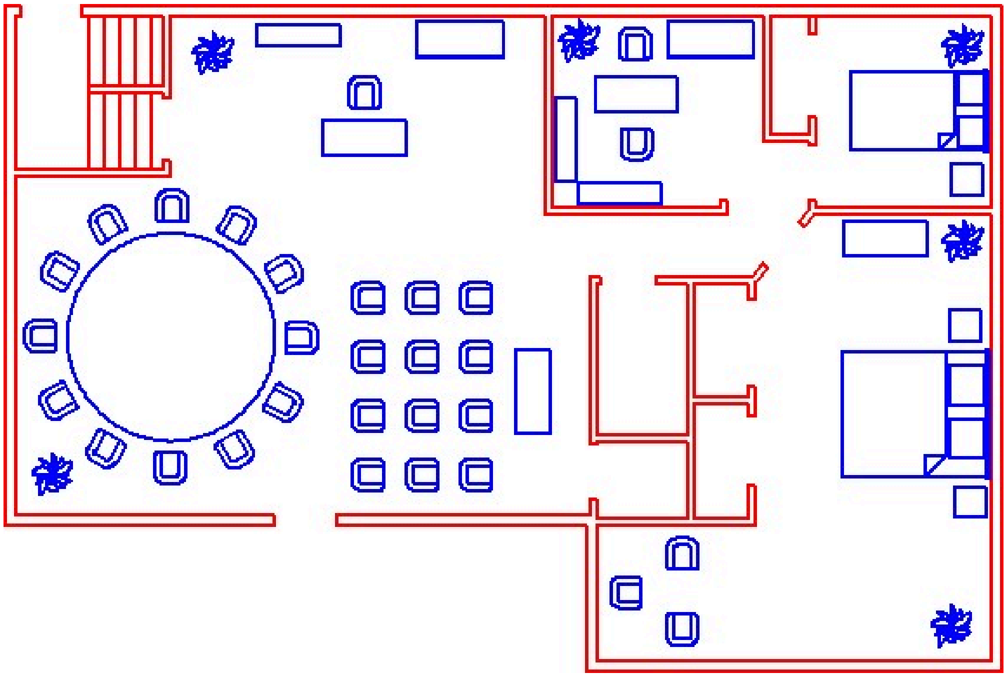


Figure Step 1

Step 2

Using the SAVEAS command, save the drawing with the name: AutoCAD 2D Lab 34-2.

Step 3

Reset the elapsed timer to zero.

Step 4

Delete all existing blocks named: File Cabinet that are on the drawing. (Figure Step 4)

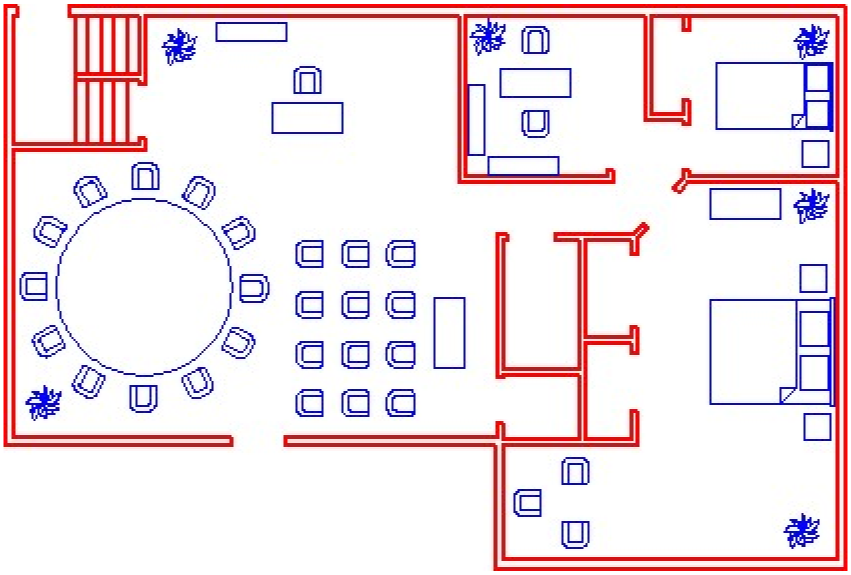


Figure Step 4

Step 5

Purge the block: File Cabinet from your drawing.

Step 6

Rename the following blocks:

A From: Desk to 48X28 Desk

B From: Conference Table to 10 Ft. Diameter Table

Step 7

On layer: 0, draw the plant shown in the figure. (Figure Step 7A and 7B)

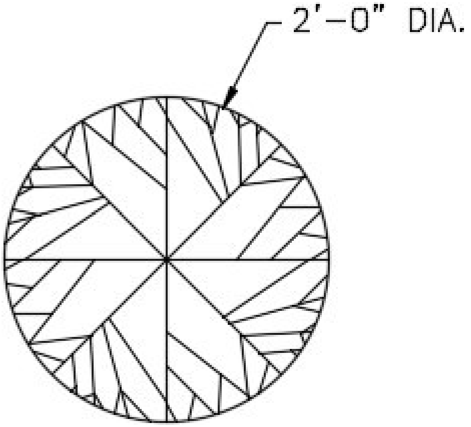


Figure Step 7A

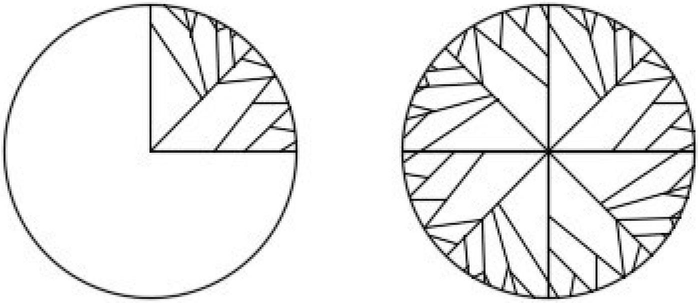


Figure Step 7B

AUTHOR'S COMMENTS: Draw a circle and one-quarter of the inside. Then array it. Draw it by eye, using lines.

Step 8

Redefine the block in the drawing named: Plant with the plant you drew in Step 7. All existing plant blocks should change to the new plant block. (Figure Step 8)

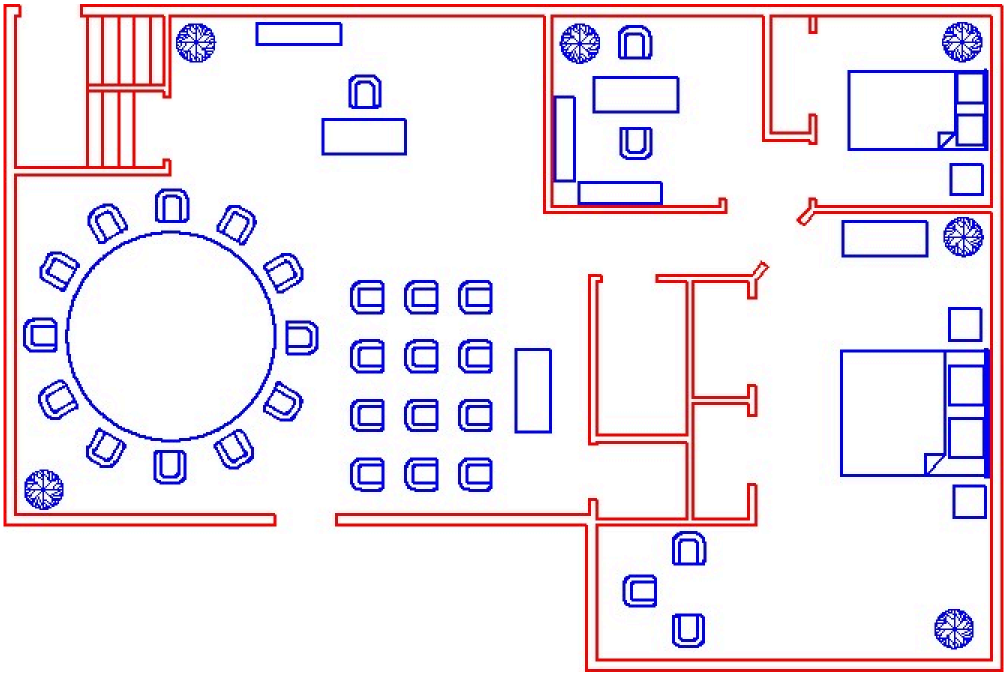


Figure Step 8

Step 9

Using the elapsed timer, record the amount of time you took to do this lab exercise.

Step 10

Save and close the drawing.

Module 35: Reference Files

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe reference files and explain how they are used in an AutoCAD drawing.
2. Apply the XREF command.

Reference Files

An *external reference file* is a drawing file linked to the current drawing. It acts as one object similar to a block and appears to be part of the current drawing. Usually called a *reference file* or a *xref*, it is quite different from a block. A reference file updates itself whenever the drawing it is linked to is opened. It can also be updated manually by the CAD operator. If the drawing that is attached as an xref is changed, the changes will appear when the drawing it is attached to is opened or reloaded. Since it is only linked to the drawing, it does not add a significant amount of data to the drawing's database.

The XREF command is used to manage the reference files that are linked to the current drawing. You can attach more than one reference file at the same time. All objects in the reference file must be in Model space. Reference files can be scaled or rotated when they are attached with the XREF command or after they are attached with the SCALE and/or ROTATE commands. You can snap to the object snap modes of objects in the reference files. A reference file can be edited in the current drawing with the changes going back to the original drawing file. This is done with the REFEDIT command.

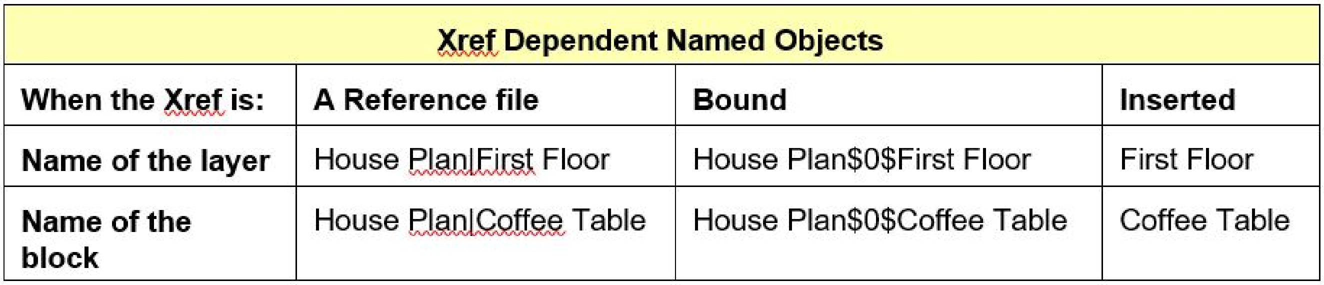
Xref Dependant Named Objects

Xref dependent named objects are the layers, linetypes, dimension styles, blocks, and text styles of the reference file. They are added to the drawing they are attached to as non-graphical object information. The names are changed so that no two names are the same. Table 35-1 shows how the non-graphical names are dealt with by AutoCAD.

This table demonstrates how AutoCAD names the xref dependent named objects for layers, linetypes, dimension styles, blocks, and text styles. In this table, the name of the attached xref file drawing is House Plan and a layer in that drawing is First Floor and a block named Coffee Table. For a reference file, the name of the non-graphical object is changed to the name ‘ drawing name|object ‘. If it is bound to the drawing, it is renamed ‘ name$n$objectname ‘. The ‘ n ‘ starts with zero and increases by one if the same name is encountered.

Binding a Reference File

A reference file can be bound to the current drawing it is attached to with the XREF command. When a reference drawing is bound to the drawing, the XREF command converts it into a block. The xref dependent named objects are renamed according to Table 35-1. A block can be exploded to convert the objects into single objects and they become part of the drawing. When a reference file is bound, it can either be bound or inserted. Inserting a reference file converts it into a block but it strips the named non-graphical objects back to their original names. See Table 35-1



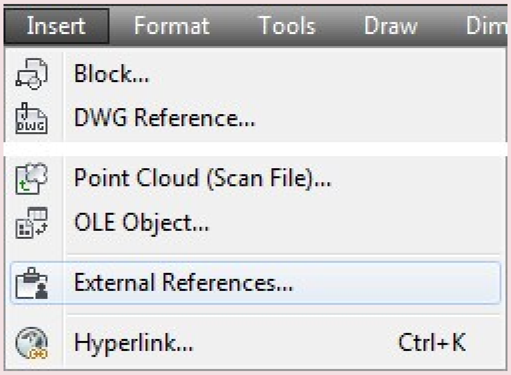
Xref Dependent Named Objects			
When the Xref is:	**A Reference file**	**Bound**	**Inserted**
Name of the layer	House Plan|First Floor	House Plan$0$First Floor	First Floor
Name of the block	House Plan|Coffee Table	House Plan$0$Coffee Table	Coffee Table

Table 35-1
Naming of Xref Dependent Named Objects

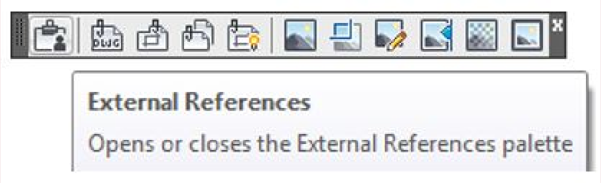
AutoCAD Command: XREF

The XREF command is used to manage reference files in the current drawing.

Shortcut: **XREF** or **XR**



Insert Pull-down



External References
Opens or closes the External References palette

Reference Toolbar

WORKALONG: Using Reference Files

Step 1

Open drawing: AutoCAD 2D Lab 32-3.

Step 2

Open the Units dialogue box and set the Insertion scale to millimeters.

Step 3

Using the SAVEAS command, save and name the drawing four times to create four identical drawings with the following names:

AutoCAD 2D Workalong 35-1A

AutoCAD 2D Workalong 35-1B

AutoCAD 2D Workalong 35-1C

AutoCAD 2D Workalong 35-1D

Step 4

Close drawing: AutoCAD 2D Lab 32-3.

Step 5

Open drawing: AutoCAD 2D Workalong 35-1A. (Figure Step 5)

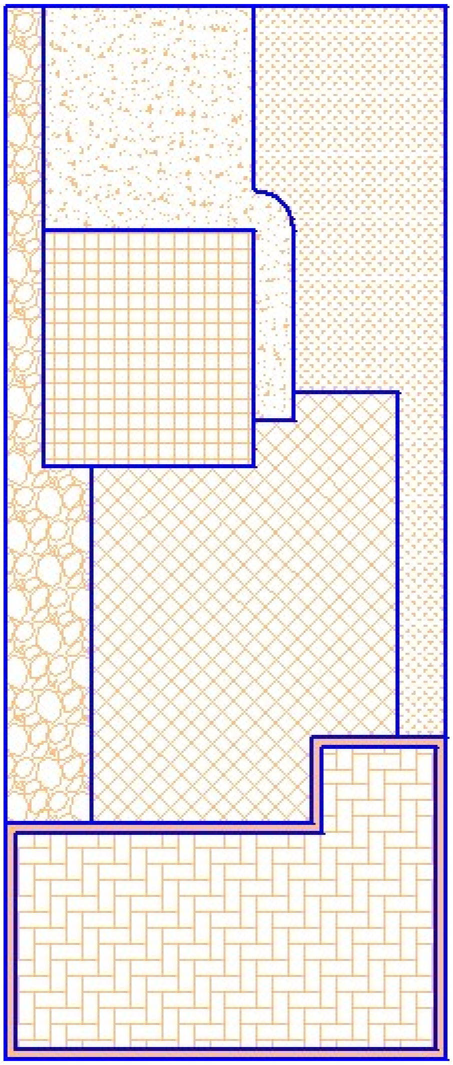


Figure Step 5

Step 6

Ensure that all layers are on and thawed.

Step 7

Turn layer: Lot Lines off. Enter the ZOOM Extents command. Erase all visible objects on the drawing. The drawing should appear blank.

Step 8

Turn layer: Lot Lines on. Your drawing should match the figure. (Figure Step 8)

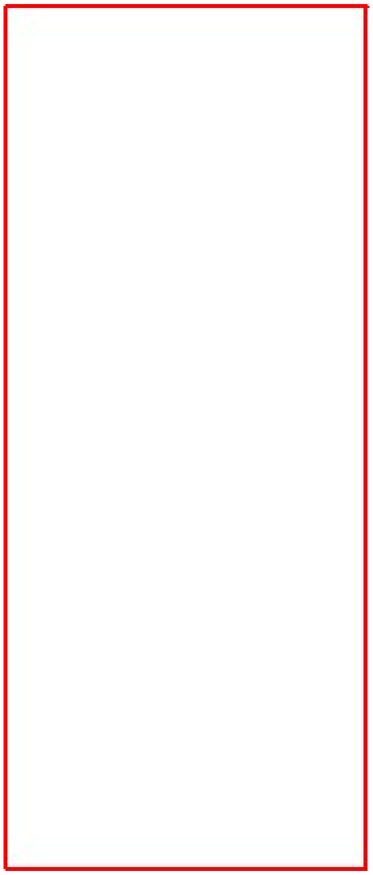


Figure Step 8

Step 9

Purge all layers and blocks that can be purged from the drawing.

Step 10

Save and close the drawing.

Step 11

Open drawing: AutoCAD 2D Workalong 35-1B.

Step 12

Ensure that all layers are on and thawed.

Step 13

Turn layer: Object off. Enter the ZOOM Extents command. Erase all visible objects on the drawing. The drawing should appear blank.

Step 14

Turn layer: Object on. The drawing should match the figure. (Figure Step 14)

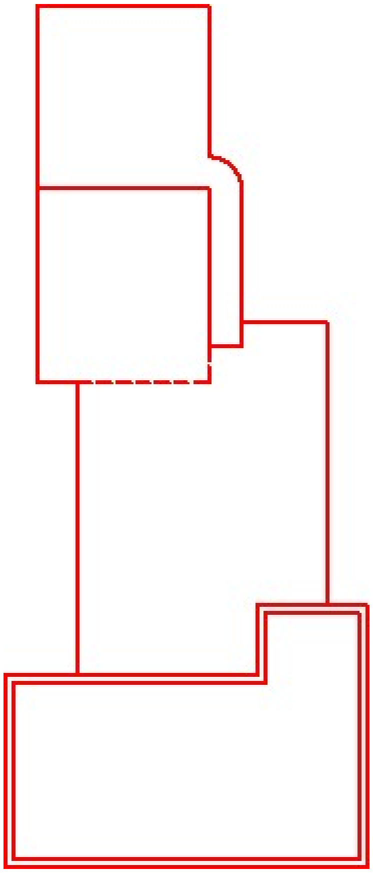


Figure Step 14

Step 15

Purge all layers and blocks that can be purged from the drawing.

Step 16

Save and close the drawing.

Step 17

Open drawing: AutoCAD 2D Workalong 35-1C.

Step 18

Ensure that all layers are on and thawed.

Step 19

Turn layer: Boundary off. Enter the ZOOM Extents command. Erase all visible objects on the drawing. The drawing should appear blank.

Step 20

Turn layer: Boundary on. The drawing should match the figure. (Figure Step 20)

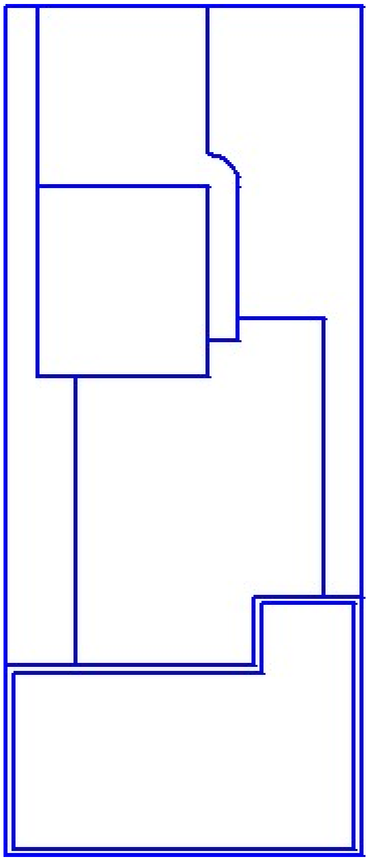


Figure Step 20

Step 21

Purge all layers and blocks that can be purged from the drawing.

Step 22

Save and close the drawing.

Step 23

Open drawing: AutoCAD 2D Workalong 35-1D.

Step 24

Ensure that all layers are on and thawed.

Step 25

Turn layer: Lot Hatch off. Enter the ZOOM Extents command. Erase all visible objects in the drawing. The drawing should appear blank.

Step 26

Turn layer: Lot Hatch on. The drawing should match the figure. (Figure Step 26)

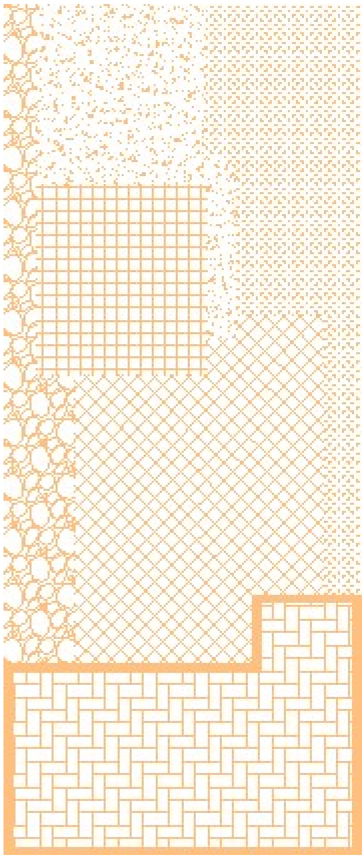


Figure Step 26

Step 27

Save and close the drawing.

Step 28

Start a new drawing using the template: 2D Layout Metric.

Step 29

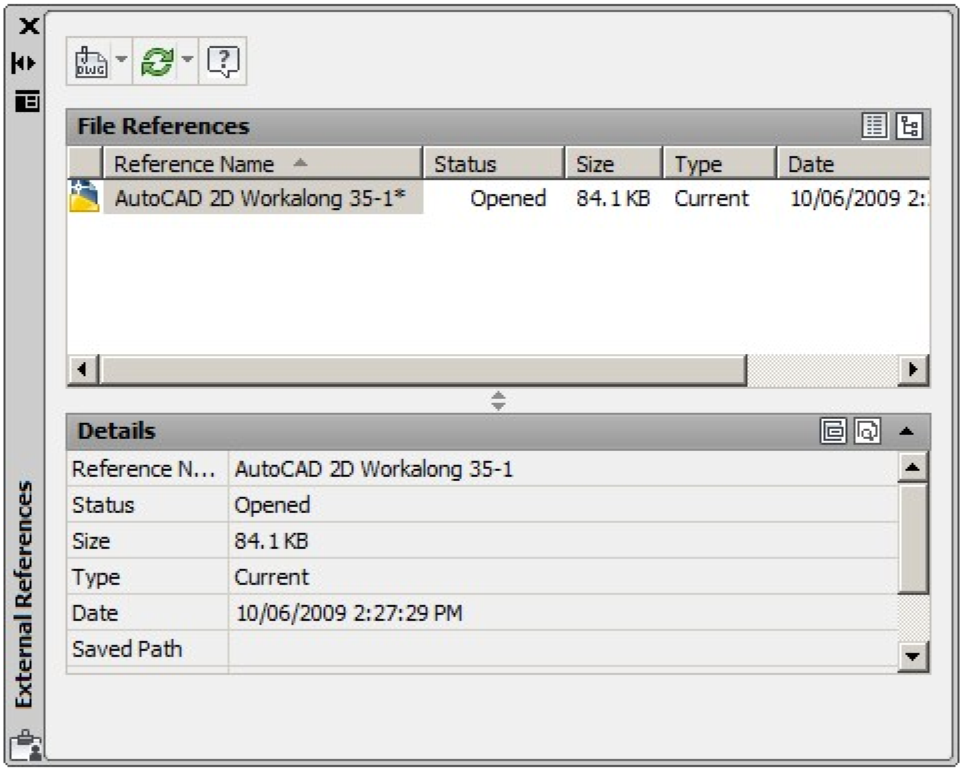
Save and name the drawing: AutoCAD 2D Workalong 35-1.

Step 30

Open the Units dialogue box and set the Insertion scale to millimeters.

Step 31

Enter the XREF command to open the External References window. (Figure Step 31)



External References

File References

Reference Name	Status	Size	Type	Date
AutoCAD 2D Workalong 35-1*	Opened	84.1 KB	Current	10/06/2009 2:

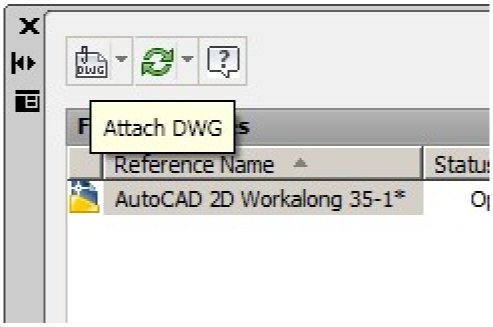
Details

Reference N...	AutoCAD 2D Workalong 35-1
Status	Opened
Size	84.1 KB
Type	Current
Date	10/06/2009 2:27:29 PM
Saved Path	

Figure Step 31

Step 32

Click the Attach DWG icon. (Figure Step 32)

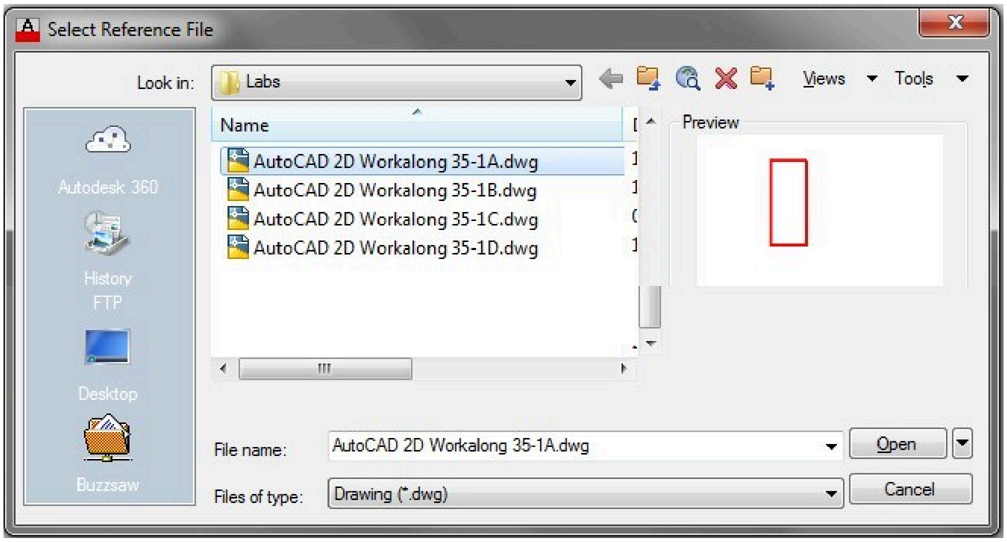


Attach DWG
F s
Reference Name
Statu
AutoCAD 2D Workalong 35-1*
O

Figure Step 32

Step 33

In the Select Reference File dialogue box, select the drawing: AutoCAD 2D Workalong 35-1A.dwg. (Figure Step 33)


Select Reference File
Look in:
Labs
Views
Tools
Name
Preview
Autodesk 360
History
FTP
Desktop
Buzzsaw
Open
Cancel

AutoCAD 2D Workalong 35-1A.dwg
AutoCAD 2D Workalong 35-1B.dwg
AutoCAD 2D Workalong 35-1C.dwg
AutoCAD 2D Workalong 35-1D.dwg

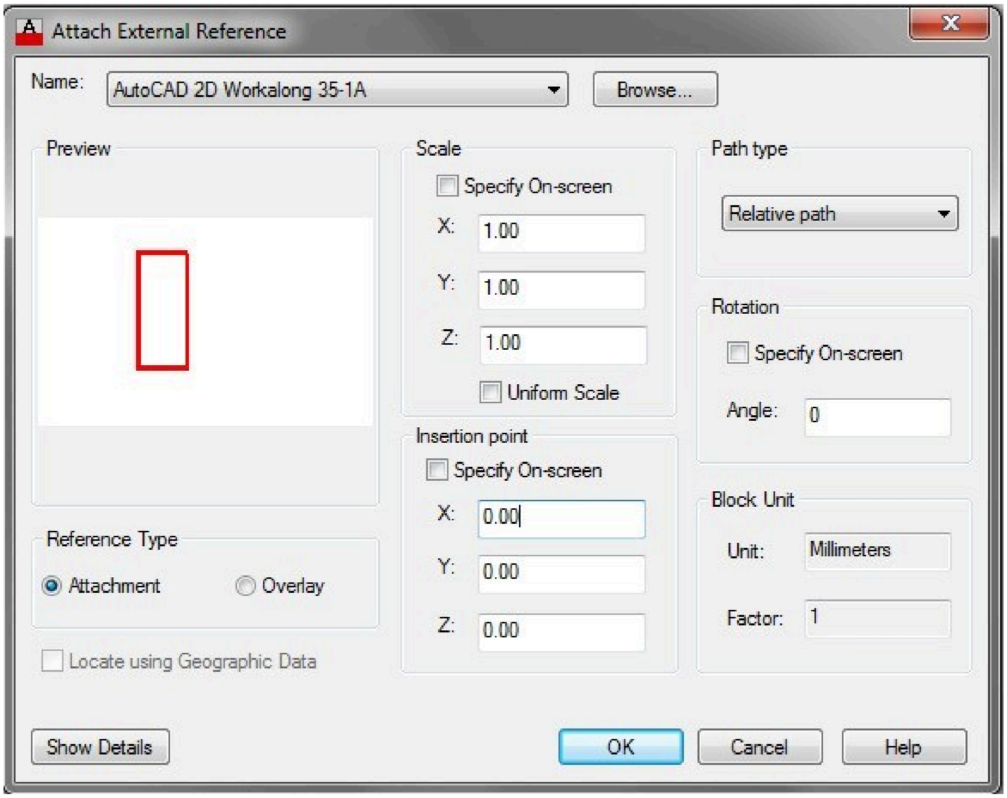
File name: AutoCAD 2D Workalong 35-1A.dwg

Files of type: Drawing (*.dwg)

Figure Step 33

Step 34

In the Attach External Reference dialogue box, set your dialogue box to match the figure. (Figure Step 34)



Attach External Reference

Name: AutoCAD 2D Workalong 35-1A

Preview

Reference Type

● Attachment ○ Overlay

☐ Locate using Geographic Data

Scale

☐ Specify On-screen

X: 1.00

Y: 1.00

Z: 1.00

☐ Uniform Scale

Insertion point

☐ Specify On-screen

X: 0.00

Y: 0.00

Z: 0.00

Path type

Relative path

Rotation

☐ Specify On-screen

Angle: 0

Block Unit

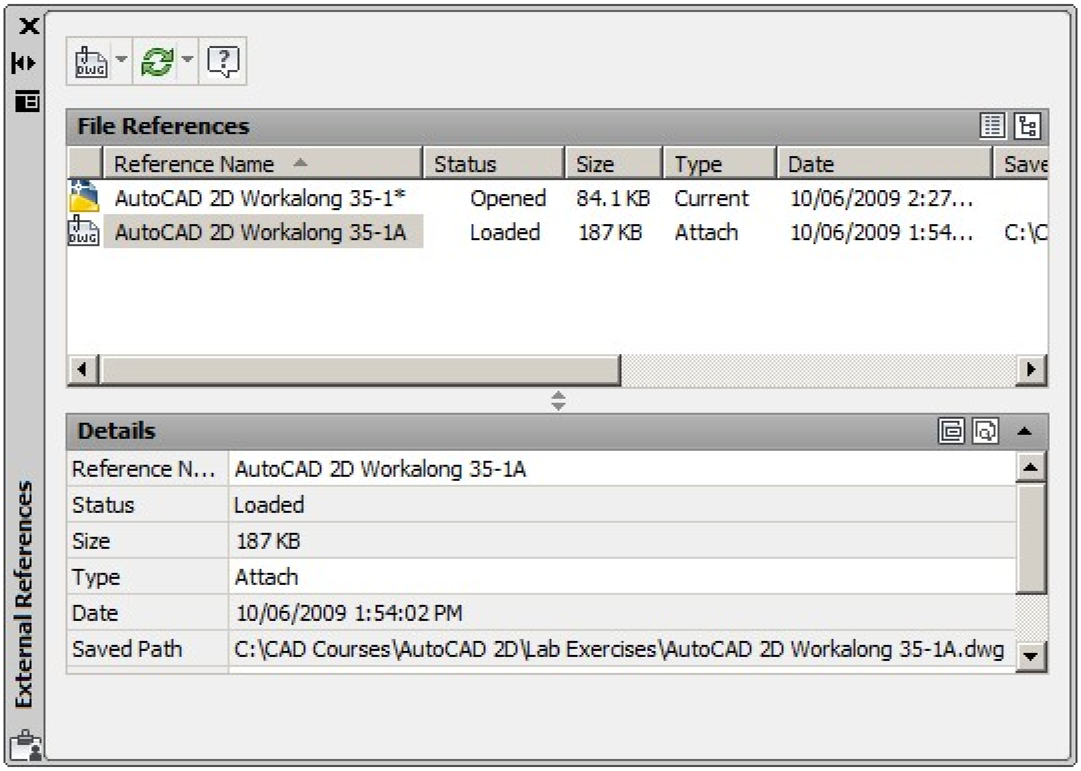
Unit: Millimeters

Factor: 1

Figure Step 34

Step 35

In the External References window, you should see the drawing you just attached and the drawing you currently have open. (Figure Step 35)



Reference Name	Status	Size	Type	Date	Save
AutoCAD 2D Workalong 35-1*	Opened	84.1 KB	Current	10/06/2009 2:27...	
AutoCAD 2D Workalong 35-1A	Loaded	187 KB	Attach	10/06/2009 1:54...	C:\C

Reference N...	AutoCAD 2D Workalong 35-1A
Status	Loaded
Size	187 KB
Type	Attach
Date	10/06/2009 1:54:02 PM
Saved Path	C:\CAD Courses\AutoCAD 2D\Lab Exercises\AutoCAD 2D Workalong 35-1A.dwg

Figure Step 35

Step 36

Enter the ZOOM Extents command. (Figure Step 36)

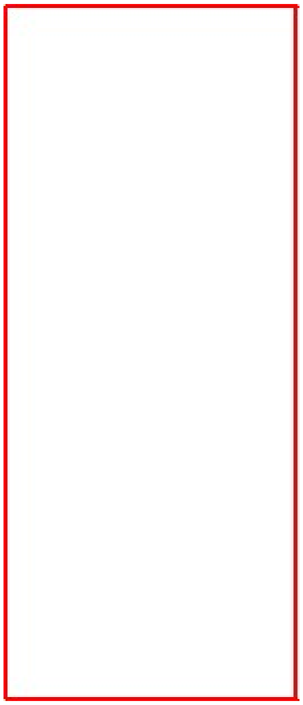


Figure Step 36

Step 37

Using what you just learned, attach the drawing: AutoCAD 2D Workalong 35-1B as a reference file. (Figure Step 37)

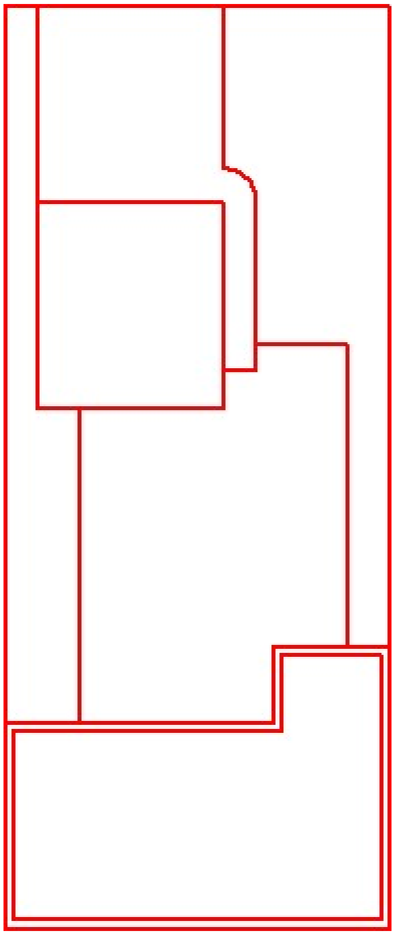


Figure Step 37

Step 38

Attach the drawing: AutoCAD 2D Workalong 35-1C as a reference file. (Figure Step 38)

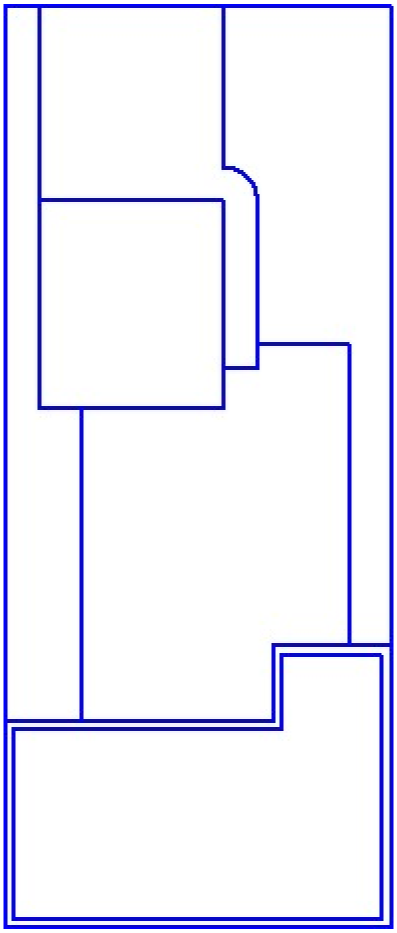


Figure Step 38

Step 39

Attach the drawing: AutoCAD 2D Workalong 35-1D as a reference file. (Figure Step 39)

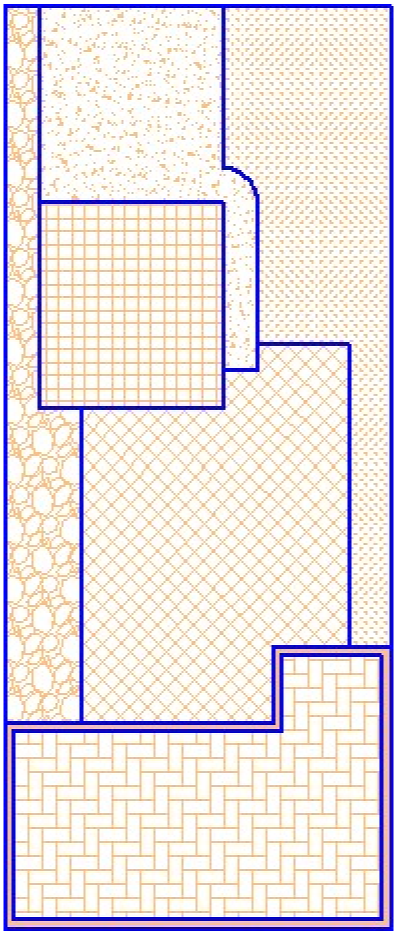
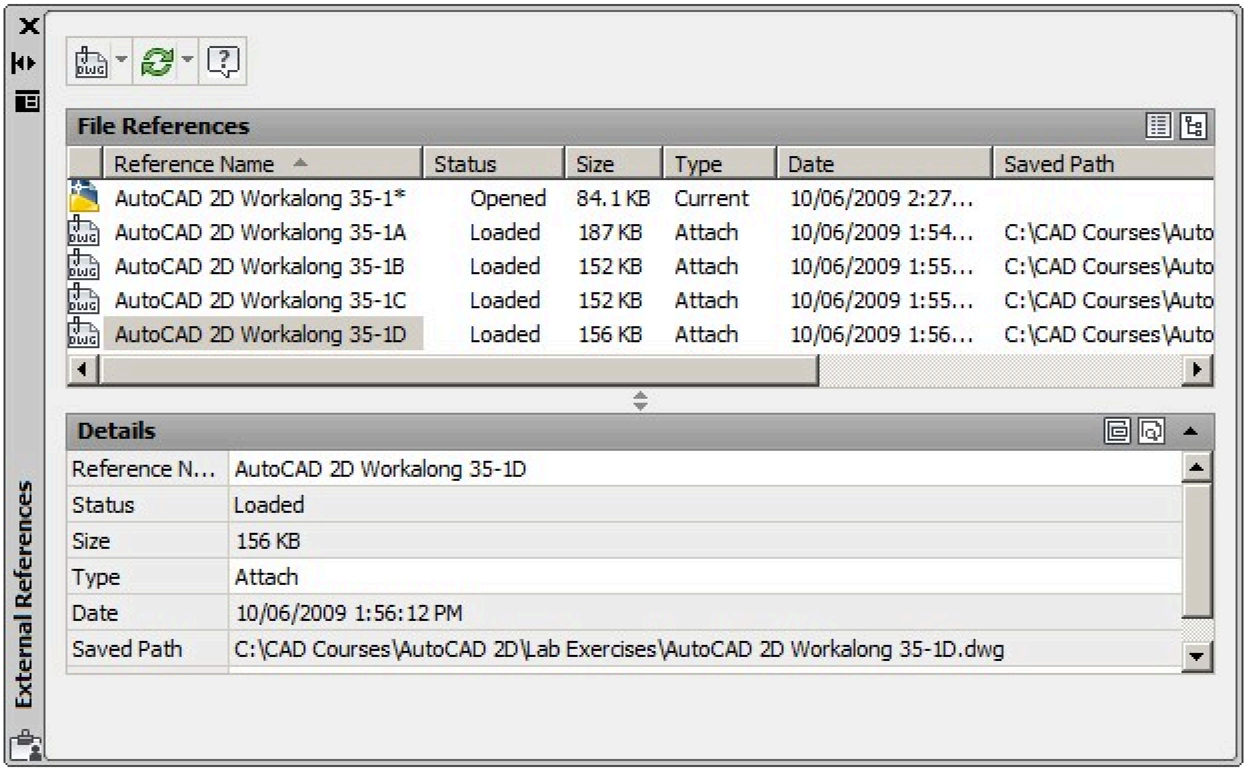


Figure Step 39

Step 40

The External Reference window should match the figure. (Figure Step 40)



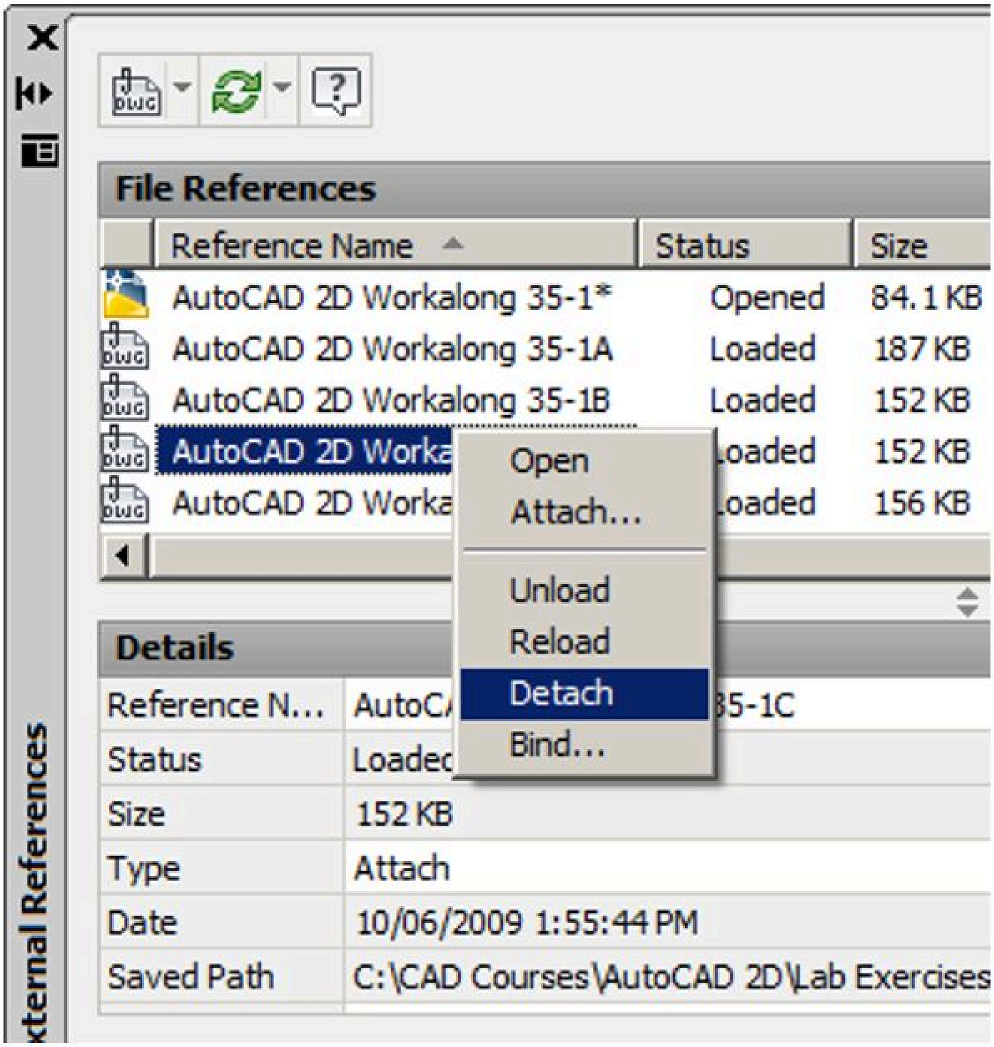
Reference Name	Status	Size	Type	Date	Saved Path
AutoCAD 2D Workalong 35-1*	Opened	84.1 KB	Current	10/06/2009 2:27...	
AutoCAD 2D Workalong 35-1A	Loaded	187 KB	Attach	10/06/2009 1:54...	C:\CAD Courses\Auto
AutoCAD 2D Workalong 35-1B	Loaded	152 KB	Attach	10/06/2009 1:55...	C:\CAD Courses\Auto
AutoCAD 2D Workalong 35-1C	Loaded	152 KB	Attach	10/06/2009 1:55...	C:\CAD Courses\Auto
AutoCAD 2D Workalong 35-1D	Loaded	156 KB	Attach	10/06/2009 1:56...	C:\CAD Courses\Auto

Reference N...	AutoCAD 2D Workalong 35-1D
Status	Loaded
Size	156 KB
Type	Attach
Date	10/06/2009 1:56:12 PM
Saved Path	C:\CAD Courses\AutoCAD 2D\Lab Exercises\AutoCAD 2D Workalong 35-1D.dwg

Figure Step 40

Step 41

In the External Reference window, select the drawing: AutoCAD 2D Workalong 35-1C. Right click the mouse. In the Right-click menu, click Detach. (Figure Step 41A and 41B)



External References
File References

Reference Name	Status	Size
AutoCAD 2D Workalong 35-1*	Opened	84.1 KB
AutoCAD 2D Workalong 35-1A	Loaded	187 KB
AutoCAD 2D Workalong 35-1B	Loaded	152 KB
AutoCAD 2D Worka	Loaded	152 KB
AutoCAD 2D Worka	Loaded	156 KB

Open
Attach...
Unload
Reload
Detach
Bind...
Details

Reference N...	AutoC 35-1C
Status	Loadec
Size	152 KB
Type	Attach
Date	10/06/2009 1:55:44 PM
Saved Path	C:\CAD Courses\AutoCAD 2D\Lab Exercises

Figure Step 41A

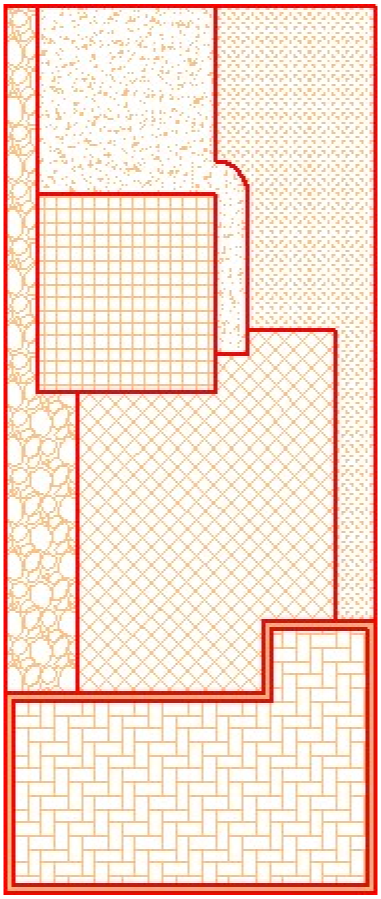
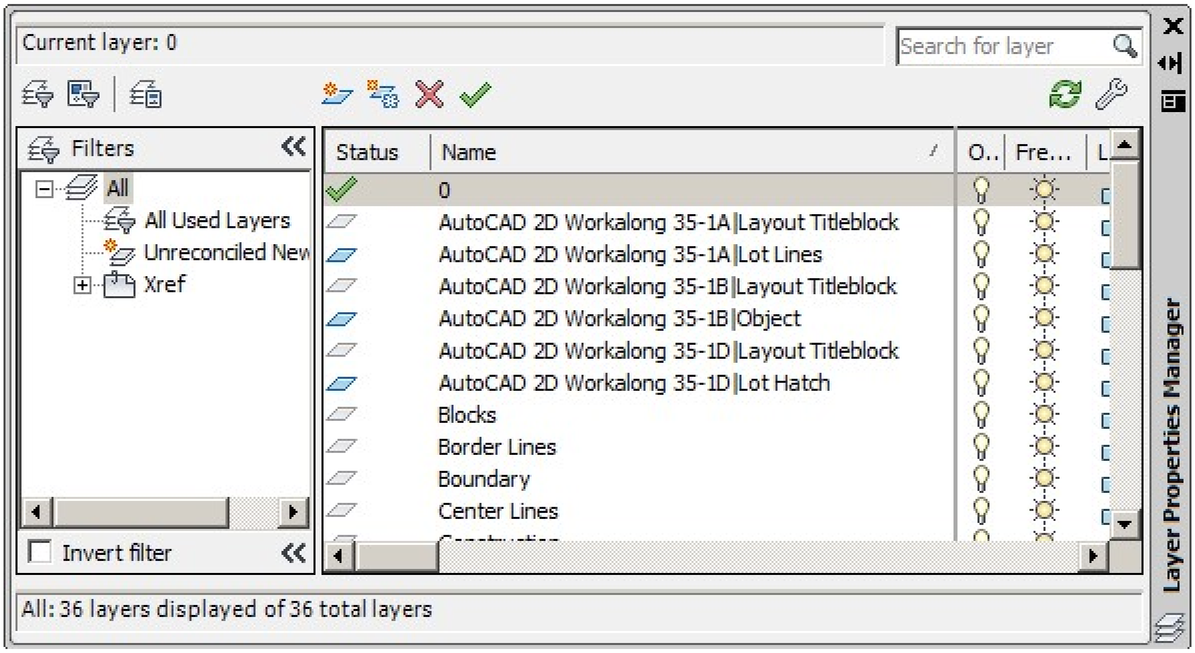


Figure Step 41B

AUTHOR’S COMMENTS: When a drawing is detached, it is removed as a reference file.

Step 42

Open the Layer Properties Manager window. Take note of the layer names of the layers of the attached referenced files. (Figure Step 42)


Search for layer
Filters
All
All Used Layers
Unreconciled New
Xref
Invert filter
Layer Properties Manager

Current layer: 0

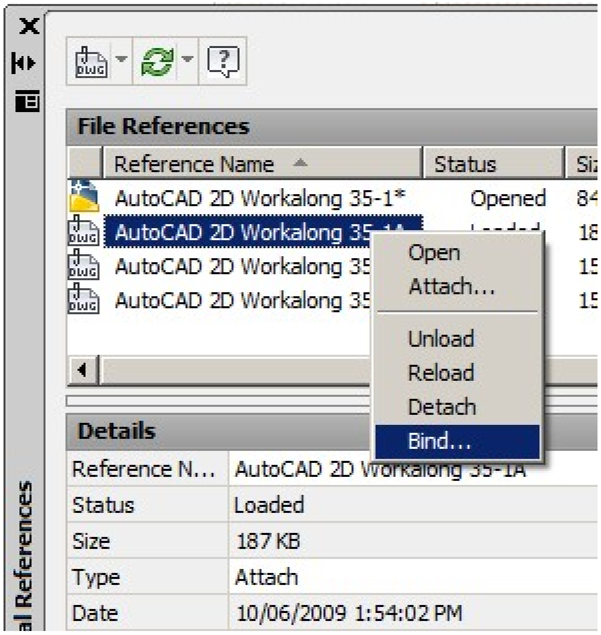
Status	Name	O..	Fre...
✓	0		
	AutoCAD 2D Workalong 35-1A|Layout Titleblock		
	AutoCAD 2D Workalong 35-1A|Lot Lines		
	AutoCAD 2D Workalong 35-1B|Layout Titleblock		
	AutoCAD 2D Workalong 35-1B|Object		
	AutoCAD 2D Workalong 35-1D|Layout Titleblock		
	AutoCAD 2D Workalong 35-1D|Lot Hatch		
	Blocks		
	Border Lines		
	Boundary		
	Center Lines		

All: 36 layers displayed of 36 total layers

Figure Step 42

Step 43

In the External Reference window, select the drawing: AutoCAD 2D Workalong 35-1A. Right click the mouse. In the Right-click menu, click Bind. (Figure Step 43)


File References
Open
Attach...
Unload
Reload
Detach
Bind...
Details
al References

Reference Name	Status	Si
AutoCAD 2D Workalong 35-1*	Opened	84
AutoCAD 2D Workalong 35-1A	Loaded	18
AutoCAD 2D Workalong 35		15
AutoCAD 2D Workalong 35		15

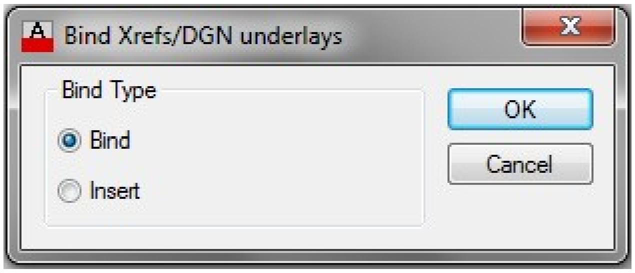
Details	
Reference N...	AutoCAD 2D Workalong 35-1A
Status	Loaded
Size	187 KB
Type	Attach
Date	10/06/2009 1:54:02 PM

Figure Step 43

AUTHOR'S COMMENTS: When a drawing is bound into the current drawing, it is inserted as a block.

Step 44

In the Bind Xrefs dialogue box, enable Bind and click OK. (Figure Step 44)



Bind Xrefs/DGN underlays

Bind Type

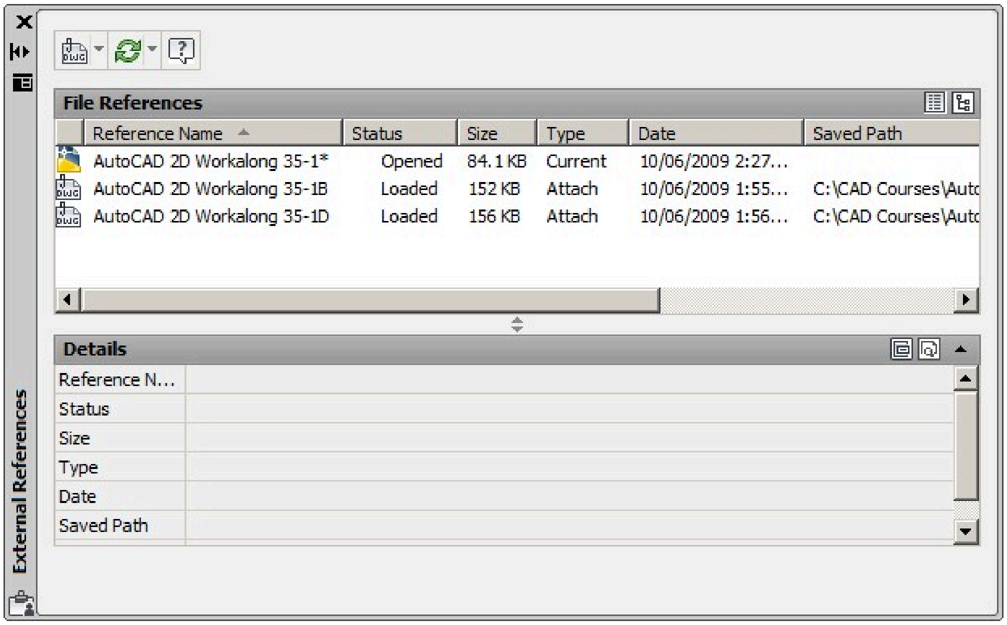
● Bind

○ Insert

Figure Step 44

Step 45

The External Reference window should match the figure. (Figure Step 45)



File References

Reference Name	Status	Size	Type	Date	Saved Path
AutoCAD 2D Workalong 35-1*	Opened	84.1 KB	Current	10/06/2009 2:27...	
AutoCAD 2D Workalong 35-1B	Loaded	152 KB	Attach	10/06/2009 1:55...	C:\CAD Courses\Aut
AutoCAD 2D Workalong 35-1D	Loaded	156 KB	Attach	10/06/2009 1:56...	C:\CAD Courses\Aut

Details

Reference N...	
Status	
Size	
Type	
Date	
Saved Path	

Figure Step 45

Step 46

Your drawing should as shown in the figure. (Figure step 46)

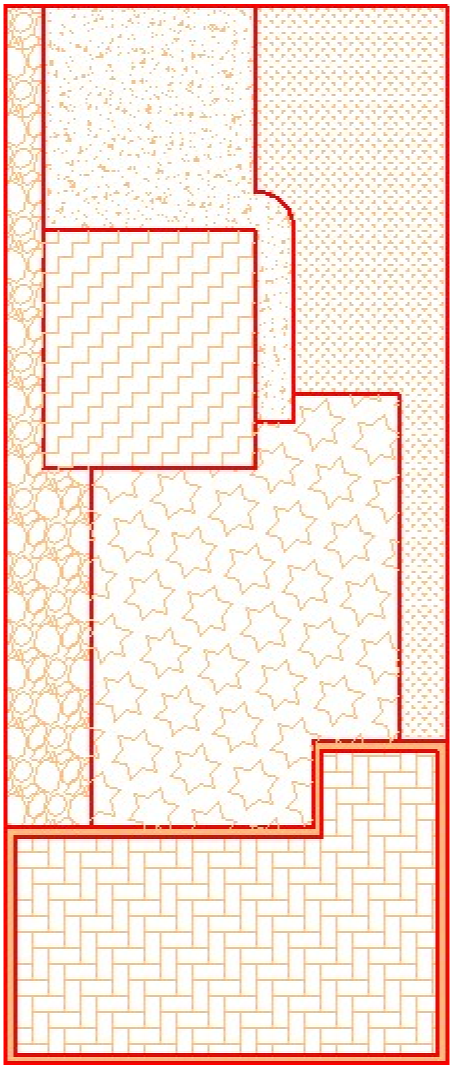
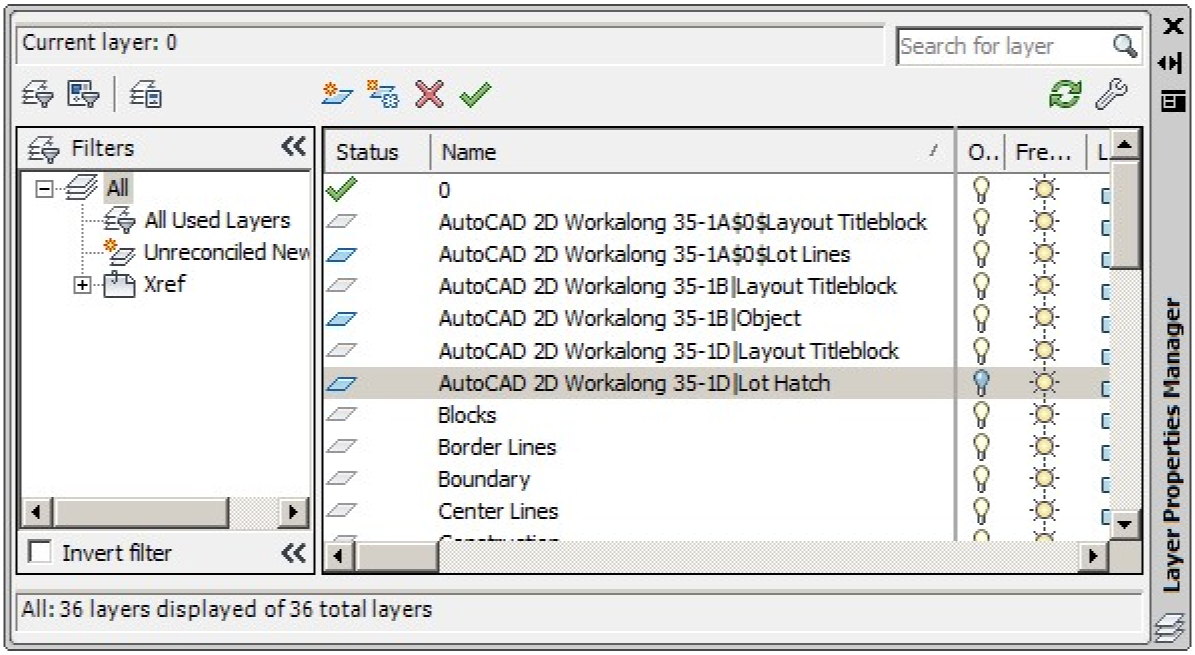


Figure Step 46

Step 47

In Layer Properties Manager window, turn off layer: AutoCAD 2D Workalong 35-1D|Lot Hatch. (Figure Step 47)



Search for layer
Filters
All
All Used Layers
Unreconciled New
Xref
Invert filter
Layer Properties Manager

Current layer: 0

Status	Name
✓	0
	AutoCAD 2D Workalong 35-1A$0$Layout Titleblock
	AutoCAD 2D Workalong 35-1A$0$Lot Lines
	AutoCAD 2D Workalong 35-1B|Layout Titleblock
	AutoCAD 2D Workalong 35-1B|Object
	AutoCAD 2D Workalong 35-1D|Layout Titleblock
	AutoCAD 2D Workalong 35-1D|Lot Hatch
	Blocks
	Border Lines
	Boundary
	Center Lines

All: 36 layers displayed of 36 total layers

Figure Step 47

Step 48

Your completed drawing should match the figure. (Figure step 48)

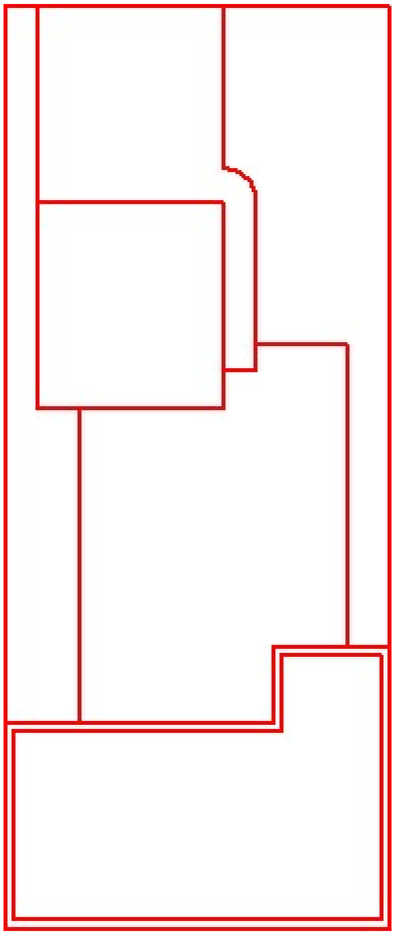


Figure Step 48

Step 49

Save and close the drawing.

MUST KNOW: To completely remove a reference file from the current drawing, you must detach it using the XREF command. Erasing a reference file does not remove it completely from the drawing. All the named objects definitions associated with an erased xref remain. Using the Detach option removes the selected xref and all associated information.

Key Principles

Key Principles in Module 35

1. An external reference file is a drawing file linked to the current drawing. It acts as one object similar to a block and appears to be part of the current drawing.
2. A reference file can be bound to the drawing it is attached to with the XREF command. When a reference drawing is bound to the drawing, the XREF command converts it into a block.
3. To completely remove a reference file from the current drawing, it must be detached using the XREF command.

Lab Exercise 35-1

Time allowed: 50 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 35-1	2D Layout English	Inches

Step 1

On layer: Object, draw the Polygon Multi-Tool shown in the figure. (Figure Step 1A, 1B, and 1C)

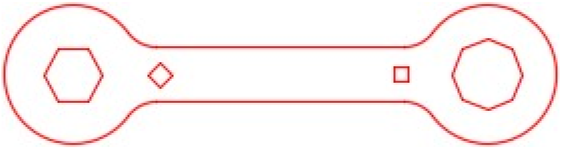


Figure Step 1A
Completed Drawing

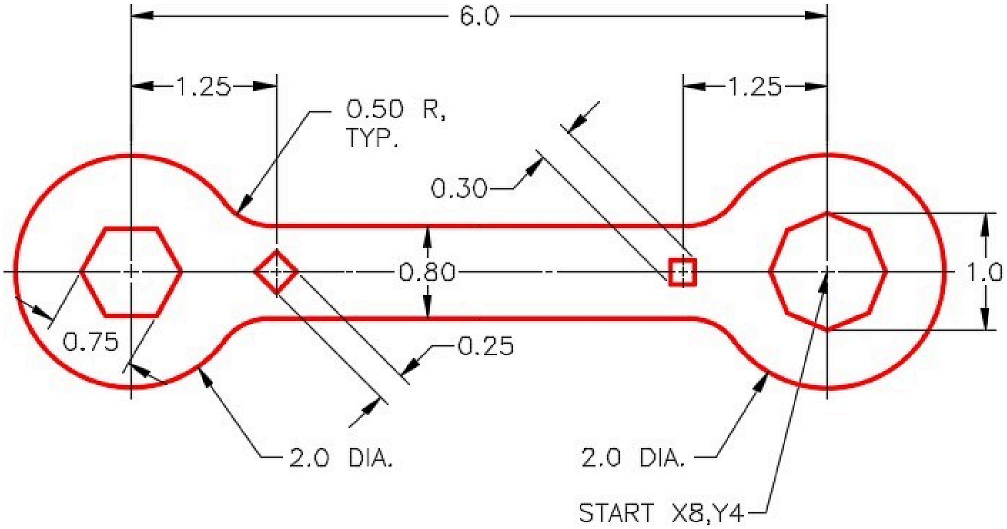

6.0
1.25
1.25
0.50 R,
TYP.
0.30
0.80
1.0
0.75
0.25
2.0 DIA.
2.0 DIA.
START X8,Y4

Figure Step 1B

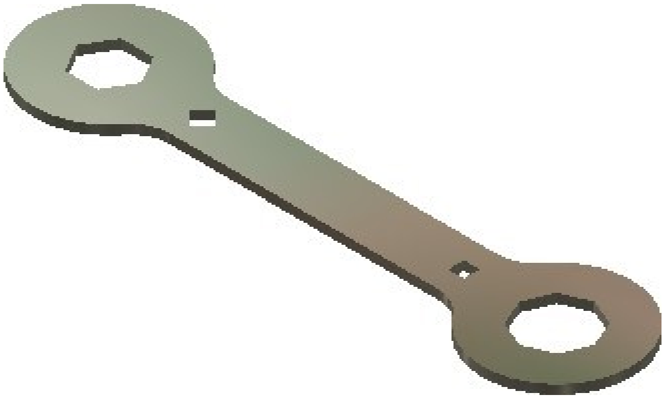


Figure Step 1C
Polygon Multi-Tool

Step 2

Set the insertion units and check your drawing with the key. Turn layer: Key off and freeze layer: Construction.

Lab Exercise 35-2

Time allowed: 50 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 35-2	2D Layout Metric	Millimeters

Step 1

On layer: Object, draw the Double Hexagon Wrench shown in the figure. (Figure Step 1A, 1B, 1C, and 1D)

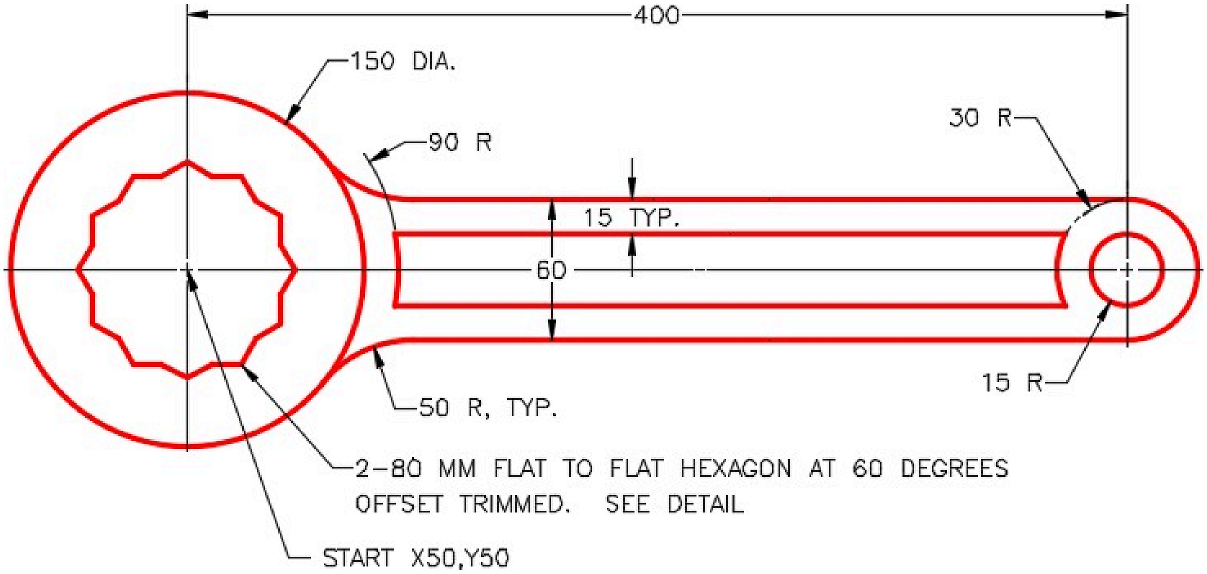

400
150 DIA.
90 R
30 R
15 TYP.
60
15 R
50 R, TYP.
2–80 MM FLAT TO FLAT HEXAGON AT 60 DEGREES
OFFSET TRIMMED. SEE DETAIL
START X50,Y50

Figure Step 1A

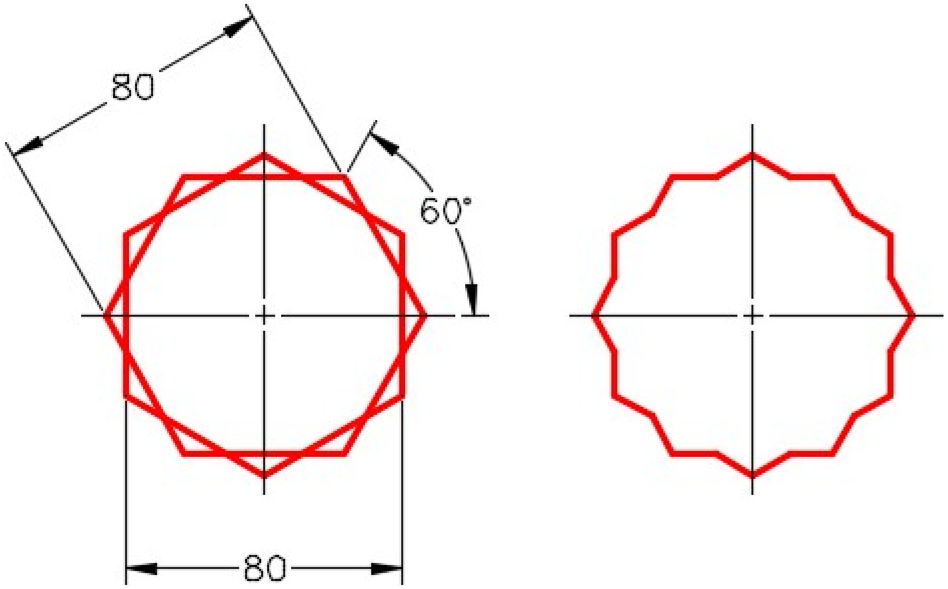


Figure Step 1B
Detail

AUTHOR’S COMMENTS: The double hexagon shape is drawn with two hexagons at 60 degrees offset. The inner lines are then trimmed to complete the double hexagon.

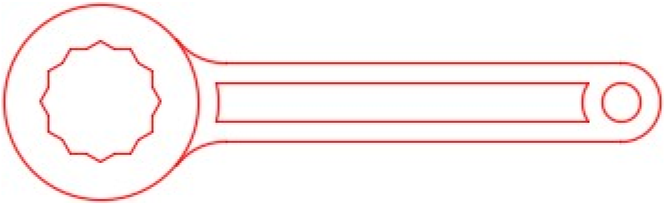


Figure Step 1C
Completed Drawing

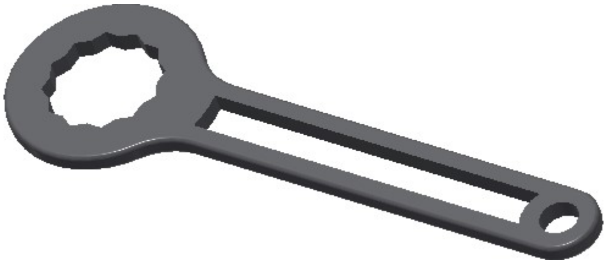


Figure Step 1D
Double Hexagon Wrench

Step 2

Set the insertion units and check your drawing with the key. Turn layer: Key off and freeze layer: Construction.

Module 36: Competency Test 7 Open Book

Learning Outcomes

When you have completed this module, you will be able to:

1. Within a three hour time limit, complete a written exam and the lab exercise without the aid of a key.

The AutoCAD 2D book was written with competency based modules. What that means is that you have not completed each module until you have mastered it. The Competency Test module contains multiple choice questions and a comprehensive lab exercise to test your mastery of the set of modules that you completed. There are no answers or keys supplied in a Competency Test module since it is meant to be checked by your instructor. If there are any parts of this module that you have trouble completing, you should go back and reread the module or modules containing the information that you are having trouble with. If necessary, redo as many lab exercises required until you fully understand the material.

If you are Completing this book:

- Without the Aid of an Instructor, complete the written test and the lab exercise.
- In a Classroom with an Instructor, the instructor will give instructions on what to do after this module has been completed.

Multiple Choice Questions

Select the BEST answer.

1. What term is used to describe a block within block?
    A. Nested object
    B. Hatched object
    C. Xref object
    D. Bound object
    E. Insert object
2. What command is used to establish a block's base point?
    A. XREF

   B. BLOCK
   C. INSERT
   D. PEDIT
   E. POINT

3. Where are block definitions stored in a drawing's database?
   A. Block location table
   B. Block definition table
   C. Block database table
   D. Block table
   E. Block database

4. What command is used to edit existing hatch patterns?
   A. EDITHATCH
   B. BHATCH
   C. EDIT
   D. PEDIT
   E. HATCHEDIT

5. From what drawing object are all hatch patterns created from?
   A. Plines
   B. Arcs
   C. Lines
   D. Splines
   E. None of the above.

6. What command is used to totally remove a block definition from the current drawing?
   A. TAKEOUT
   B. PURGE
   C. DELETE
   D. REMOVE
   E. BLOCK

7. What command is used to change the name of a block definition in the current drawing?
   A. INSERT
   B. NAME
   C. CHANGE
   D. RENAME

   E. PURGE

8. What must be done to completely remove a reference file from the current drawing?
   A. Detach it using the XREF command.
   B. Detach it using the XBIND command.
   C. Erase it using the ERASE command.
   D. Detach it using the BLOCK command.
   E. Detach it using the BIND command.

9. What two commands are used to toggle the display of the existing hatching patterns on or off?
   A. FILL and EXPLODE
   B. PEDIT and FILL
   C. REGEN and PEDIT
   D. EXPLODE and FILL
   E. FILL and REGEN

10. When a reference file is bound to a drawing that it is attached to, what drawing object type is the xref converted into?
   A. Individual objects
   B. Line
   C. Block
   D. Pline
   E. Hatch

Lab Exercise 36-1 OPEN BOOK

Drawing Name	Template	Units
AutoCAD 2D Lab 36-1	N/A	Feet and Inches

Step 1

Open the drawing: AutoCAD 2D Lab 29-3. (Figure Step 1)

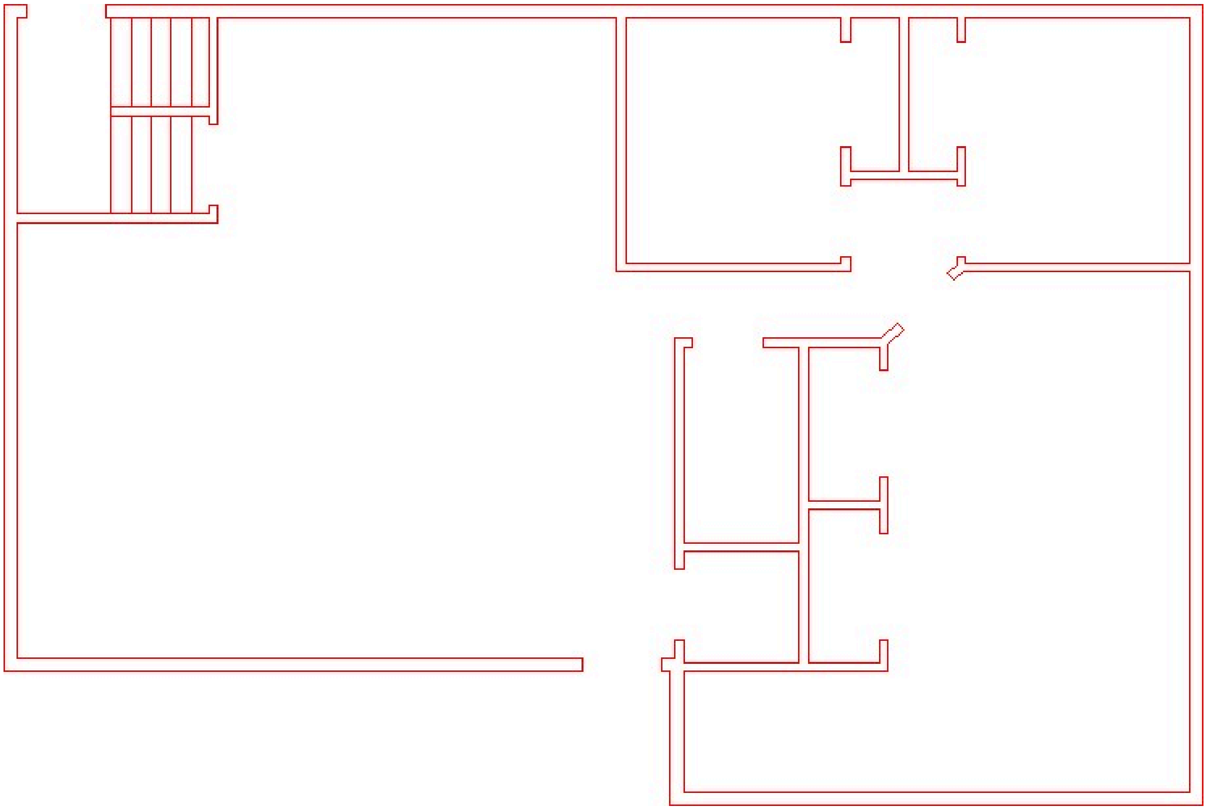


Figure Step 1

Step 2

Using the SAVEAS command, save the drawing with the name: AutoCAD 2D Lab 36-1.

Step 3

Draw the three doors, shown in the figure, anywhere in Model space. Draw them on layer: 0. Ensure that the color assigned to layer: 0 is black/white. (Figure Step 3A and 3B)

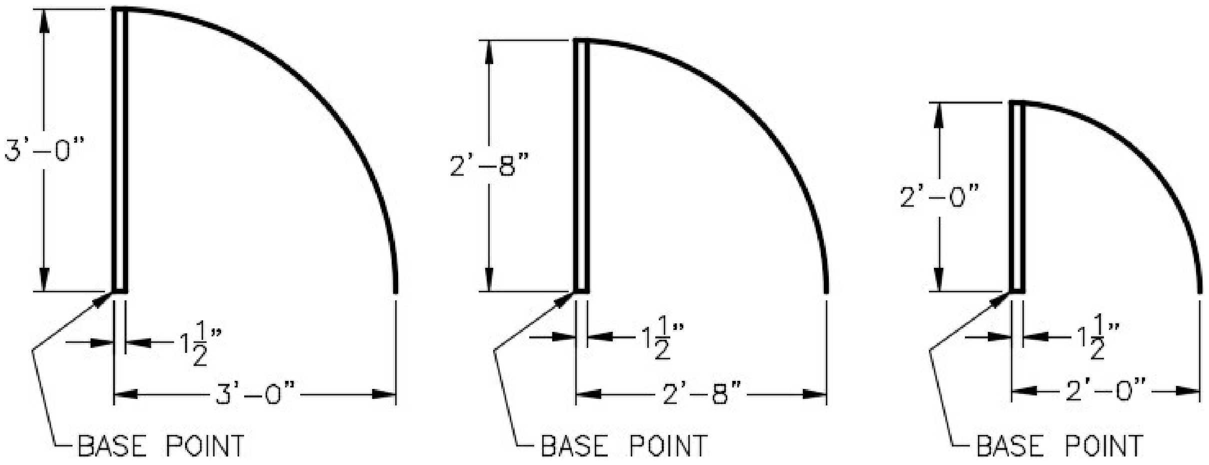

3'–0"
1½"
3'–0"
BASE POINT
2'–8"
1½"
2'–8"
BASE POINT
2'–0"
1½"
2'–0"
BASE POINT

Figure Step 3A

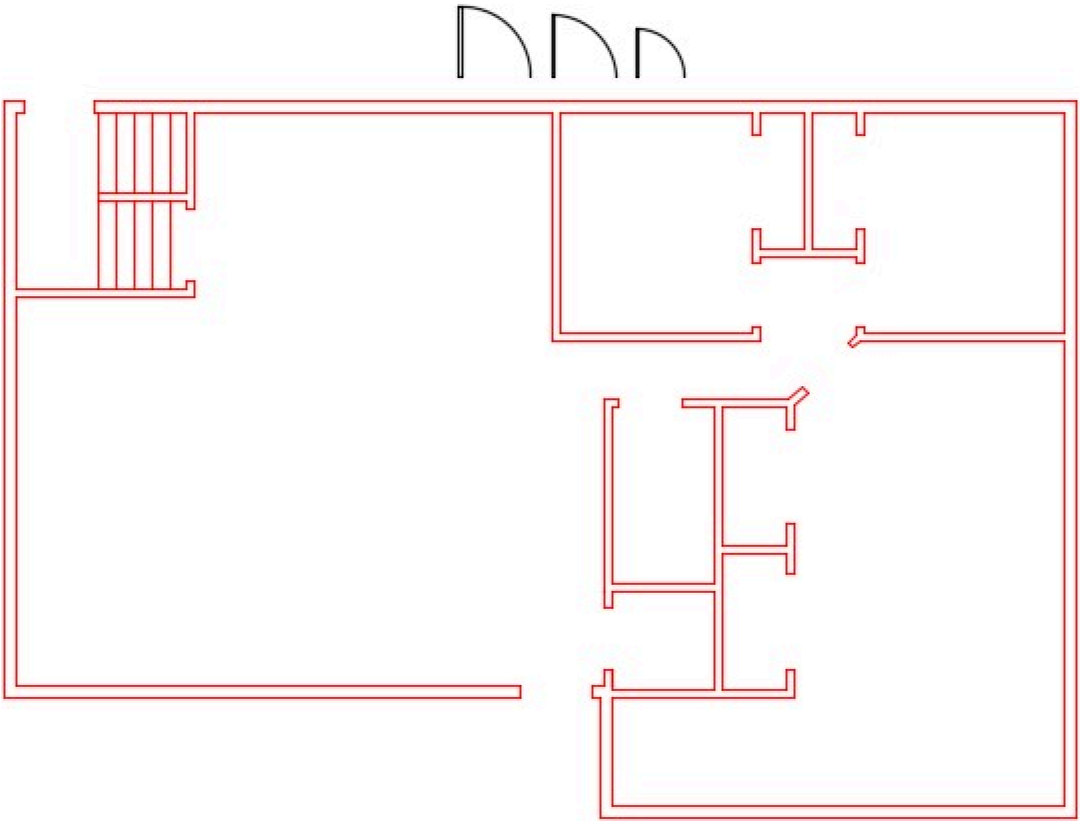


Figure Step 3B

Step 4

Create a block from each of the three doors that you drew in Step 3. Use the following:

A Name each block as shown below the drawing.

B Set the BLOCK command to delete the objects after creation.

C Set the base point of the blocks as shown on the drawing.

D Give each block a description and set the Block units to Inches.

Step 5

On layer Doors, insert the door blocks to match the door opening size. (Figure Step 5)

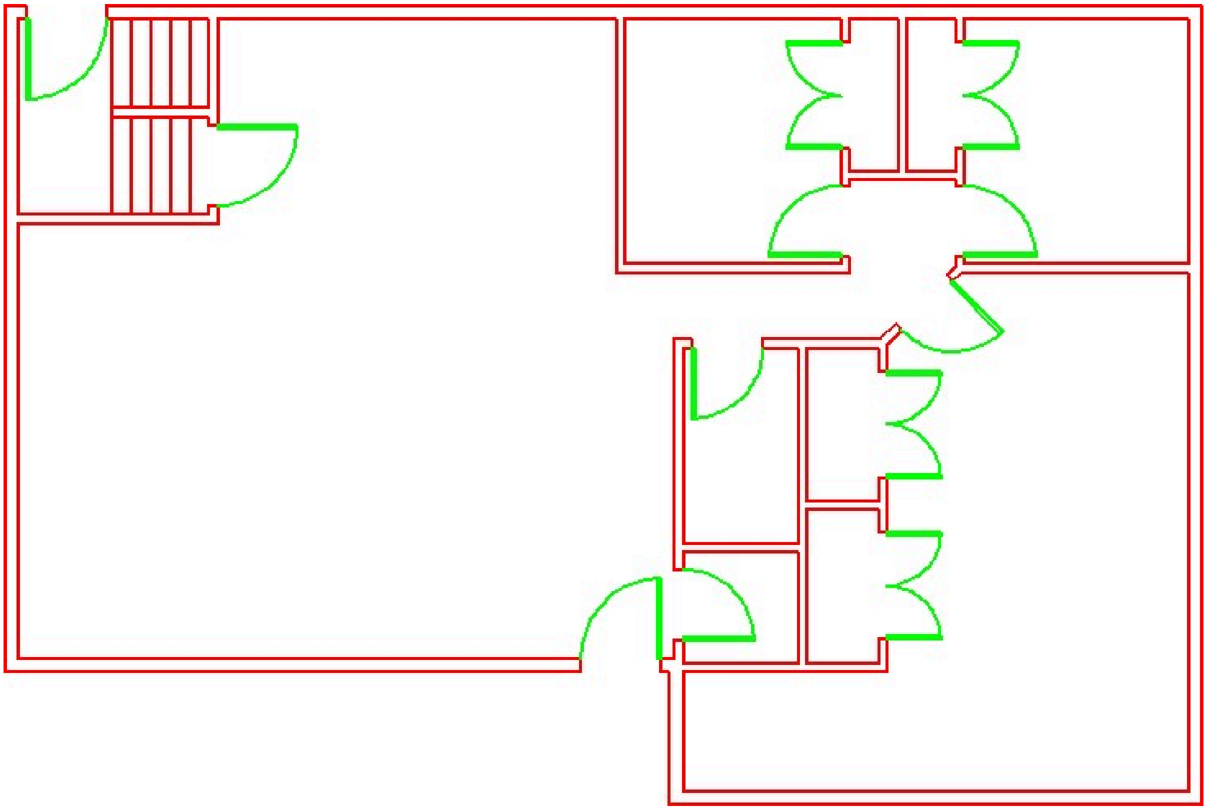


Figure Step 5

AUTHOR'S COMMENTS: Insert the blocks near the door openings and then use the ROTATE and MOVE commands to locate them.

Step 6

On layer Hatch, insert the pattern: Net in the areas shown in the figure. Set the scale to match the hatch to the figure the best you can by eye. (Figure Step 6)

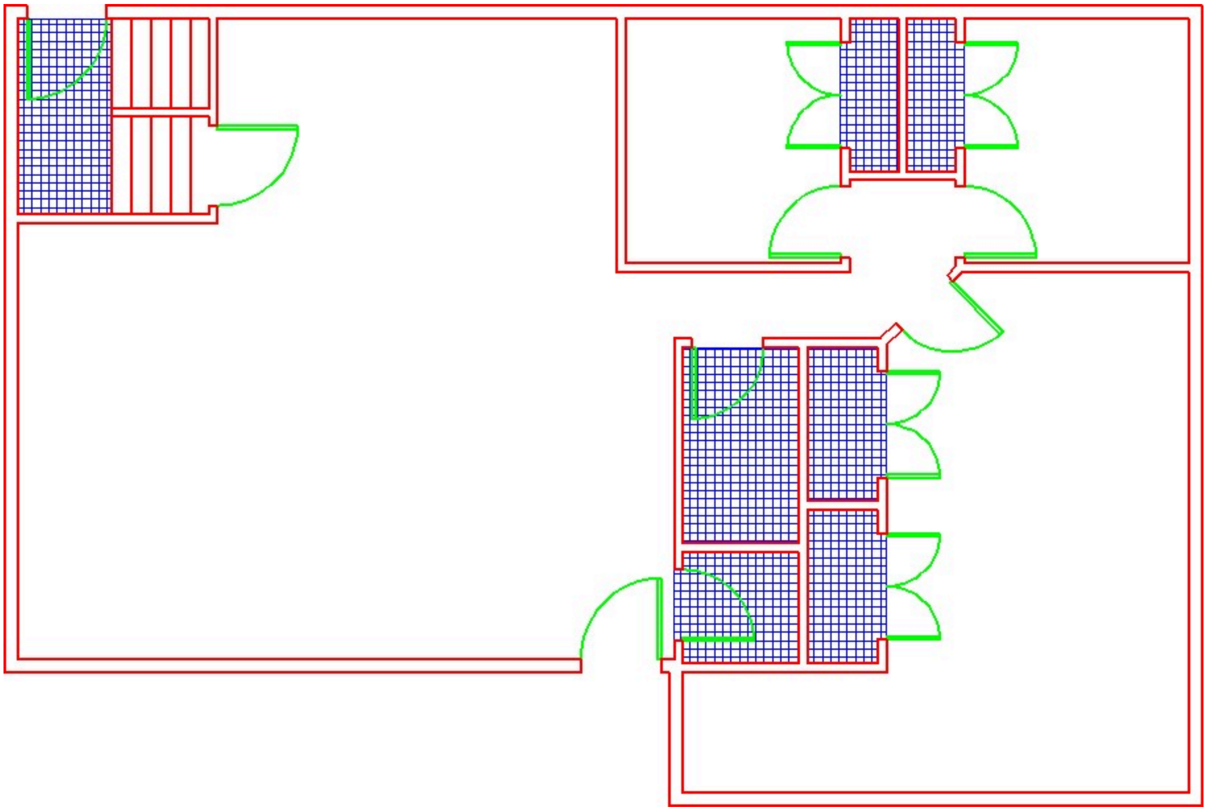


Figure Step 6

AUTHOR'S COMMENTS: Insert the blocks near the door openings and then use the ROTATE and MOVE commands to locate them.

Step 7

On layer: Boundary, draw a closed pline, at a width of 1″, around the 4 rooms and the hallway. Ensure that you snap to the corners of the walls. Find the area of each closed pline in Sq Ft to two decimal places of precision. Record the results. (Figure Step 7)

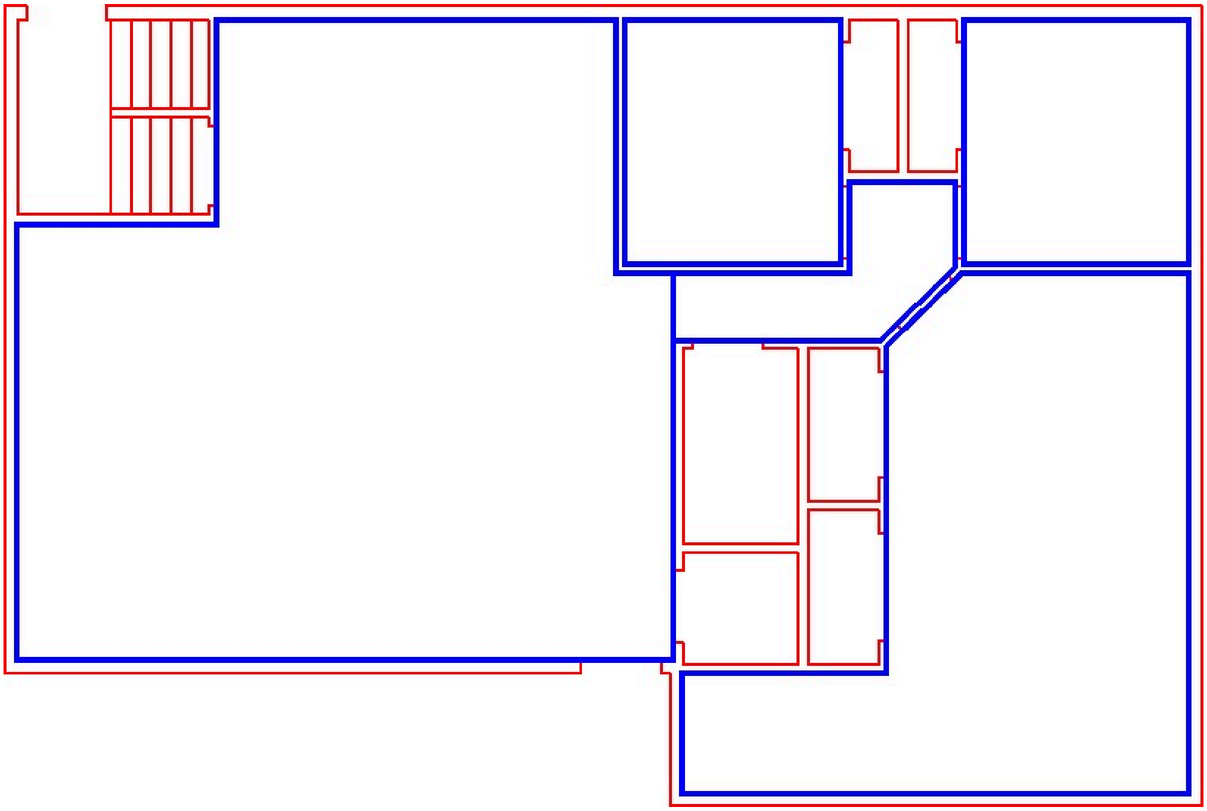


Figure Step 7

Step 8

Enable layout: Module Layout C. On layer: Viewport, create a viewport at a scale of 3/8″=1′-0″ and lock the display. In Paper space, on layer: Text, add the room name and square footage that you found in Step 7. Using the standards in Module 20, fill in the titleblock. (Figure Step 8)

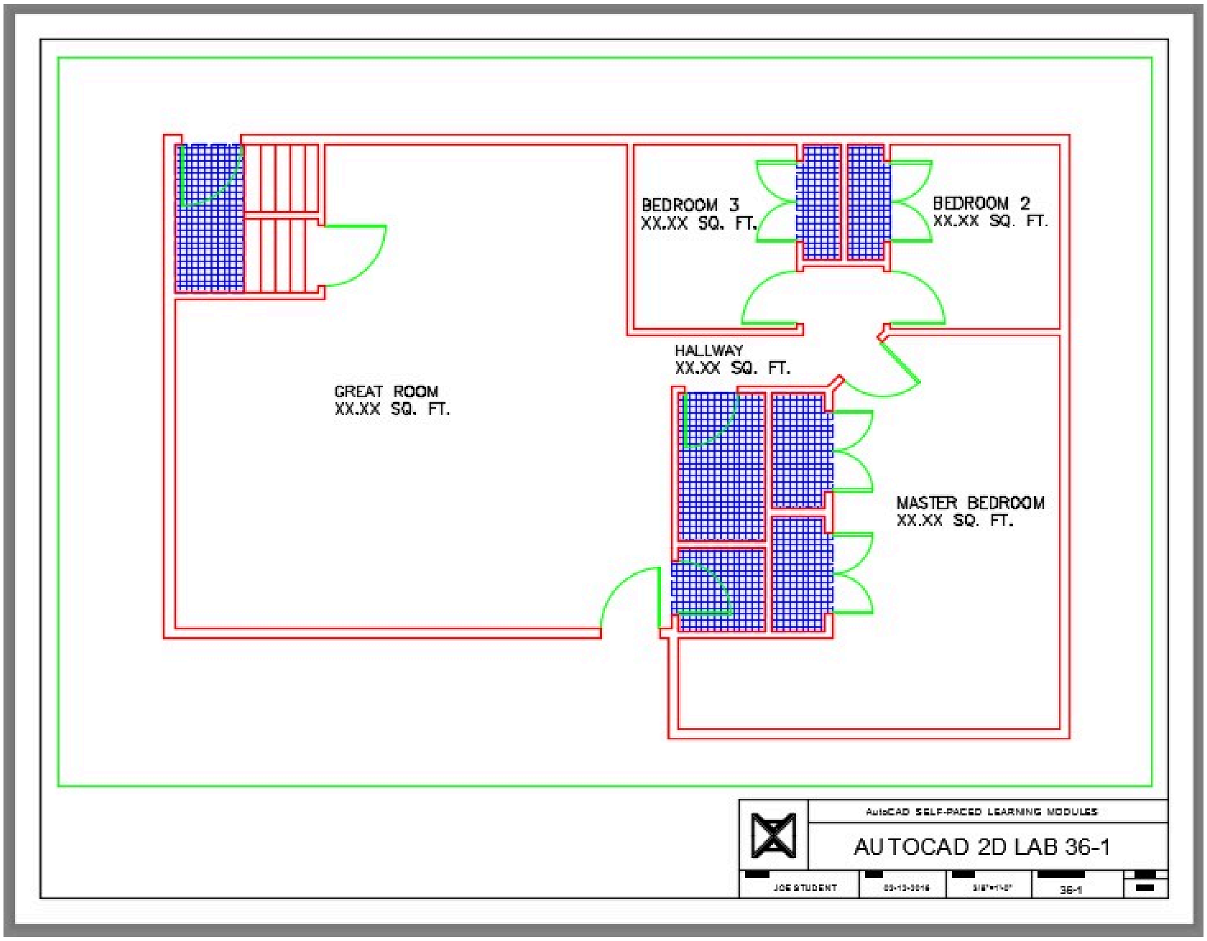

BEDROOM 3
XX.XX SQ. FT.
BEDROOM 2
XX.XX SQ. FT.
HALLWAY
XX.XX SQ. FT.
GREAT ROOM
XX.XX SQ. FT.
MASTER BEDROOM
XX.XX SQ. FT.
AutoCAD SELF-PACED LEARNING MODULES
AUTOCAD 2D LAB 36-1
JOE STUDENT
3/8"=1'-0"
36-1

Figure Step 8

AUTHOR’S COMMENTS: Ensure that you insert the text for both the rooms and titleblock in Paper space.

Step 9

Turn layers: Viewport and Boundary off to complete the drawing. (Figure Step 9)

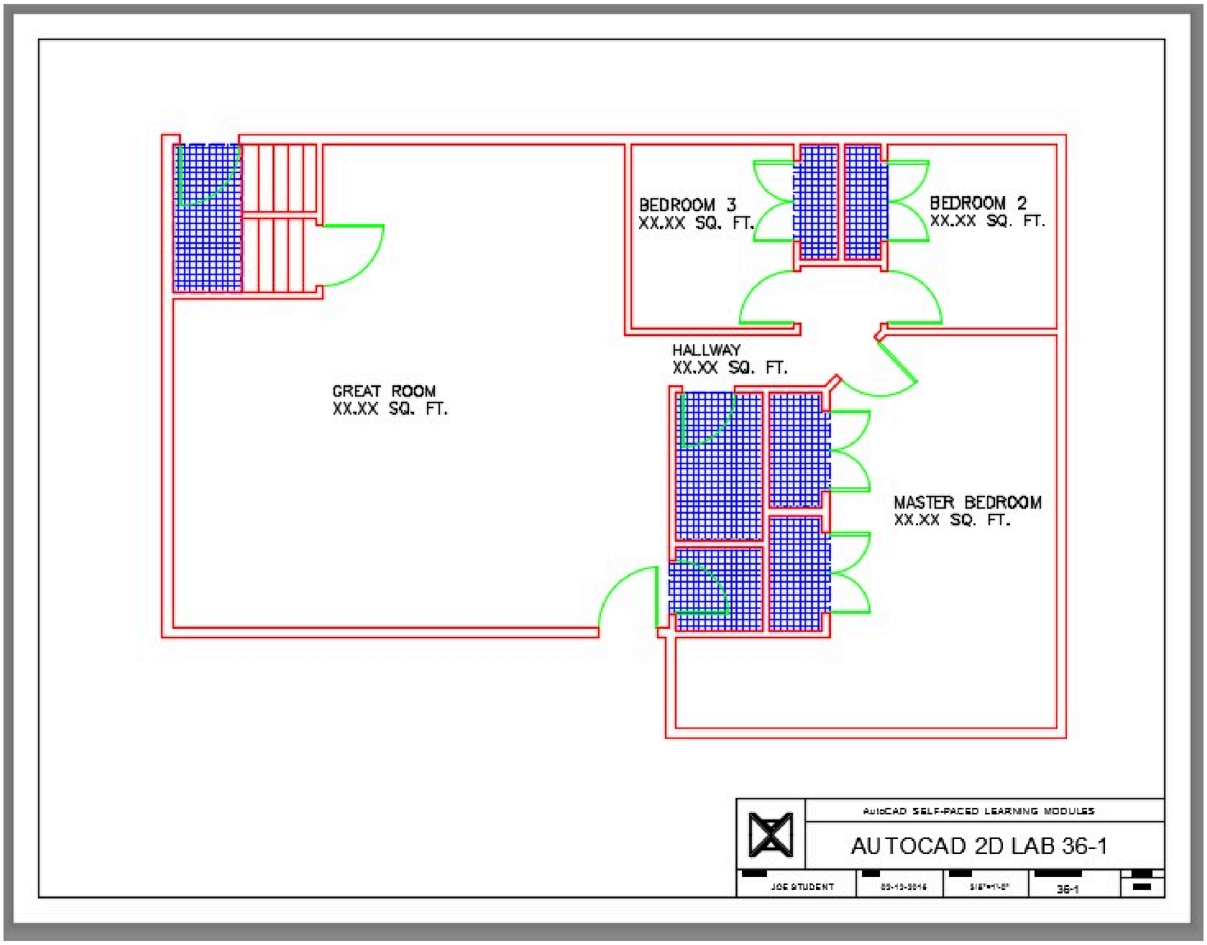

BEDROOM 3
XX.XX SQ. FT.
BEDROOM 2
XX.XX SQ. FT.
HALLWAY
XX.XX SQ. FT.
GREAT ROOM
XX.XX SQ. FT.
MASTER BEDROOM
XX.XX SQ. FT.
AutoCAD SELF-PACED LEARNING MODULES
AUTOCAD 2D LAB 36-1
JOE STUDENT
36-1

Figure Step 9

Step 10

Change the current space to Model. You drawing should match the figure. (Figure Step 10)

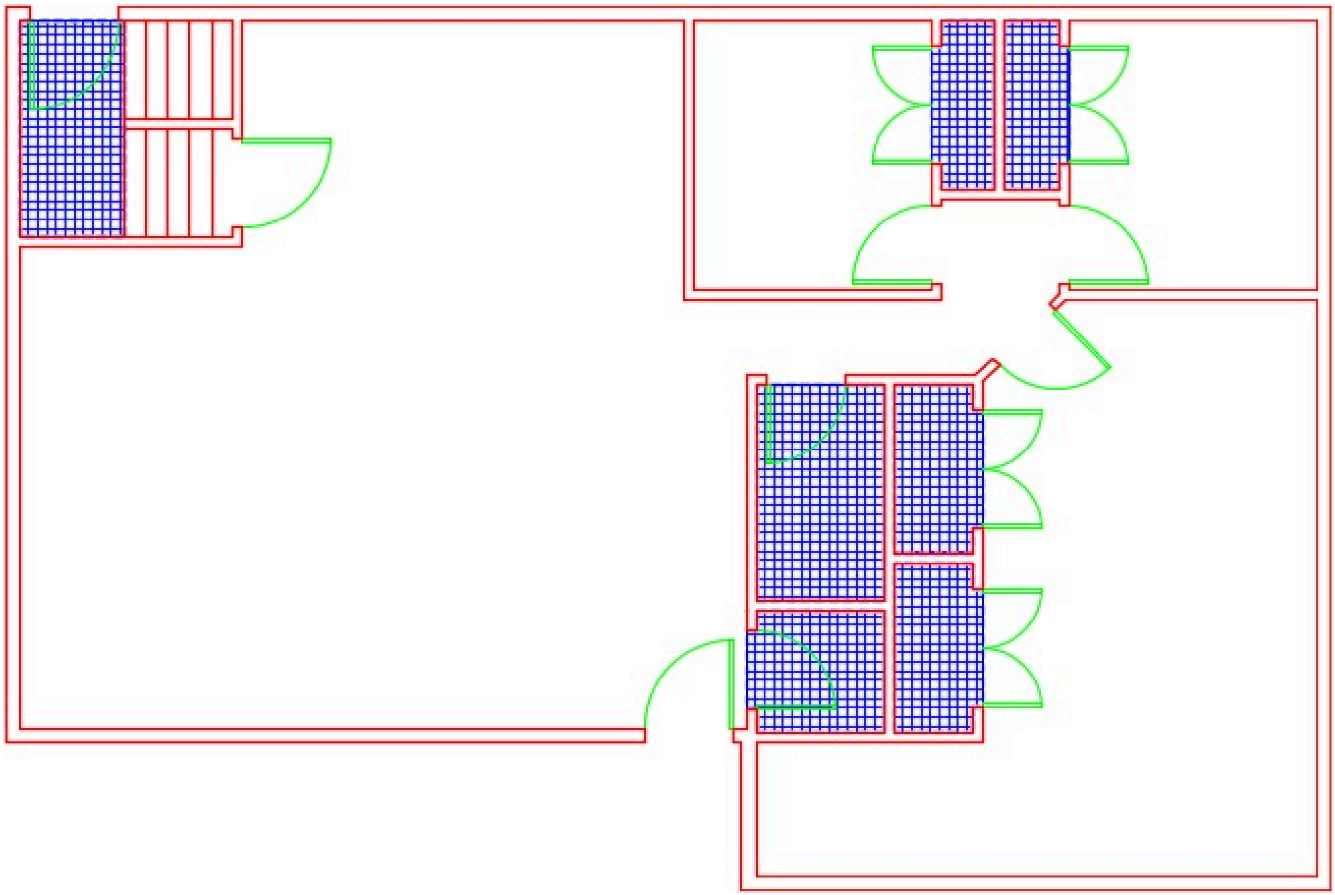


Figure Step 10

Step 11

Saved and close the drawing.

Part 8

Module 37: Dimensioning- Part 1

Learning Outcomes

When you have completed this module, you will be able to:

1. Define the 11 basic dimensioning terms.
2. Describe the properties of associative and exploded dimensions.
3. Apply the DIMASSOC and DIMSTYLE commands to control and modify the appearance of dimensions.
4. Create a dimension style and set the dimension variables for both an english and metric drawing.
5. Apply the BREAK and DIMCENTER commands to draw center lines.

Dimensioning

Dimensioning is the process of adding size descriptions to objects in the drawing. After the dimensioned are inserted, the layout can be plotted and used for construction or reference. Up to this point in the book, only shape description have been defined to the object by describing its size and location. Since AutoCAD knows the exact size of the objects in the drawing, you only have to enter the type and location of the dimension you want to insert and AutoCAD will do the rest.

Since there is a lot to learn, dimensioning is a complex but not a difficult subject. Therefore, in the AutoCAD 2D book, only learning to control the appearance, inserting, and editing the basic dimension types is taught. When you complete the AutoCAD 2D book, you will be able to completely dimension a drawing for most applications.

Associative verses Exploded Dimensions

An *associative dimension* is intelligent, similar to an associative hatch pattern. They are tied to the object they are dimensioning. An *exploded dimension* is simply a collection of AutoCAD drawing objects indicating a measurement on the object. It has no relation to the object it is dimensioning. An associative dimension and exploded dimension do not appear any different on the drawing and you cannot tell them apart by looking at them. The differences between them are as follows:

1. The dimensions of objects dimensioned with associative dimensions, when modified with most editing commands, will change to reflect the changes to the size of the object. The dimensions of objects that are dimensioned with an exploded dimension, when modified, will not change.
2. Associative dimensions are harder to edit than exploded dimensions because they are inserted

as blocks. An exploded dimension is inserted with individual drawing objects. If an associative dimension is exploded, it will be converted into an exploded dimension.

3. If the properties of the dimensioning style of an associative dimension is modified, all dimensions on the drawing, using that style, will change. For example, if the user changes the height of dimension text, change it once in the dimensioning style and all existing associative dimensions, using that style, will automatically display the new size.

Definition Points

Definition points are points that are automatically inserted on the drawing when an associative dimension is inserted. One definition point is inserted for each end of an extension line or on opposite sides of a circle or an arc. The points are automatically placed on layer Defpoints. If the Defpoints layer does not exist in the drawing, it will be created automatically when the first associative dimension is inserted on that drawing. The Defpoints layer cannot be deleted or purged.

These inserted points are the intelligent part of the associative dimension. It is these points that cause the dimension to automatically update since they are moved in the object editing commands and the dimension follows them. In an associative dimension, the points are attached to the object. Therefore, almost all editing commands move the points with the object.

The definition points will not plot and are not affected by the point type and size appearance. If you explode an associative dimension, the points attached to it will automatically be erased.

Dimensioning Styles

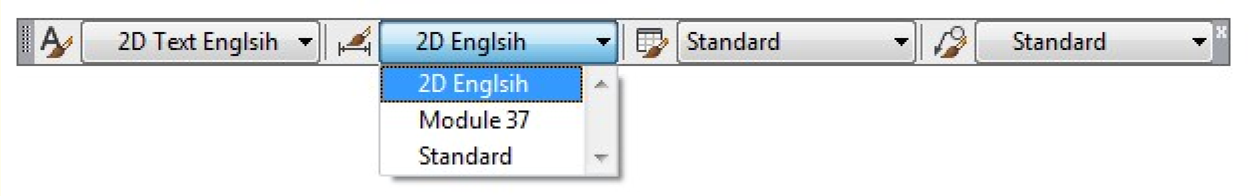
A dimensioning style is similar to a text style. A *dimensioning style* controls the way dimensions appear on your drawing. There are many different settings in a dimensioning style so it will take you a lot of practice to get used to setting them.

New styles can be created or existing dimensioning styles edited with the DIMSTYLE command. AutoCAD uses the current dimensioning style properties when inserting dimensions with any dimensioning command. By modifying a dimension style, you can update all existing associative dimensions on the drawing inserted with that style.

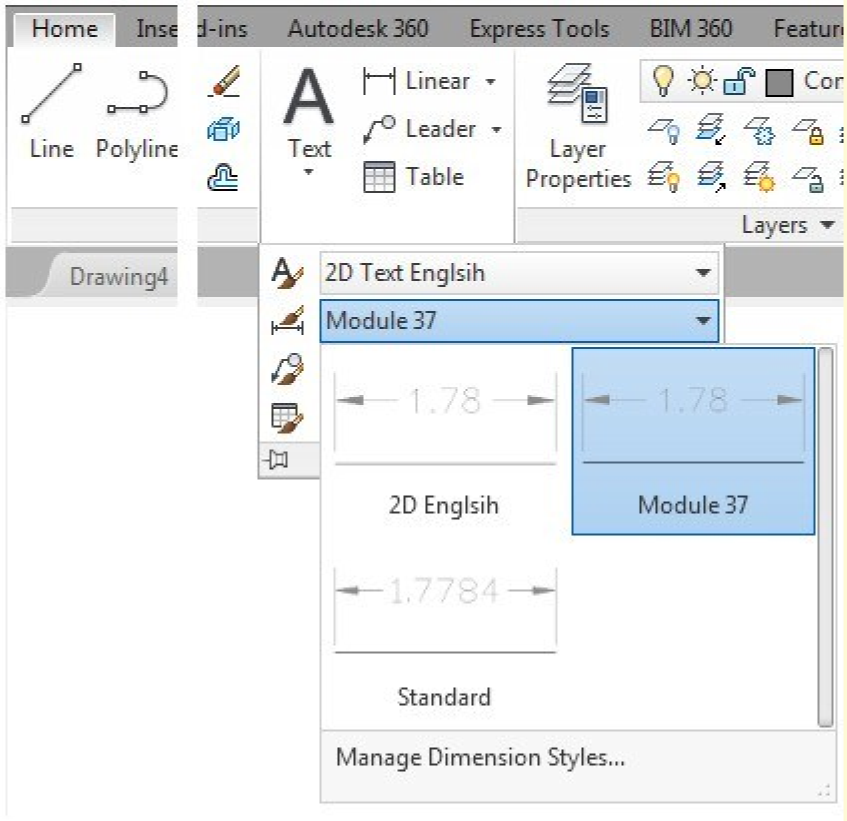
Standard Dimensioning Style

The Standard dimensioning style exists in every drawing and cannot be deleted. Even though the Standard style can be used, it is not a good practice to use it. Create and name your own styles.

USER TIP: Use the Styles toolbar or the Home ribbon to pull down a list and select a name to change the current dimensioning style for the current drawing. When the list is not currently being pulled down, it displays the current dimension style.


2D Text Englsih
2D Englsih
Standard
Standard
2D Englsih
Module 37
Standard

Styles Toolbar


Home
Inse
d-ins
Autodesk 360
Express Tools
BIM 360
Featur
Line
Polyline
Text
Linear
Leader
Table
Layer Properties
Cor
Layers
Drawing4
2D Text Englsih
Module 37
1.78
2D Englsih
1.78
Module 37
1.7784
Standard
Manage Dimension Styles...

Home Ribbon

Drafting Lesson: Basic Dimensioning Terms

Figure 37-1 shows the 11 basic dimensioning terms that you should know when setting dimensioning styles.

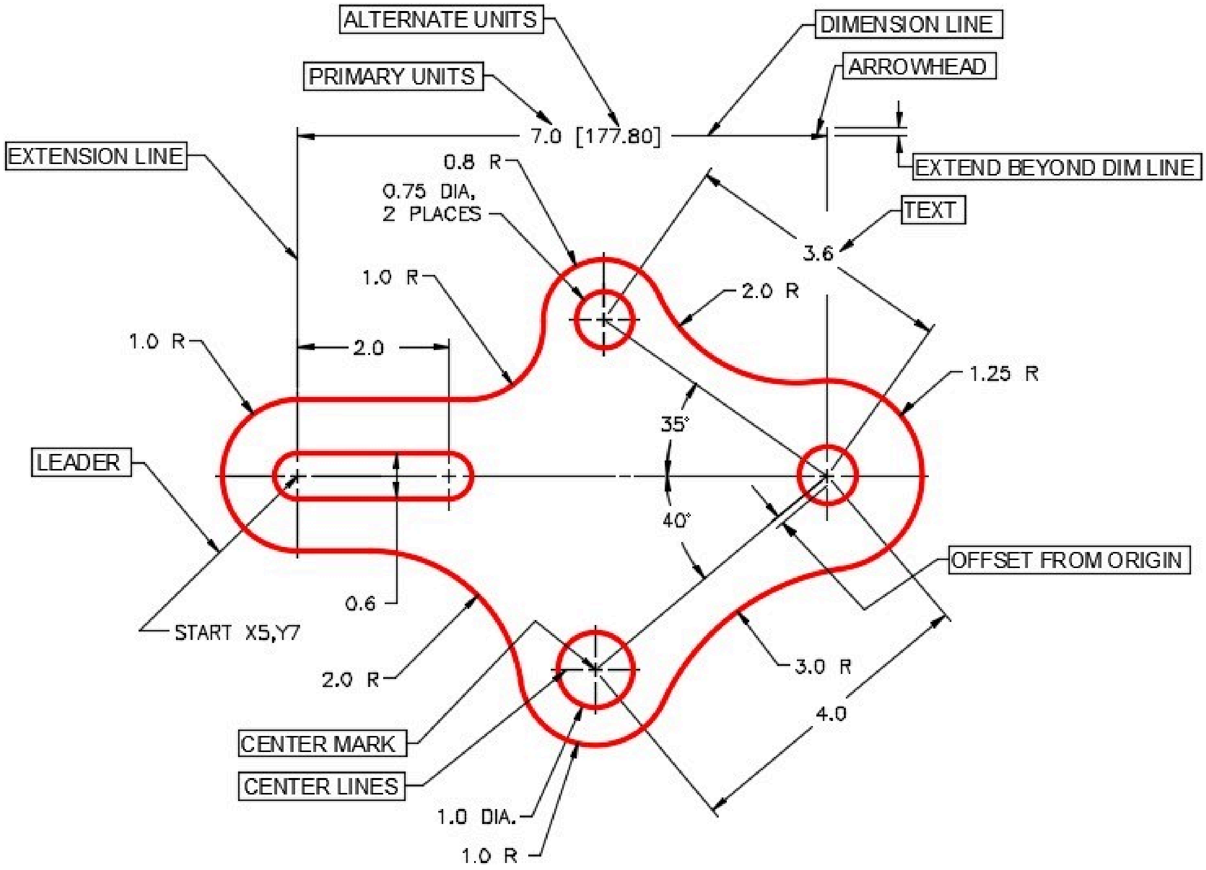

ALTERNATE UNITS
DIMENSION LINE
ARROWHEAD
PRIMARY UNITS
EXTENSION LINE
7.0 [177.80]
0.8 R
EXTEND BEYOND DIM LINE
0.75 DIA,
2 PLACES
TEXT
3.6
1.0 R
2.0 R
1.0 R
2.0
1.25 R
35°
LEADER
40°
OFFSET FROM ORIGIN
0.6
START X5,Y7
3.0 R
2.0 R
4.0
CENTER MARK
CENTER LINES
1.0 DIA.
1.0 R

Figure 37-1 Basic Dimensioning Terms

AutoCAD Command: DIMASSOC

The DIMASSOC system variable is used to enable either associative, non-associative, or exploded dimensions being inserted by dimensioning commands.

Command: **DIMASSOC**

Enter new value for DIMASSOC <2>: 0

Command:

0 – Dimensions will be inserted as exploded dimensions.

1 – Dimensions will be inserted as non-associative dimensions.

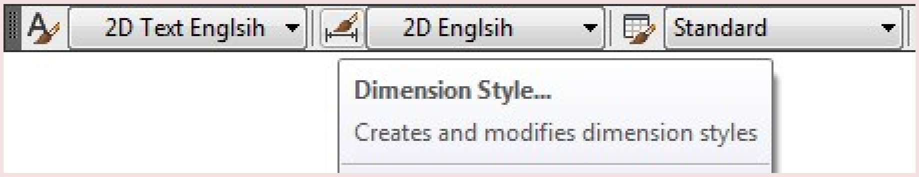
2 – Dimensions will be inserted as associative dimensions.

AUTHOR'S COMMENTS: Every AutoCAD drawing contains the Standard dimensioning style. It cannot be deleted and must be present so that additional styles can be created. I recommend that you never use the Standard dimensioning style. It is best to create your own styles, assigning them suitable names. Although the Standard style could be used, there is a very good reason not to. It is a common practice in industry to combine two or more drawings to form one drawing. In most cases, they are drawings created by different operators and a lot of the time, working at different companies. If the drawings, to be combined, each assign the Standard dimensioning style to dimensions on the drawings, all dimensions on the drawing, being added, would take the style properties of the drawing that it is added to. That can produce very unwanted results.

AutoCAD Command: DIMSTYLE

The DIMSTYLE command is used to create or modify dimensioning styles.

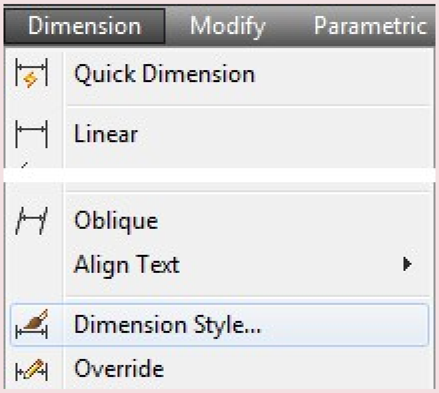
Shortcut: **DDIM**


2D Text Englsih
2D Englsih
Standard

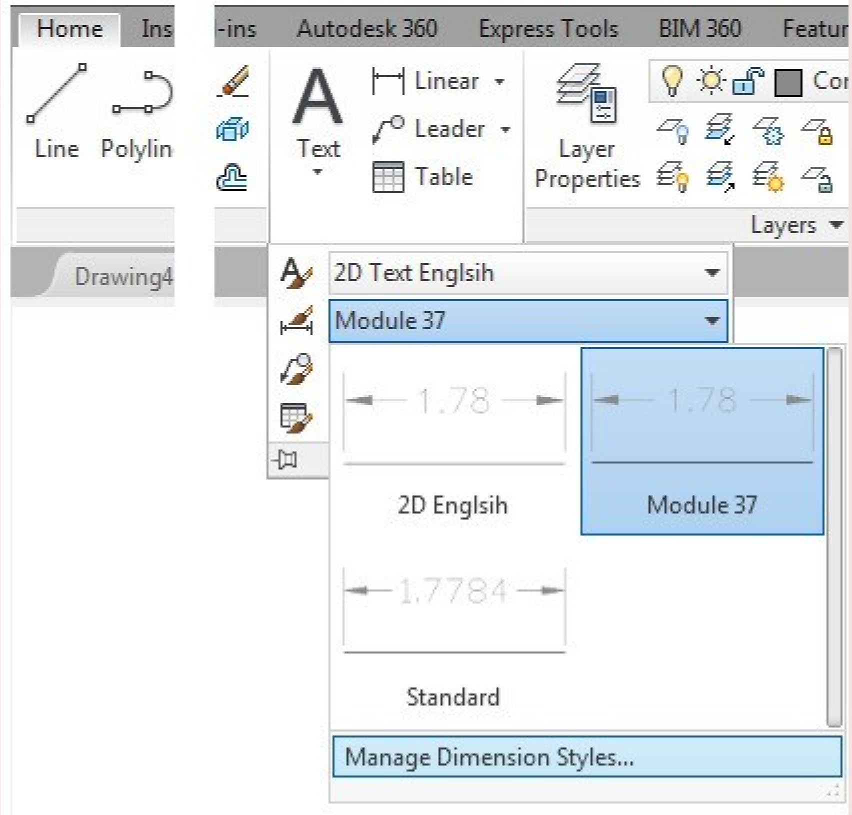
Dimension Style...

Creates and modifies dimension styles

Styles Toolbar


Dimension
Modify
Parametric
Quick Dimension
Linear
Oblique
Align Text
Dimension Style...
Override

Dimension Pull-down


Home
Ins
-ins
Autodesk 360
Express Tools
BIM 360
Featur
Line
Polylin
Text
Linear
Leader
Table
Layer Properties
Cor
Layers
Drawing4
2D Text Englsih
Module 37
1.78
1.78
2D Englsih
Module 37
1.7784
Standard
Manage Dimension Styles...

Home Ribbon

WORKALONG: Creating and Editing Dimension Styles

Step 1

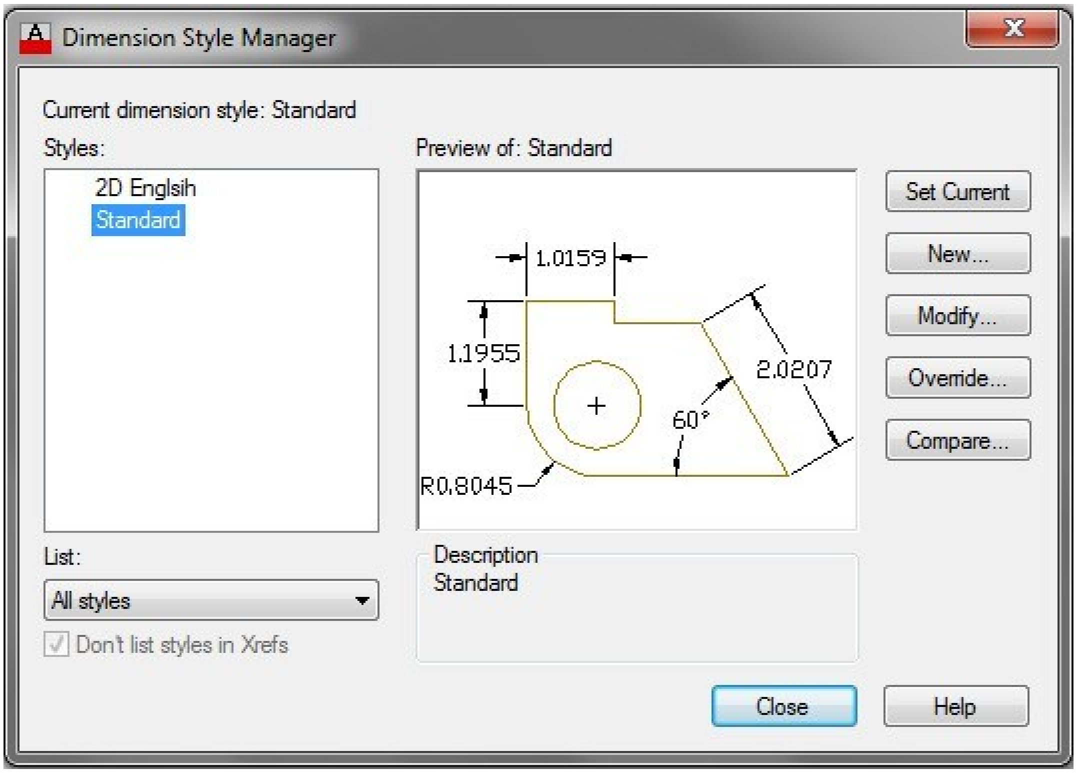
Start a new drawing using the template: 2D Layout English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 37-1.

Step 3

Enter the DIMSTYLE command. It will open the Dimension Style Manager dialogue box. (Figure Step 3)


Current dimension style: Standard

Styles:
2D Englsih
Standard

List: All styles

☑ Don't list styles in Xrefs

Description
Standard
Dimension Style Manager
Preview of: Standard
1.0159
1.1955
2.0207
60°
R0.8045
Set Current
New...
Modify...
Override...
Compare...
Close
Help

Figure Step 3

Step 4

Notice the graphic in the Preview of: window. It displays a pictorial view of how the dimensions, using current dimensioning style, would appear on a drawing. As changes are made to the current dimensioning style, the preview window will change to reflect those changes. (Figure Step 4)

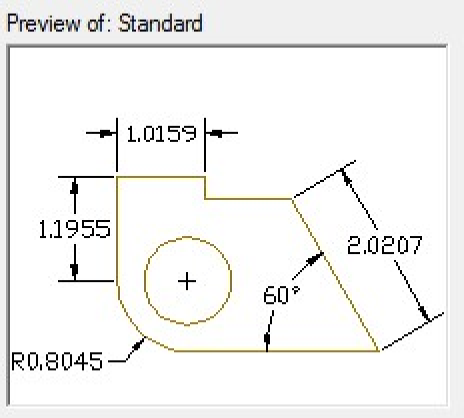
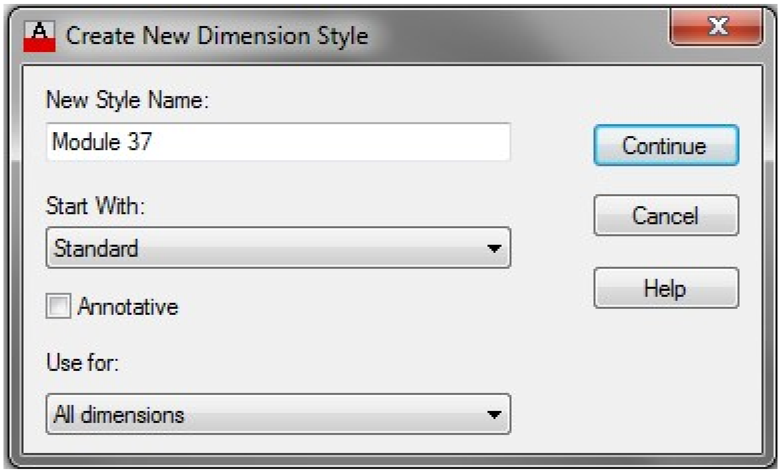

Preview of: Standard
1.0159
1.1955
2.0207
60°
R0.8045

Figure Step 4

Step 5

Click the New button. It will open the Create New Dimension Style dialogue box. In the New Style Name: box, enter the name: Module 37 and then click the Continue button. (Figure Step 5)



New Style Name: Module 37

Start With: Standard

☐ Annotative

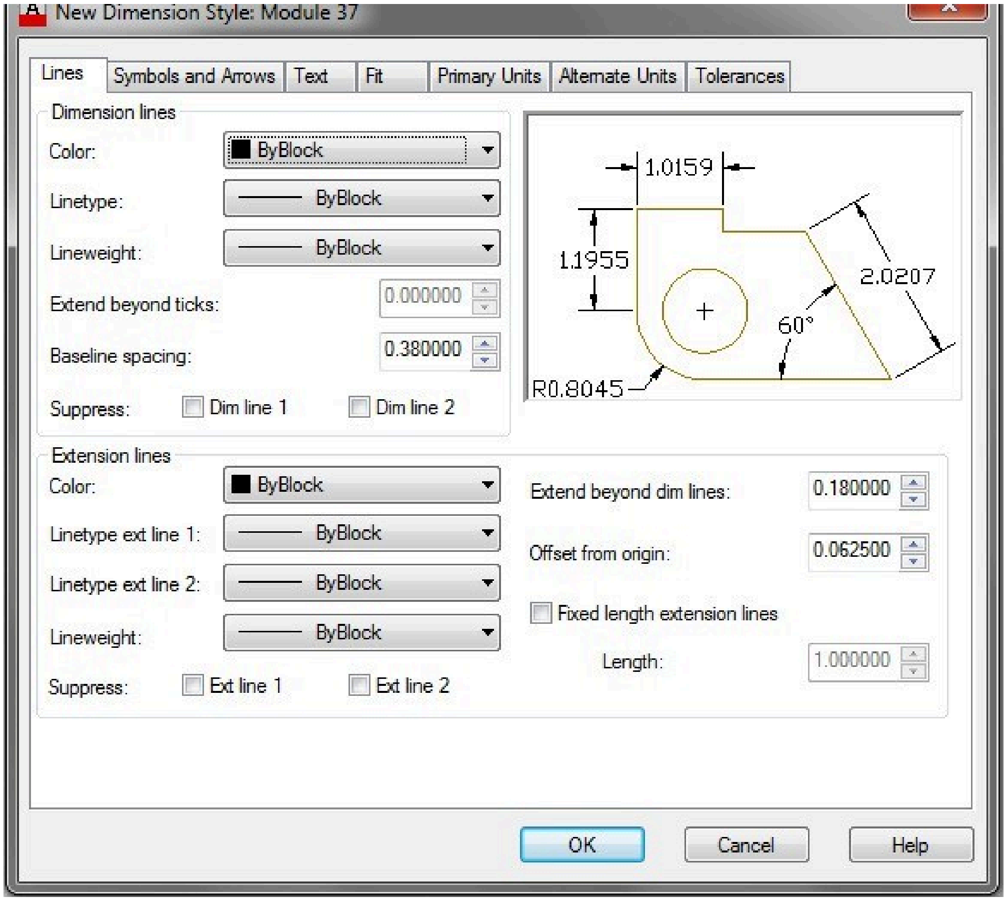
Use for: All dimensions

Figure Step 5

AUTHOR'S COMMENTS: The AutoCAD 2D book only covers the first five tabs in the Dimensioning Styles dialogue box. When you can set the variables in these five tabs, you will be able to use the styles to dimension any drawing using standards dimensioning techniques.

Step 6

In the Dimension Style dialogue box, enable the Lines tab. (Figure Step 6)


New Dimension Style: Module 37
Lines
Symbols and Arrows
Text
Fit
Primary Units
Alternate Units
Tolerances
1.0159
1.1955
2.0207
60°
R0.8045
OK
Cancel
Help

Dimension lines

Color: ByBlock

Linetype: ByBlock

Lineweight: ByBlock

Extend beyond ticks: 0.000000

Baseline spacing: 0.380000

Suppress: ☐ Dim line 1 ☐ Dim line 2

Extension lines

Color: ByBlock

Linetype ext line 1: ByBlock

Linetype ext line 2: ByBlock

Lineweight: ByBlock

Suppress: ☐ Ext line 1 ☐ Ext line 2

Extend beyond dim lines: 0.180000

Offset from origin: 0.062500

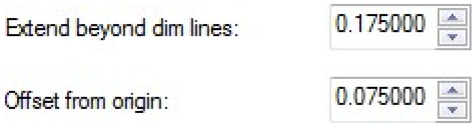
☐ Fixed length extension lines

Length: 1.000000

Figure Step 6

Step 7

In the Extension lines area, change the Extend beyond dim lines: box to 0.1750 and the Offset from origin: box to 0.075. Figure Step 7B shows which dimension style variables you are setting. (Figure Step 7A and 7B)



Extend beyond dim lines: 0.175000

Offset from origin: 0.075000

Figure Step 7A

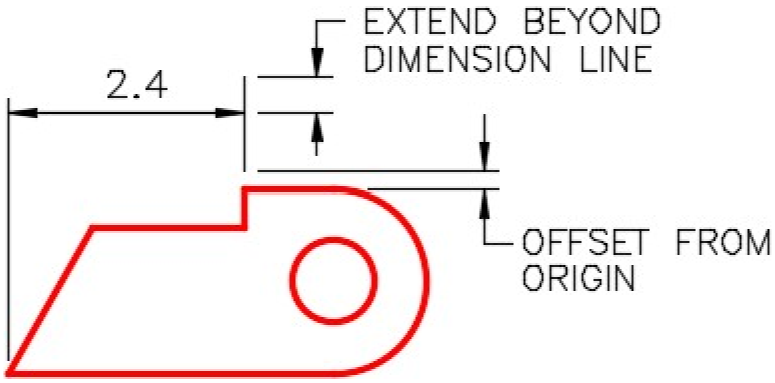
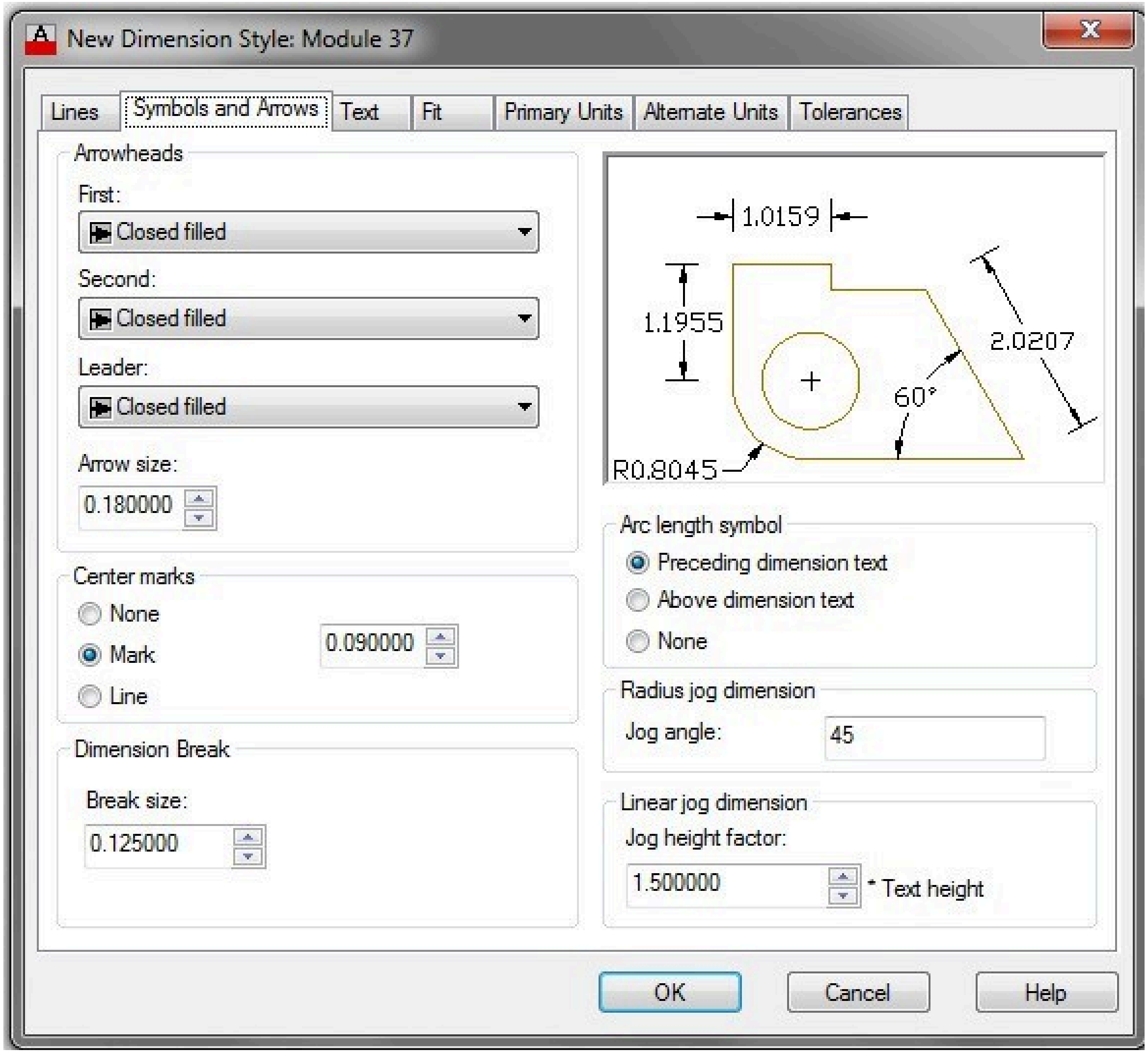

2.4
EXTEND BEYOND DIMENSION LINE
OFFSET FROM ORIGIN

Figure Step 7B

Step 8

Enable the Symbols and Arrows tab. (Figure Step 8)


New Dimension Style: Module 37
Lines
Symbols and Arrows
Text
Fit
Primary Units
Alternate Units
Tolerances
1.0159
1.1955
2.0207
60°
R0.8045
OK
Cancel
Help

Arrowheads

First: Closed filled

Second: Closed filled

Leader: Closed filled

Arrow size: 0.180000

Center marks

○ None

● Mark 0.090000

○ Line

Dimension Break

Break size: 0.125000

Arc length symbol

● Preceding dimension text

○ Above dimension text

○ None

Radius jog dimension

Jog angle: 45

Linear jog dimension

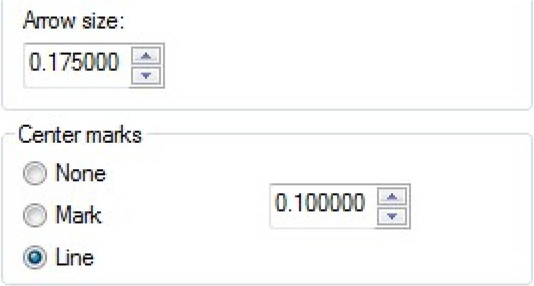
Jog height factor:

1.500000 * Text height

Figure Step 8

Step 9

In the Arrowheads area, set the Arrow size: to 0.1750. (Figure Step 9A and 9B)



Arrow size: 0.175000

Center marks

○ None

○ Mark

● Line

0.100000

Figure Step 9A

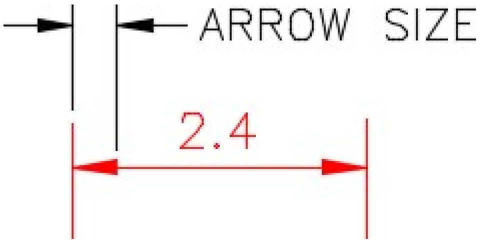

ARROW SIZE
2.4

Figure Step 9B

Step 10

In the Center marks area, enable Line and set it to 0.1000 as shown in Figure Step 9A. (Figure Step 10)

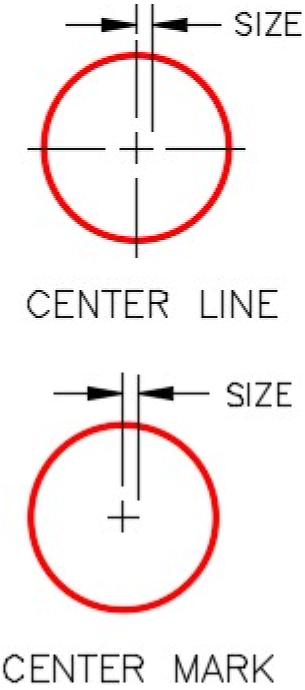
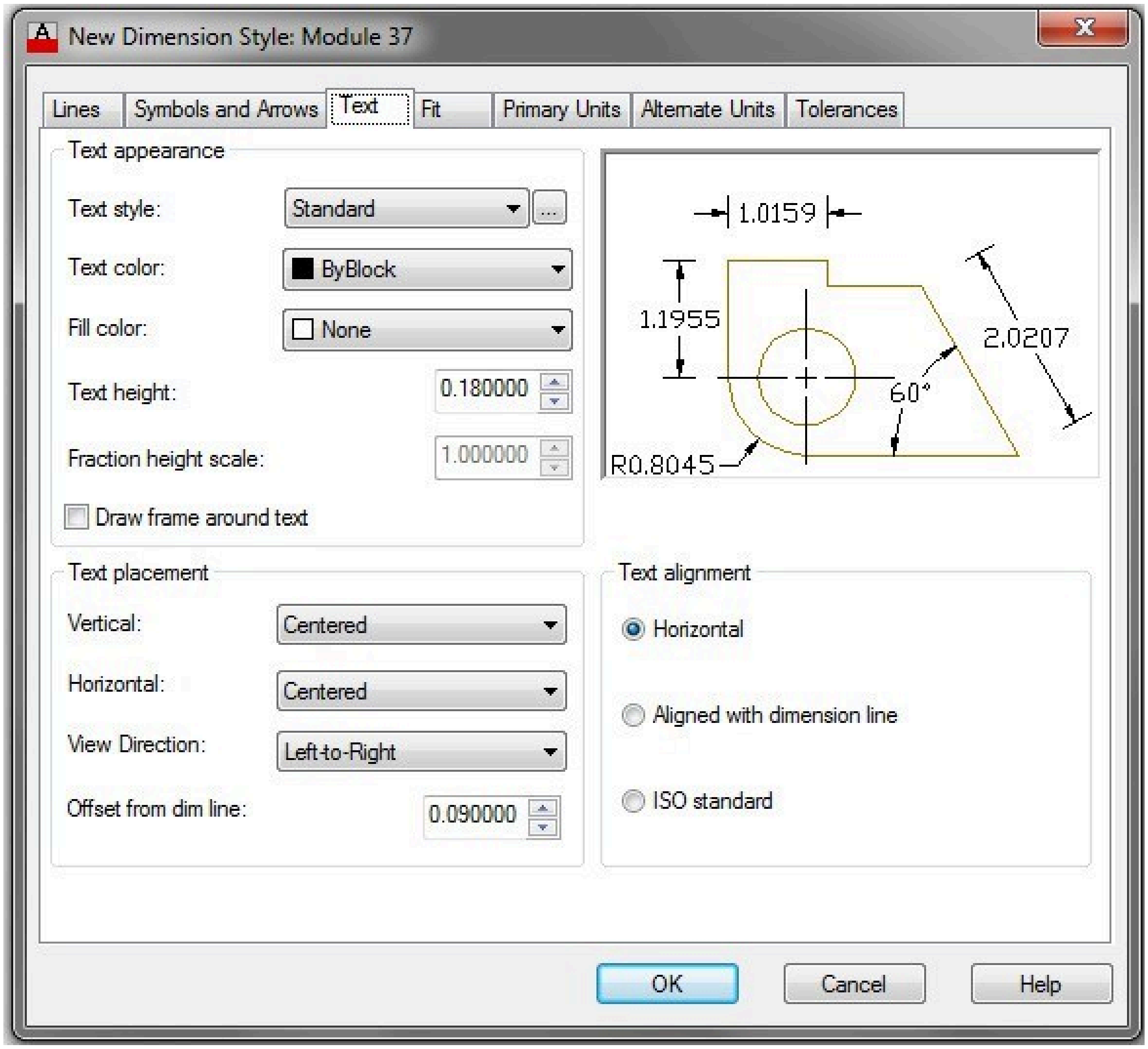

SIZE
CENTER LINE
SIZE
CENTER MARK

Figure Step 10


New Dimension Style: Module 37
Lines
Symbols and Arrows
Text
Fit
Primary Units
Alternate Units
Tolerances
1.0159
1.1955
2.0207
60°
R0.8045
OK
Cancel
Help

Text appearance

Text style: Standard

Text color: ByBlock

Fill color: None

Text height: 0.180000

Fraction height scale: 1.000000

☐ Draw frame around text

Text placement

Vertical: Centered

Horizontal: Centered

View Direction: Left-to-Right

Offset from dim line: 0.090000

Text alignment

● Horizontal

○ Aligned with dimension line

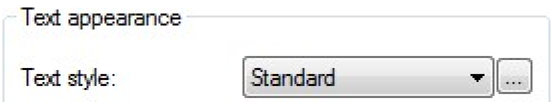
○ ISO standard

Figure Step 11

Enable the Text tab. (Figure Step 11)

Step 12

In the Text appearance area and Text style box, select the … box. (Figure Step 12)



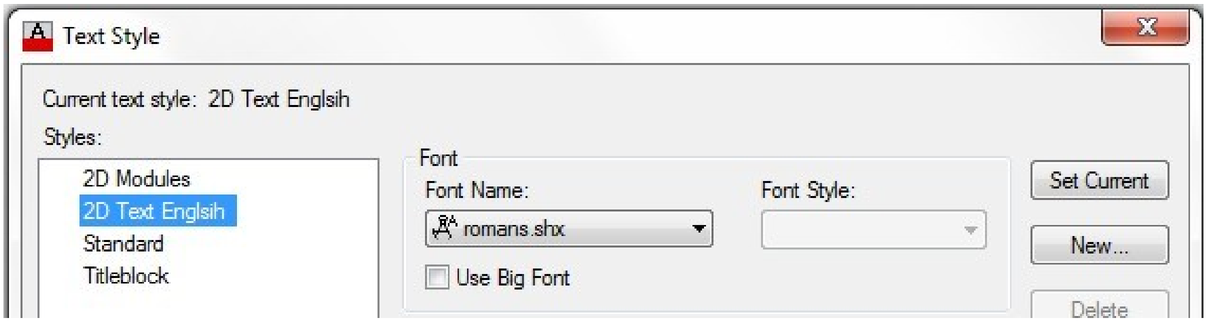
Text appearance

Text style: Standard

Figure Step 12

Step 13

Step 12 will open the Text Style dialogue box. This is the same dialogue box that you open when you use the STYLE command. Using what you already learned, set the current text style to: 2D Text English with the font: romans.shx. (Figure Step 13)


Text Style

Current text style: 2D Text Englsih

Styles:
- 2D Modules
- 2D Text Englsih
- Standard
- Titleblock

Font

Font Name: romans.shx

Font Style:

☐ Use Big Font

Figure Step 13

Step 14

In the Text appearance area, pull down the list of available text styles in the Text style box and select style: 2D Text English. Set the text height to 0.1500 in the Text height box. (Figure Step 14A and 14B)

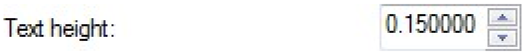
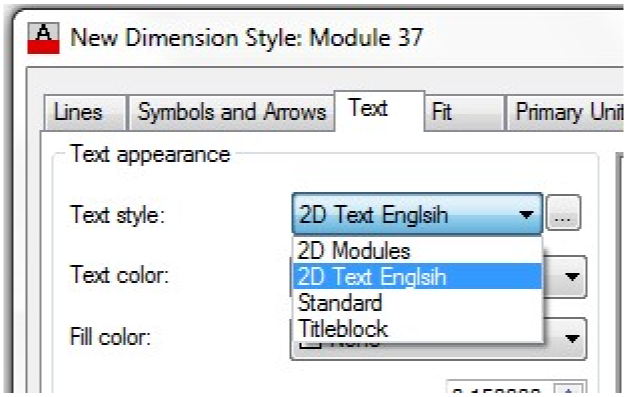

Text height: 0.150000

Figure Step 14A


New Dimension Style: Module 37

Text appearance

Text style: 2D Text Englsih
- 2D Modules
- 2D Text Englsih
- Standard
- Titleblock

Text color:

Fill color:

Figure Step 14B

Step 15

In the Text placement area, set the Offset from dim line to 0.1000. (Figure Step 15A and 15B)

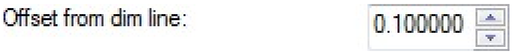

Offset from dim line: 0.100000

Figure Step 15A

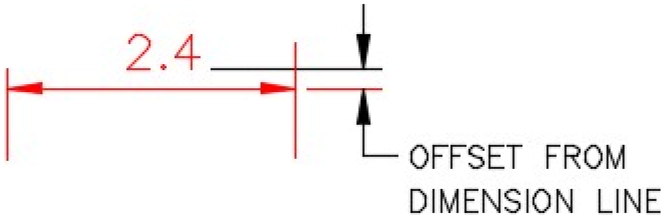
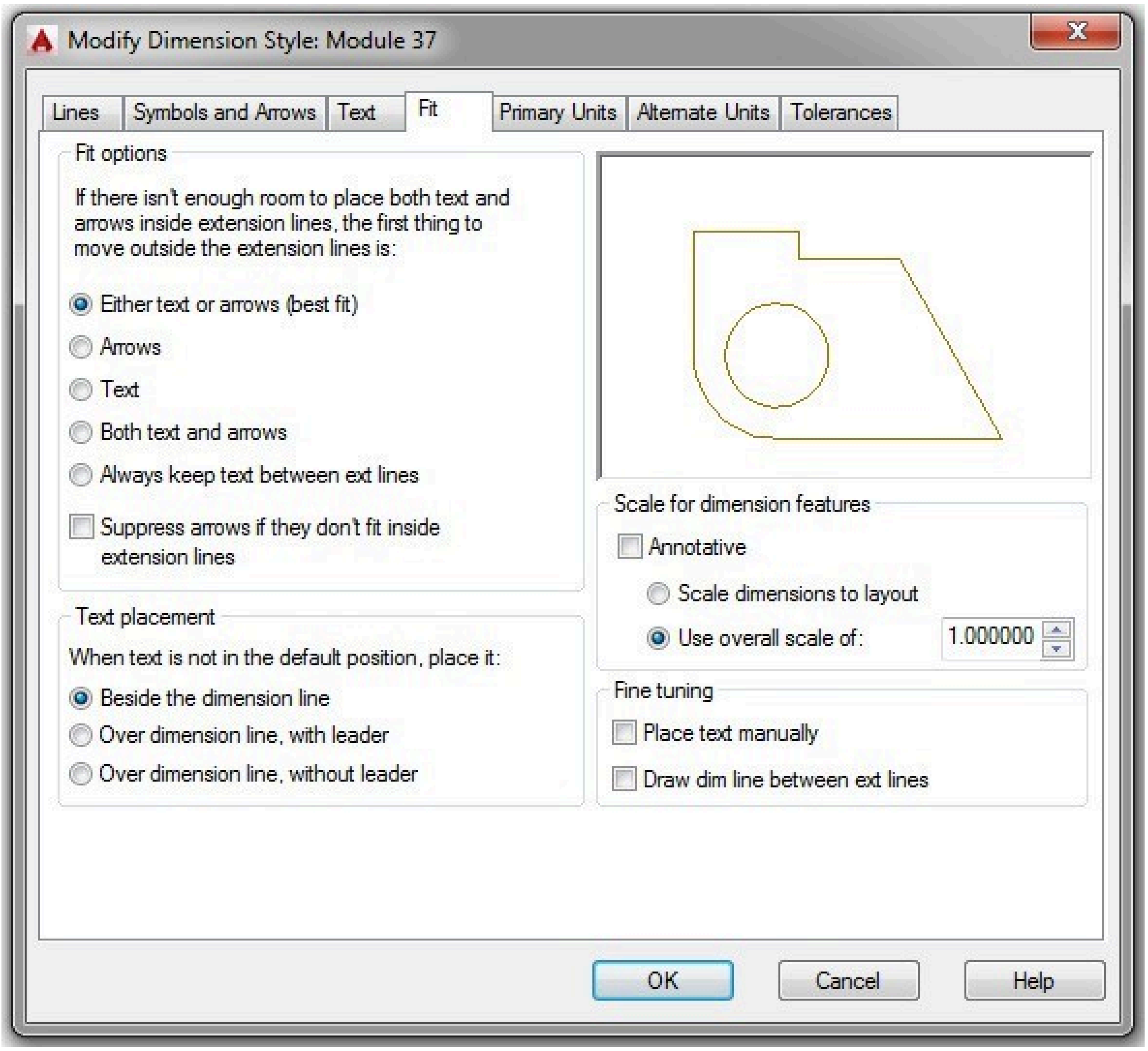

2.4
OFFSET FROM
DIMENSION LINE

Figure Step 15B

Step 16

Enable the Fit tab. Ensure that the settings in your dialogue box match the figures. Ensure that the Use of Overall scale of is set to 1.00000. (Figure Step 16A and 16B)


Lines
Symbols and Arrows
Text
Fit
Primary Units
Alternate Units
Tolerances
OK
Cancel
Help

Modify Dimension Style: Module 37

Fit options

If there isn't enough room to place both text and arrows inside extension lines, the first thing to move outside the extension lines is:

● Either text or arrows (best fit)
○ Arrows
○ Text
○ Both text and arrows
○ Always keep text between ext lines

☐ Suppress arrows if they don't fit inside extension lines

Text placement

When text is not in the default position, place it:

● Beside the dimension line
○ Over dimension line, with leader
○ Over dimension line, without leader

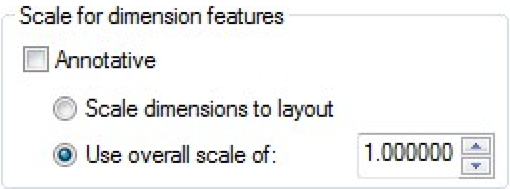
Scale for dimension features

☐ Annotative
○ Scale dimensions to layout
● Use overall scale of: 1.000000

Fine tuning

☐ Place text manually
☐ Draw dim line between ext lines

Figure Step 16A



Scale for dimension features

☐ Annotative

○ Scale dimensions to layout

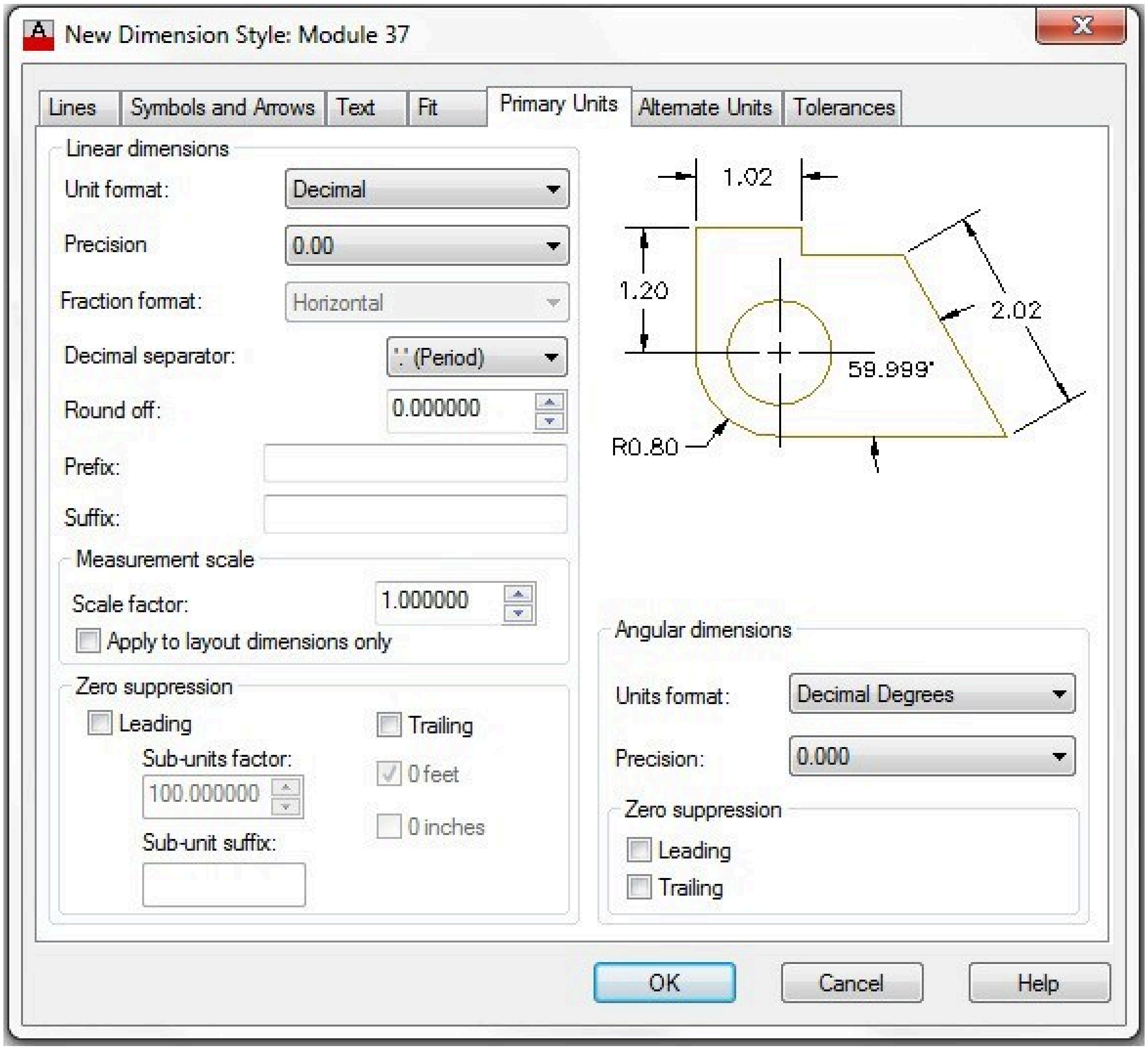
● Use overall scale of: 1.000000

Figure Step 16B

AUTHOR'S COMMENTS: The scale of dimension features is the scale applied to features only. For example, arrowheads, text, and offsets. It has no effect to the scale of the model itself.

Step 17

Enable the Primary Units tab. (Figure Step 17)



New Dimension Style: Module 37

Lines
Symbols and Arrows
Text
Fit
Primary Units
Alternate Units
Tolerances
1.02
1.20
2.02
59.999°
R0.80
OK
Cancel
Help

Linear dimensions

Unit format: Decimal

Precision: 0.00

Fraction format: Horizontal

Decimal separator: '.' (Period)

Round off: 0.000000

Prefix:

Suffix:

Measurement scale

Scale factor: 1.000000

☐ Apply to layout dimensions only

Zero suppression

☐ Leading

Sub-units factor: 100.000000

Sub-unit suffix:

☐ Trailing

☑ 0 feet

☐ 0 inches

Angular dimensions

Units format: Decimal Degrees

Precision: 0.000

Zero suppression

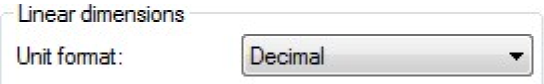
☐ Leading

☐ Trailing

Figure Step 17

Step 18

In the Linear dimensions area, set the Precision to two decimal places. (Figure Step 18)



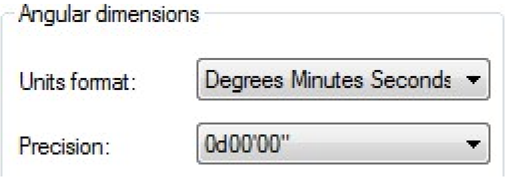
Linear dimensions

Unit format: Decimal

Figure Step 18

Step 19

In the Angular dimensions area, set the Units format to Degrees Minutes Seconds and set the Precision to 0d00’00”. (Figure Step 19)



Angular dimensions

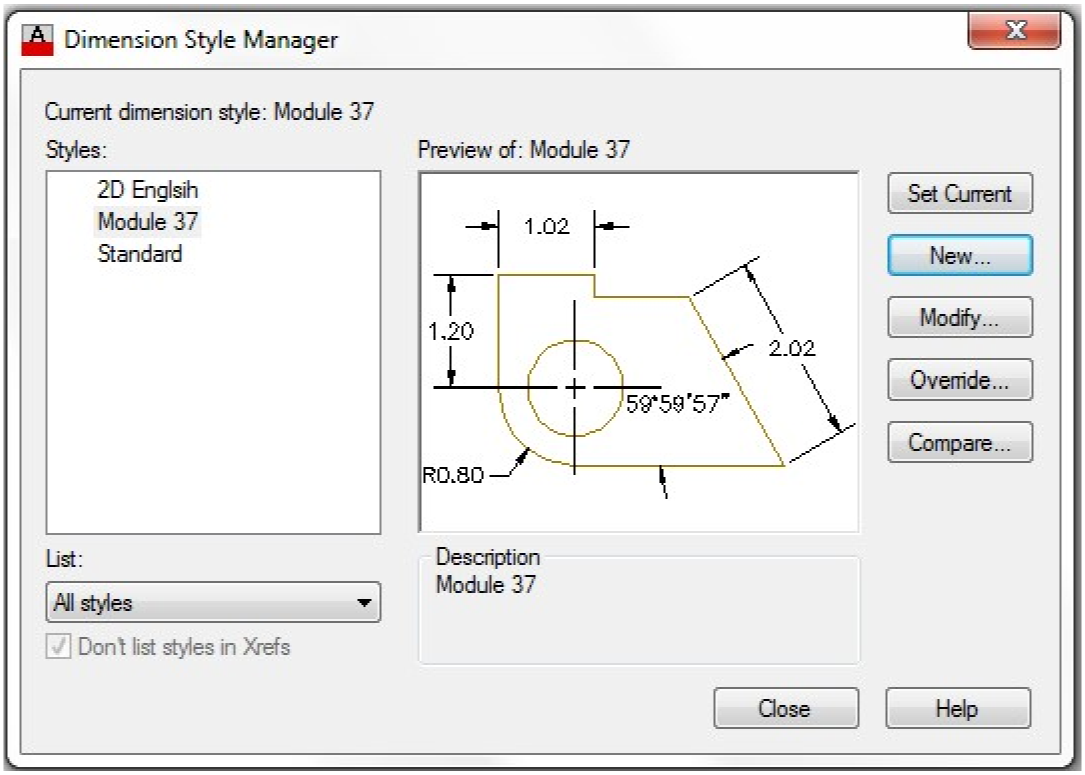
Units format: Degrees Minutes Seconds

Precision: 0d00'00"

Figure Step 19

Step 20

Click OK and you will return to the Dimension Style Manager dialogue box. Click Close. (Figure Step 20)



Dimension Style Manager

Current dimension style: Module 37

Styles:

2D Englsih
Module 37
Standard

Preview of: Module 37

List: All styles

☑ Don't list styles in Xrefs

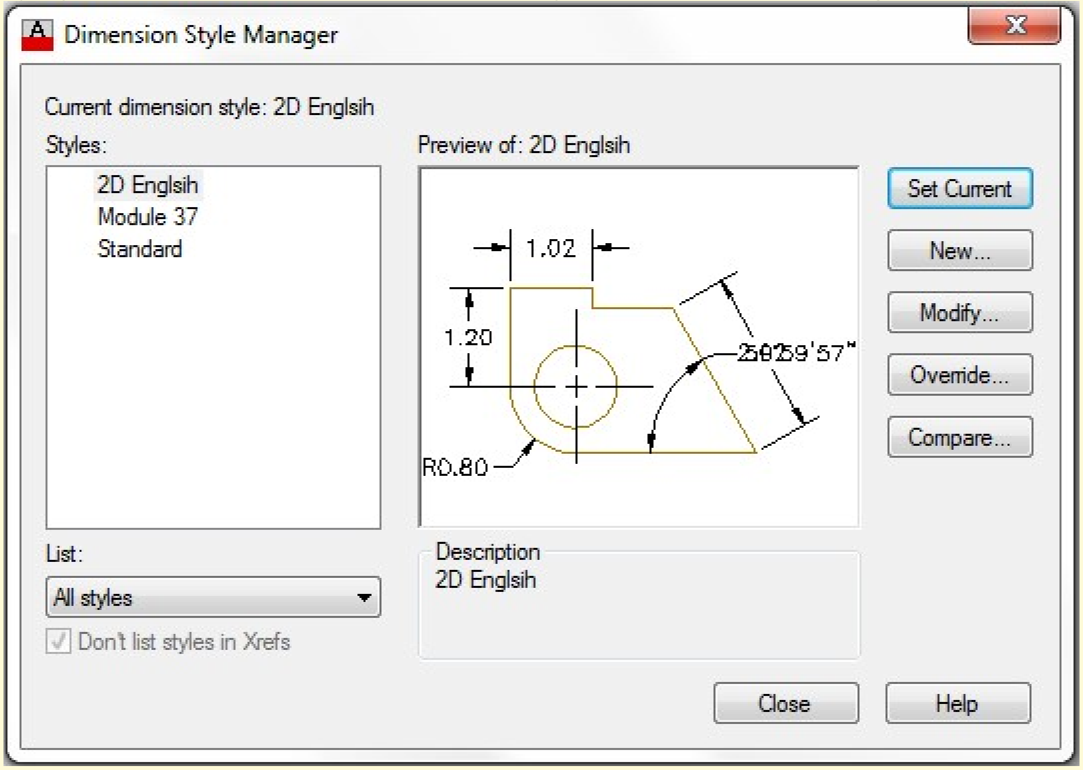
Description
Module 37

Figure Step 20

Step 21

Save and close the drawing.

USER TIP: A very common error, when using the DDIM command, is not setting the current style. Ensure that the style you want to use is the current dimension style. In this case, it is 2D English.



Dimension Style Manager

Current dimension style: 2D Englsih

Styles:

2D Englsih

Module 37

Standard

Preview of: 2D Englsih

List: All styles

☑ Don't list styles in Xrefs

Description

2D Englsih

Drafting Lesson: Center Lines

In multiview drawings, the *center line* is used to indicate the location of the axis of symmetry. Placing center lines on all objects that have a symmetrical shape will help the reader and save the you from inserting a lot of dimensions as you will see in future modules. Below are some examples of typical applications of the use of center lines in a multiview drawing.

A ‘ C ‘ with an ‘ L ‘ through it is the symbol for a center line. Center lines are drawn as repeating long and short lines. See Figure37-2.

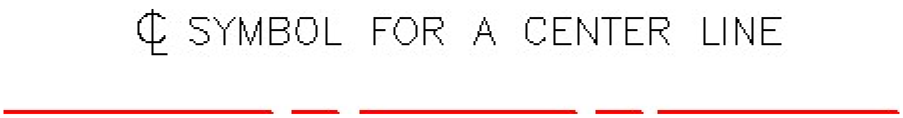


Figure 37-2

Figure 37-3 shows a center line indicating the center of the circle with two short lines called a *center* mark intersecting at the center. Note in the right sideview, the center line follows the length of the cylinder.

When the symmetry ends, so does the center line.

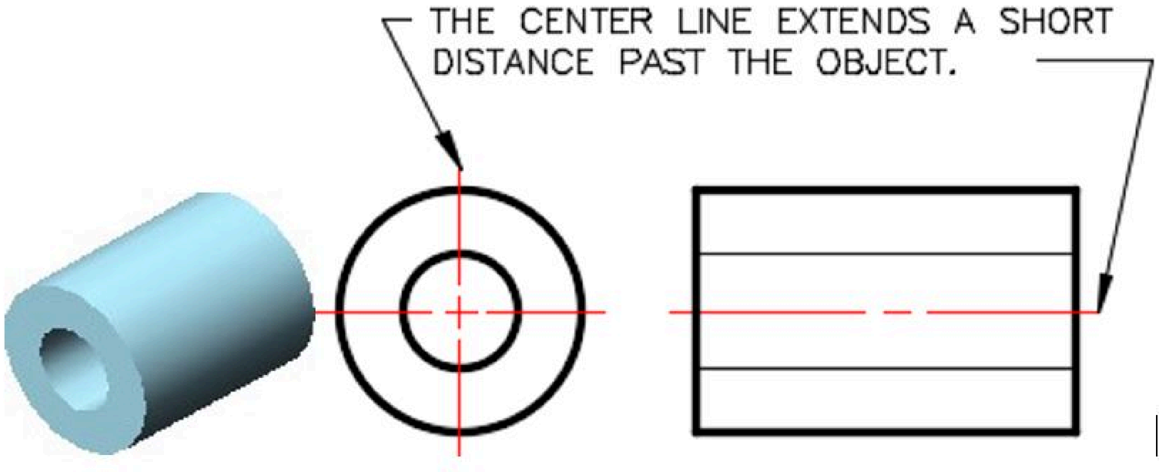

THE CENTER LINE EXTENDS A SHORT DISTANCE PAST THE OBJECT.

Figure Step 37-3

Note how the center line ends on the left side of the arc. See Figure 37-4

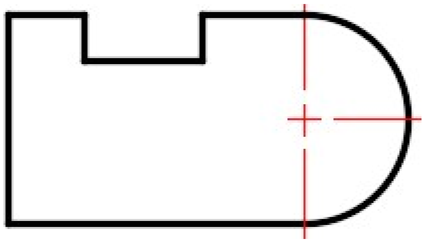


Figure Step 37-4

Figure 37-5 shows the center lines along with the hidden lines that indicate a hole going through the object. The center line on the top and bottom circles stops at the circle.

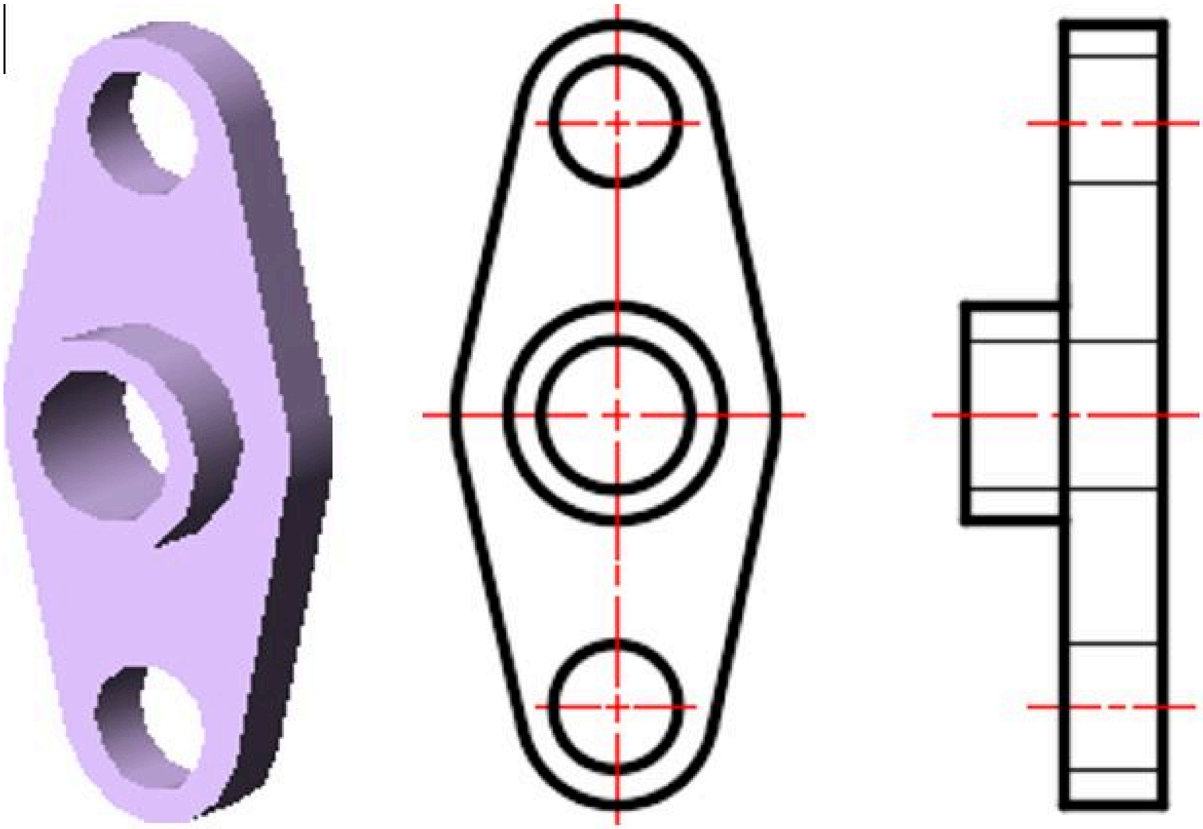


Figure Step 37-5

In Figure 37-6, note how center lines are drawn for an array of circles.

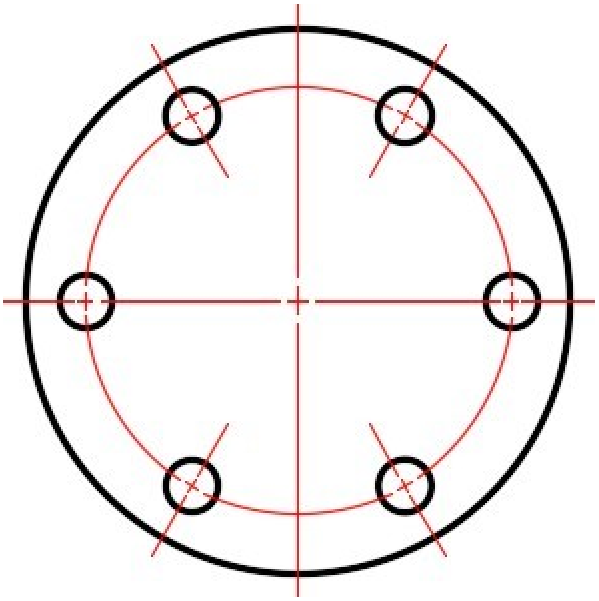
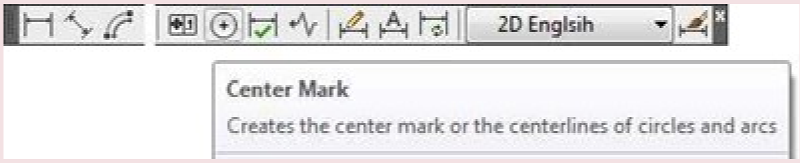


Figure Step 37-6

AutoCAD Command: DIMCENTER

The DIMCENTER command is used to insert center lines or center marks on circles or arcs.

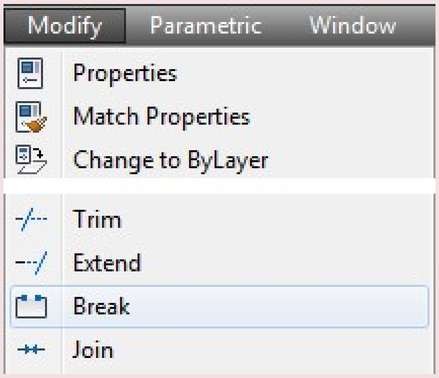
Shortcut: none



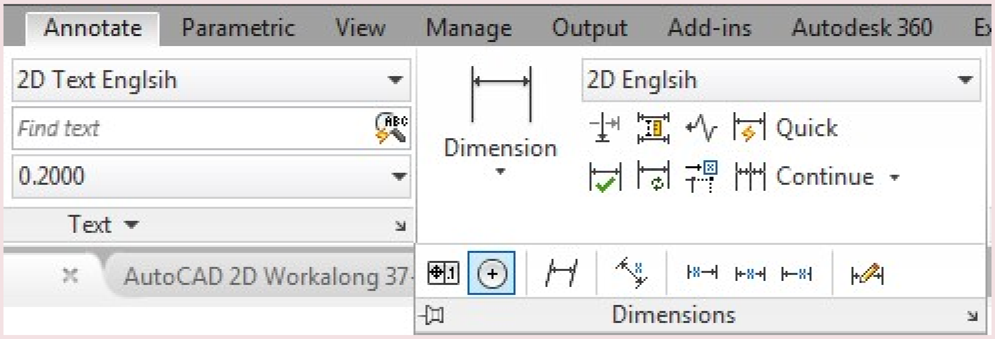
Center Mark

Creates the center mark or the centerlines of circles and arcs

Dimension toolbar


Modify
Parametric
Window
Properties
Match Properties
Change to ByLayer
Trim
Extend
Break
Join

Dimension Pull-down

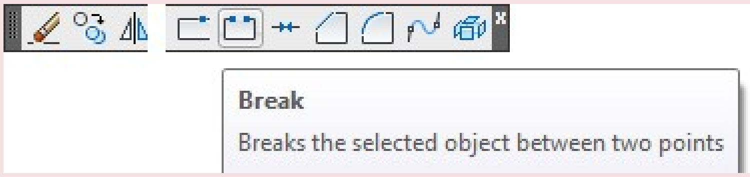

Annotate
Parametric
View
Manage
Output
Add-ins
Autodesk 360
2D Text Englsih
Find text
0.2000
Text
Dimension
2D Englsih
Quick
Continue
AutoCAD 2D Workalong 37-
Dimensions

Annotate Ribbon

AutoCAD Command: BREAK

The BREAK command is used to break an object into two objects.

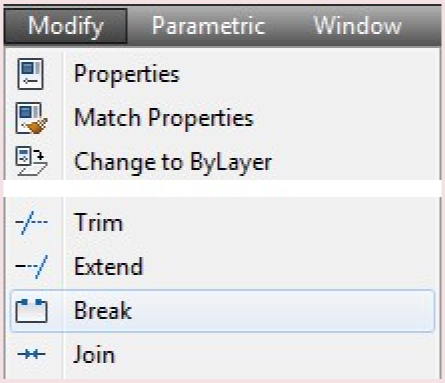
Shortcut: **BR**



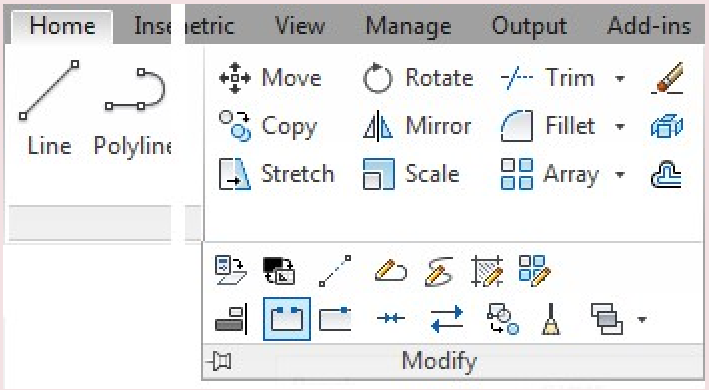
Break

Breaks the selected object between two points

Modify Toolbar


Modify
Parametric
Window
Properties
Match Properties
Change to ByLayer
Trim
Extend
Break
Join

Modify Pull-down


Home
Inse
etric
View
Manage
Output
Add-ins
Line
Polyline
Move
Rotate
Trim
Copy
Mirror
Fillet
Stretch
Scale
Array
Modify

Home Ribbon

USER TIP: To avoid unwanted results, disable object snap when using the BREAK command.

WORKALONG: Drawing Center Lines

Step 1

Open the drawing: AutoCAD 2D Workalong 37-1. At the same time, open the drawing: AutoCAD 2D Workalong 15-2. With the drawing: AutoCAD 2D Workalong 15-2 as the current drawing, using the COPYBASE command and the base point of the 0,0, use a window and select all drawing objects.

Step 2

Change the current drawing to: AutoCAD 2D Workalong 37-1. Enter the PASTECLIP command and paste the objects using the insert point of 0,0. (Figure Step 2)

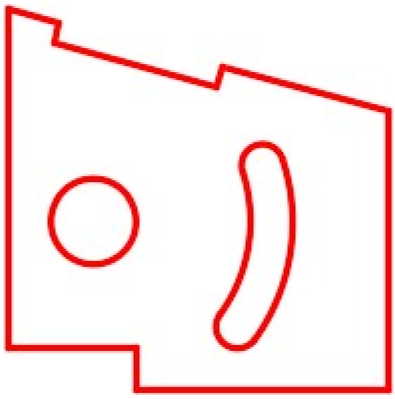


Figure Step 2

Step 3

Using the SAVEAS command, save the drawing with the name: AutoCAD 2D Workalong 37-2.

Step 4

Enable layout: Module Layout A. Change the current space to Paper. On layer: Viewport, create a viewport. Set the scale of the viewport to 0.75:1 and lock the display. (Figure Step 4)

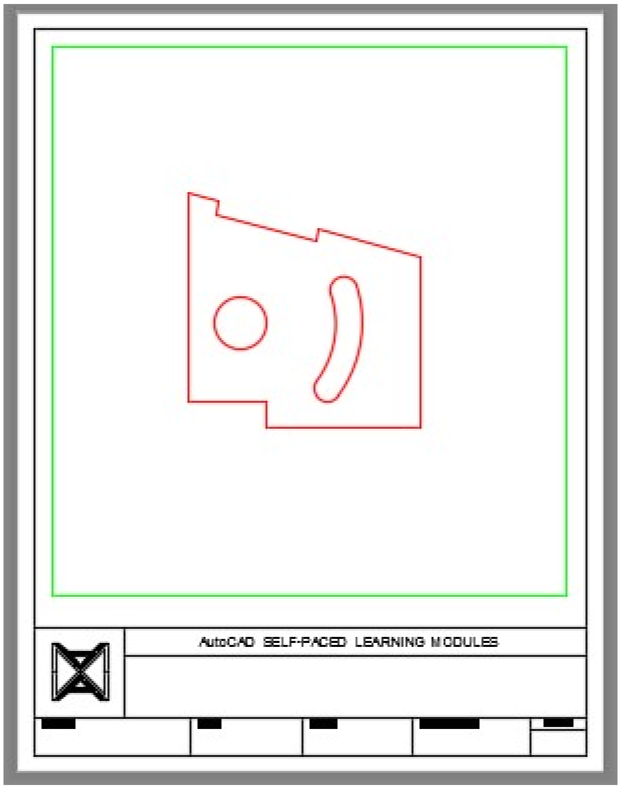
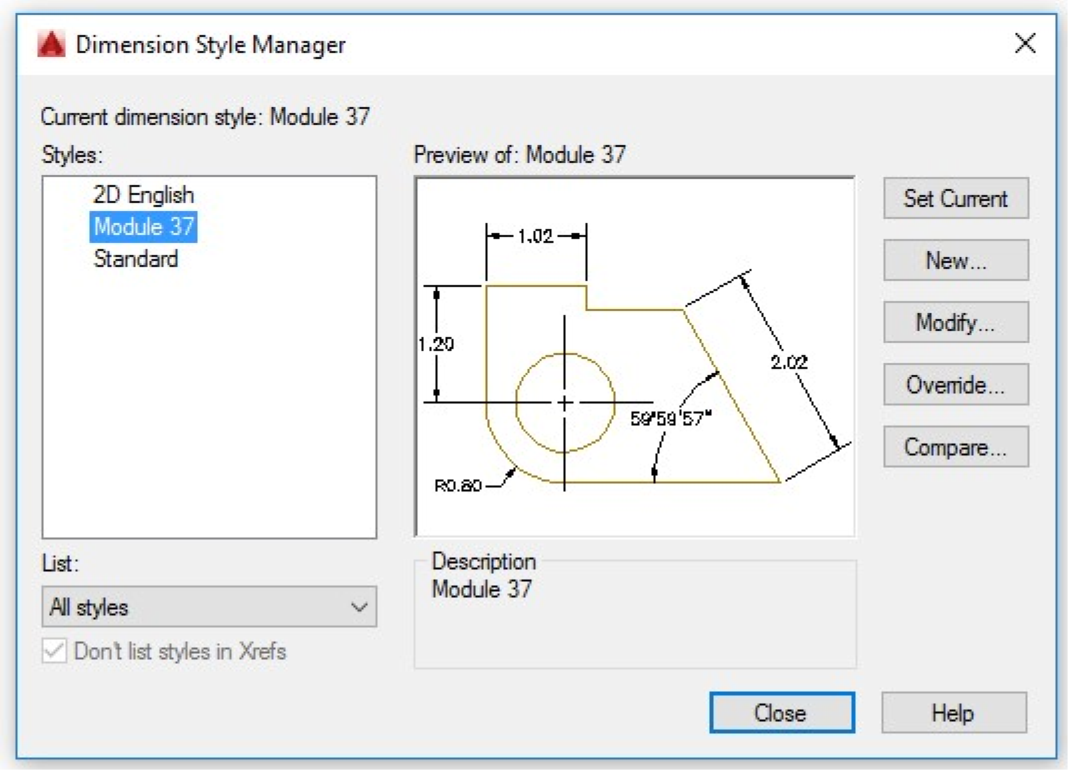


Figure Step 4

AUTHOR’S COMMENTS: Complete the remainder of this workalong by drawing in Paper space on the layout.

Step 5

Using the DDIM command, open the Dimension Style Manager dialogue box. Set the current dimensioning style: Module 37. (Figure Step 5)



Dimension Style Manager

Current dimension style: Module 37

Styles:

- 2D English
- Module 37
- Standard

Preview of: Module 37

List: All styles

☑ Don't list styles in Xrefs

Description

Module 37

1.02
1.20
2.02
59°59'57"
R0.80
Set Current
New...
Modify...
Override...
Compare...
Close
Help

Figure Step 5

AUTHOR’S COMMENTS: **Important** Ensure that the current space is set to Paper.

Step 6

On layer: Construction, draw a construction circle using the center of the small circle as the center and select the radius by eye. Make it a short distance outside of the arc. (Figure Step 6)

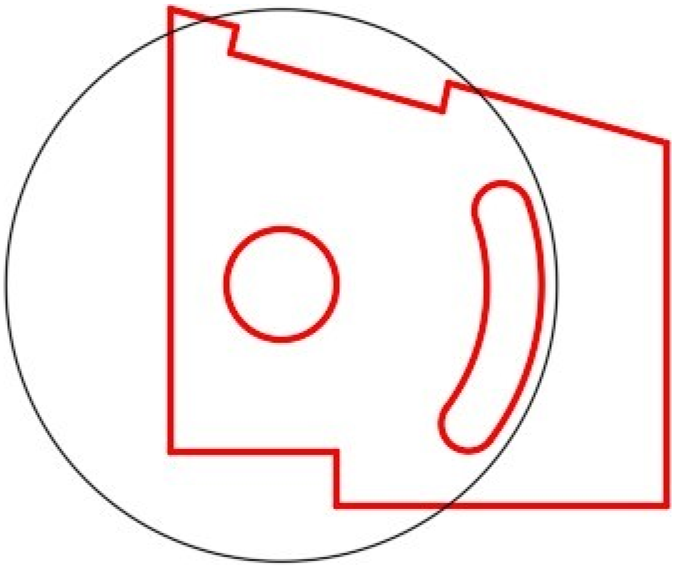


Figure Step 6

Step 7

Set layer: Center Lines as the current layer.

Step 8

Enter the DIMCENTER command. When prompted, select the circle. It will insert center lines in the circle. (Figure Step 8)

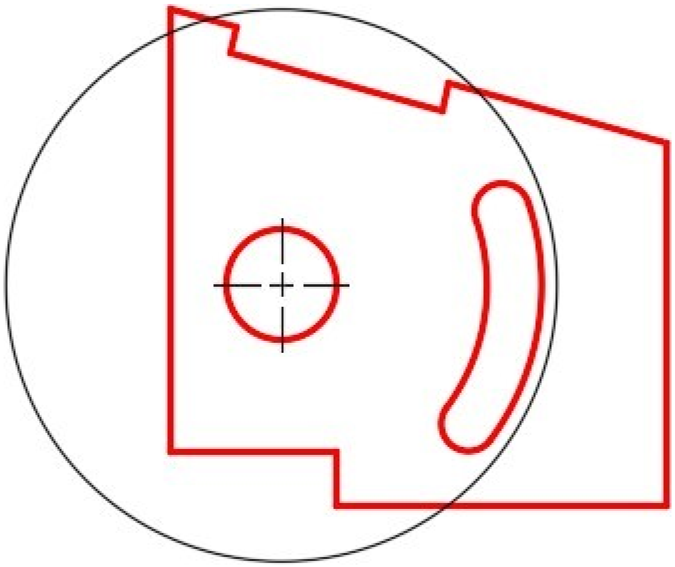


Figure Step 8

Step 9

Draw two lines from the center of the small circle to the center of the small arcs as shown in the figure. (Figure Step 9)

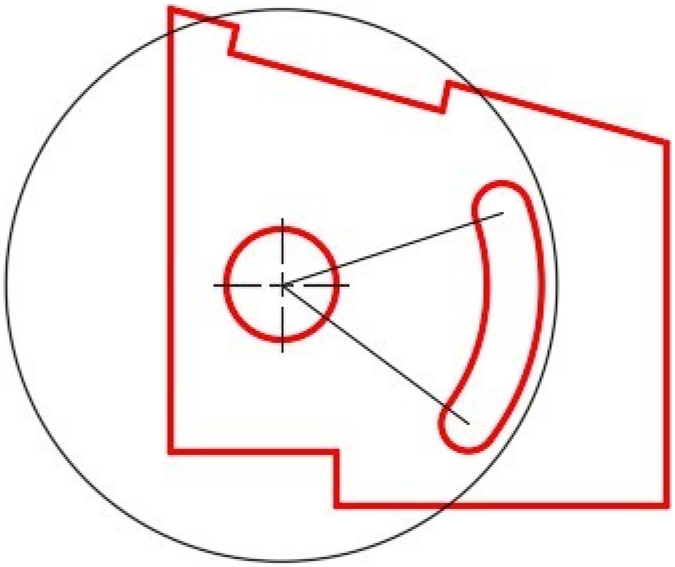


Figure Step 9

Step 10

Using the EXTEND command, extend the lines to the large construction circle. (Figure Step 10)

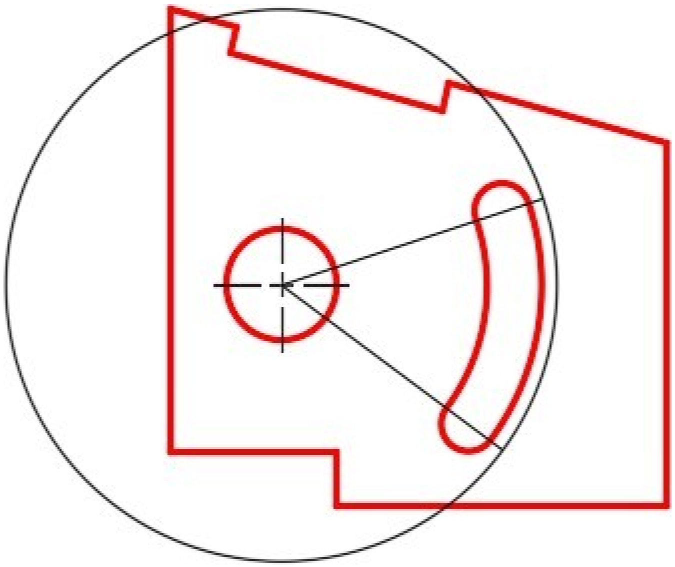


Figure Step 10

Step 11

Turn layer: Construction off. Draw a circle on layer: Center Lines with the center at the center of the small circle and the radius as the center of the arc. Ensure that you use snap mode to snap to the exact centers of the circle and the arc. (Figure Step 11)

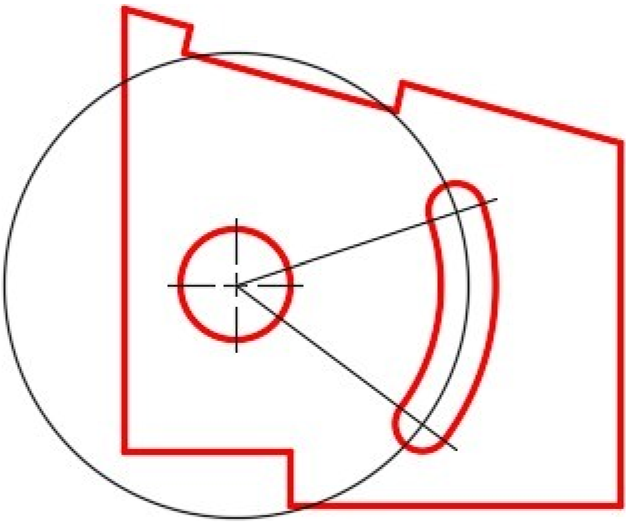


Figure Step 11

Step 12

Ensure that object snap is disabled and enter the BREAK command as shown below. Make sure that you select location P1 first and then P2. Since arcs are constructed counterclockwise, you also have to break them counterclockwise. (Figures Step 12A and 12B)

Command: **BREAK**

select object: **P1**

Specify second break point or [First point]: **P2**

(The locations P1 and P2 can be selected by eye. This is a center line so the exact length is not that important. Try to match the figure as best you can.)

Command:

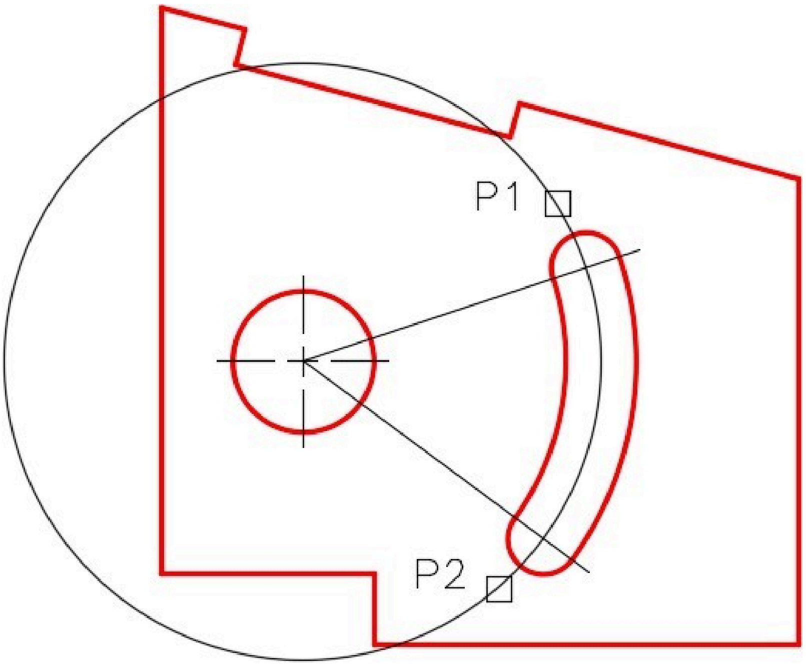


Figure Step 12A

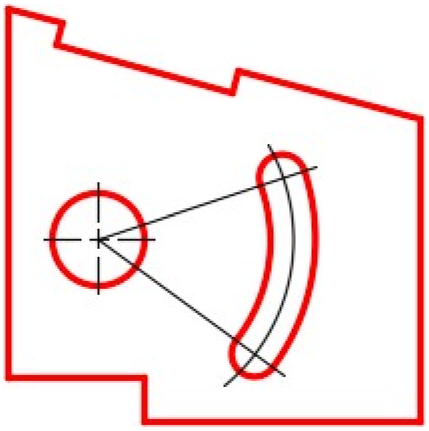
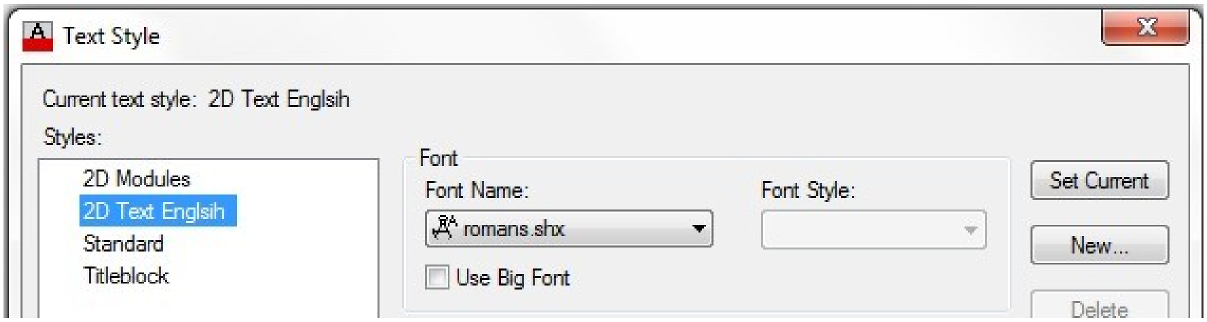


Figure Step 12B

AUTHOR'S COMMENTS: Ensure that you have object snap is disabled when you use the BREAK command.

Step 13

Using the BREAK command, break the end off of the center line. Select the location P3 for the first point and for the second point, select anywhere past the end of the line. This will remove everything to the end of the line from point P3. (Figure Step 13)



Current text style: 2D Text Englsih

Styles:

- 2D Modules
- 2D Text Englsih
- Standard
- Titleblock

Font Name: romans.shx

☐ Use Big Font

Figure Step 13

Step 14

Using the BREAK command, break the end off of the other center line. (Figure Step 14)

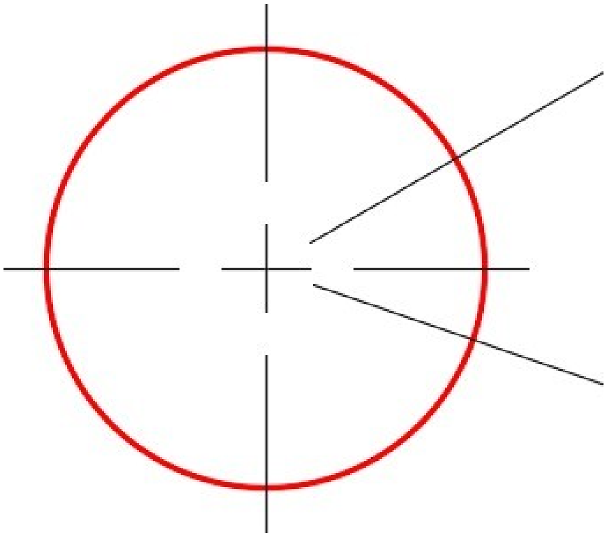


Figure Step 14

Step 15

Zoom in on the small arc and break a small part of the line as shown in the figure. Do that 7 more times to draw the center lines for the arcs. It is best to zoom in close and break the lines by eye. The more you practice this, the better you will get doing it. (Figure Step 15A, 15B, and 15C)

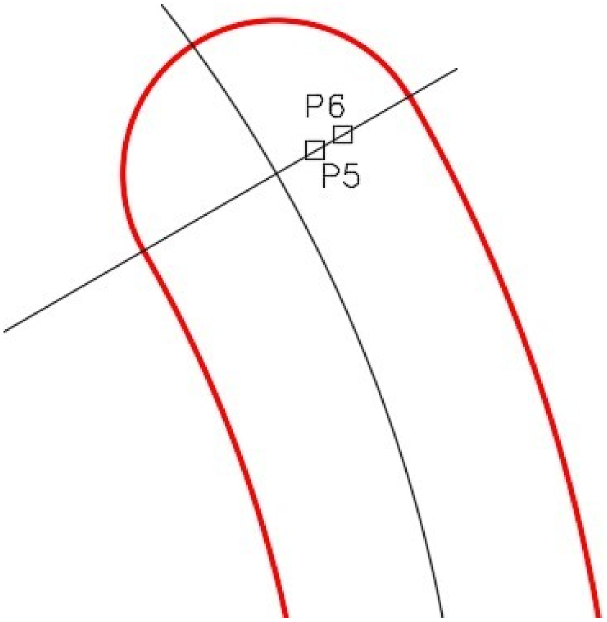


Figure Step 15A

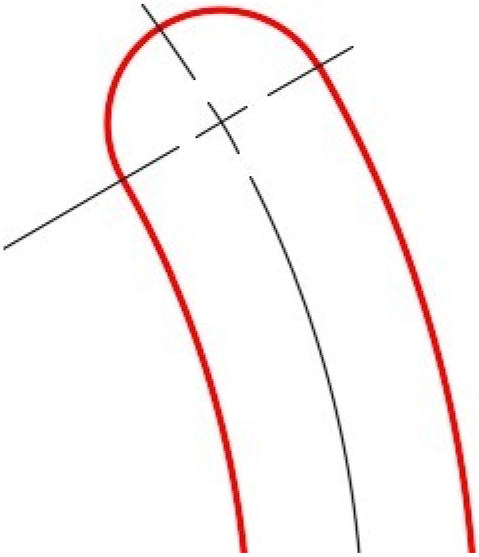


Figure Step 15B

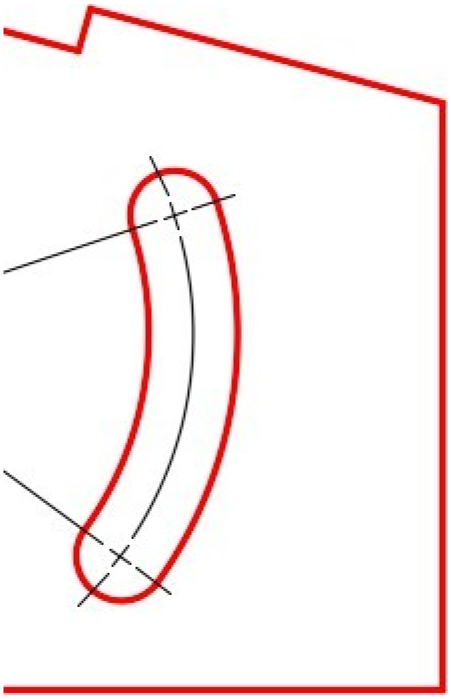


Figure Step 15C

Step 16

Using what you learned in this workalong, draw the center line for the slot at the top. The center line runs from the center of the circle perpendicular to the line, then extend it. You will have to add a construction line to extend to. (Figure Step 16)

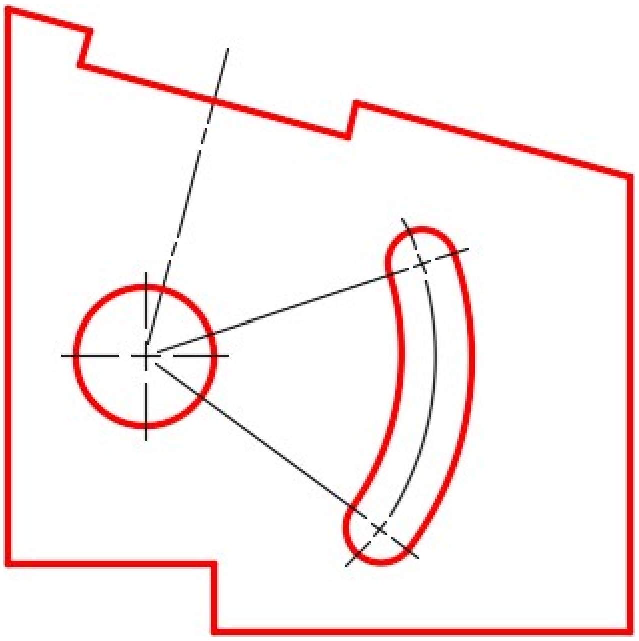


Figure Step 16

AUTHOR’S COMMENTS: If you use the perp object snap mode it is easy to draw the line.

Step 17

The layout should appear as shown in the figure. (Figure Step 17)

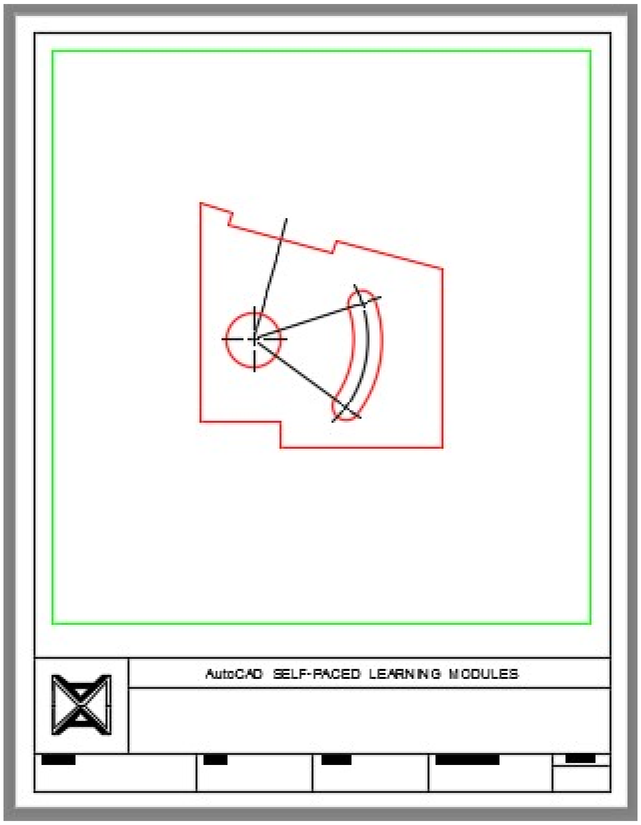

AutoCAD SELF-PACED LEARNING MODULES

Figure Step 17

Step 18

Change the current space to Model. Your drawing should appear as shown in the figure. Since the center lines are in Paper space, the should not display. (Figure Step 18)

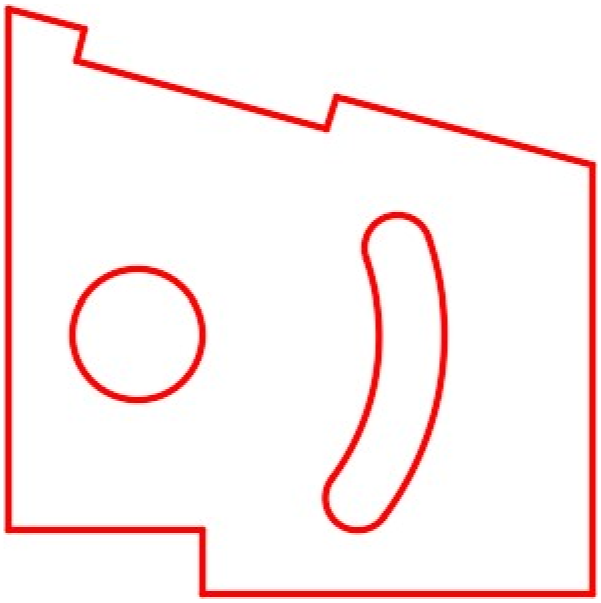


Figure Step 18

Step 19

Change the current space back to Paper space. (Figure Step 19)

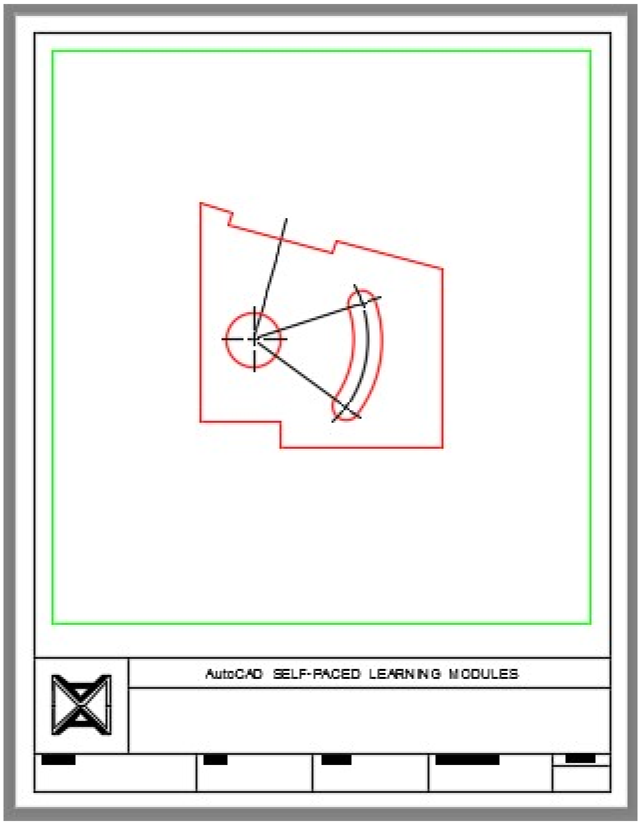


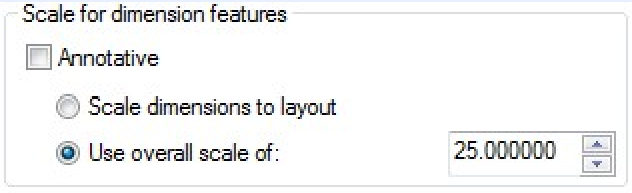
Figure Step 19

Step 20

Save and close the drawing.

USER TIP: The Standard dimensioning style is a special style and exists in every drawing. It cannot be deleted or purged from the drawing. Even though the Standard dimensioning style can be used, it is not a good practice to do this. It is much better to create your own styles, giving them descriptive names. If you are everrequired to merge your drawings together with a client's drawings (this is a very common practice), you will not have conflicting style names and all dimensions on the merged drawing will appear as they did in the original drawings.

MUST KNOW: The Scale for Dimension Features area, located on the Fit tab, allows you to select the scale factor you want applied to the dimensioning features when a dimension is inserted. This can be a very useful item and will save you time. For example, if you set a dimension style for drawing units in inches and you want the samefeature sizes for a millimeter drawing, set the Use of overall scale of 25 since there are 25.4 mm in an inch. Experiment with this number until you like the look of your dimensions. This number does not effect the actual dimensions, only the dimension features.

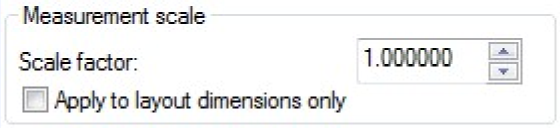

Scale for dimension features

☐ Annotative

○ Scale dimensions to layout

● Use overall scale of: 25.000000

MUST KNOW: The Measurement Scale box allows you to set the scale you want to apply to the object measurements when inserting dimensions. This is a very important setting and in almost cases it should be set to 1. If the drawing objects are constructed at full scale (actual size), then a setting of 1 inserts dimensions of the real size of the object.


Measurement scale

Scale factor: 1.000000

☐ Apply to layout dimensions only

Key Principles

Key Principles in Module 37

1. Associative dimensions are dimensions that are intelligent, similar to an associative hatch pattern and are tied to the object they are dimensioning. An exploded dimension is simply a collection of AutoCAD objects indicating a measurement on the object and has no relation to the object it is dimensioning.
2. Definition points are points that are automatically inserted, on layer Defpoints, on the drawing by dimensioning commands for associative dimensions. The Defpoints layer cannot be deleted or purged from the drawing.
3. A dimensioning style controls the way your dimensions appear in your drawing. When a dimension is inserted, it is assigned the property of the current dimension style and uses the style properties.

Lab Exercise 37-1

Time allowed: 30 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 37-1	2D Layout English	Inches

Step 1

Start a new drawing with the template shown above. Save and name the drawing: AutoCAD 2D Lab 37-1. At the same time, open the drawing: AutoCAD 2D Lab 13-1. Set drawing: AutoCAD 2D Lab 13-1 as the current drawing. Using the COPYBASE command and the base point of the 0,0, select all of the objects in the drawing.

Step 2

Change the current drawing to: AutoCAD 2D Lab 37-1. Enter the PASTECLIP command and paste the objects using the insert point of 0,0. (Figure Step 2)

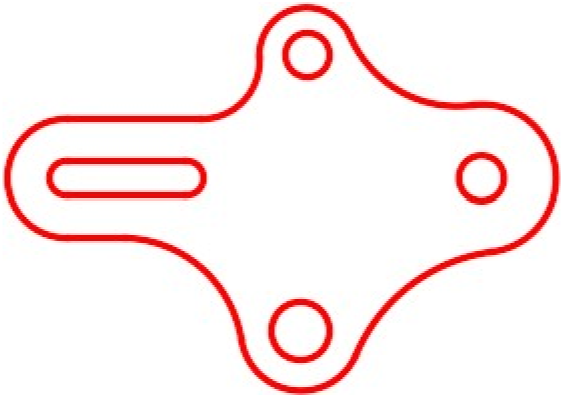


Figure Step 2

Step 3

Enable layout: Module Layout C. On layer: Viewport, create a viewport. Set the scale of the viewport to 1.5:1 and lock the display. (Figure Step 3)

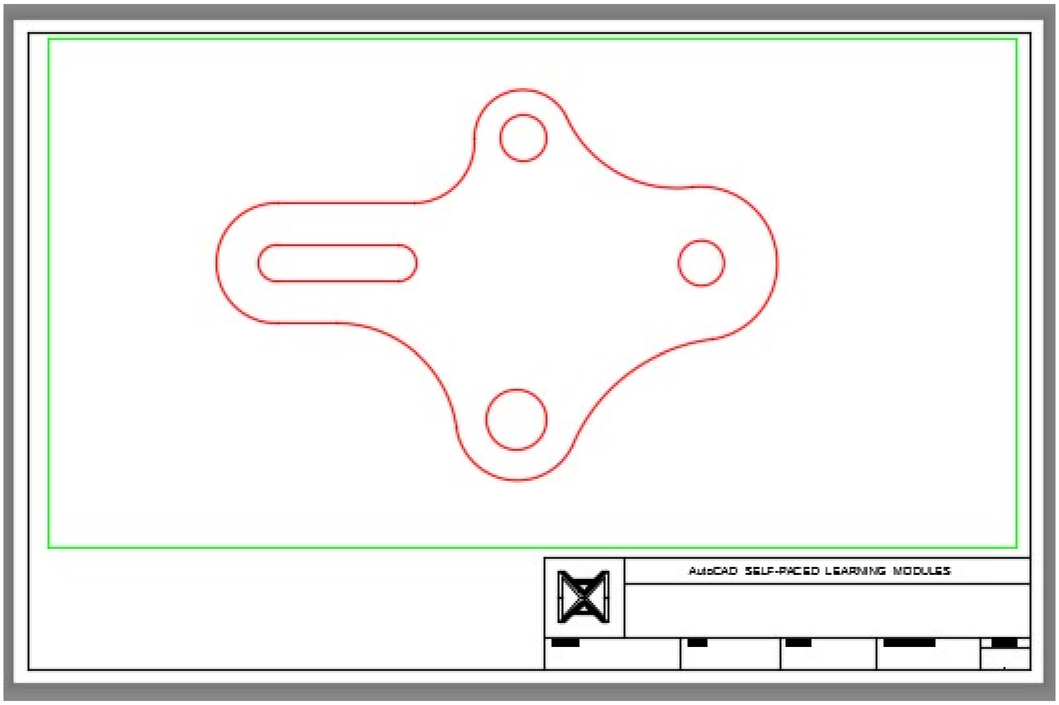

AutoCAD SELF-PACED LEARNING MODULES

Figure Step 3

AUTHOR'S COMMENTS: **Important** Ensure that the current space is Paper

Step 4

In Paper space, on layer: Center Lines, draw the center lines as shown in the figure. (Figure Step 4A and 4B)

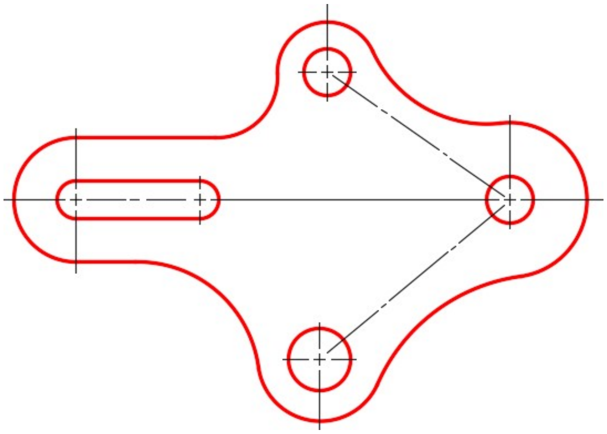


Figure Step 4A

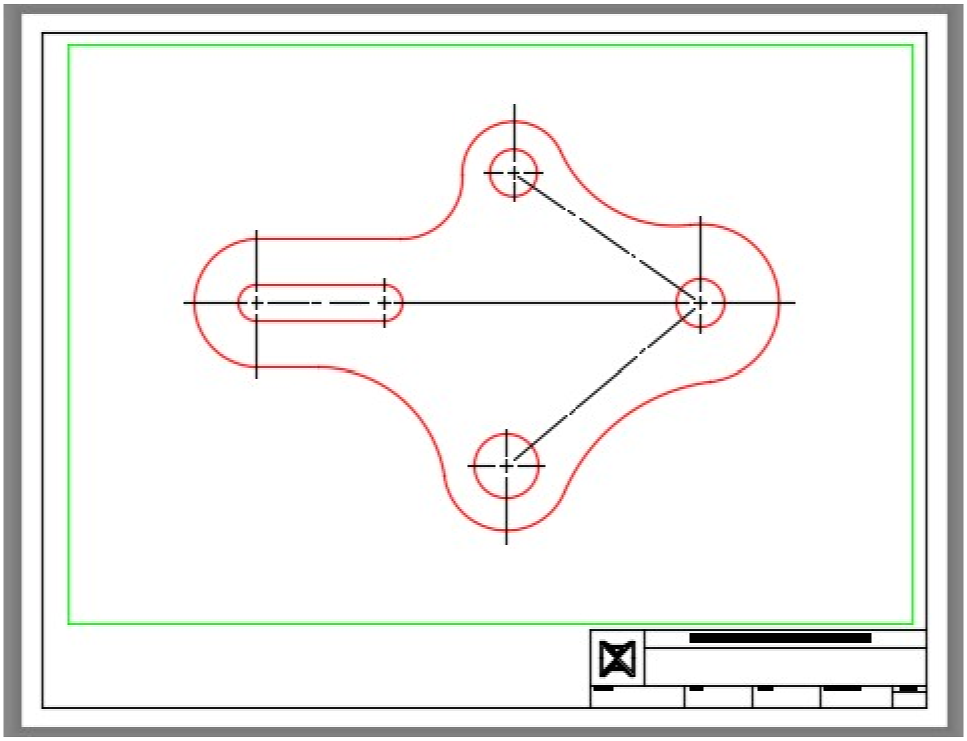


Figure Step 4B

Step 5

Save and close the drawing.

Lab Exercise 37-2

Time allowed: 30 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 37-2	2D Layout Metric	Millimeters

Step 1

Start a new drawing with the template shown above. Save and name the drawing: AutoCAD 2D Lab 37-2. At the same time, open the drawing: AutoCAD 2D Lab 12-2. Set drawing: AutoCAD 2D Lab 12-2 as the current drawing. Using the COPYBASE command and the base point of the 0,0, select all of the objects in the drawing.

Step 2

Change the current drawing to: AutoCAD 2D Lab 37-2. Enter the PASTECLIP command and paste the objects using the insert point of 0,0. (Figure Step 2)

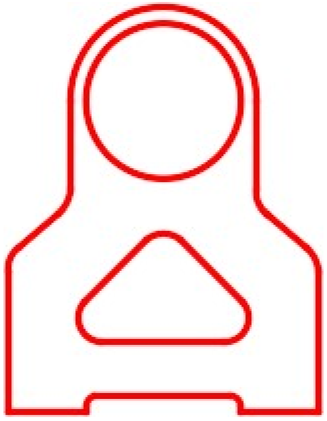


Figure Step 2

Step 3

Enable layout: Module Layout A3. On layer: Viewport, create a viewport. Set the scale of the viewport to 1.2:1 and lock the display. (Figure Step 3)

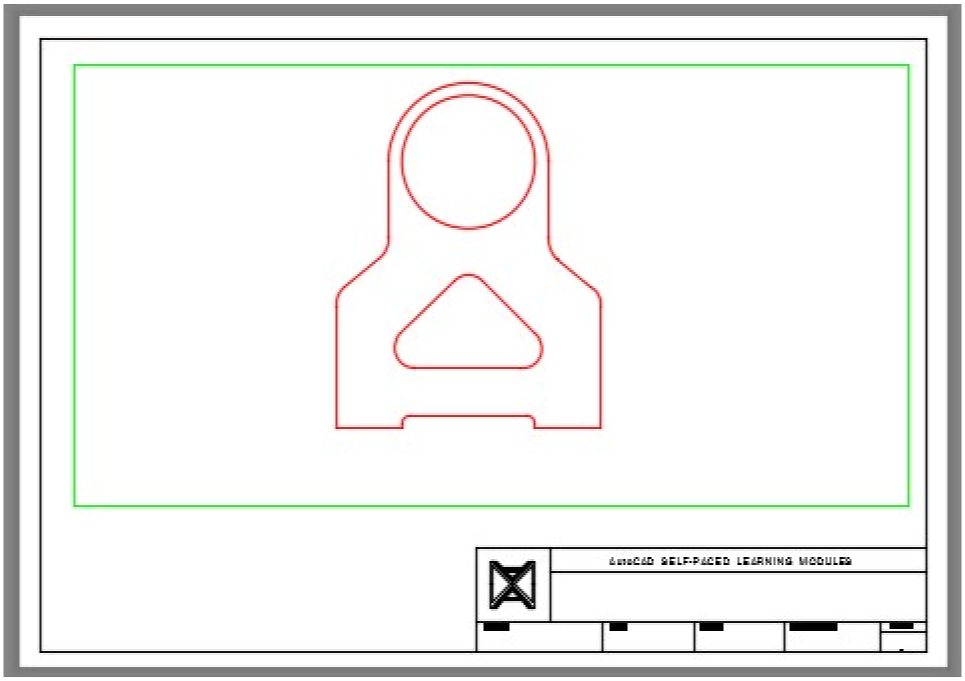

AutoCAD SELF-PACED LEARNING MODULES

Figure Step 3

AUTHOR'S COMMENTS: Important Ensure that the current space is Paper

Step 4

In Paper space, on layer: Center Lines, draw the center lines as shown in the figure. Figure Step 4A and 4B)

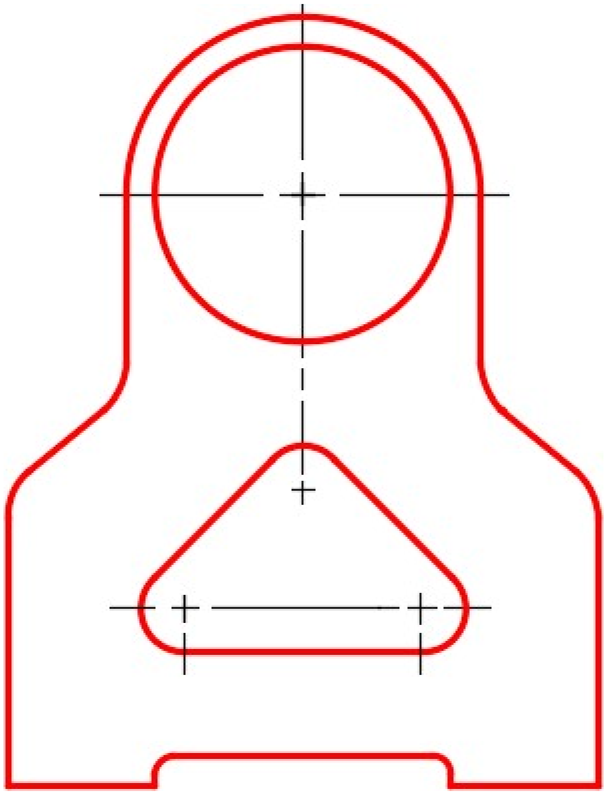


Figure Step 4A

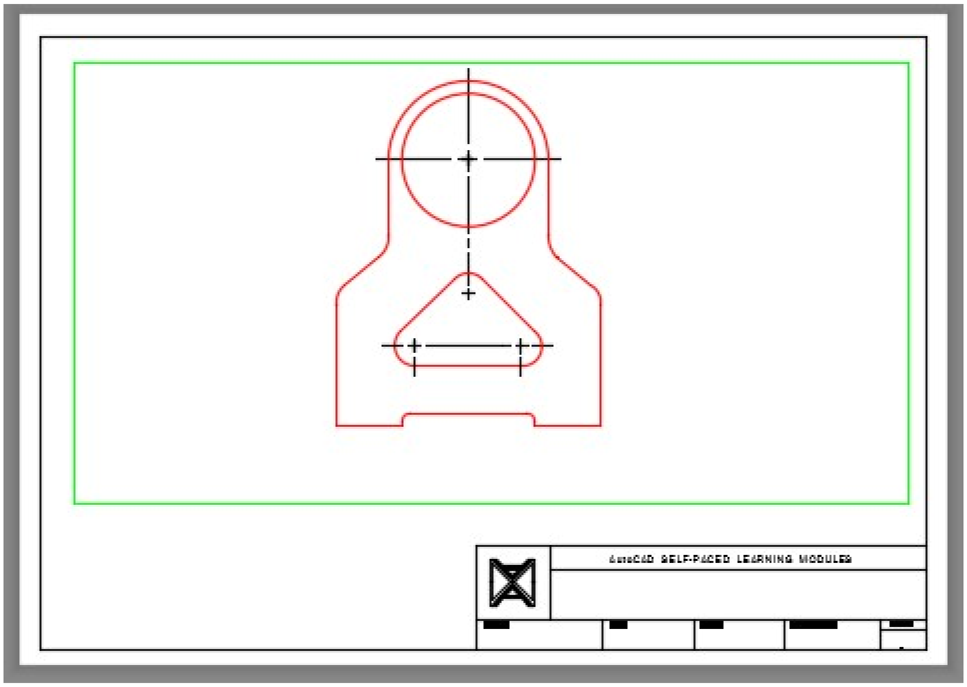

AutoCAD SELF-PACED LEARNING MODULES

Figure Step 4B

Step 5

Save and close the drawing.

Lab Exercise 37-3

Time allowed: 30 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 37-3	2D Layout English	Inches

Step 1

Start a new drawing with the template shown above. Save and name the drawing: AutoCAD 2D Lab 37-3. At the same time, open the drawing: AutoCAD 2D Lab 15-1. Set drawing: AutoCAD 2D Lab 15-1 as the current drawing. Using the COPYBASE command and the base point of the 0,0, select all of the objects in the drawing.

Step 2

Change the current drawing to: AutoCAD 2D Lab 37-3. Enter the PASTECLIP command and paste the objects using the insert point of 0,0. (Figure Step 2)

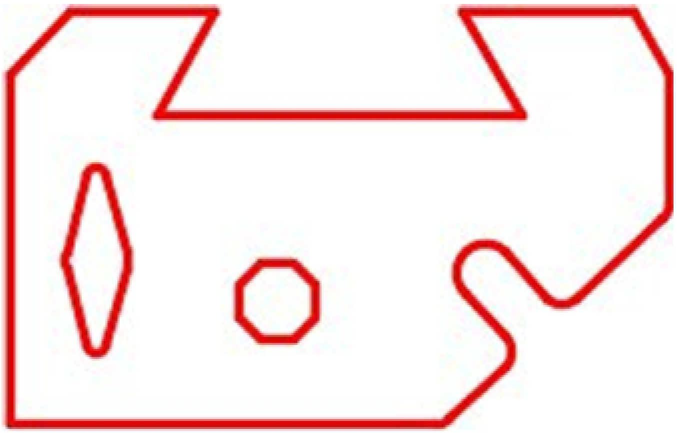


Figure Step 2

Step 3

Enable layout: Module Layout C. On layer: Viewport, create a viewport. Set the scale of the viewport to 1.25:1 and lock the display. (Figure Step 3)

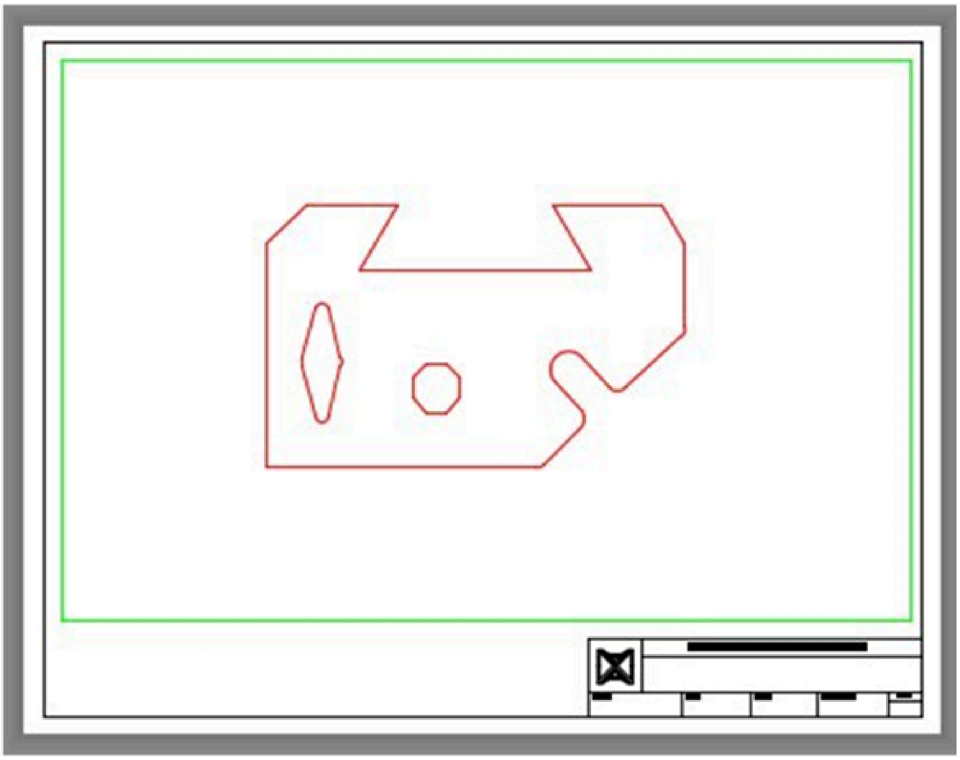


Figure Step 3

AUTHOR'S COMMENTS: Important Ensure that the current space is Paper.

Step 4

In Paper space, on layer: Center Lines, draw the center lines as shown in the figure.

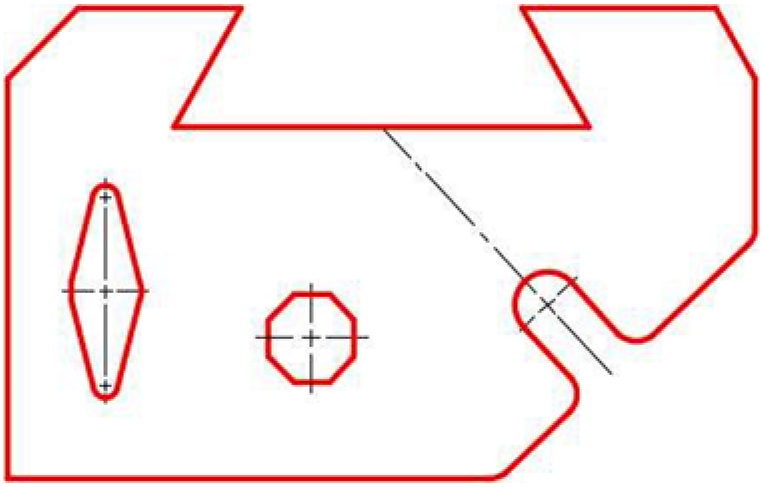


Figure Step 4A

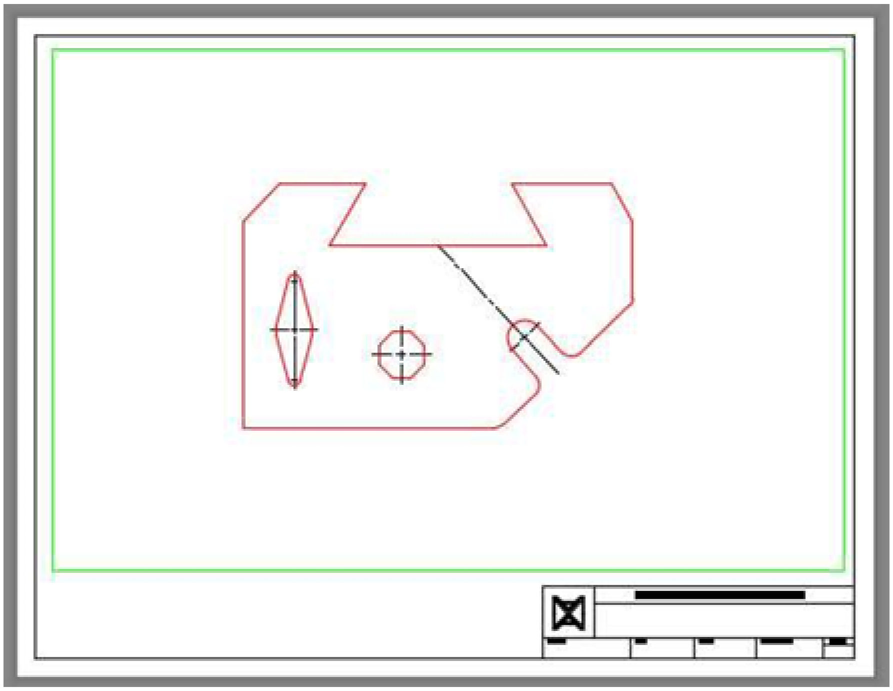


Figure Step 4B

Step 5

Save and close the drawing.

Module 38: Dimensioning – Part 2

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe the six basic dimensioning types.
2. Apply the DIMLINEAR, DIMALIGNED, DIMANGULAR, DIMDIAMETER, DIMRADIUS, and QLEADER commands to dimension a drawing.
3. Edit inserted dimensions using the Properties window.

Drafting Lesson Basic Dimension Types

Figure 38-1 shows the six basic dimension types that will be taught in this module. They are the linear, aligned, angular, diameter, radius, and leader dimensions.

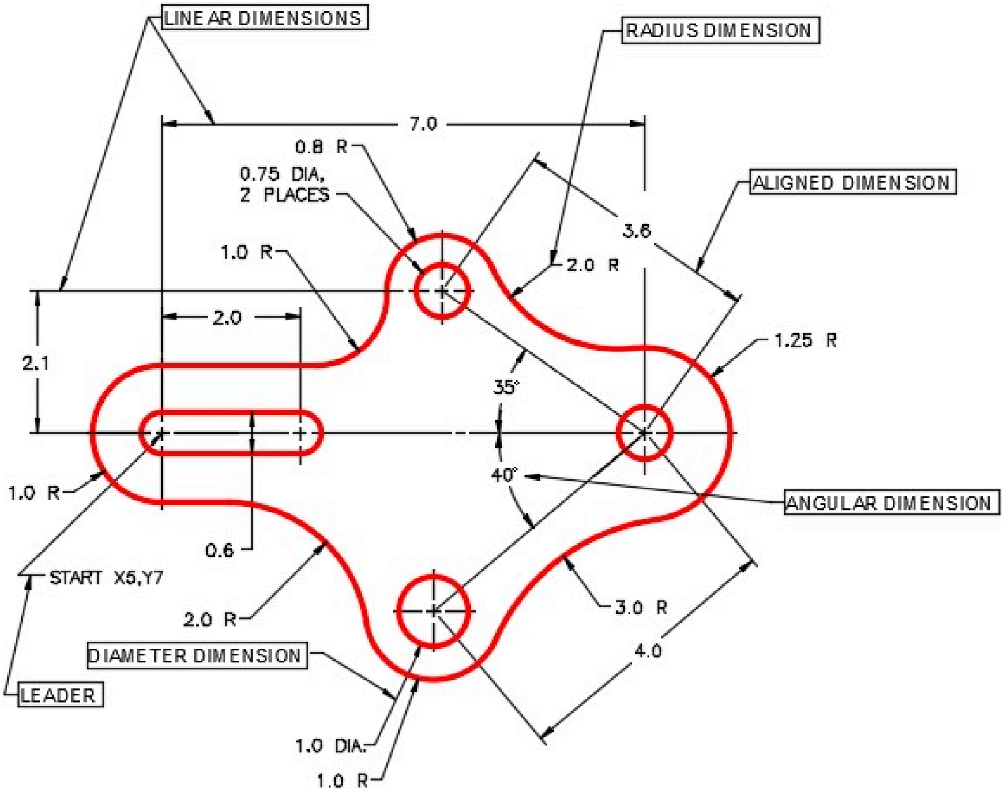
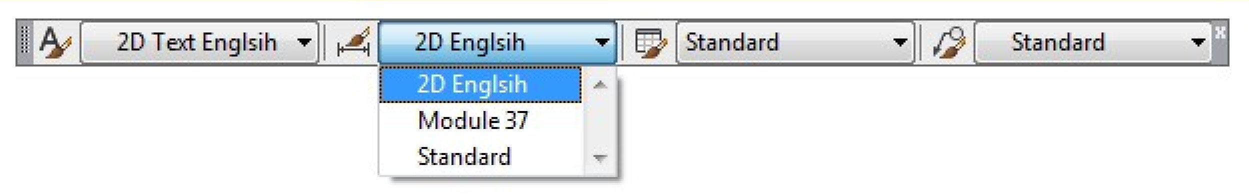

LINEAR DIMENSIONS
RADIUS DIMENSION
7.0
0.8 R
0.75 DIA,
2 PLACES
ALIGNED DIMENSION
3.6
1.0 R
2.0 R
1.25 R
2.0
2.1
35°
40°
ANGULAR DIMENSION
1.0 R
0.6
START X5,Y7
2.0 R
3.0 R
4.0
DIAMETER DIMENSION
LEADER
1.0 DIA.
1.0 R

Figure 38-1
Basic Dimensioning Types

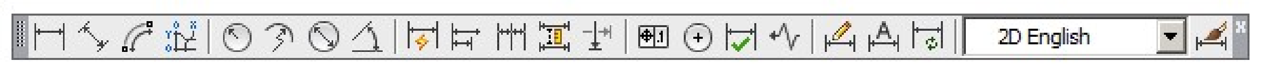
Inserting Dimensions

In this module, inserting and editing *linear, aligned, angular, radius, diameter*, and *leader* dimensions will be taught. When a dimension is inserted, it resides on the current layer and assumes the color of that layer.

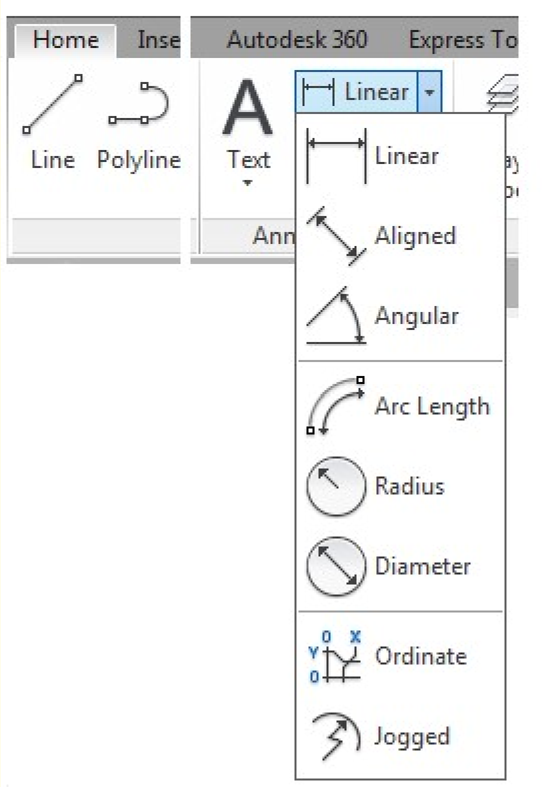
USER TIP: The current dimensioning style will appear in the Dimensioning toolbar, the Styles toolbar, and Home ribbon. A quick way to check or change the current dimensioning style is with one of these drop-down menus. Try to get into the habit of using the Dimensioning toolbar or the Home ribbon to insert dimensions. It will speed your drawing productivity.


2D Text Englsih
2D Englsih
Standard
Standard
2D Englsih
Module 37
Standard

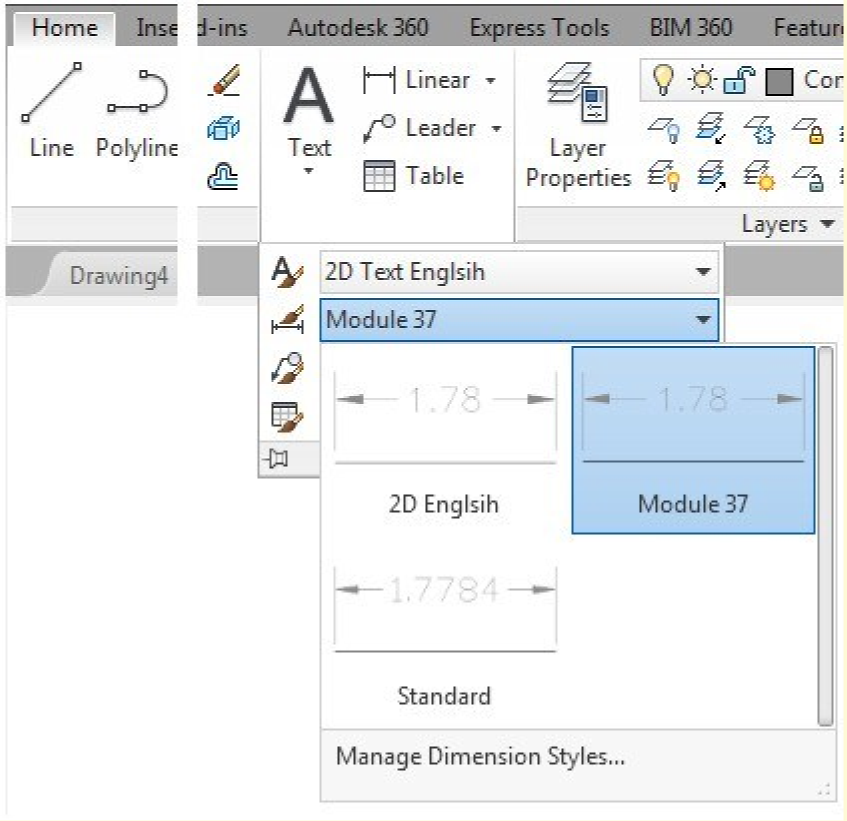
Style Toolbar



Dimension Toolbar


Home
Inse
Autodesk 360
Express To
Line
Polyline
Text
Ann
Linear
Linear
Aligned
Angular
Arc Length
Radius
Diameter
Ordinate
Jogged

Home Ribbon


Home
Inse
d-ins
Autodesk 360
Express Tools
BIM 360
Featur
Line
Polyline
Text
Linear
Leader
Table
Layer Properties
Cor
Layers
Drawing4
2D Text Englsih
Module 37
1.78
2D Englsih
1.78
Module 37
1.7784
Standard
Manage Dimension Styles...

Home Ribbon

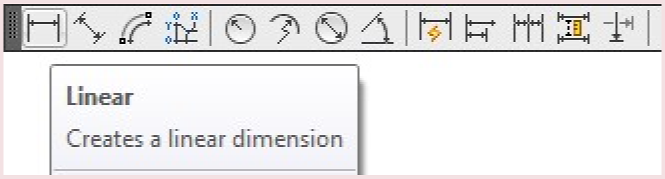
MUST KNOW: Before dimensioning a drawing, it is important to check the following settings:

1. The current layer. That is the layer that the inserted dimensions will reside on.
2. The current dimensioning style. The dimensions that you insert, with any dimensioning command, will use the properties of the current dimensioning style.
3. The DIMASSOC setting. This setting controls whether the inserted dimensions will be inserted as associative dimensions or exploded dimensions.

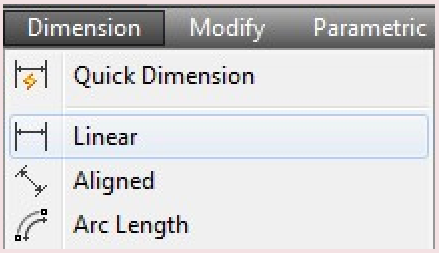
AutoCAD Command: **DIMLINEAR**

The DIMLINEAR command is used to insert a linear dimension. A linear dimension is a dimension that is either horizontal or vertical.

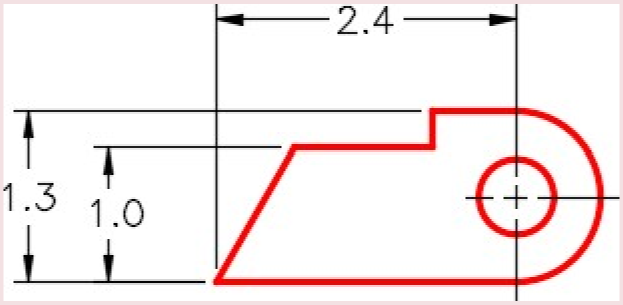
Shortcut: **DIMLIN**


Linear
Creates a linear dimension

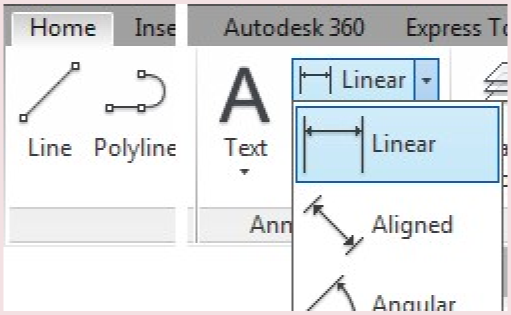
Dimension Toolbar


Dimension
Modify
Parametric
Quick Dimension
Linear
Aligned
Arc Length

Dimension Pull-down



Linear Dimension

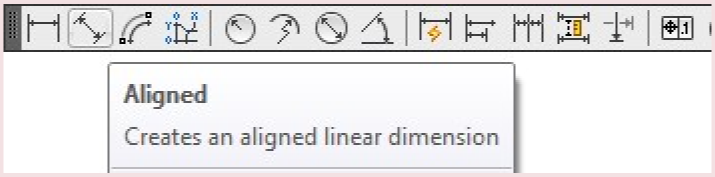

Home
Inse
Autodesk 360
Express To
Line
Polyline
Text
Ann
Linear
Linear
Aligned
Angular

Home Ribbon

AutoCAD Command: DIMALIGNED

The DIMALIGNED command is used to insert an aligned dimension.

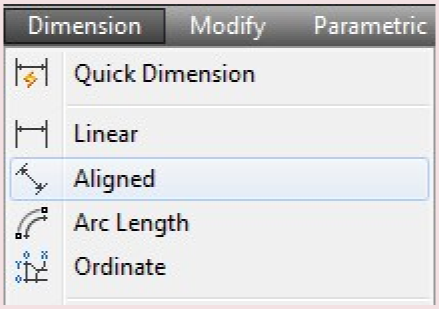
Shortcut: **DIMALI**



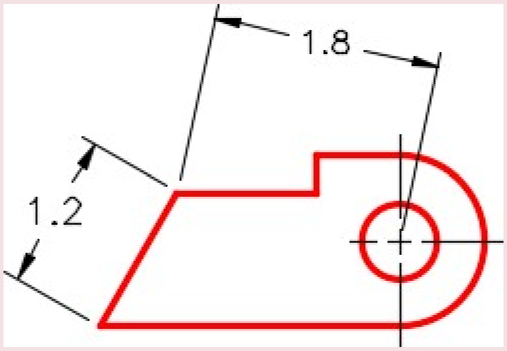
Aligned

Creates an aligned linear dimension

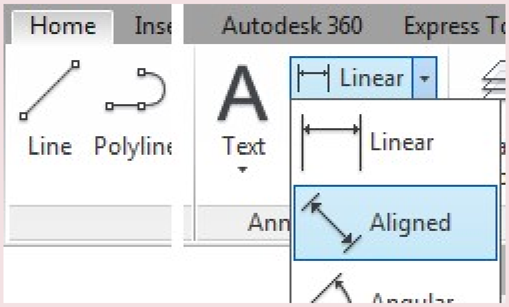
Dimension Toolbar


Dimension
Modify
Parametric
Quick Dimension
Linear
Aligned
Arc Length
Ordinate

Dimension Pull-down



Aligned Dimension

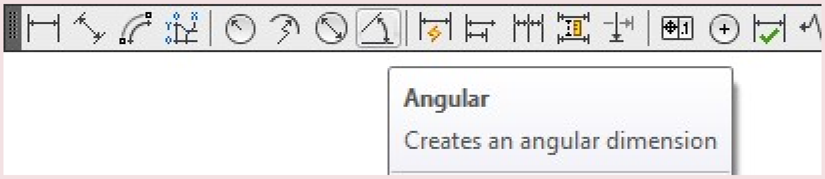

Home
Inse
Autodesk 360
Express To
Line
Polyline
Text
Ann
Linear
Linear
Aligned
Angular

Home Ribbon

AutoCAD Command: DIMANGULAR

The DIMANGULAR command is used to insert an angular dimension.

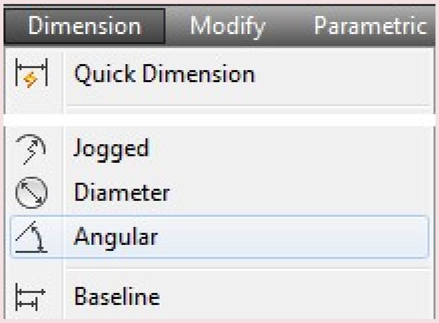
Shortcut: **DIMANG**



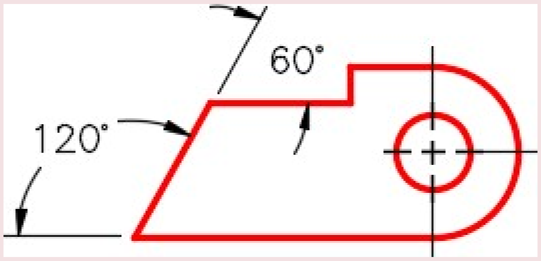
Angular

Creates an angular dimension

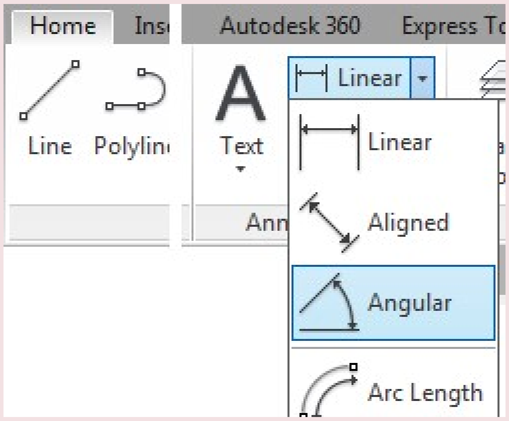
Dimension Toolbar


Dimension
Modify
Parametric
Quick Dimension
Jogged
Diameter
Angular
Baseline

Dimension Pull-down



Angular Dimension

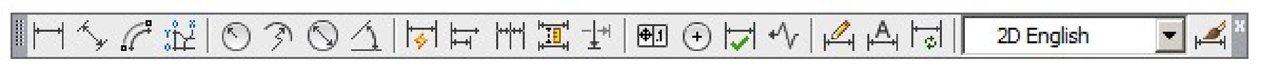

Home
Ins
Autodesk 360
Express To
Line
Polylin
Text
Ann
Linear
Linear
Aligned
Angular
Arc Length

Home Ribbon

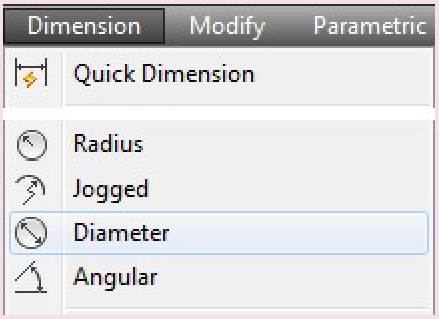
AutoCAD Command: DIMDIAMETER

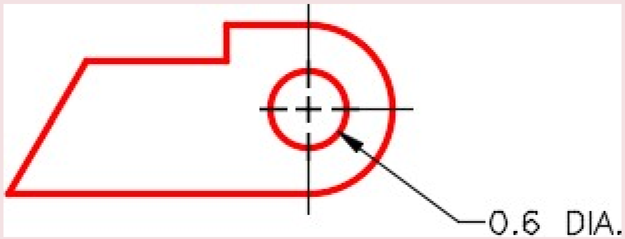
The DIMDIAMETER command is used to insert a diameter dimension.

Shortcut: **DIMDIA**

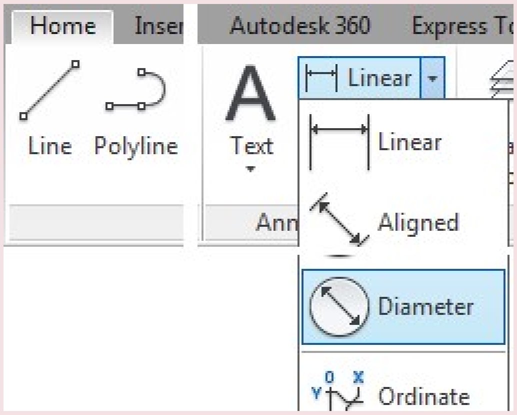


Dimension Toolbar


Dimension
Modify
Parametric
Quick Dimension
Radius
Jogged
Diameter
Angular



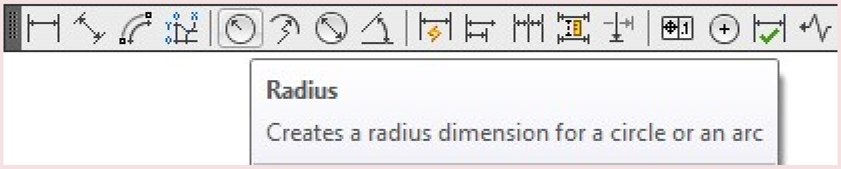
Diameter Dimension


Home
Inser
Autodesk 360
Express To
Line
Polyline
Text
Ann
Linear
Linear
Aligned
Diameter
Ordinate

AutoCAD Command: DIMRADIUS

The DIMRADIUS command is used to insert a radius dimension.

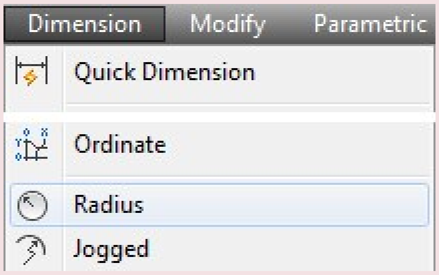
Shortcut: **DIMRAD**

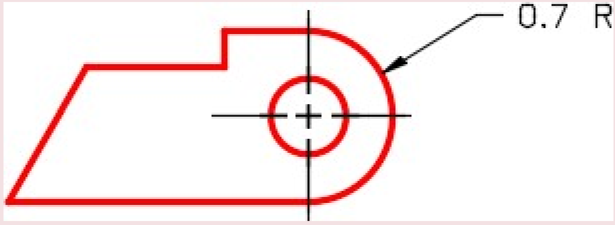


Radius

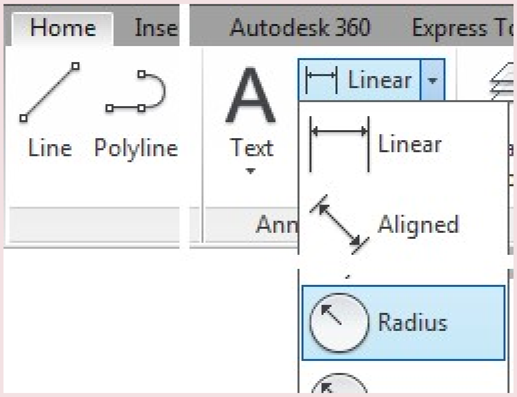
Creates a radius dimension for a circle or an arc

Dimension Toolbar


Dimension
Modify
Parametric
Quick Dimension
Ordinate
Radius
Jogged



Radius Dimension


Home
Inse
Autodesk 360
Express T
Line
Polyline
A
Text
Ann
Linear
Linear
Aligned
Radius

WORKALONG: Inserting Linear, Aligned, Angular, Diameter and Radius Dimensions

Step 1

Open the drawing: AutoCAD 2D Workalong 37-1. (Figure Step 1)

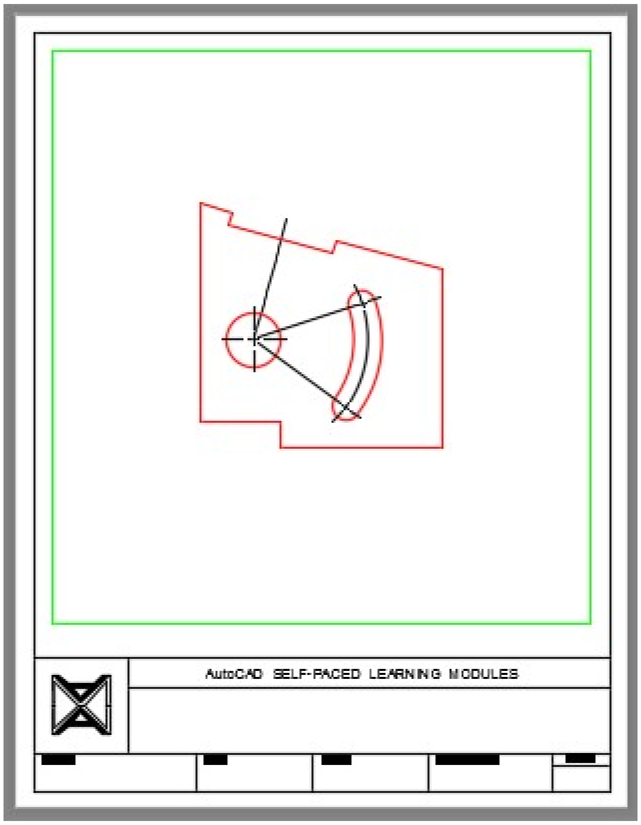


Figure Step 1

Step 2

Using the SAVEAS command, save and name the drawing: AutoCAD 2D Workalong 38-1.

Step 3

Enter the DIMASSOC system variable, as shown below, and ensure that it is set to 2. This will enable associative dimensions.

Command: **DIMASSOC**

Enter new value for DIMASSOC <1>: **2**

Command:

Step 4

Enable the display of the Dimension toolbar. Set the current dimensioning style to: Module 37. (Figure Step 4)

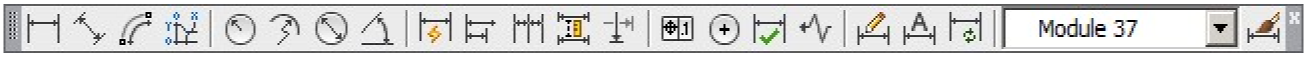


Figure Step 4

Step 5

Set layer: Dimensions as the current layer and set Paper space as the current space.

> **AUTHOR'S COMMENTS:** Ensure that you are working in Paper space when inserting dimensions and snap to Model space objects when placing the dimensions. Do not snap to Paper space objects.

Step 6

Delete the center lines for the circle. (Figure Step 6)

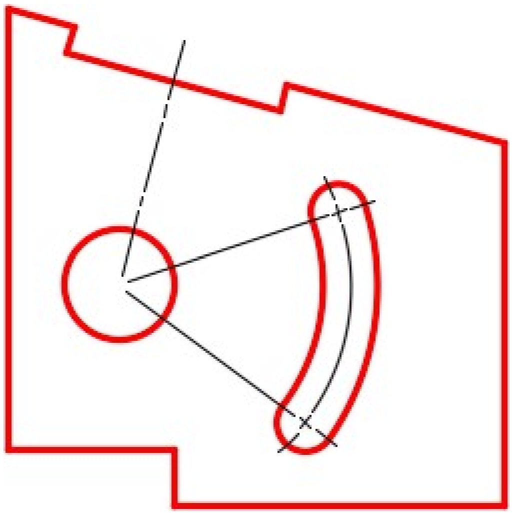


Figure Step 6

> **AUTHOR'S COMMENTS:** You will see later in the workalong why you are doing this.

Step 7

Turn layer: Center Lines off.

Step 8

Enter the DIMLIN command, as shown below, to insert a linear dimension. (Figure Step 8)

Command: **DIMLIN**

Specify first extension line origin or <select object>: *(end)* **P1**

Specify second extension line origin: (end) **P2**

(Ensure to snap to the ends of the lines.)

Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]: **P3**

(You can place the location of P3 by eye. Try to match the figure the best you can.)

Dimension text = 4.50

Command:

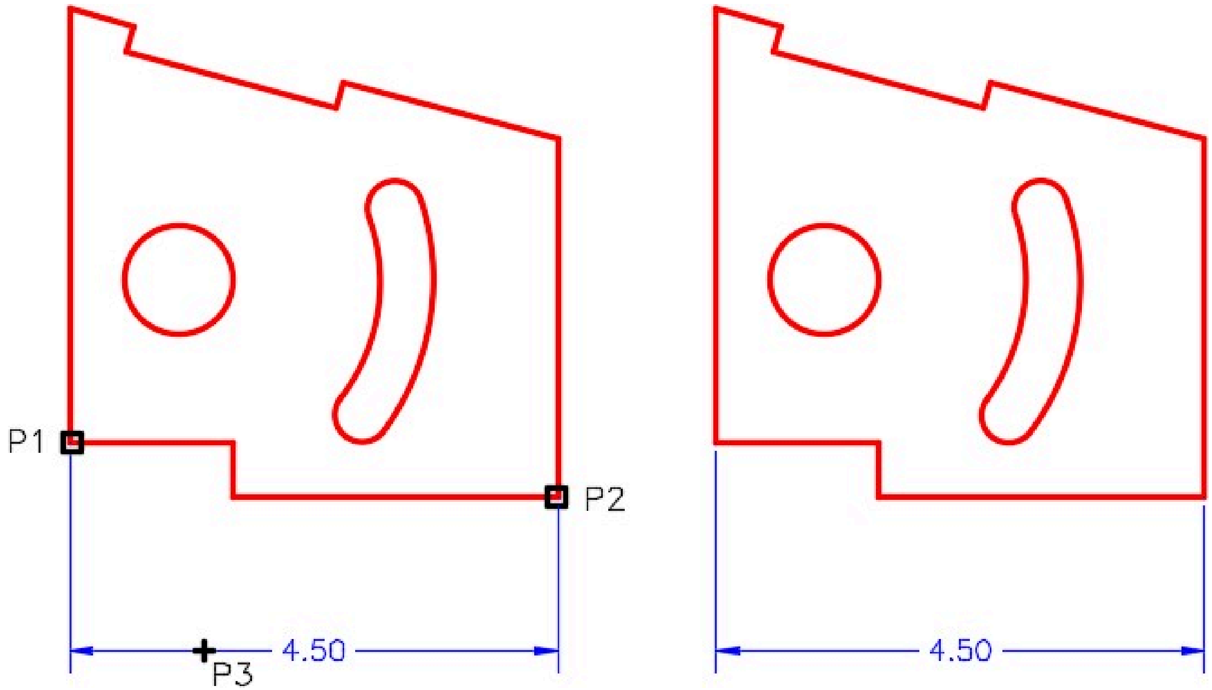


Figure Step 8

AUTHOR'S COMMENTS: Since you should always snap to Model space drawing objects when dimensioning a model, I had you temporarily turn off layer Center Lines.

Step 9

Using what you just learned, add the linear dimensions to match the figure. (Figure Step 9)

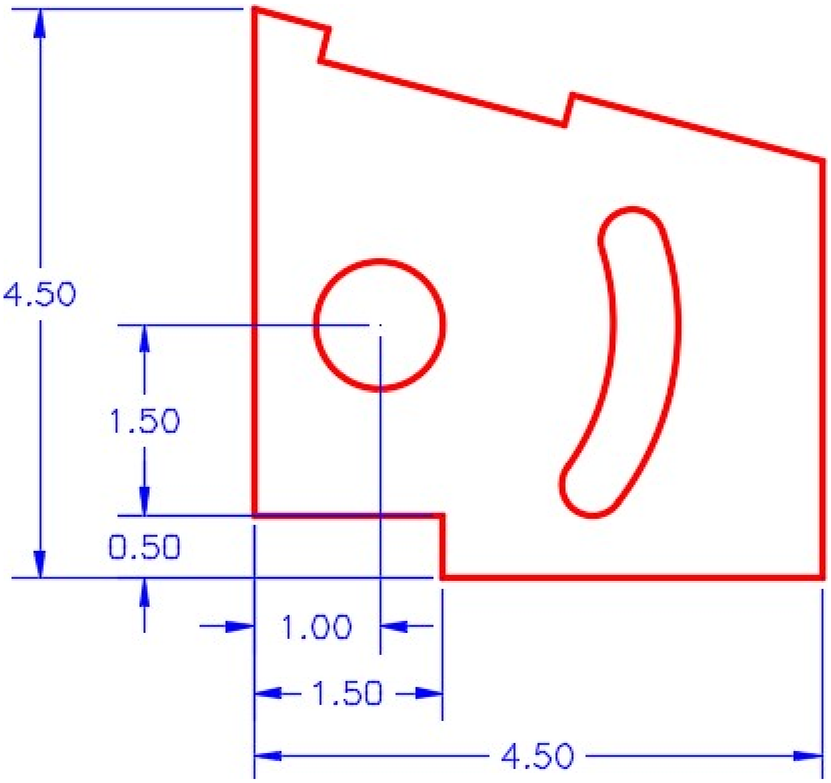

4.50
1.50
0.50
1.00
1.50
4.50

Figure Step 9

Step 10

Enter the DIMALI command, as shown below, to insert an aligned dimension. (Figure Step 10)

Command: **DIMALI**

Specify first extension line origin or <select object>: *(end)* **P4**

Specify second extension line origin: *(end)* **P5**

Specify dimension line location or [Mtext/Text/Angle]: **P6**

Dimension text = 4.66

Command:

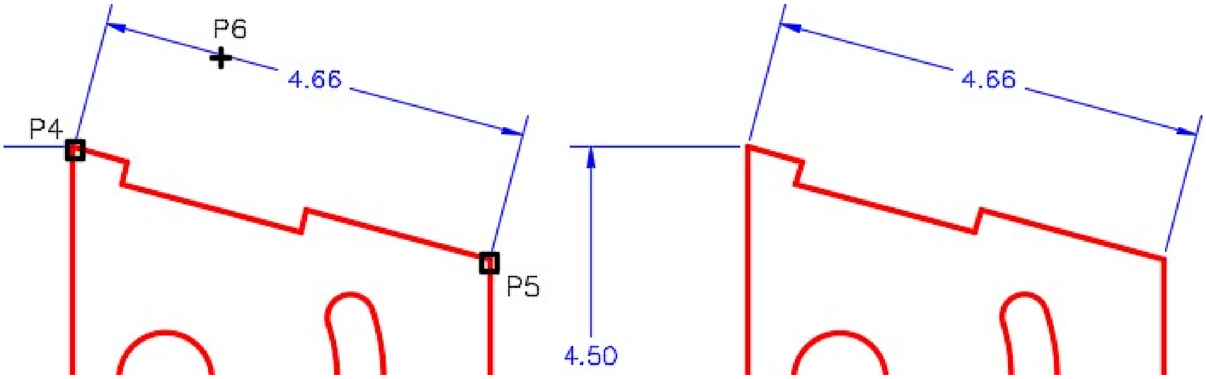


Figure Step 10

Step 11

Using what you just learned, add the aligned dimensions to match the figure. (Figure Step 11).

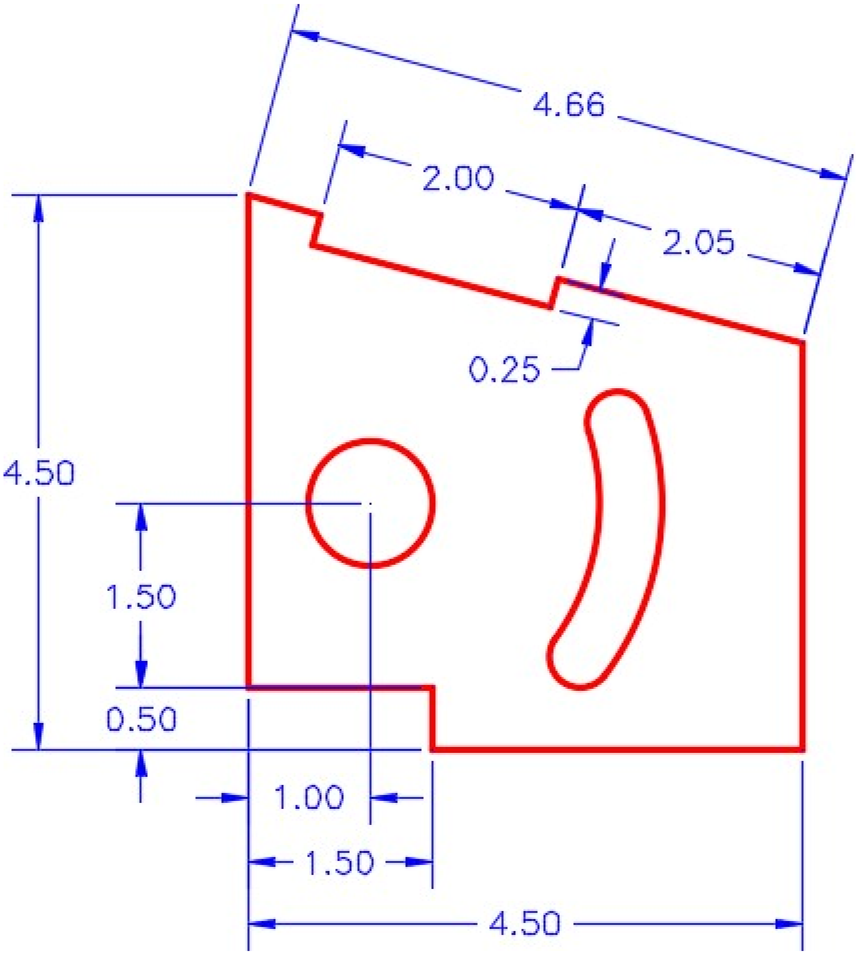

4.66
2.00
2.05
0.25
4.50
1.50
0.50
1.00
1.50
4.50

Figure Step 11

Step 12

Enter the DIMDIA command, as shown below, to insert a diameter dimension as shown in the figure. (Figure Step 12)

Command: **DIMDIA**

Select arc or circle: **P7**

Dimension text = 1.00

Specify dimension line location or [Mtext/Text/Angle]: **P8**

Command:

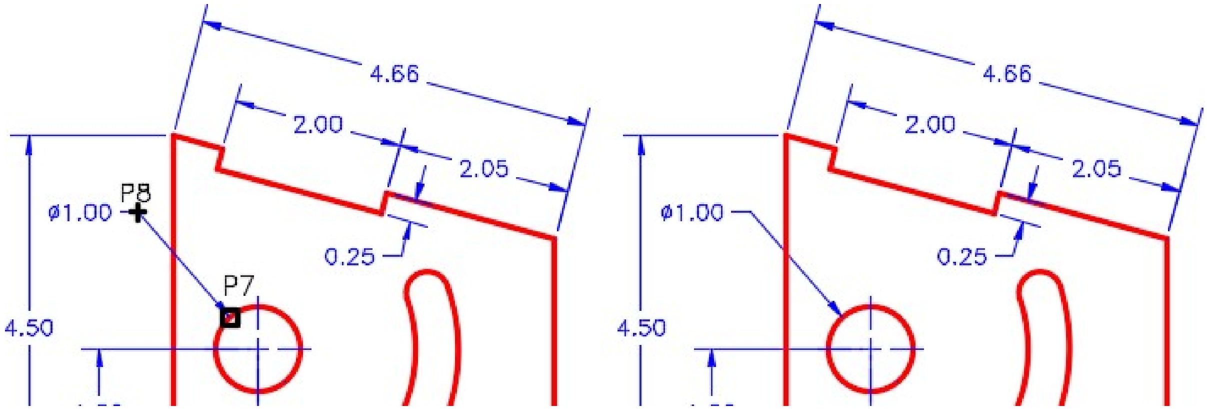


Figure Step 12

Step 13

Enter the DIMRAD command, as shown below, to insert a radius dimension. (Figure Step 13)

Command: **DIMRAD** Select arc or circle: **P9**

Dimension text = 2.56

Specify dimension line location or [Mtext/Text/Angle]: **P10**

Command:

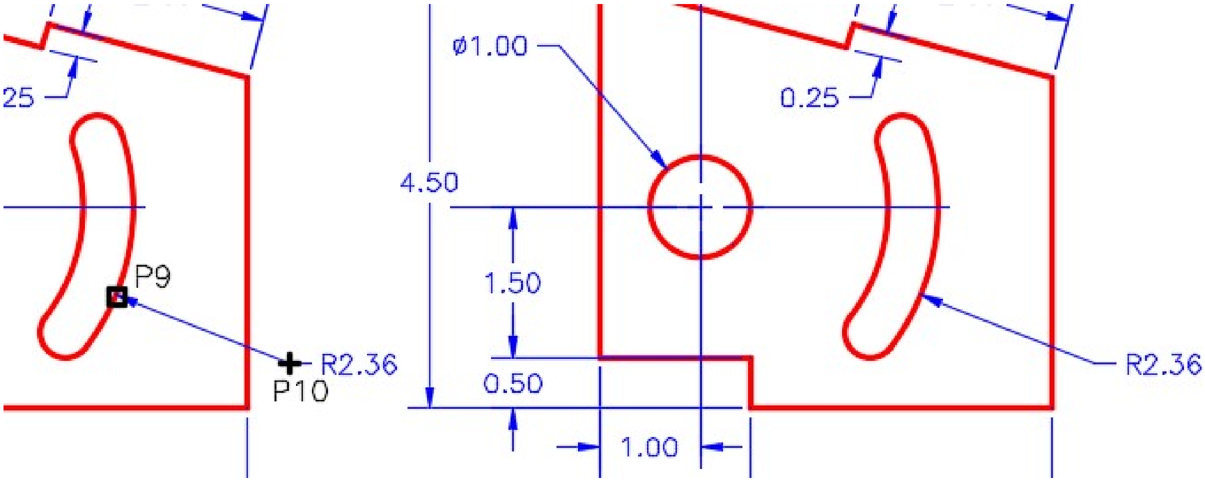

25
P9
R2.36
P10
ø1.00
4.50
1.50
0.50
1.00
0.25
R2.36

Figure Step 13

Step 14

Using what you just learned, add a second radius dimension to match the figure. (Figure Step 14)

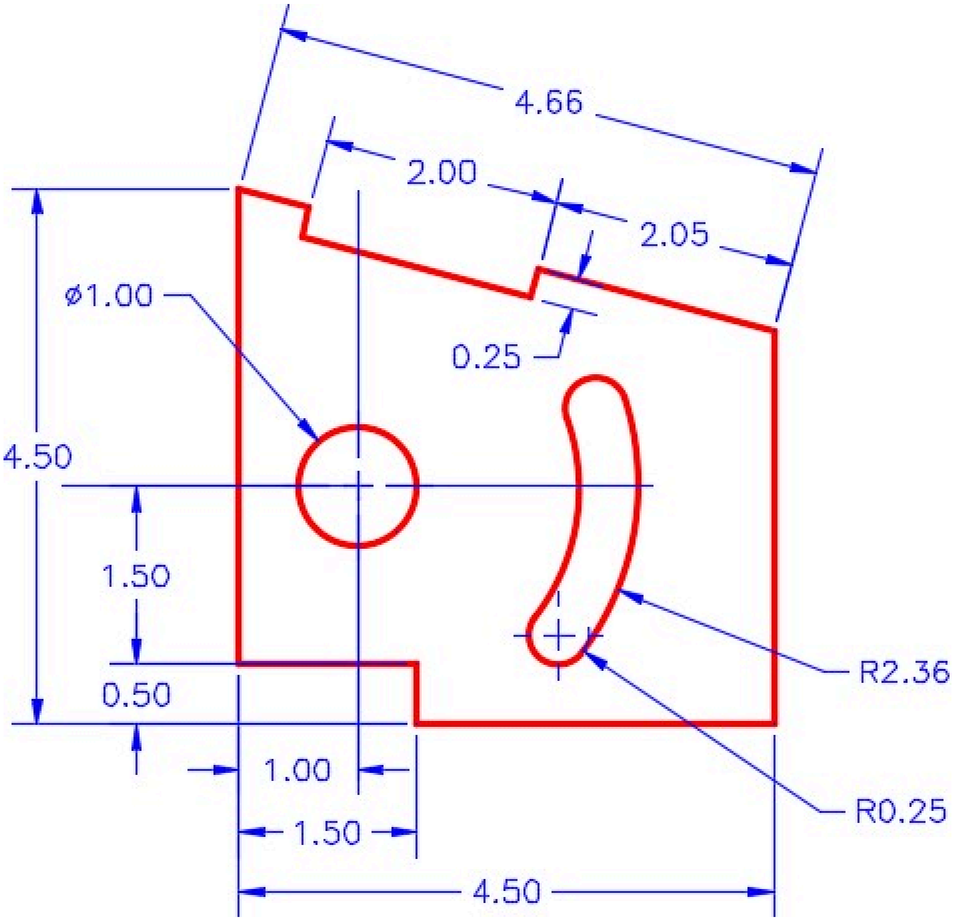

4.66
2.00
2.05
0.25
⌀1.00
4.50
1.50
0.50
R2.36
R0.25
1.00
1.50
4.50

Figure Step 14

Step 15

Enter the DIMANG command, as shown below, to insert an angular dimension. (Figure Step 15)

```
Command: DIMANG

Select arc, circle, line, or <specify vertex>: P11

Select second line: P12

Specify dimension arc line location or [Mtext/Text/Angle]: P13

Dimension text = 75d0’0″

Command:
```

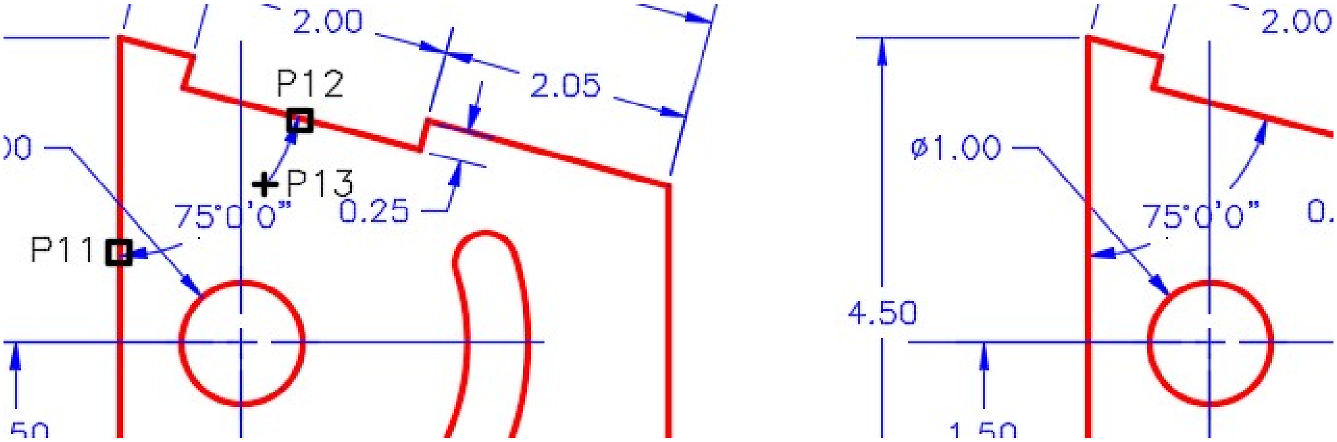

2.00
P12
2.05
00
P13
75°0'0"
0.25
P11
50
2.00
ø1.00
75°0'0"
0.
4.50
1.50

Figure Step 15

Step 16

Your drawing should be similar to the figure. (Figure Step 16)

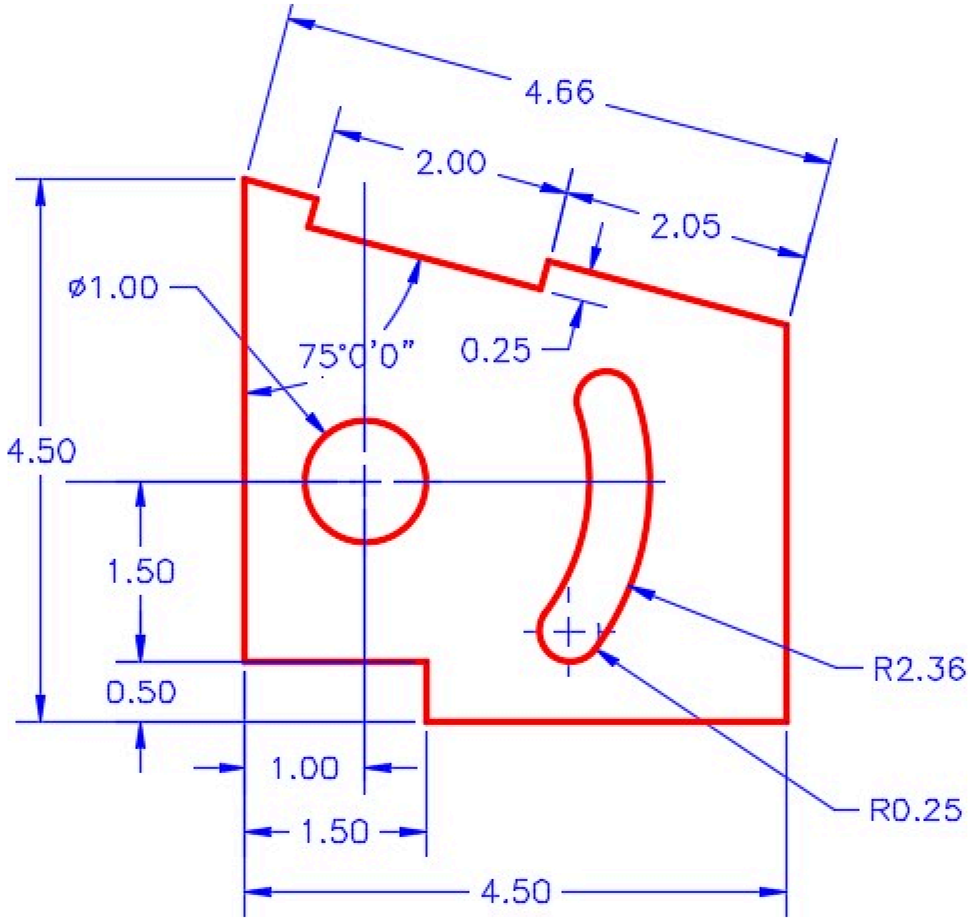
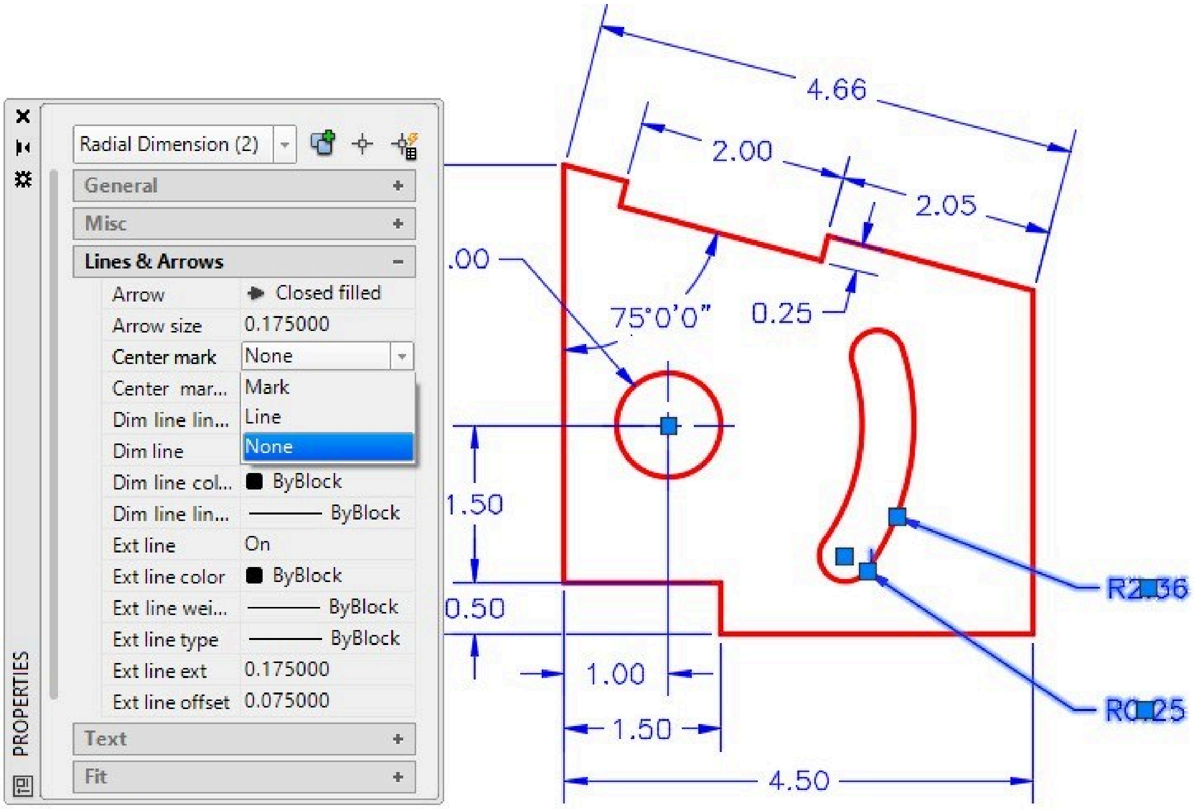

4.66
2.00
2.05
ø1.00
75°0'0"
0.25
4.50
1.50
R2.36
0.50
R0.25
1.00
1.50
4.50

Figure Step 16

AUTHOR'S COMMENTS: After you have inserted all the required associative dimensions, make any corrections that are required to them by changing the properties of the dimensions that need correcting using the Properties window.

Step 17

Open the Properties window. Select the two radius dimensions shown in the figure. Set the Center mark property to None. (Figure Step 17)



Radial Dimension (2)

General

Misc

Lines & Arrows

Arrow	Closed filled
Arrow size	0.175000
Center mark	None
Center mar...	
Dim line lin...	
Dim line	
Dim line col...	ByBlock
Dim line lin...	ByBlock
Ext line	On
Ext line color	ByBlock
Ext line wei...	ByBlock
Ext line type	ByBlock
Ext line ext	0.175000
Ext line offset	0.075000

Text

Fit

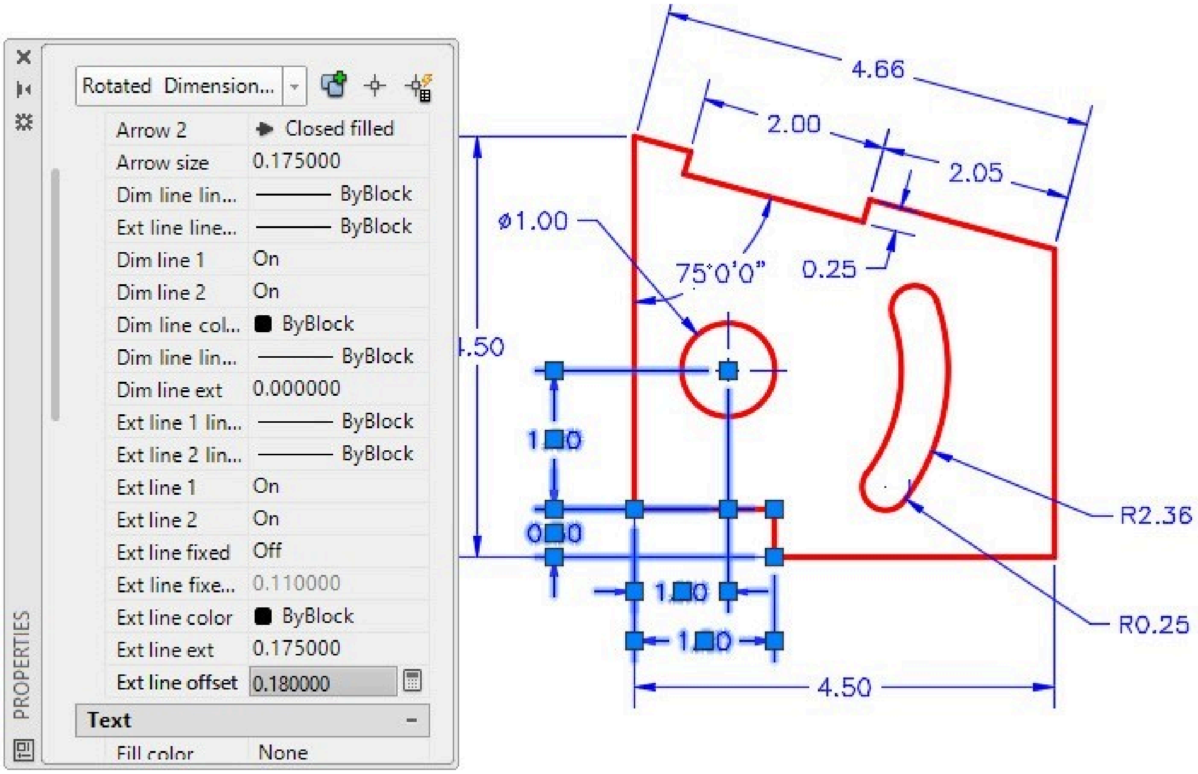
PROPERTIES
Mark
Line
None
4.66
2.00
2.05
.00
75°0'0"
0.25
1.50
0.50
1.00
1.50
4.50
R2.36
R0.25

Figure Step 17

AUTHOR'S COMMENTS: This step will remove the center line inserted with the radius dimension.

Step 18

Select the two linear dimensions shown in the figure. Set the Ext line offset property to 0.1800. (Figure Step 18)



Rotated Dimension...

Arrow 2	Closed filled
Arrow size	0.175000
Dim line lin...	ByBlock
Ext line line...	ByBlock
Dim line 1	On
Dim line 2	On
Dim line col...	ByBlock
Dim line lin...	ByBlock
Dim line ext	0.000000
Ext line 1 lin...	ByBlock
Ext line 2 lin...	ByBlock
Ext line 1	On
Ext line 2	On
Ext line fixed	Off
Ext line fixe...	0.110000
Ext line color	ByBlock
Ext line ext	0.175000
Ext line offset	0.180000

Text

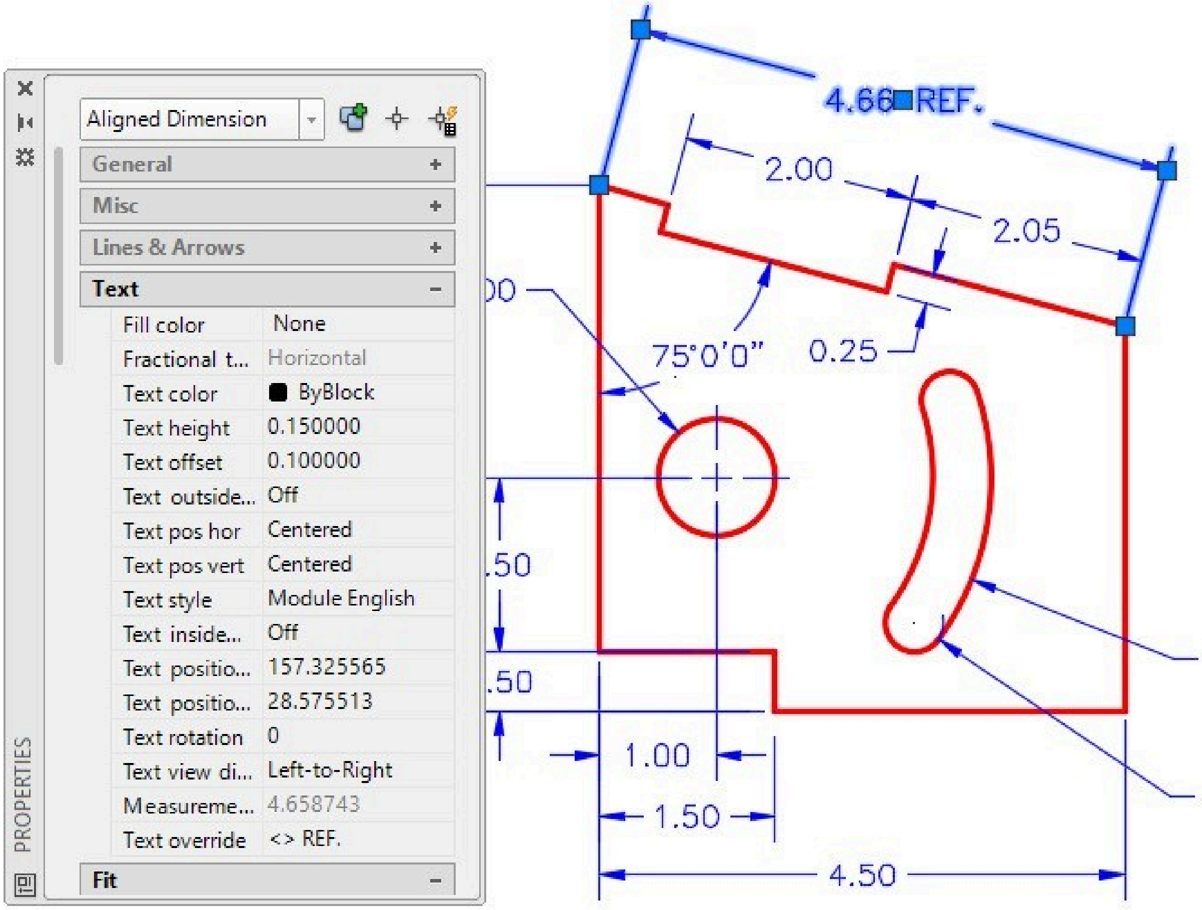
Fill color	None

Figure Step 18

MUST KNOW: When AutoCAD finds the dimension or measures the object in a dimensioning command, it does not display the actual number. It displays the symbol <>. This symbol means default or the actual measurement. To add the text REF after the actual measurement, enter it as: <> REF.

Step 19

Select the aligned dimension shown in the figure. Set the Text override property to < > REF. (Figure Step 19)



Aligned Dimension

General +

Misc +

Lines & Arrows +

Text –

Fill color	None
Fractional t...	Horizontal
Text color	ByBlock
Text height	0.150000
Text offset	0.100000
Text outside...	Off
Text pos hor	Centered
Text pos vert	Centered
Text style	Module English
Text inside...	Off
Text positio...	157.325565
Text positio...	28.575513
Text rotation	0
Text view di...	Left-to-Right
Measureme...	4.658743
Text override	<> REF.

Fit –

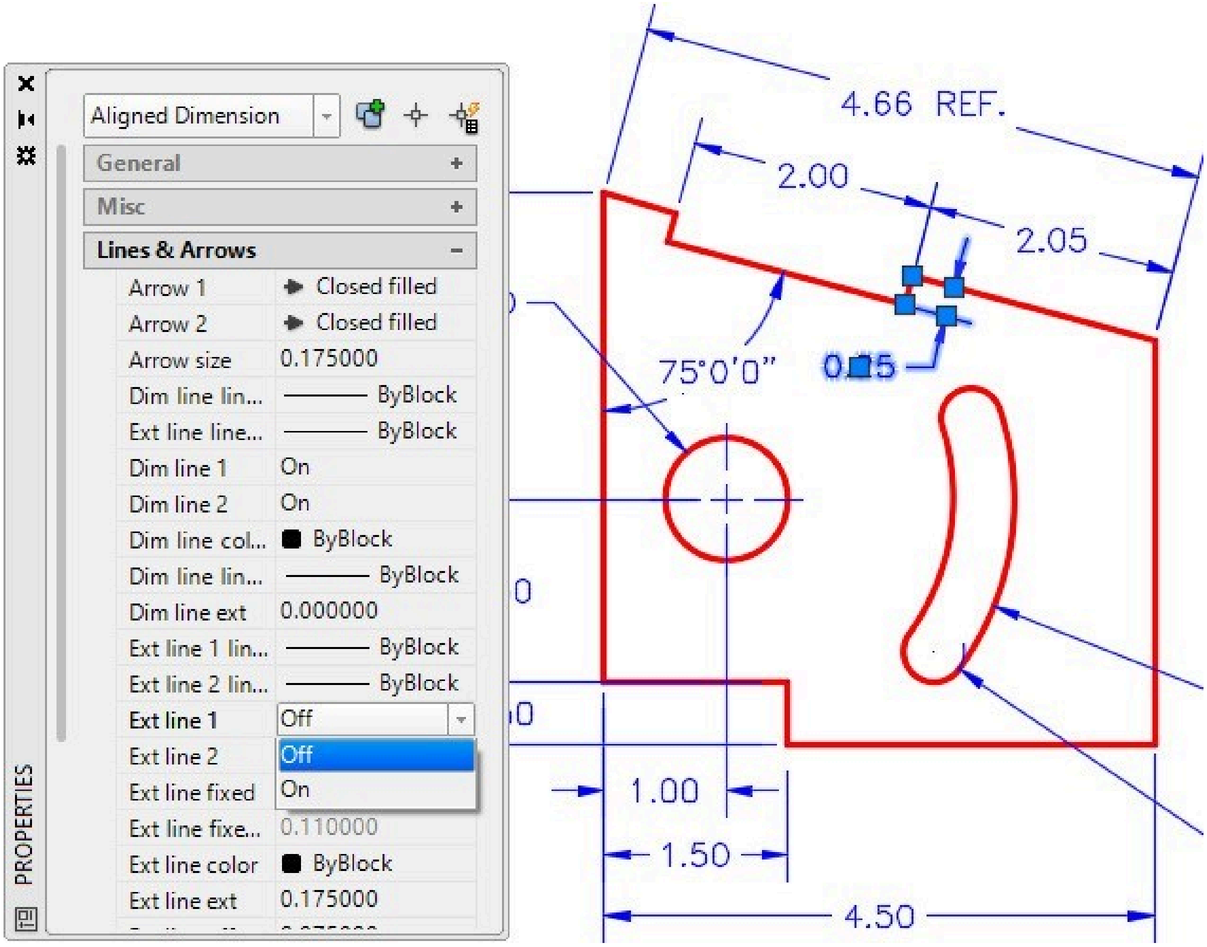
Figure Step 19

AUTHOR’S COMMENTS: The <> means default or the actual dimension that AutoCAD measured. If sometime in the future the size changes, the dimension will also change.

USER TIP: The fastest way to dimension an object is to dimension it very quickly, inserting all the required dimensions. When all the dimensions are inserted, change them using the Properties window, if necessary. This is much faster than fixing the dimensions as they are inserted.

Step 20

Select the aligned dimension shown in the figure. Change the property Ext Line 2 to Off. This will turn off the display of one of the extension lines since it is on object line. Depending on the order that you used when inserting the dimension, you may have to set Ext Line 1 instead. (Figure Step 20)



Aligned Dimension

General	+
Misc	+
Lines & Arrows	−
Arrow 1	Closed filled
Arrow 2	Closed filled
Arrow size	0.175000
Dim line lin...	ByBlock
Ext line line...	ByBlock
Dim line 1	On
Dim line 2	On
Dim line col...	ByBlock
Dim line lin...	ByBlock
Dim line ext	0.000000
Ext line 1 lin...	ByBlock
Ext line 2 lin...	ByBlock
Ext line 1	Off
Ext line 2	Off On
Ext line fixed	
Ext line fixe...	0.110000
Ext line color	ByBlock
Ext line ext	0.175000

Figure Step 20

Step 21

Turn layer: Center Lines on. Your drawing should match the figures. (Figure Step 21)

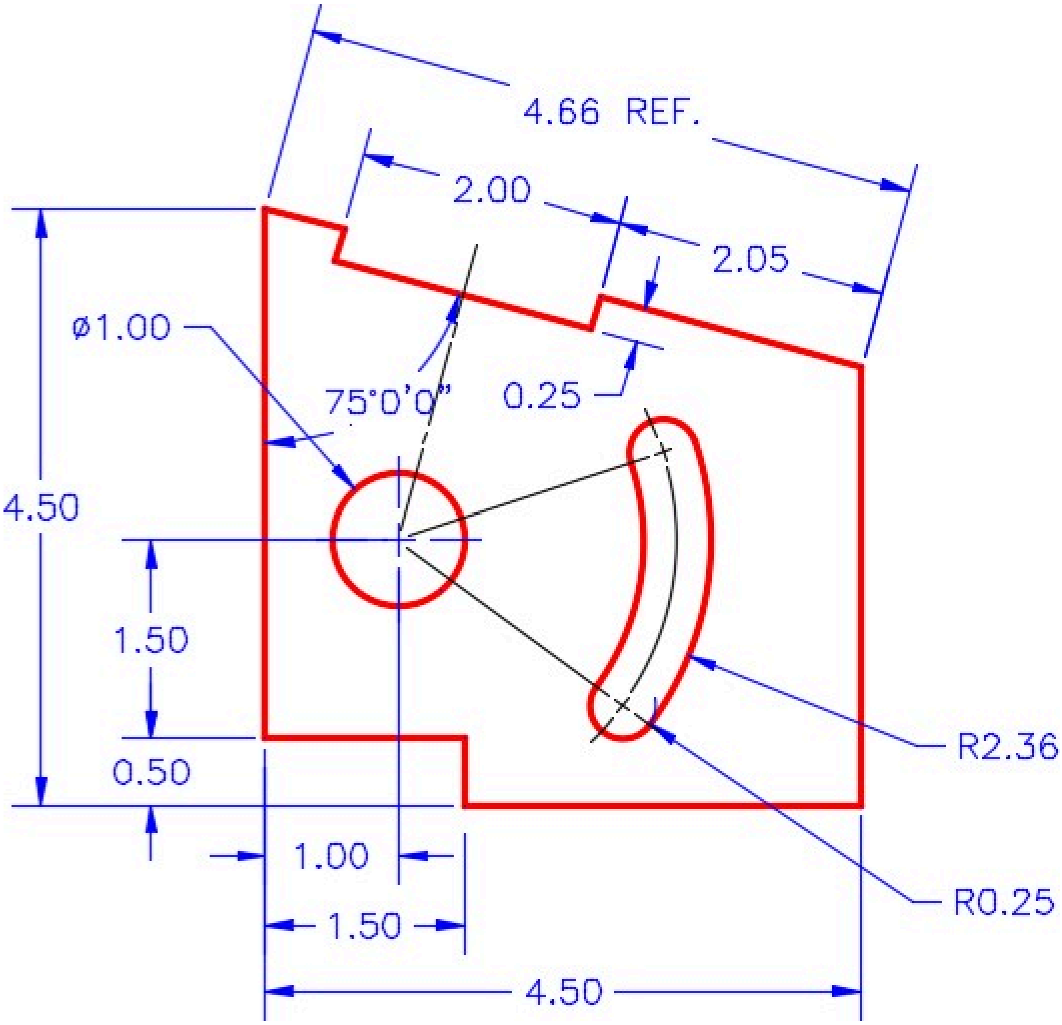

4.66 REF.
2.00
2.05
ø1.00
75°0'0"
0.25
4.50
1.50
0.50
1.00
1.50
4.50
R2.36
R0.25

Figure Step 21

Step 22

Using what you learned earlier, insert the angular dimension as shown in the figure. (Figure Step 22)

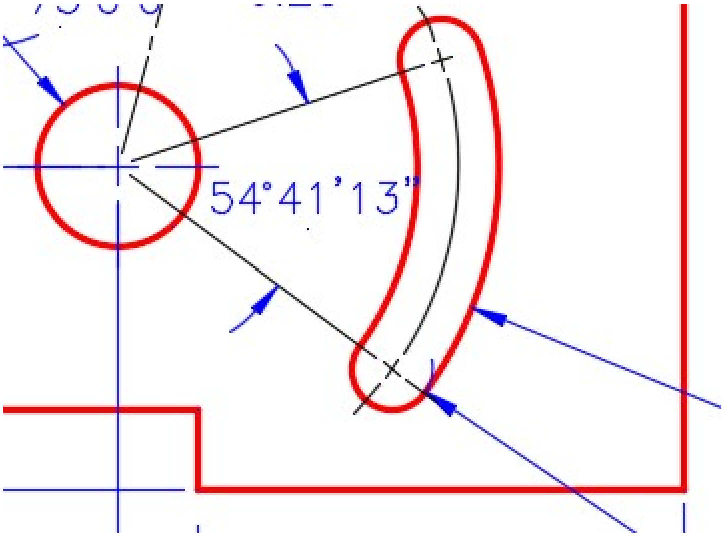


Figure Step 22

Step 23

Your drawing should closely match the figure. (Figure Step 23A and 23B)

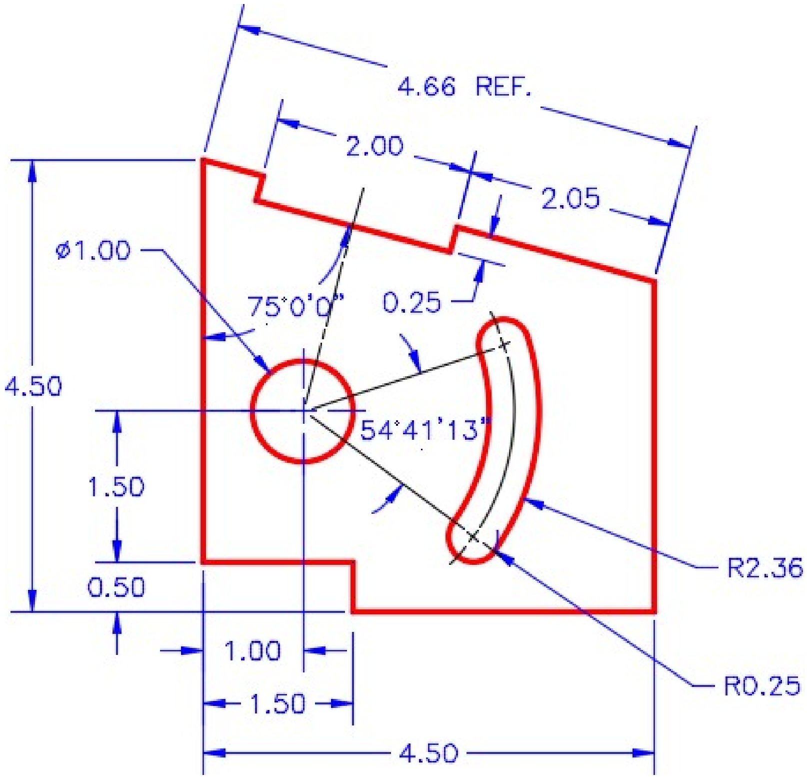

4.66 REF.
2.00
2.05
⌀1.00
75°0'0"
0.25
4.50
54°41'13"
1.50
0.50
R2.36
R0.25
1.00
1.50
4.50

Figure Step 23A

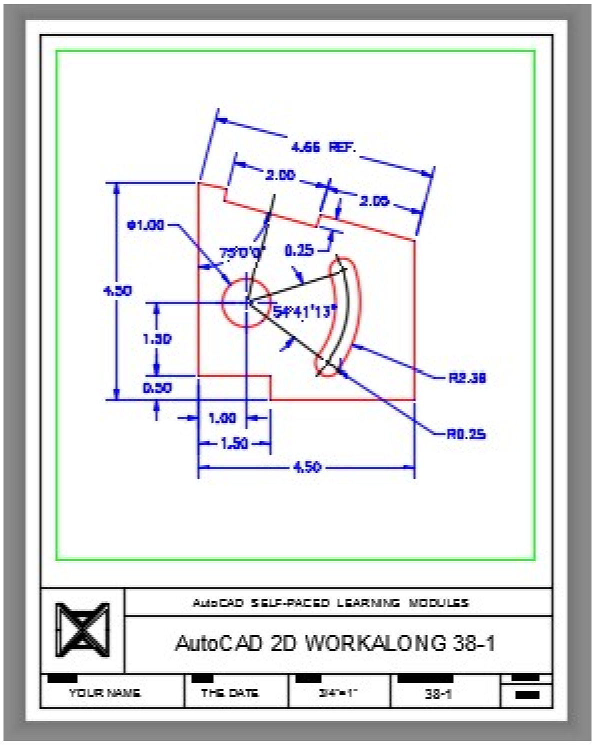
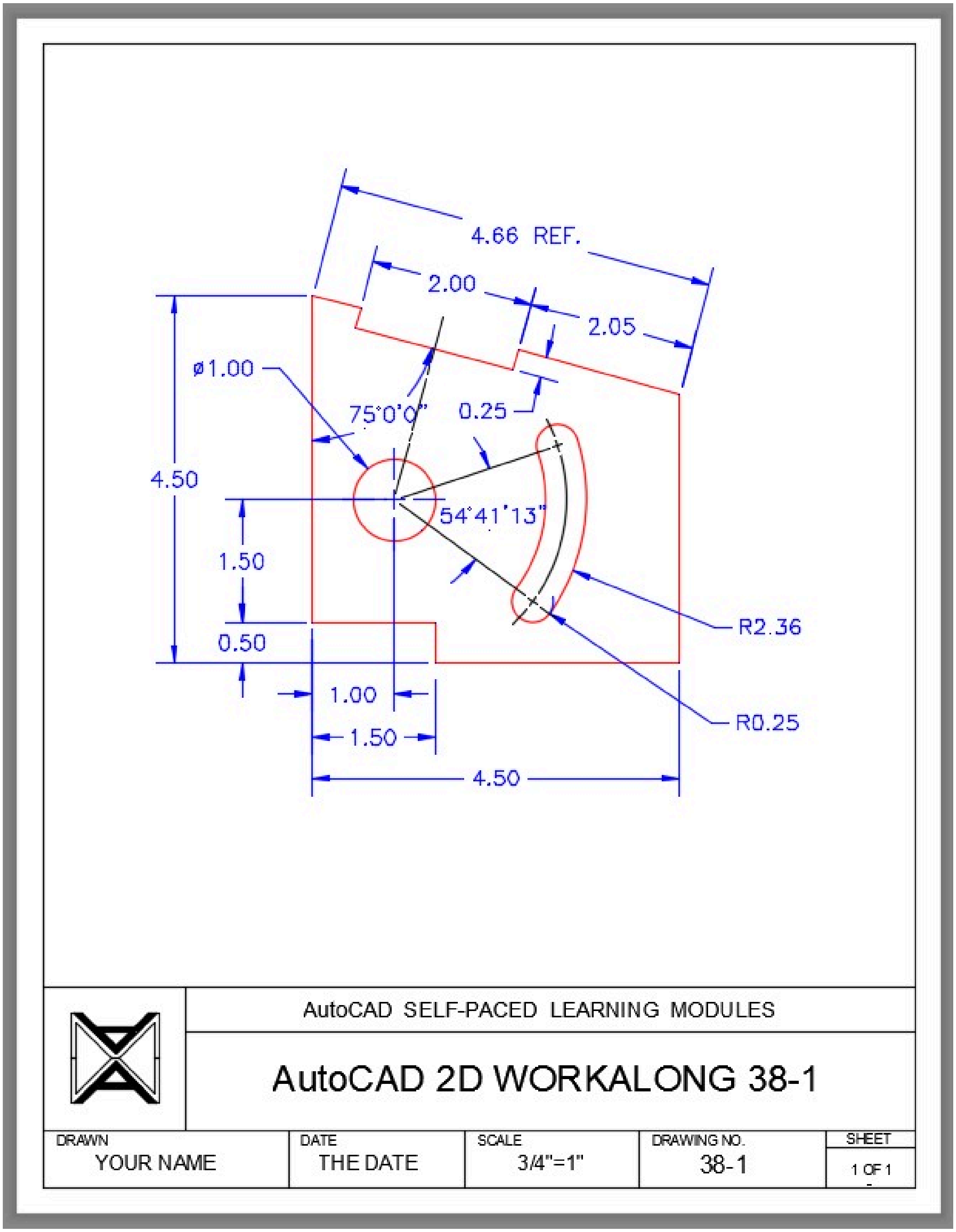

4.66 REF.
2.00
2.00
Ø1.00
75°0'0"
0.25
4.50
54°41'13"
1.50
0.50
R2.38
R0.25
1.00
1.50
4.50
AutoCAD SELF-PACED LEARNING MODULES
AutoCAD 2D WORKALONG 38-1
YOUR NAME
THE DATE
3/4"=1"
38-1

Figure Step 23B

Step 24

Turn layer: Viewport off. Fill in the titleblock. (Figure Step 24)


4.66 REF.
2.00
2.05
ø1.00
75°0'0"
0.25
4.50
54°41'13"
1.50
R2.36
0.50
1.00
R0.25
1.50
4.50

AutoCAD SELF-PACED LEARNING MODULES

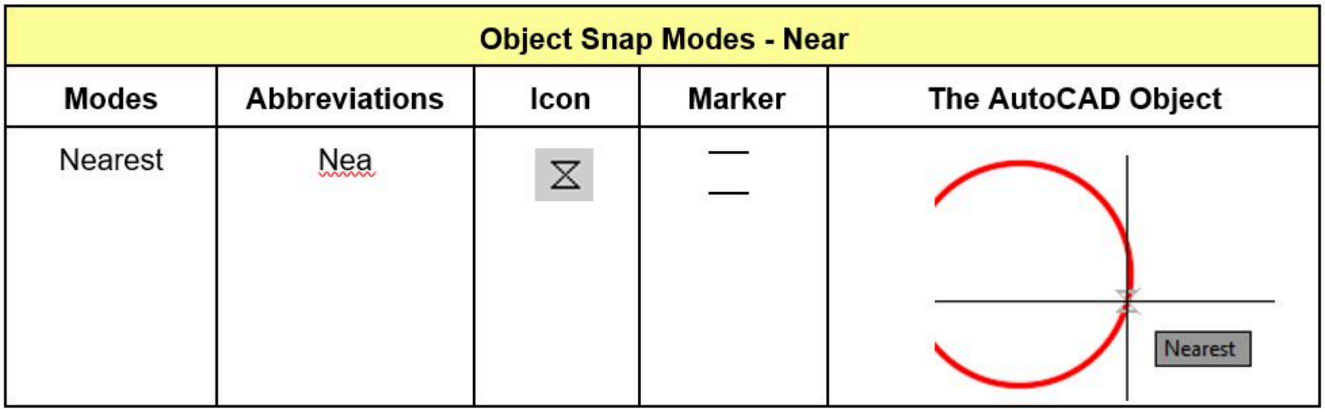
AutoCAD 2D WORKALONG 38-1

DRAWN	DATE	SCALE	DRAWING NO.	SHEET
YOUR NAME	THE DATE	3/4"=1"	38-1	1 OF 1

Figure Step 24

Step 25

Save and close the drawing.



Object Snap Modes - Near				
Modes	**Abbreviations**	**Icon**	**Marker**	**The AutoCAD Object**
Nearest	Nea	⧖	=	Nearest

Figure 38-2
Object Snap Mode – Near

AutoCAD Command: QLEADER

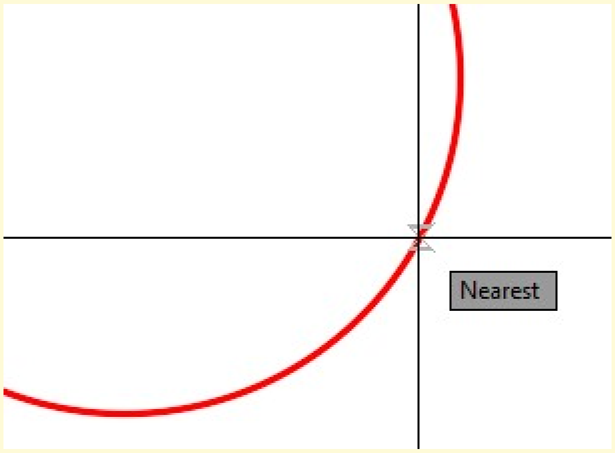
The QLEADER command is used to a leader dimension.

Shortcut: none

USER TIP: The snap mode nearest can be used for many purposes while drawing in AutoCAD. To snap to an object, but not any particular location on that object, use the nearest snap mode. It is best to manually type the near snap mode, when required, rather then including it in the Autosnap settings. i.e.

Command: **LINE**

Specify first point: **near** to



WORKALONG: Inserting Leaders

Step 1

Open the drawing: AutoCAD 2D Workalong 38-1. (Figure Step 1)

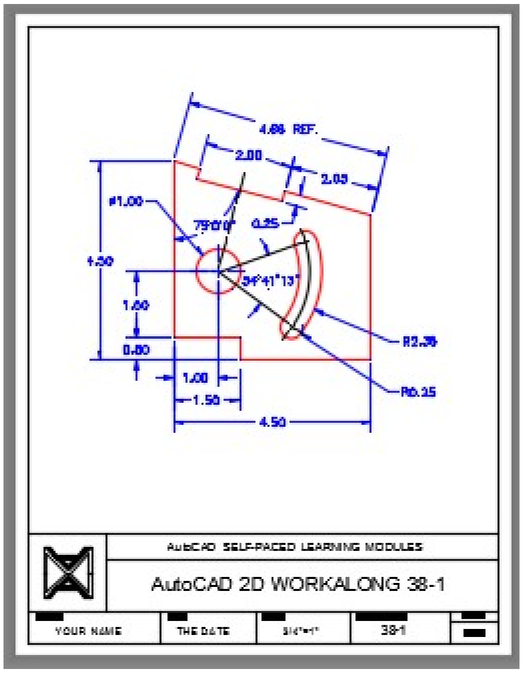

4.86 REF.
2.00
2.03
Ø1.00
79°0'0"
0.25
4.50
34°41'13"
1.60
0.80
R2.38
1.00
R0.25
1.50
4.50
AutoCAD SELF-PACED LEARNING MODULES
AutoCAD 2D WORKALONG 38-1
YOUR NAME
THE DATE
3/4"=1"
38-1

Figure Step 1

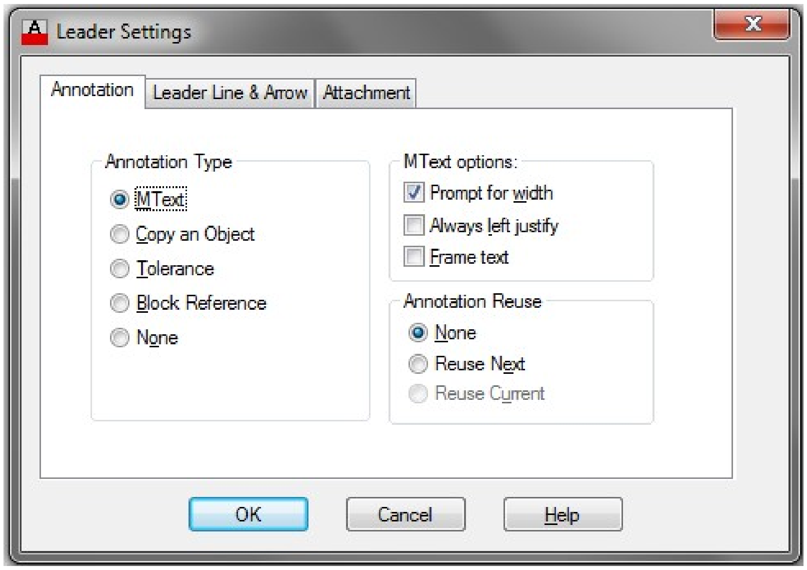
Step 2

Enter the QLEADER command, as shown below, and enter the Settings option. It will open the Leader Settings dialogue box. Enable the Annotation tab. Ensure that the dialogue box matches the settings in the figure. (Figure Step 2)

Command: **QLEADER**

Specify first leader point, or [Settings]: **S**

Command:


Leader Settings
Annotation
Leader Line & Arrow
Attachment
OK
Cancel
Help

Annotation Type

● MText

○ Copy an Object

○ Tolerance

○ Block Reference

○ None

MText options:

☑ Prompt for width

☐ Always left justify

☐ Frame text

Annotation Reuse

● None

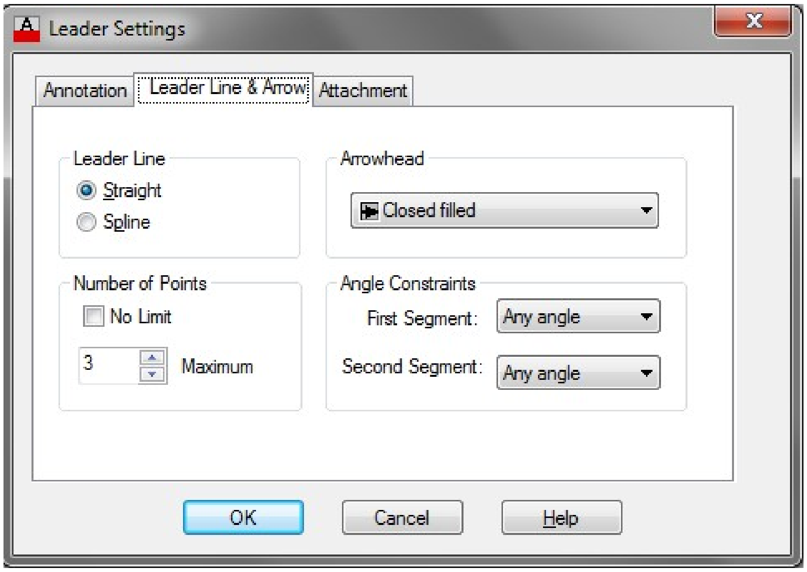
○ Reuse Next

○ Reuse Current

Figure Step 2

Step 3

Enable the Leader Line and Arrow tab. Ensure that your dialogue box matches the settings in the figure. (Figure Step 3)


Leader Settings
Annotation
Leader Line & Arrow
Attachment
OK
Cancel
Help

Leader Line

● Straight

○ Spline

Arrowhead: Closed filled

Number of Points

☐ No Limit

3 Maximum

Angle Constraints

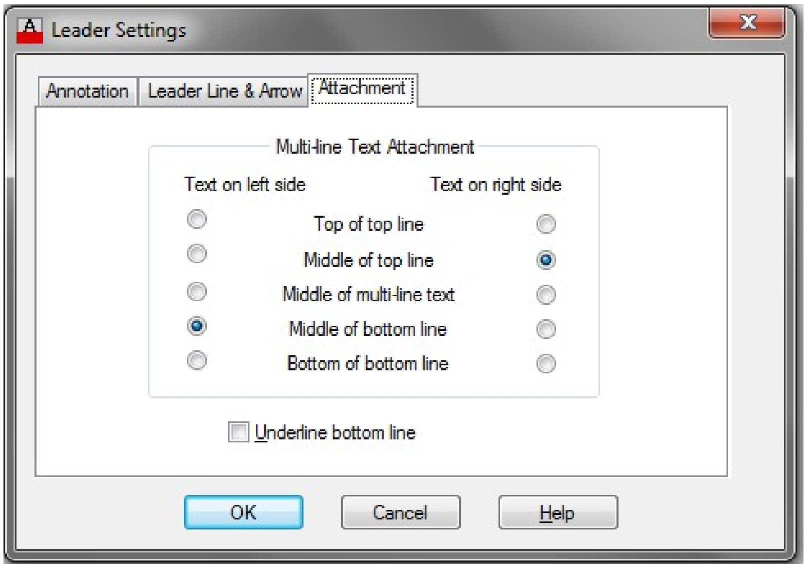
First Segment: Any angle

Second Segment: Any angle

Figure Step 3

Step 4

Enable the Attachment tab. Ensure that your dialogue box matches the settings in the figure. (Figure Step 4)


Leader Settings
Annotation
Leader Line & Arrow
Attachment
OK
Cancel
Help

Multi-line Text Attachment

Text on left side		Text on right side
○	Top of top line	○
○	Middle of top line	●
○	Middle of multi-line text	○
●	Middle of bottom line	○
○	Bottom of bottom line	○

☐ Underline bottom line

Figure Step 4

Step 5

Enter the QLEADER command, as shown below, to insert the leader. (Figure Step 5)

Command: **QLEADER**

Specify first leader point, or [Settings] <Settings>: *(near)* **P13**

(Use the Near osnap mode)

Specify next point: **P14**

Specify next point: <Ortho on> **P15**

(I enabled Ortho mode to ensure the line is horizontal.)

Specify text width <0.0000>:

Enter first line of annotation text <Mtext>: **MATCHING**

Enter next line of annotation text <Mtext>: **EDGE**

Enter next line of annotation text:

Command:

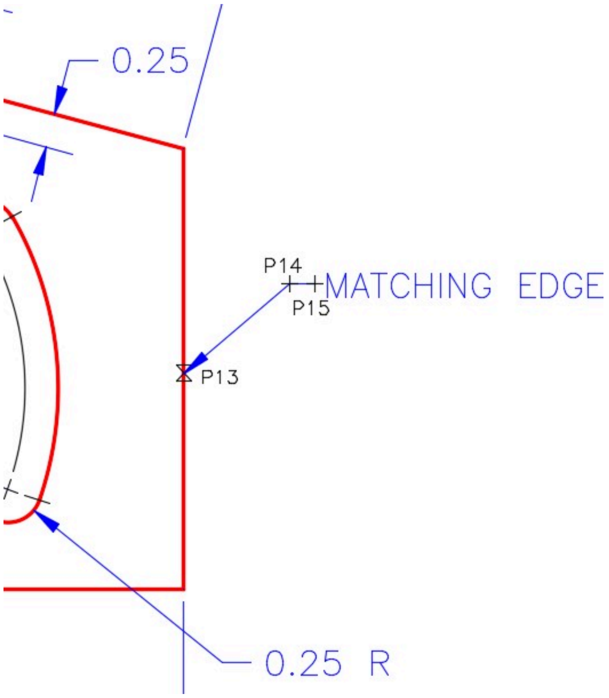


Figure Step 5

Step 6

Using what you just learned, add the leader START X5.5Y4.5 as shown in the figure. Your completed drawing shown now appear similar to the figure. (Figure Step 6)

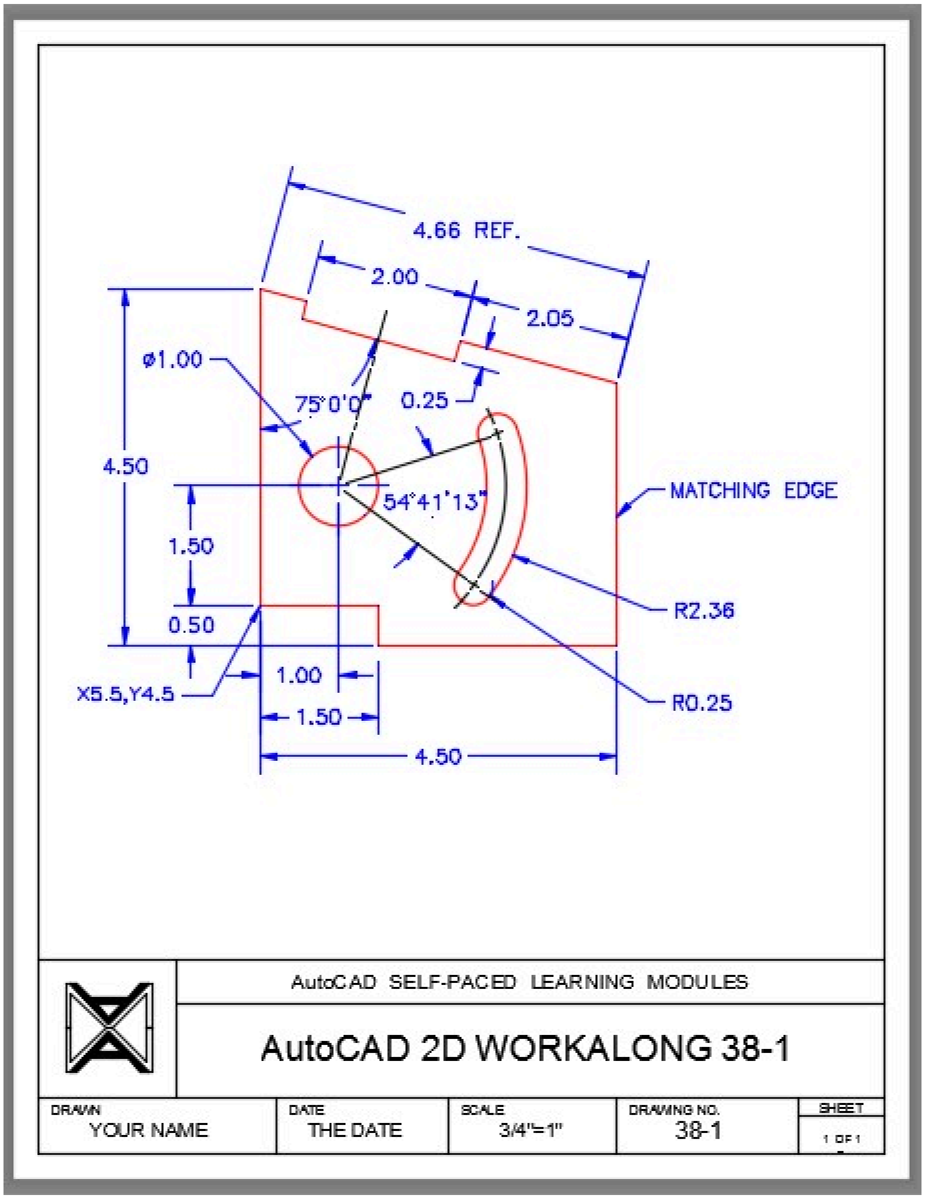

4.66 REF.
2.00
2.05
ø1.00
75°0'0"
0.25
4.50
1.50
54°41'13"
MATCHING EDGE
R2.36
0.50
X5.5,Y4.5
1.00
1.50
4.50
R0.25
AutoCAD SELF-PACED LEARNING MODULES
AutoCAD 2D WORKALONG 38-1
DRAWN
YOUR NAME
DATE
THE DATE
SCALE
3/4"=1"
DRAWING NO.
38-1
SHEET
1 OF 1

Figure Step 6

Step 7

Save and close the drawing.

Key Principles

Key Principles in Module 38

1. When AutoCAD finds the dimension or measures the object in a dimensioning command, it does not display the actual number. The symbol <> is displayed. This symbol means default or the actual measurement.
2. To make corrections to inserted associative dimensions, change the properties of the dimension using the Properties window.

Lab Exercise 38-1

Time allowed: 30 minutes.

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 38-1	N/A	Inches

Step 1

Open the drawing: AutoCAD 2D Lab 37-1. Using the SAVEAS command, save and name it: AutoCAD 2D Lab 38-1.

Step 2

Set the current dimension style to: 2D English.

Step 3

Ensure that associate dimensioning is enabled by checking the setting of the DIMASSOC system variable.

Step 4

In Paper space, on layer: Dimensions, insert the dimensions shown in the figure. Match the figure the best you can. (Figure step 4A and 4B)

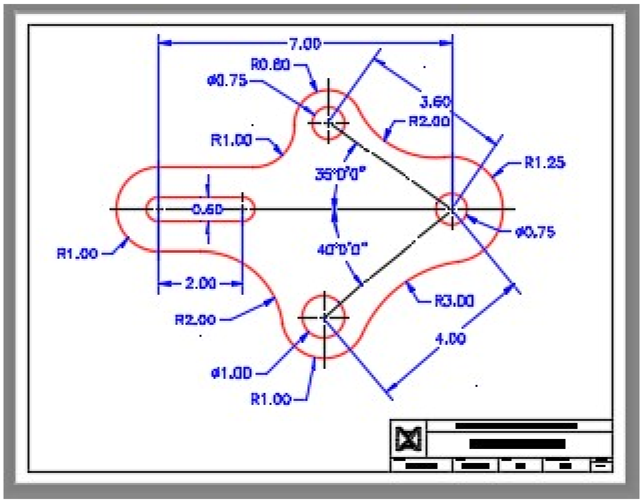

7.00
R0.80
ø0.75
3.60
R2.00
R1.00
R1.25
35°0'0"
0.60
ø0.75
R1.00
40°0'0"
2.00
R3.00
R2.00
4.00
ø1.00
R1.00

Figure Step 4A
Completed Drawing

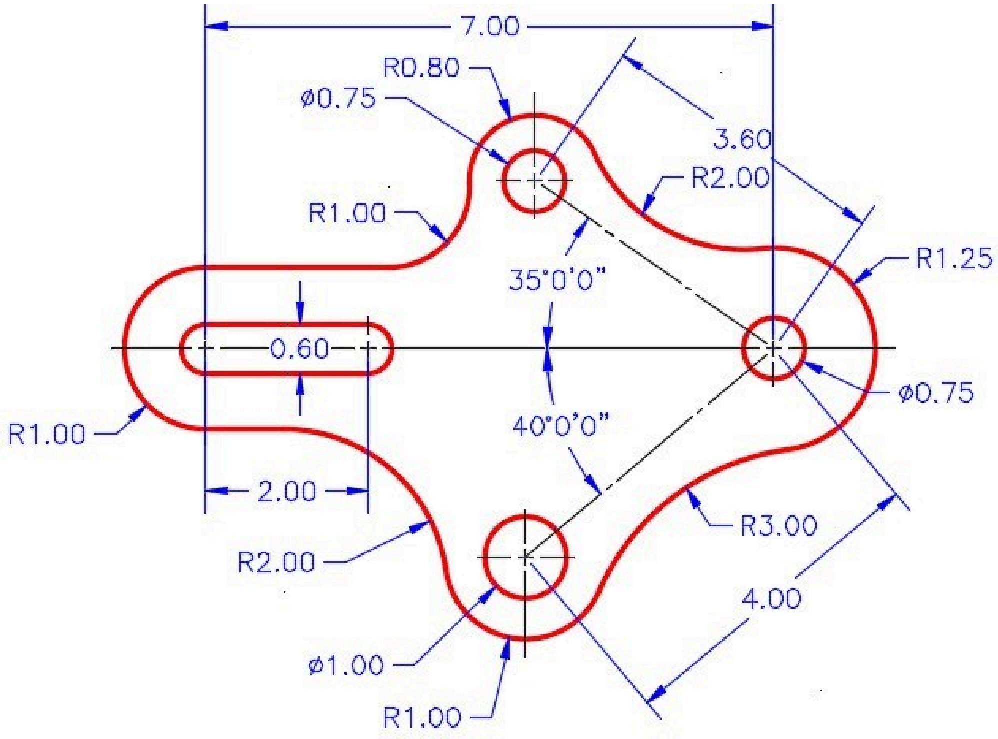

7.00
R0.80
ø0.75
3.60
R2.00
R1.00
R1.25
35°0'0"
0.60
ø0.75
R1.00
40°0'0"
2.00
R3.00
R2.00
4.00
ø1.00
R1.00

Figure Step 4B

Step 5

Turn layer: Viewport off.

Step 6

Using the standards in Module 20, complete the titleblock.

Lab Exercise 38-2

Time allowed: 30 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 38-2	N/A	Millimeters

Step 1

Open the drawing: AutoCAD 2D Lab 37-2. Using the SAVEAS command, save and name it: AutoCAD 2D Lab 38-2.

Step 2

Set the current dimension style to: 2D Metric.

Step 3

Ensure that associate dimensioning is enabled by checking the setting of the DIMASSOC system variable.

Step 4

In Paper space, on layer: Dimensions, insert the dimensions shown in the figure. Match the figure the best you can. Figure step 4A and 4B)

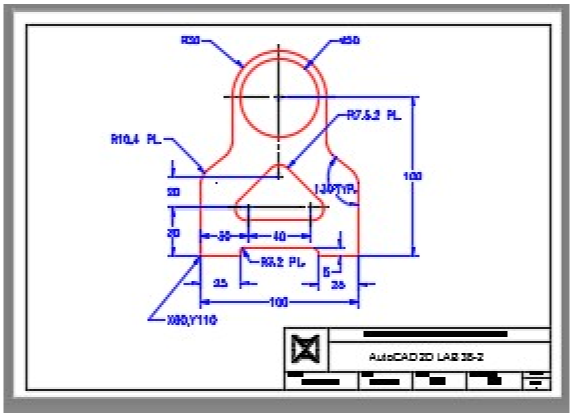

R30
⌀50
R7.5 2 PL
R10.4 PL
100
50
120 TYP.
20
30
40
R8.2 PL
5
25
25
100
X80,Y110
AutoCAD 2D LAB 38-2

Figure Step 4A

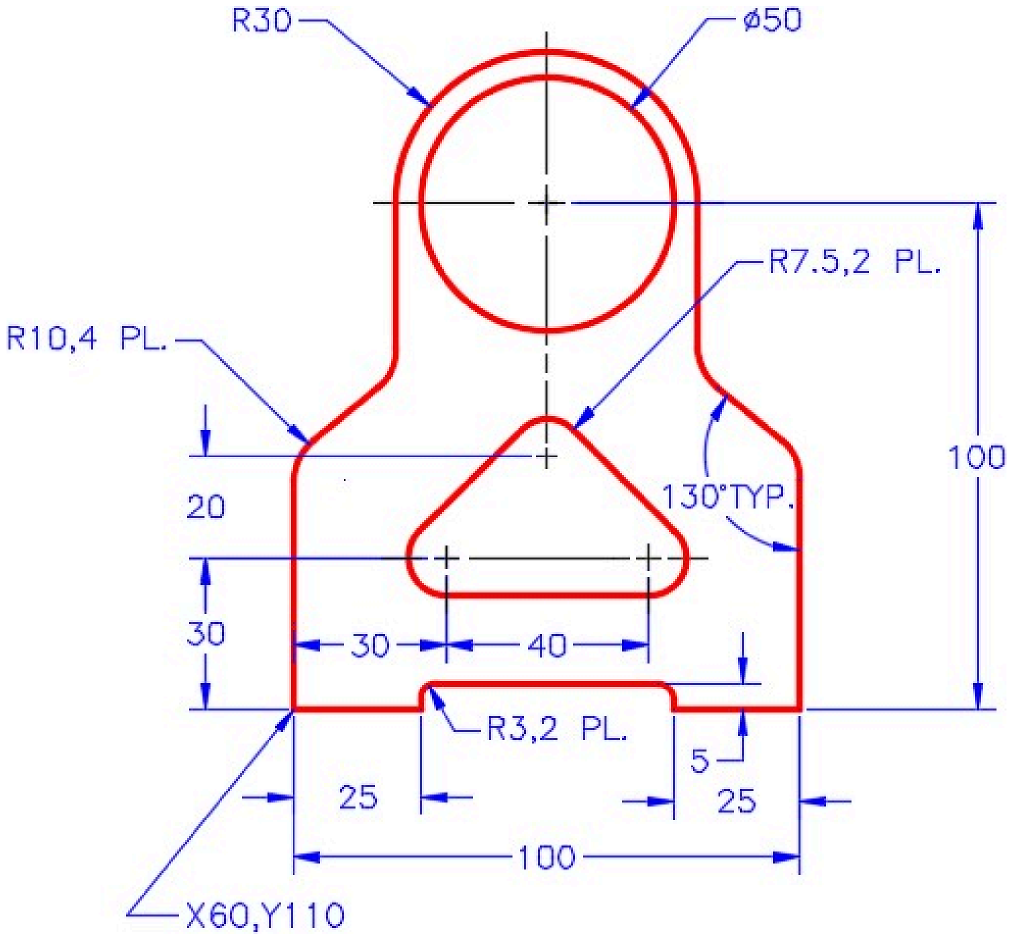

R30
ø50
R7.5,2 PL.
R10,4 PL.
20
130°TYP.
100
30
30
40
R3,2 PL.
5
25
25
100
X60,Y110

Figure Step 4B

Step 5

Turn layer: Viewport off.

Step 6

Using the standards in Module 20, complete the titleblock.

Lab Exercise 38-3

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 2D Lab 38-3	N/A	Inches

Step 1

Open the drawing: AutoCAD 2D Lab 37-3. Using the SAVEAS command, save and name it: AutoCAD 2D Lab 38-3.

Step 2

Set the current dimension style to: 2D English.

Step 3

Ensure that associate dimensioning is enabled by checking the setting of DIMASSOC.

Step 4

In Paper space, on layer: Dimensions, insert the dimensions shown in the figure. Match the figure the best you can. (Figure step 4A and 4B)

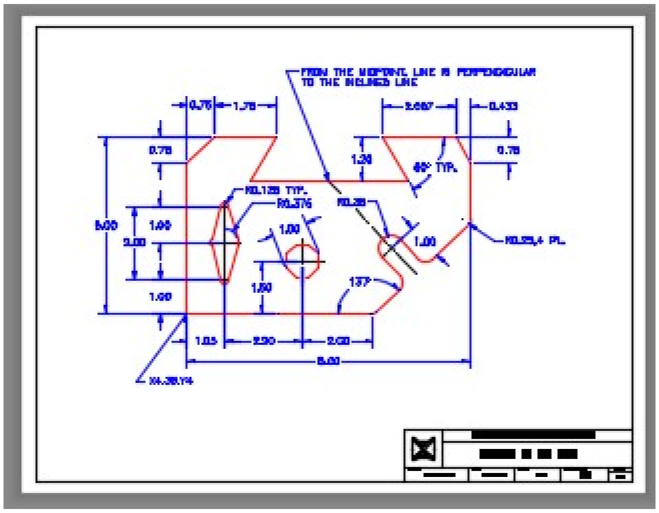

FROM THE MIDPOINT, LINE IS PERPENDICULAR TO THE INCLINED LINE
0.75
1.75
2.667
0.433
0.75
1.25
0.75
45° TYP.
R0.125 TYP.
R0.375
R0.25
5.00
1.00
3.00
1.00
1.00
1.00
R0.25,4 PL.
1.50
137°
1.05
2.50
2.00
8.00
X4.38,Y4

Figure Step 4A

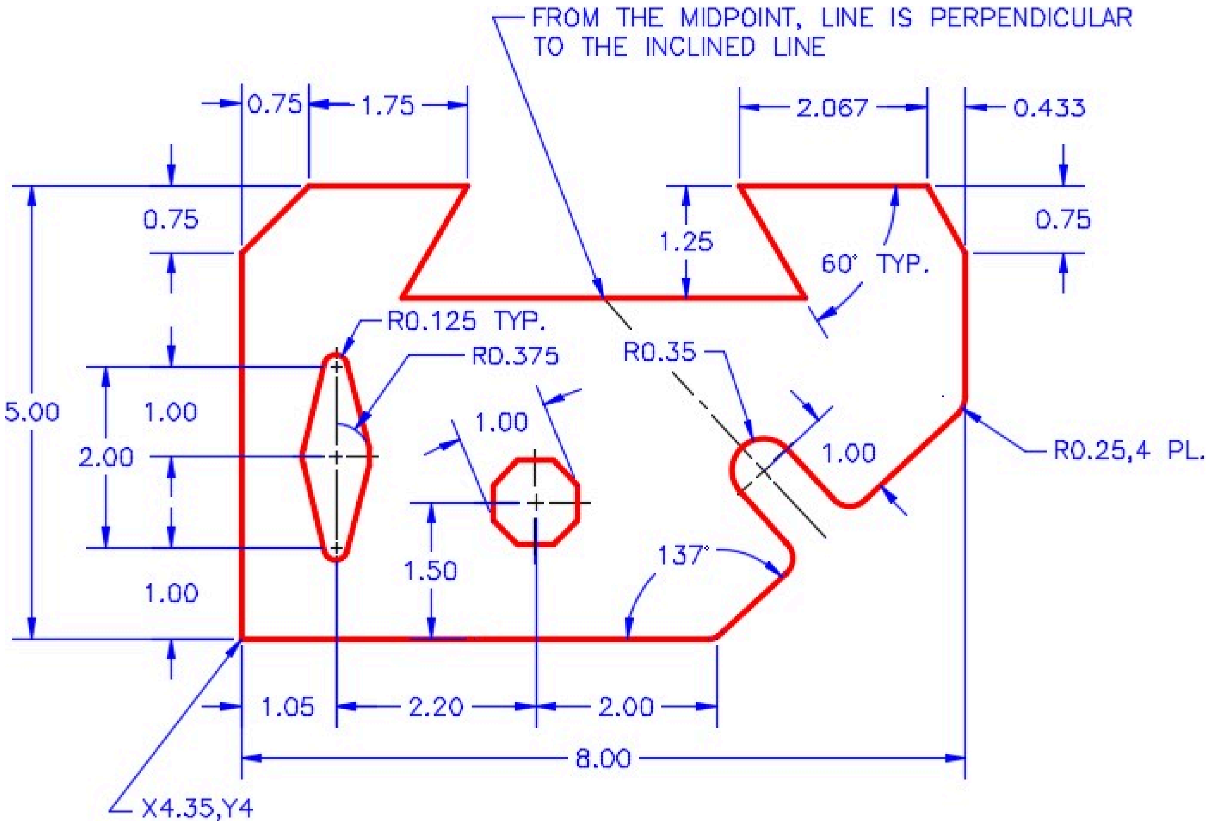

FROM THE MIDPOINT, LINE IS PERPENDICULAR TO THE INCLINED LINE
0.75
1.75
2.067
0.433
0.75
1.25
0.75
60° TYP.
5.00
R0.125 TYP.
R0.375
R0.35
1.00
1.00
2.00
1.00
R0.25,4 PL.
1.00
1.50
137°
1.00
1.05
2.20
2.00
8.00
X4.35,Y4

Figure Step 4B

Step 5

Turn layer: Viewport off.

Step 6

Using the standards in Module 20, complete the titleblock.

Module 39: DesignCenter®

Learning Outcomes

When you have completed this module, you will be able to:

1. Describe how DesignCenter can be used to increase drawing productivity,
2. Apply the ADCENTER command.

DesignCenter

DesignCenter, Figure 39-1, organizes access to drawings, blocks, layers, and other drawing content. From DesignCenter, you can insert, copy, or drag and drop objects from open drawings, filed drawings, a network, or even the internet into your current drawing. *Drag and drop* is the process of dragging an object, such as a block, from one drawing and dropping it into another.

Another important feature of DesignCenter is being able to copy and paste or drag and drop layer definitions, dimensioning styles, linetypes, layouts, text styles, and xrefs between drawings. A important feature of DesignCenter is that the drawings being copying from do not have to be open at the time, however, the drawing that the items are pasted into, must be open and the current drawing. DesignCenter can also be used to open drawing files.

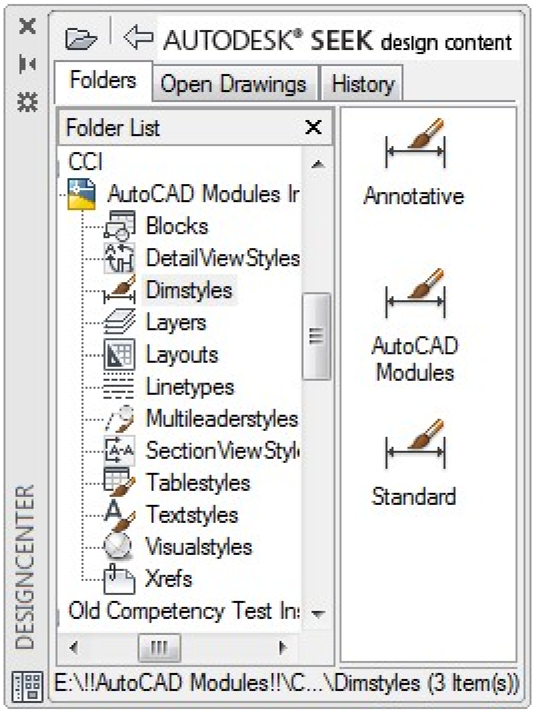
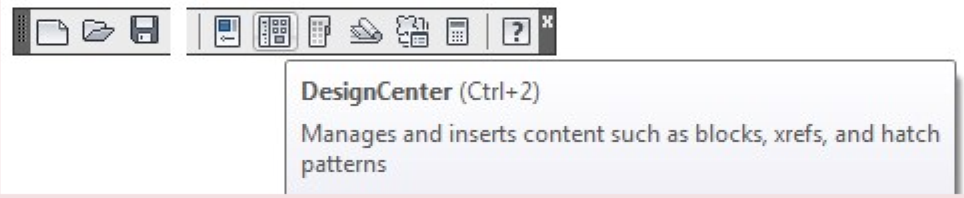

AUTODESK® SEEK design content
Folders
Open Drawings
History
Folder List
CCI
AutoCAD Modules Ir
Blocks
DetailViewStyles
Dimstyles
Layers
Layouts
Linetypes
Multileaderstyles
SectionViewStyl
Tablestyles
Textstyles
Visualstyles
Xrefs
Old Competency Test In
Annotative
AutoCAD Modules
Standard
DESIGNCENTER
E:\!!AutoCAD Modules!!\C...\Dimstyles (3 Item(s))

Figure 39-1
DesignCenter

AutoCAD Command: ADCENTER

The ADCENTER command is used to open the DesignCenter window.

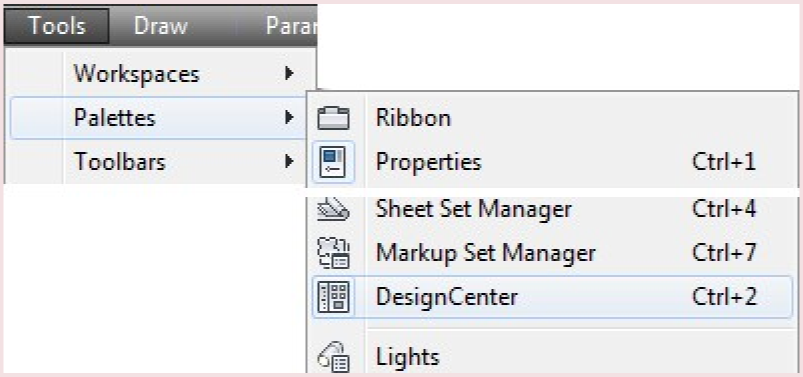
Shortcut: **ADC** or **CTRL+2**



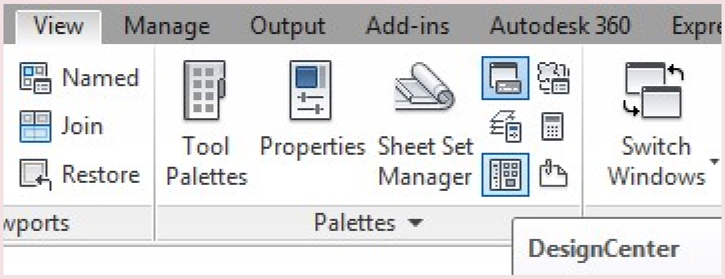
DesignCenter (Ctrl+2)

Manages and inserts content such as blocks, xrefs, and hatch patterns

Standard Toolbar


Tools
Draw
Para
Workspaces
Palettes
Toolbars
Ribbon
Properties Ctrl+1
Sheet Set Manager Ctrl+4
Markup Set Manager Ctrl+7
DesignCenter Ctrl+2
Lights

Tools Pull-down


View
Manage
Output
Add-ins
Autodesk 360
Expre
Named
Join
Restore
wports
Tool Palettes
Properties
Sheet Set Manager
Palettes
Switch Windows
DesignCenter

View Ribbon

WORKALONG: Using DesignCenter

Step 1

Start a new drawing using the template: 2D Layout English.

Step 2

Save and name the drawing: AutoCAD 2D Workalong 39-1.

Step 3

Open the drawing: AutoCAD 2D Lab 33-1. You now have two drawings open at the same time. To confirm that, click the Windows pull-down. (Figure Step 3)

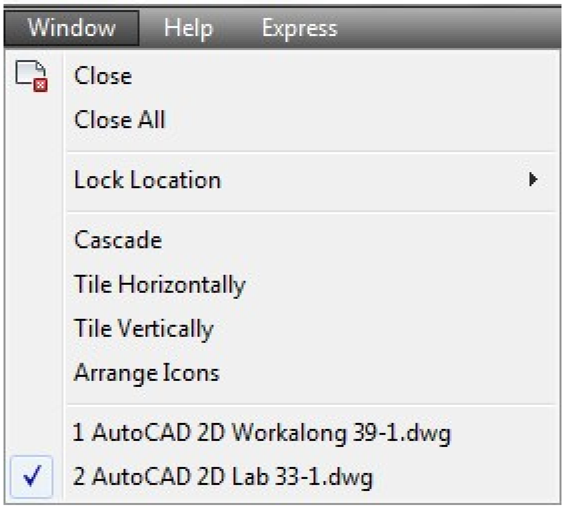

Window
Help
Express
Close
Close All
Lock Location
Cascade
Tile Horizontally
Tile Vertically
Arrange Icons
1 AutoCAD 2D Workalong 39-1.dwg
✓ 2 AutoCAD 2D Lab 33-1.dwg

Figure Step 3

Step 4

Set the drawing: AutoCAD 2D Lab 33-1 as the current drawing. (Figure Step 4)

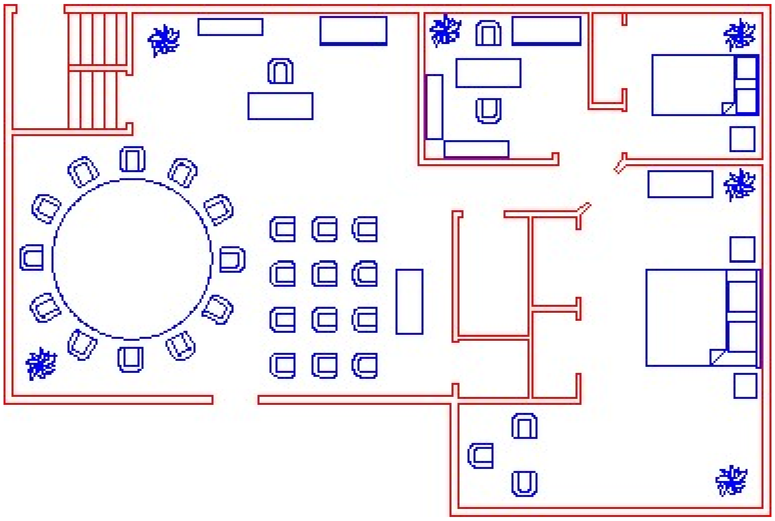


Figure Step 4

AUTHOR'S COMMENTS: The current drawing's name is preceded with the checkmark icon as shown in Figure Step 3. The drawing AutoCAD 2D Lab 33-1 should now display in the Graphic window.

Step 5

Enter the PURGE command. In the Purge dialogue box, purge all blocks that can be purged in the current drawing.

Step 6

Enter the ADCENTER command. It will open the DesignCenter window. (Figure Step 6)

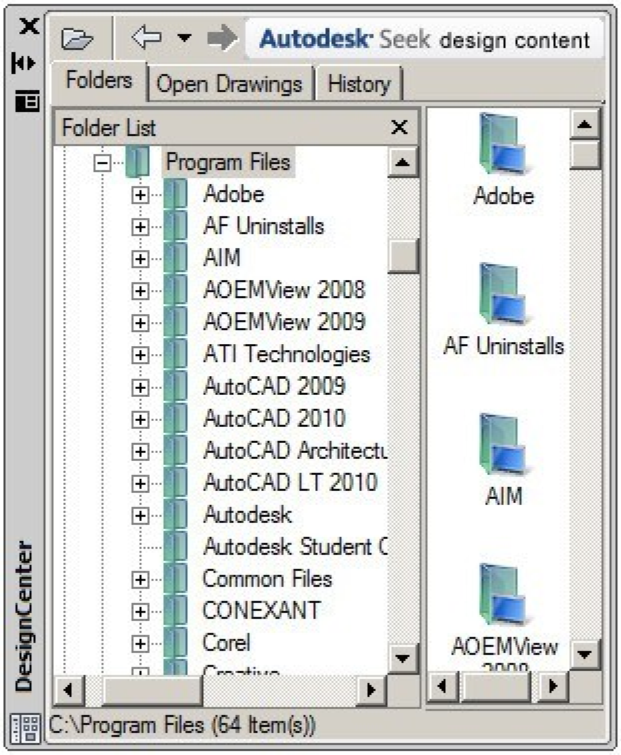
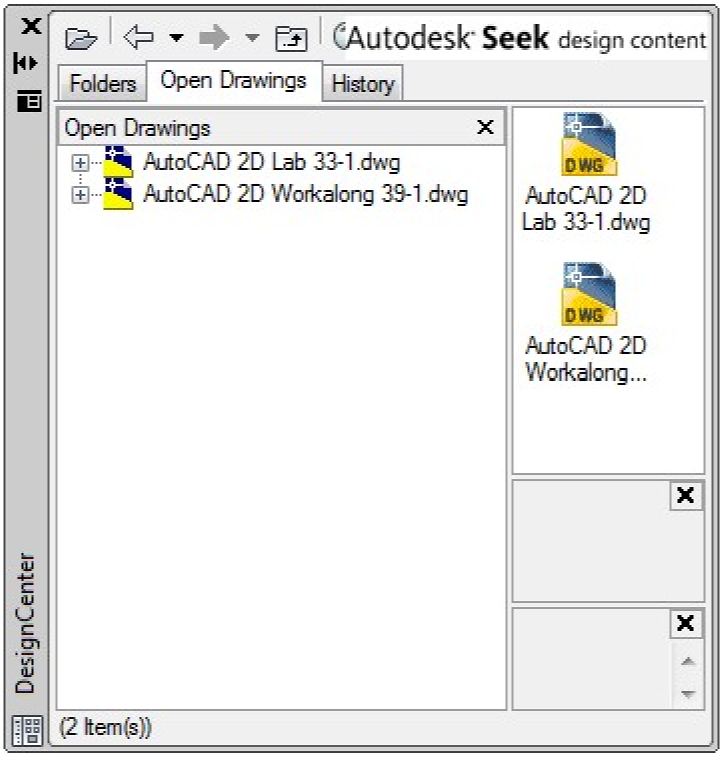

Autodesk Seek design content
Folders
Open Drawings
History
Folder List
Program Files
Adobe
AF Uninstalls
AIM
AOEMView 2008
AOEMView 2009
ATI Technologies
AutoCAD 2009
AutoCAD 2010
AutoCAD Architectu
AutoCAD LT 2010
Autodesk
Autodesk Student C
Common Files
CONEXANT
Corel
Adobe
AF Uninstalls
AIM
AOEMView
DesignCenter
C:\Program Files (64 Item(s))

Figure Step 6

AUTHOR'S COMMENTS: The folder structure will display differently on your computer than shown in Figure Step 6.

Step 7

Enable the Open Drawings tab. It will display the two drawings that are currently open in your AutoCAD session. (Figure Step 7)



Autodesk Seek design content
Folders
Open Drawings
History
Open Drawings
AutoCAD 2D Lab 33-1.dwg
AutoCAD 2D Workalong 39-1.dwg
DWG
AutoCAD 2D Lab 33-1.dwg
DWG
AutoCAD 2D Workalong...
DesignCenter
(2 Item(s))

Figure Step 7

Step 8

Click the + sign beside the drawing: AutoCAD 2D Lab 33-1.dwg to expand it as shown in the figure. It will now display the – sign preceding it to display the items in the drawing. (Figure Step 8)

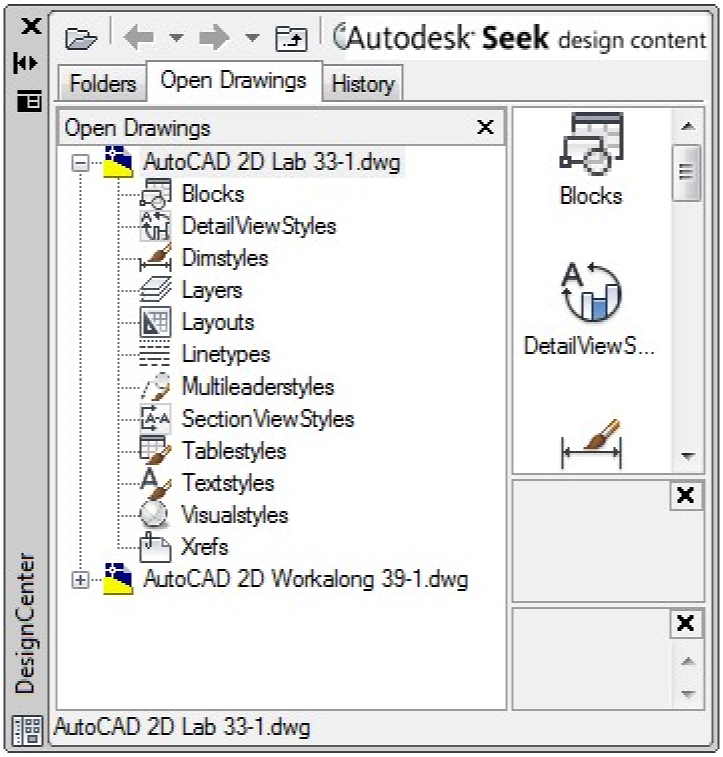
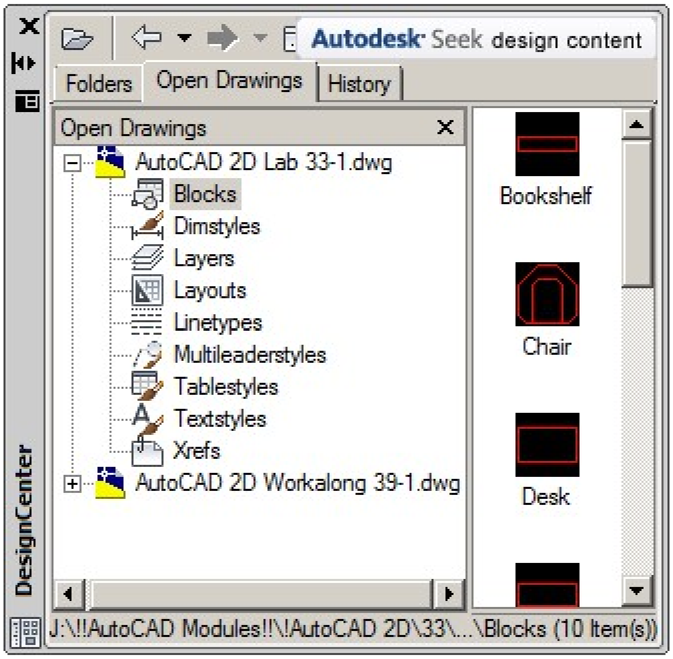

Autodesk Seek design content
Folders
Open Drawings
History
Open Drawings
AutoCAD 2D Lab 33-1.dwg
Blocks
DetailViewStyles
Dimstyles
Layers
Layouts
Linetypes
Multileaderstyles
SectionViewStyles
Tablestyles
Textstyles
Visualstyles
Xrefs
AutoCAD 2D Workalong 39-1.dwg
Blocks
DetailViewS...
DesignCenter
AutoCAD 2D Lab 33-1.dwg

Figure Step 8

Step 9

Select the item Blocks to highlight it. Along the right side of the window the icons of the blocks that are currently defined in the drawing's block definition table will display. (Figure Step 9)

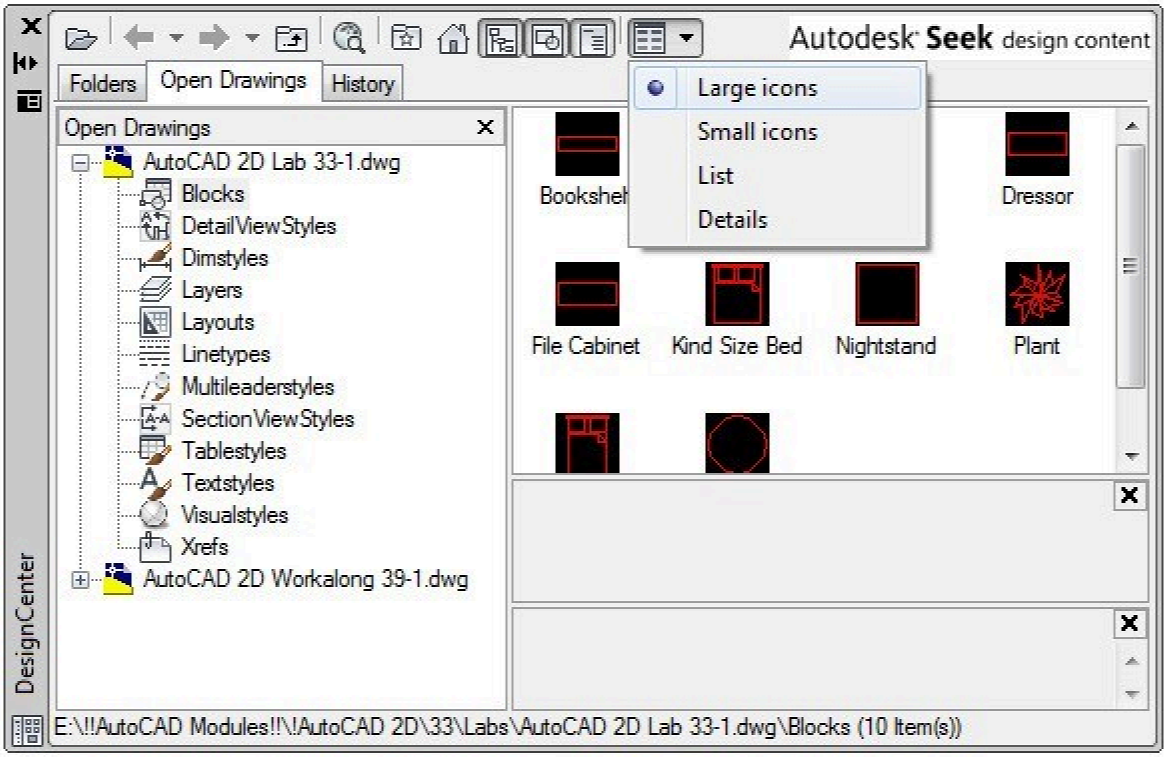

Autodesk Seek design content
Folders
Open Drawings
History
Open Drawings
AutoCAD 2D Lab 33-1.dwg
Blocks
Dimstyles
Layers
Layouts
Linetypes
Multileaderstyles
Tablestyles
Textstyles
Xrefs
AutoCAD 2D Workalong 39-1.dwg
Bookshelf
Chair
Desk
DesignCenter

J:\!!AutoCAD Modules!!\!AutoCAD 2D\33\...\Blocks (10 Item(s))

Figure Step 9

Step 10

Expand the window as shown in the figure. Click the Views icon and in the Pull-down list, enable Large Icons. (Figure Step 10)


Autodesk Seek design content
Folders
Open Drawings
History
Large icons
Small icons
List
Details
Open Drawings
AutoCAD 2D Lab 33-1.dwg
Blocks
DetailViewStyles
Dimstyles
Layers
Layouts
Linetypes
Multileaderstyles
SectionViewStyles
Tablestyles
Textstyles
Visualstyles
Xrefs
AutoCAD 2D Workalong 39-1.dwg
Bookshel
Dressor
File Cabinet
Kind Size Bed
Nightstand
Plant
DesignCenter

E:\!!AutoCAD Modules!!\!AutoCAD 2D\33\Labs\AutoCAD 2D Lab 33-1.dwg\Blocks (10 Item(s))

Figure Step 10

Step 11

Disable the Description icon and enable the Preview icon. (Figure Step 11A and 11B)

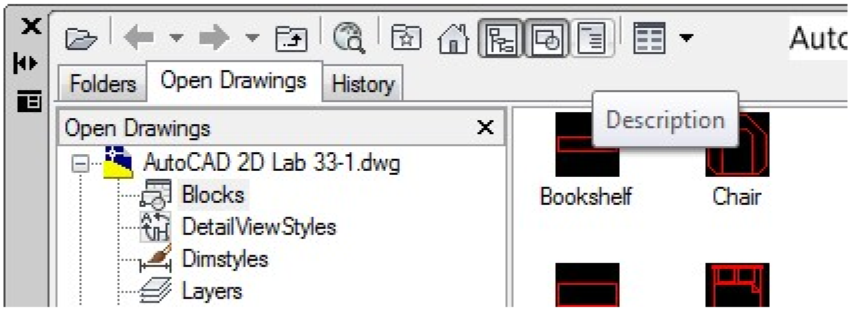

Autc
Folders
Open Drawings
History
Description
Open Drawings
AutoCAD 2D Lab 33-1.dwg
Blocks
DetailViewStyles
Dimstyles
Layers
Bookshelf
Chair

Figure Step 11A

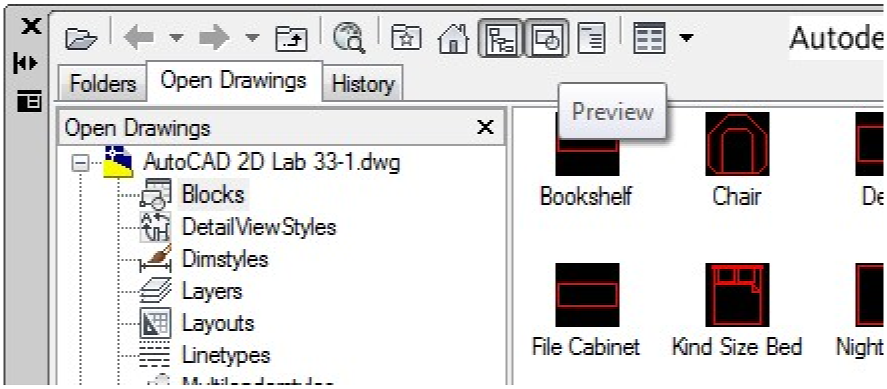

Autode
Folders
Open Drawings
History
Preview
Open Drawings
AutoCAD 2D Lab 33-1.dwg
Blocks
DetailViewStyles
Dimstyles
Layers
Layouts
Linetypes
Bookshelf
Chair
De
File Cabinet
Kind Size Bed
Night

Figure Step 11B

Step 12

Resize the window as shown in the figure. Click the Chair icon and it will display a preview of the Chair block in the lower right corner of the window. (Figure Step 12)

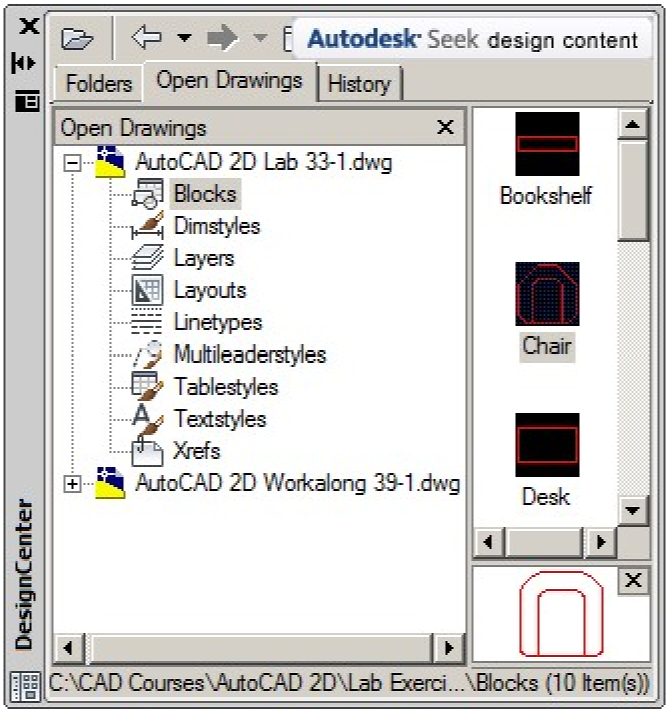

Autodesk Seek design content
Folders
Open Drawings
History
Open Drawings
AutoCAD 2D Lab 33-1.dwg
Blocks
Dimstyles
Layers
Layouts
Linetypes
Multileaderstyles
Tablestyles
Textstyles
Xrefs
AutoCAD 2D Workalong 39-1.dwg
Bookshelf
Chair
Desk
DesignCenter
C:\CAD Courses\AutoCAD 2D\Lab Exerci...\Blocks (10 Item(s))

Figure Step 12

Step 13

Set the drawing AutoCAD 2D Workalong 39-1 as the current drawing.

Step 14

Right click the Chair icon. In the Right-click menu, click Insert Block. When the Insert dialogue box opens, set it as shown in the figure. Click the OK button. (Figure Step 14A and 14B)

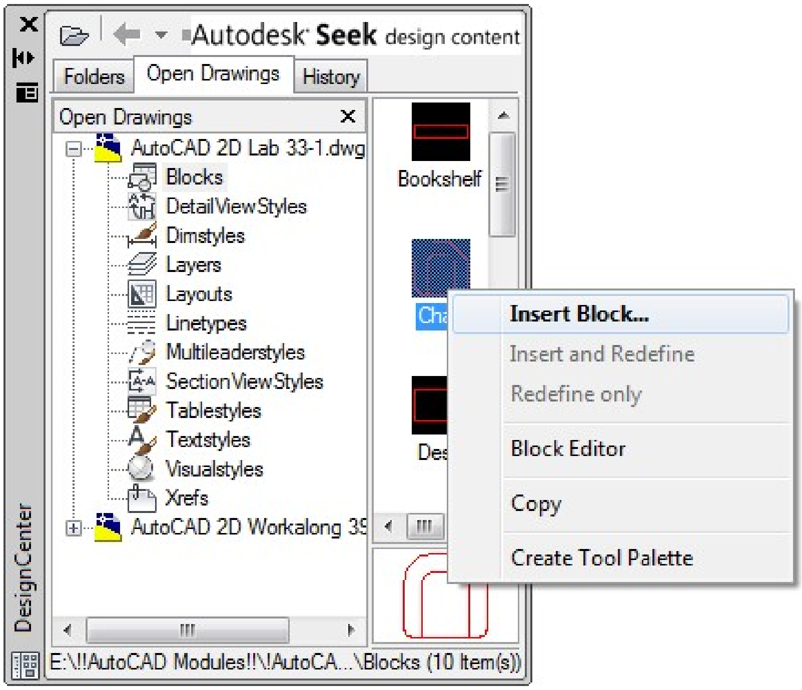
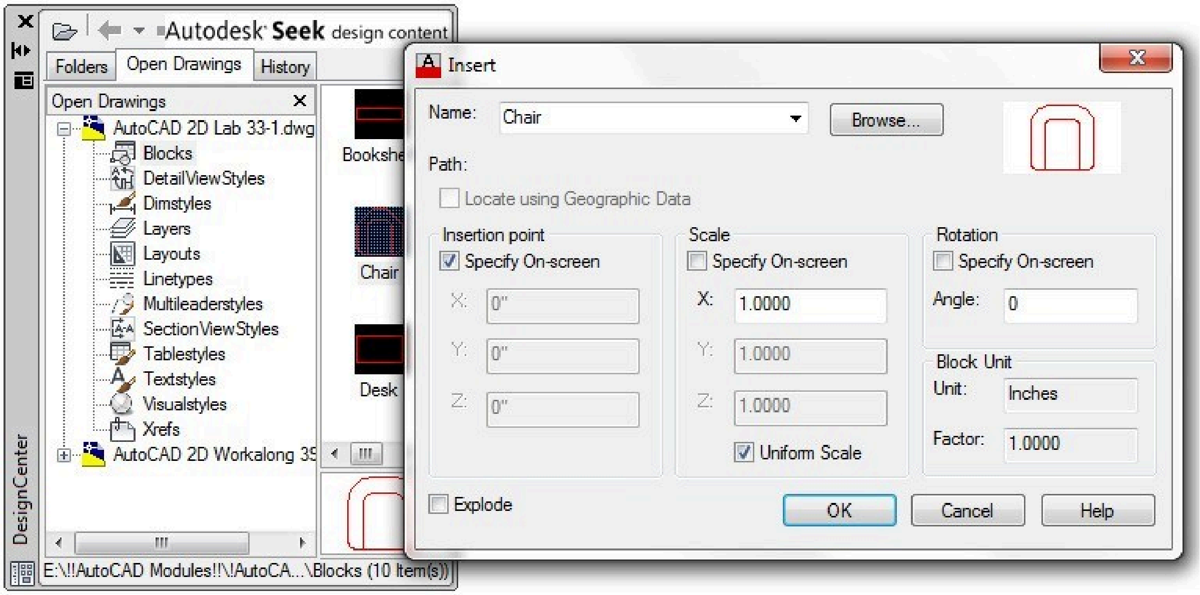

Autodesk Seek design content
Folders
Open Drawings
History
Open Drawings
AutoCAD 2D Lab 33-1.dwg
Blocks
DetailViewStyles
Dimstyles
Layers
Layouts
Linetypes
Multileaderstyles
SectionViewStyles
Tablestyles
Textstyles
Visualstyles
Xrefs
AutoCAD 2D Workalong 3
Bookshelf
Cha
Des
Insert Block...
Insert and Redefine
Redefine only
Block Editor
Copy
Create Tool Palette
DesignCenter
E:\!!AutoCAD Modules!!\!AutoCA...\Blocks (10 Item(s))

Figure Step 14A


Autodesk Seek design content
Folders
Open Drawings
History
Open Drawings
AutoCAD 2D Lab 33-1.dwg
Blocks
DetailViewStyles
Dimstyles
Layers
Layouts
Linetypes
Multileaderstyles
SectionViewStyles
Tablestyles
Textstyles
Visualstyles
Xrefs
AutoCAD 2D Workalong 3
Booksh
Chair
Desk
DesignCenter
E:\!!AutoCAD Modules!!\!AutoCA...\Blocks (10 Item(s))
Insert
Browse...
OK
Cancel
Help

Name: Chair

Path:

☐ Locate using Geographic Data

Insertion point
☑ Specify On-screen
X: 0"
Y: 0"
Z: 0"

Scale
☐ Specify On-screen
X: 1.0000
Y: 1.0000
Z: 1.0000
☑ Uniform Scale

Rotation
☐ Specify On-screen
Angle: 0

Block Unit
Unit: Inches
Factor: 1.0000

☐ Explode

Figure Step 14B

Step 15

Using the crosshairs, select any location on the drawing and press the pick button. (Figure Step 15)

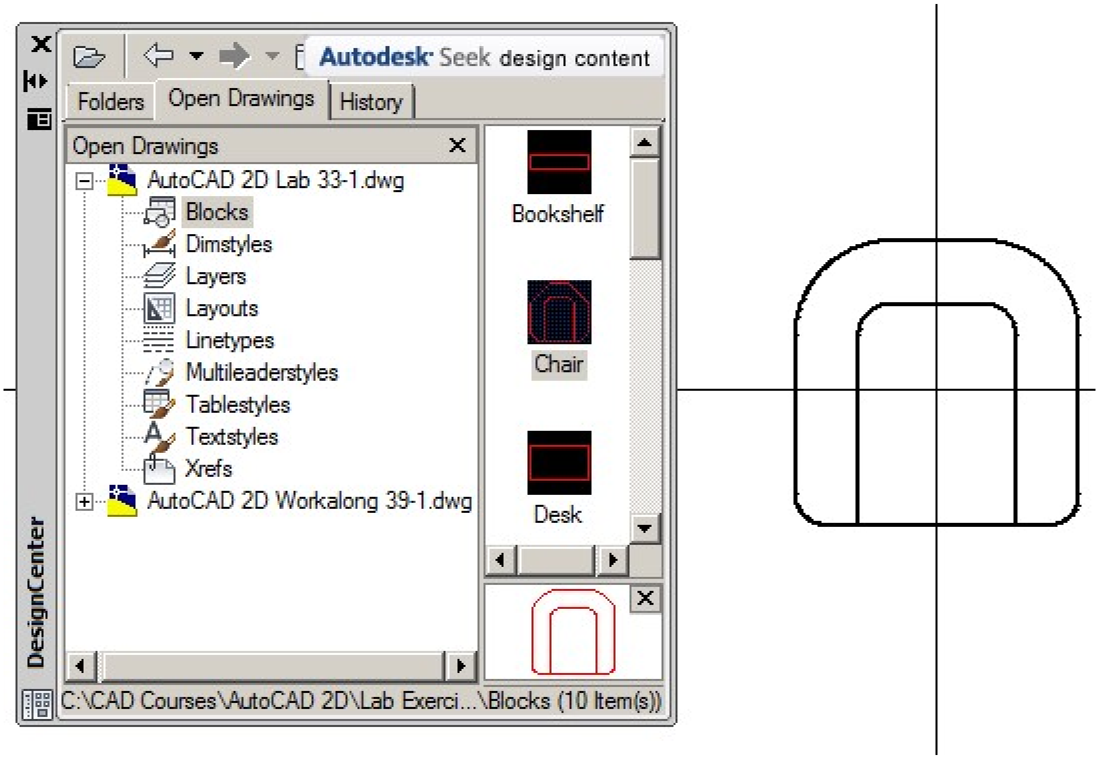

Autodesk Seek design content
Folders
Open Drawings
History
Open Drawings
AutoCAD 2D Lab 33-1.dwg
Blocks
Dimstyles
Layers
Layouts
Linetypes
Multileaderstyles
Tablestyles
Textstyles
Xrefs
AutoCAD 2D Workalong 39-1.dwg
Bookshelf
Chair
Desk
C:\CAD Courses\AutoCAD 2D\Lab Exerci...\Blocks (10 Item(s))
DesignCenter

Figure Step 15

Step 16

Use the drag and drop method of copying a block from one drawing to another. In DesignCenter, click the Desk icon and while holding down the left mouse button, drag the icon into the current drawing and release the button.

Step 17

Using the methods you just learned, copy the eight additional blocks into your drawing. Copy four with each method. Your drawing should appear similar to the figure. (Figure Step 17)

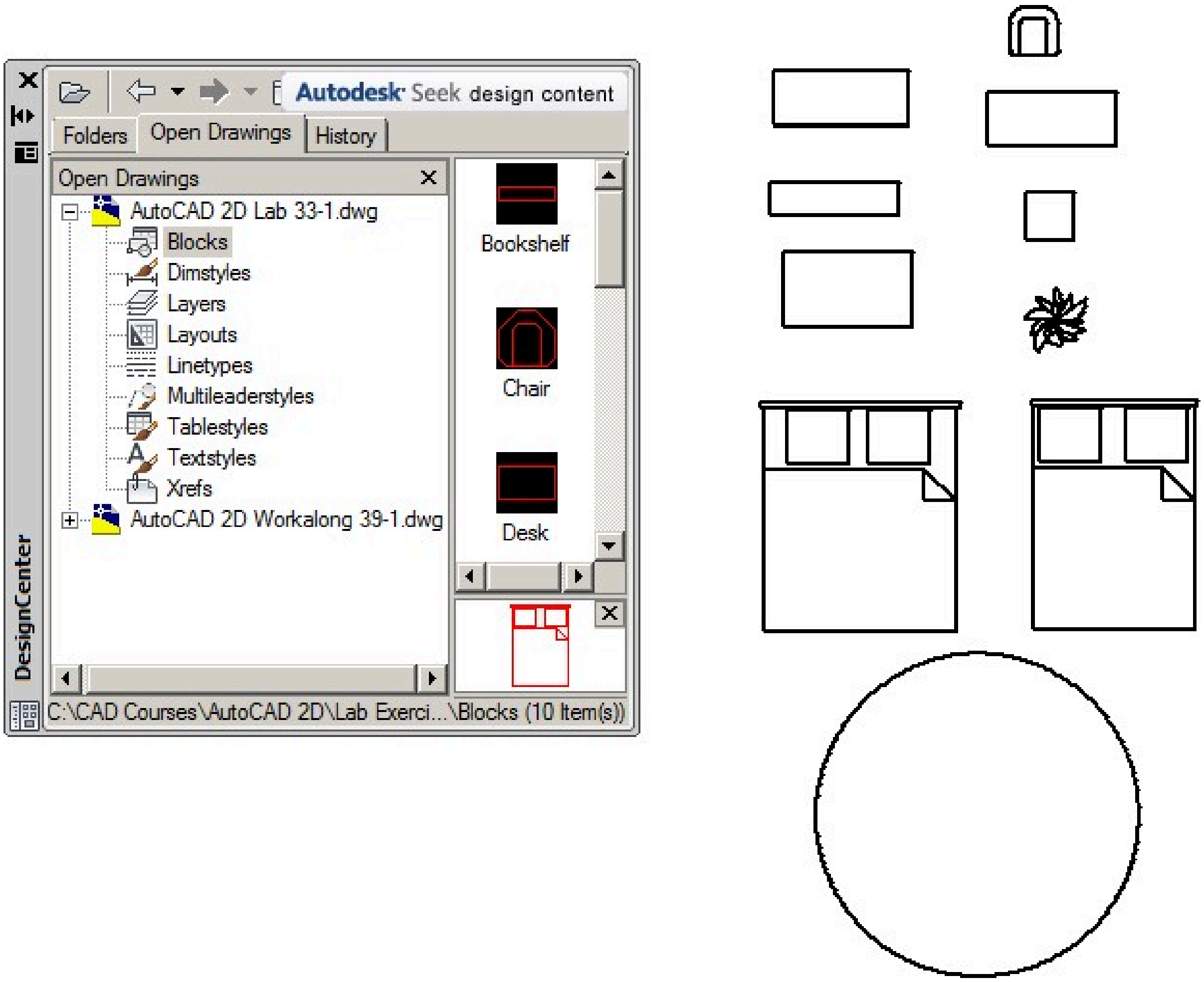

Autodesk Seek design content
Folders
Open Drawings
History
Open Drawings
AutoCAD 2D Lab 33-1.dwg
Blocks
Dimstyles
Layers
Layouts
Linetypes
Multileaderstyles
Tablestyles
Textstyles
Xrefs
AutoCAD 2D Workalong 39-1.dwg
Bookshelf
Chair
Desk
DesignCenter
C:\CAD Courses\AutoCAD 2D\Lab Exerci...\Blocks (10 Item(s))

Figure Step 17

AUTHOR’S COMMENTS: Another way to work with DesignCenter is to work using drawings that are not open at the time.

Step 18

Enable the Folders tab and then find the Lab Exercises folder as shown in the figure. (Figure Step 18)

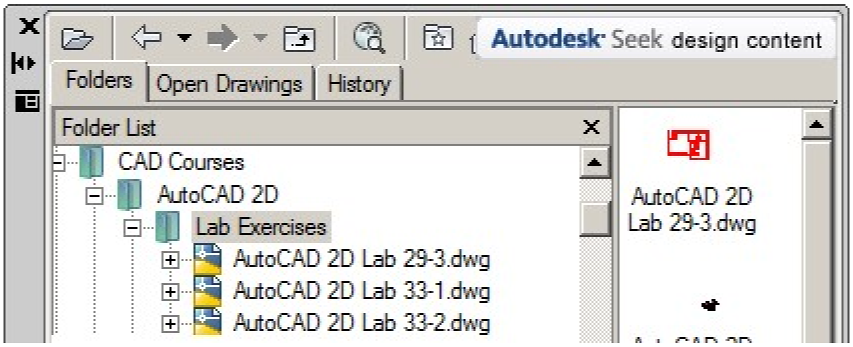

Autodesk Seek design content
Folders
Open Drawings
History
Folder List
CAD Courses
AutoCAD 2D
Lab Exercises
AutoCAD 2D Lab 29-3.dwg
AutoCAD 2D Lab 33-1.dwg
AutoCAD 2D Lab 33-2.dwg
AutoCAD 2D Lab 29-3.dwg

Figure Step 18

Step 19

In the folder, locate the drawing: AutoCAD 2D Lab 33-2.dwg. Expand the drawing and expand Blocks as shown in the figure. (Figure Step 19)

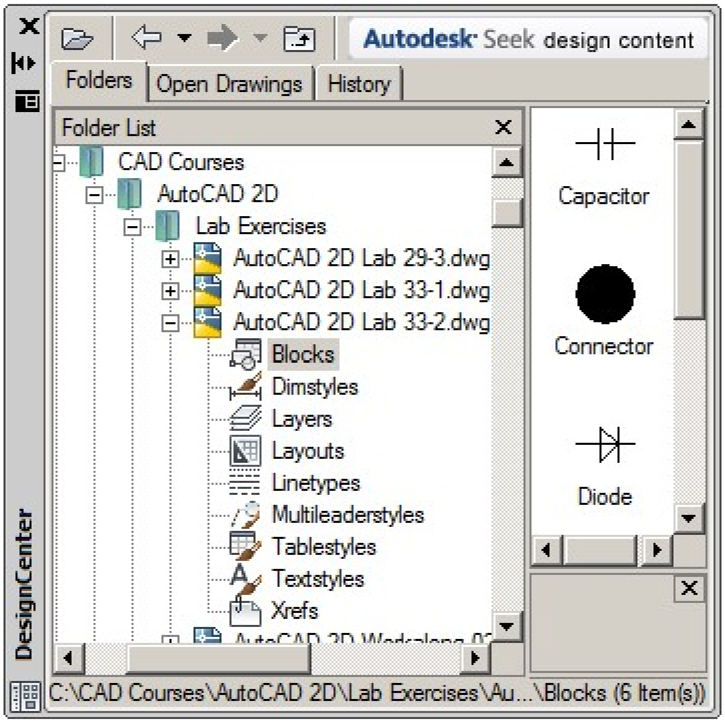
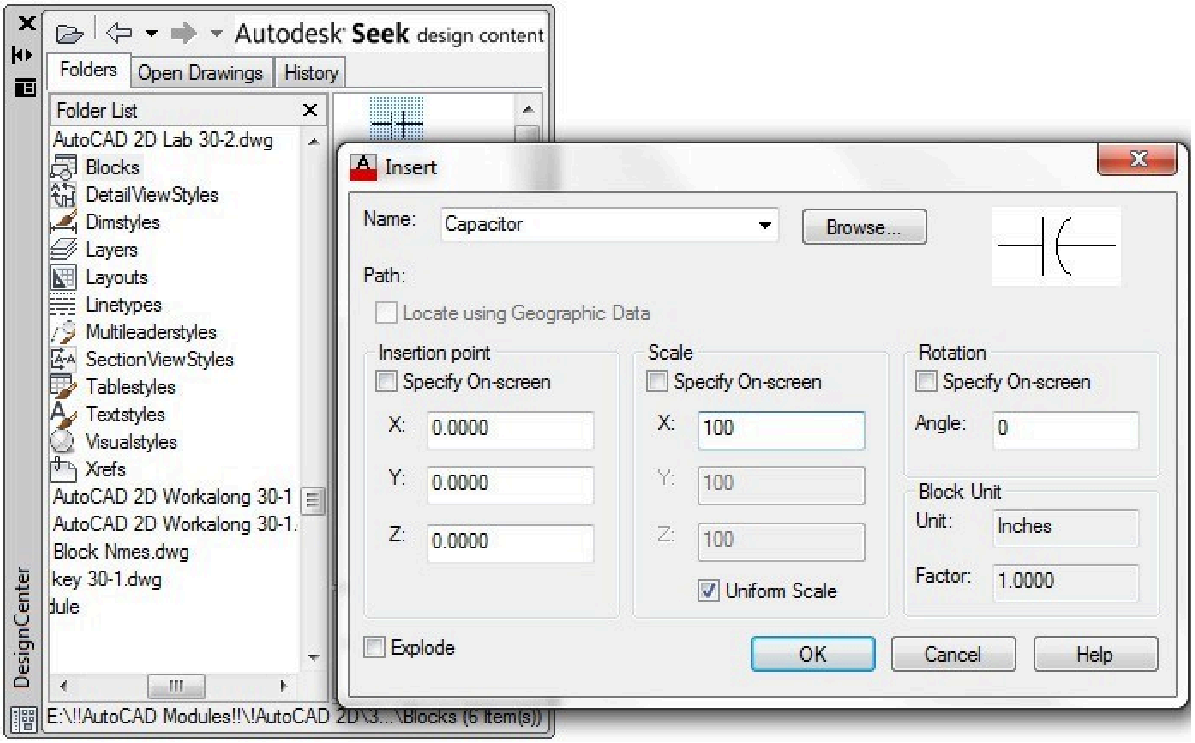

Autodesk Seek design content
Folders
Open Drawings
History
Folder List
CAD Courses
AutoCAD 2D
Lab Exercises
AutoCAD 2D Lab 29-3.dwg
AutoCAD 2D Lab 33-1.dwg
AutoCAD 2D Lab 33-2.dwg
Blocks
Dimstyles
Layers
Layouts
Linetypes
Multileaderstyles
Tablestyles
Textstyles
Xrefs
Capacitor
Connector
Diode
DesignCenter
C:\CAD Courses\AutoCAD 2D\Lab Exercises\Au...\Blocks (6 Item(s))

Figure Step 19

Step 20

Using the Insert block method, insert all of the blocks from the drawing:

AutoCAD 2D Lab 30-2.dwg into the current drawing. Insert them anywhere in the Model space at a scale of 100. (Figure Step 20A and 20B)


Autodesk Seek design content
Folders
Open Drawings
History
Folder List
AutoCAD 2D Lab 30-2.dwg
Blocks
DetailViewStyles
Dimstyles
Layers
Layouts
Linetypes
Multileaderstyles
SectionViewStyles
Tablestyles
Textstyles
Visualstyles
Xrefs
AutoCAD 2D Workalong 30-1
AutoCAD 2D Workalong 30-1.
Block Nmes.dwg
key 30-1.dwg
dule
DesignCenter
E:\!!AutoCAD Modules!!\!AutoCAD 2D\3...\Blocks (6 Item(s))
Insert
Browse...
OK
Cancel
Help

Name: Capacitor

Path:

☐ Locate using Geographic Data

Insertion point
☐ Specify On-screen
X: 0.0000
Y: 0.0000
Z: 0.0000

Scale
☐ Specify On-screen
X: 100
Y: 100
Z: 100
☑ Uniform Scale

Rotation
☐ Specify On-screen
Angle: 0

Block Unit
Unit: Inches
Factor: 1.0000

☐ Explode

Figure Step 20A

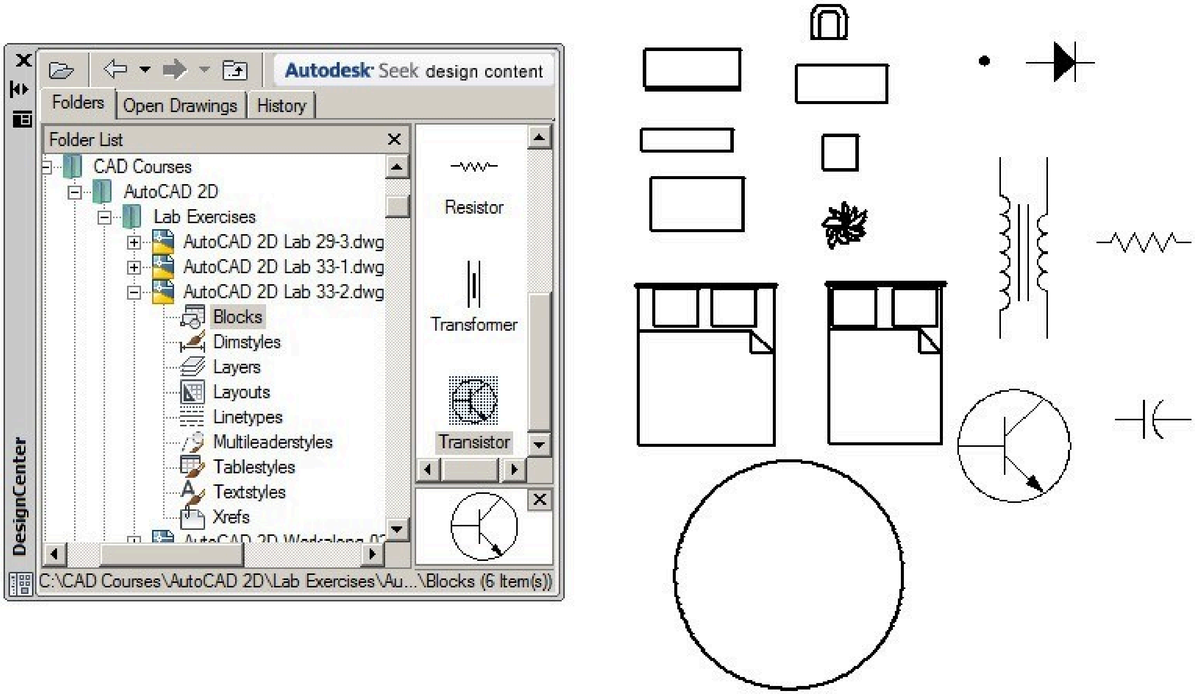

Autodesk Seek design content
Folders
Open Drawings
History
Folder List
CAD Courses
AutoCAD 2D
Lab Exercises
AutoCAD 2D Lab 29-3.dwg
AutoCAD 2D Lab 33-1.dwg
AutoCAD 2D Lab 33-2.dwg
Blocks
Dimstyles
Layers
Layouts
Linetypes
Multileaderstyles
Tablestyles
Textstyles
Xrefs
Resistor
Transformer
Transistor
DesignCenter
C:\CAD Courses\AutoCAD 2D\Lab Exercises\Au...\Blocks (6 Item(s))

Figure Step 20B

Step 21

Delete all blocks in the drawing but do not purge them. They are still in the block definition table. Your drawing should appear blank.

Step 22

Enable the Open Drawings tab and expand the drawing: AutoCAD 2D Workalong 39-1. Right-click the View icon and in the right-click menu, select List. (Figure Step 22)

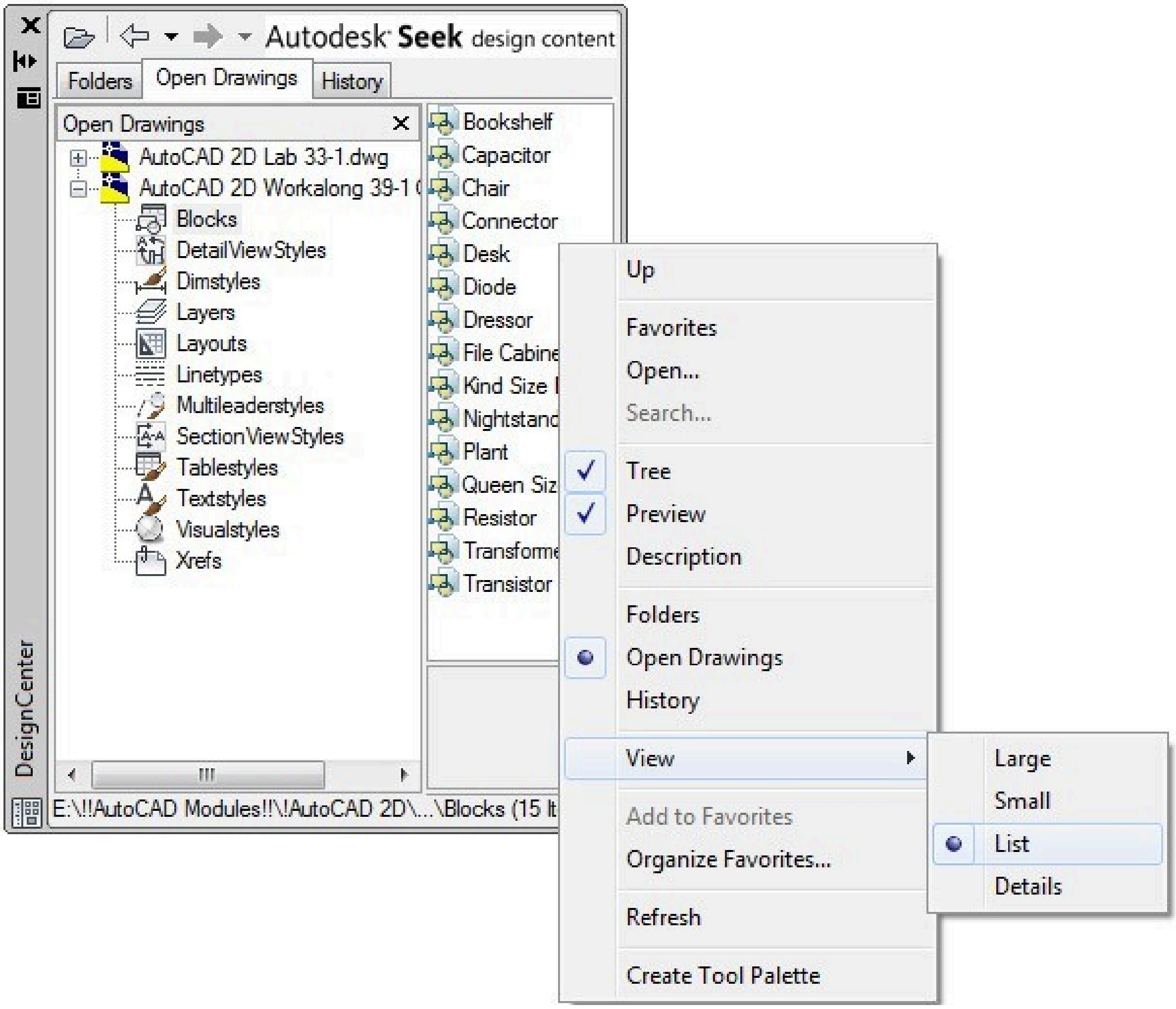

Autodesk Seek design content
Folders
Open Drawings
History
Open Drawings
AutoCAD 2D Lab 33-1.dwg
AutoCAD 2D Workalong 39-1
Blocks
DetailViewStyles
Dimstyles
Layers
Layouts
Linetypes
Multileaderstyles
SectionViewStyles
Tablestyles
Textstyles
Visualstyles
Xrefs
Bookshelf
Capacitor
Chair
Connector
Desk
Diode
Dressor
File Cabine
Kind Size
Nightstand
Plant
Queen Siz
Resistor
Transforme
Transistor
DesignCenter
E:\!!AutoCAD Modules!!\!AutoCAD 2D\...\Blocks (15 It
Up
Favorites
Open...
Search...
✓ Tree
✓ Preview
Description
Folders
● Open Drawings
History
View
Add to Favorites
Organize Favorites...
Refresh
Create Tool Palette
Large
Small
● List
Details

Figure Step 22

Step 23

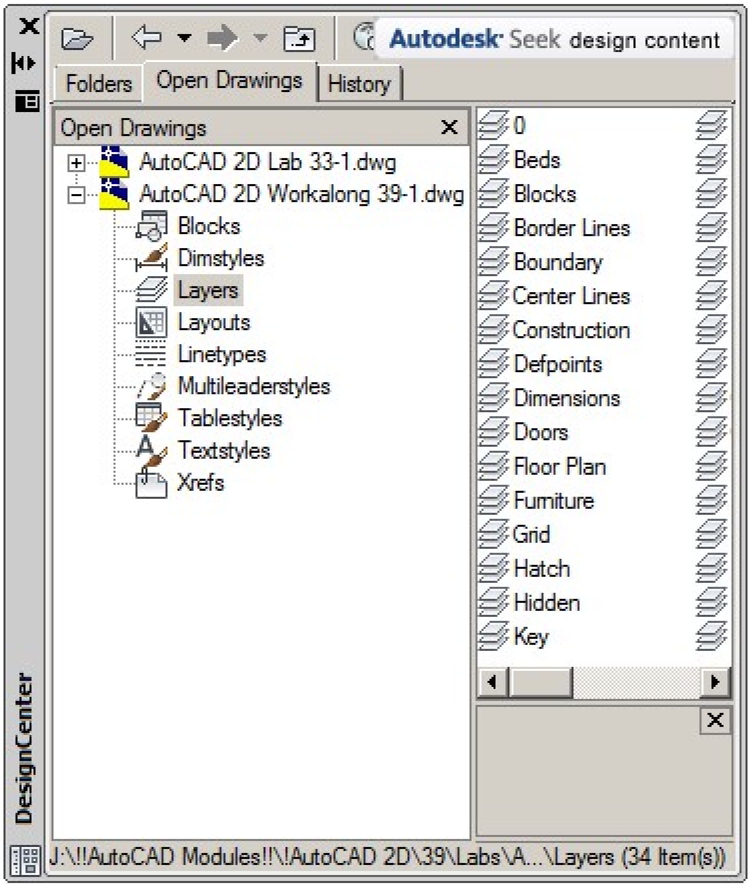
Set the drawing: AutoCAD 2D Lab 33-1 as the current drawing.

Step 24

Enter the PURGE command. In the Purge dialogue box, purge all layers that can be purged from the drawing.

Step 25

In the Open Drawings tab, expand the drawing: AutoCAD 2D Workalong 39-1 and the item Layers. (Figure Step 25)


Autodesk Seek design content
Folders
Open Drawings
History
Open Drawings
AutoCAD 2D Lab 33-1.dwg
AutoCAD 2D Workalong 39-1.dwg
Blocks
Dimstyles
Layers
Layouts
Linetypes
Multileaderstyles
Tablestyles
Textstyles
Xrefs
0
Beds
Blocks
Border Lines
Boundary
Center Lines
Construction
Defpoints
Dimensions
Doors
Floor Plan
Furniture
Grid
Hatch
Hidden
Key
DesignCenter

J:\!!AutoCAD Modules!!\!AutoCAD 2D\39\Labs\A...\Layers (34 Item(s))

Figure Step 25

Step 26

Select all layers and using the drag and drop method drag method, drag them into the current drawing. (Figure Step 26)

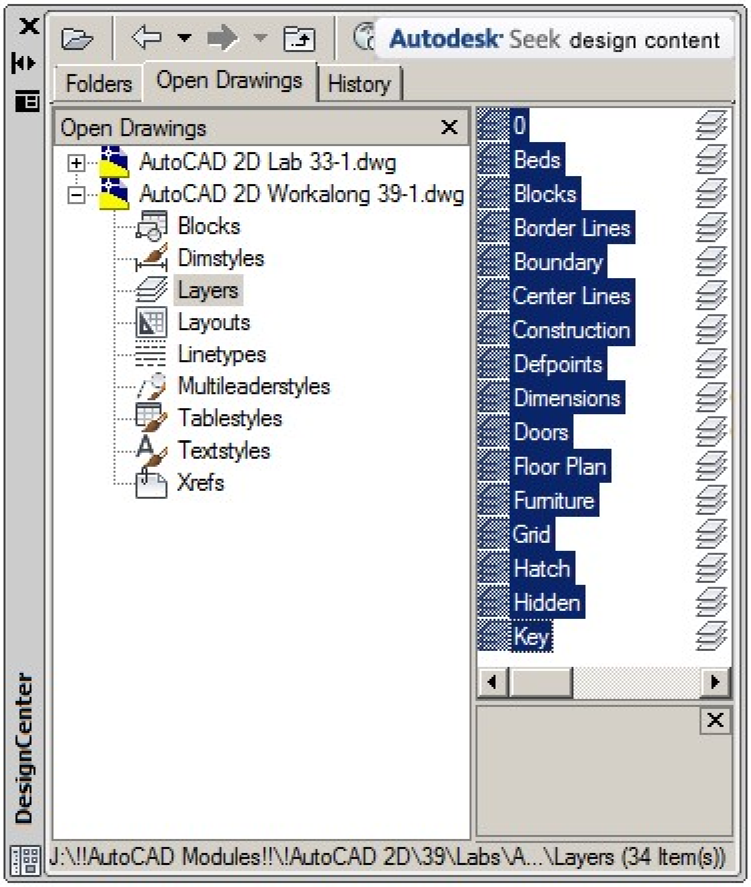

Autodesk Seek design content
Folders
Open Drawings
History
Open Drawings
AutoCAD 2D Lab 33-1.dwg
AutoCAD 2D Workalong 39-1.dwg
Blocks
Dimstyles
Layers
Layouts
Linetypes
Multileaderstyles
Tablestyles
Textstyles
Xrefs
0
Beds
Blocks
Border Lines
Boundary
Center Lines
Construction
Defpoints
Dimensions
Doors
Floor Plan
Furniture
Grid
Hatch
Hidden
Key
DesignCenter
J:\!!AutoCAD Modules!!\!AutoCAD 2D\39\Labs\A...\Layers (34 Item(s))

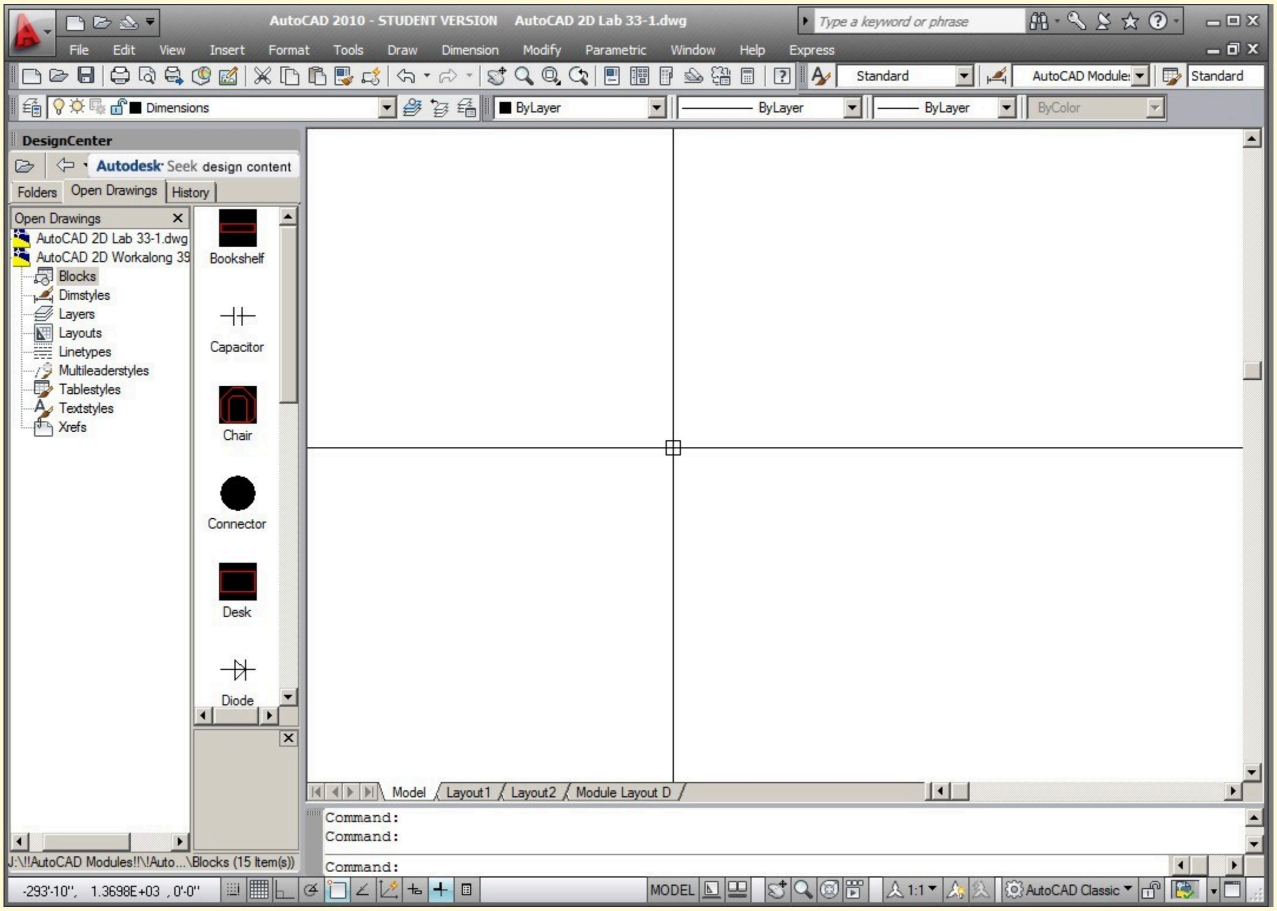
Figure Step 26

AUTHOR'S COMMENTS: As you see, items from any of the 9 areas can be copied from one drawing to another.

Step 27

Save and close the drawing.

USER TIP: A good way to use the DesignCenter window is to dock it along the left side of the graphic window as shown below.


AutoCAD 2010 - STUDENT VERSION AutoCAD 2D Lab 33-1.dwg
Type a keyword or phrase
File
Edit
View
Insert
Format
Tools
Draw
Dimension
Modify
Parametric
Window
Help
Express
Standard
AutoCAD Modules
Standard
Dimensions
ByLayer
ByLayer
ByLayer
ByColor
DesignCenter
Autodesk Seek design content
Folders
Open Drawings
History
Open Drawings
AutoCAD 2D Lab 33-1.dwg
AutoCAD 2D Workalong 39
Blocks
Dimstyles
Layers
Layouts
Linetypes
Multileaderstyles
Tablestyles
Textstyles
Xrefs
Bookshelf
Capacitor
Chair
Connector
Desk
Diode
J:\!!AutoCAD Modules!!\Auto...\Blocks (15 Item(s))
Model
Layout1
Layout2
Module Layout D
-293'-10", 1.3698E+03 , 0'-0"
MODEL
1:1
AutoCAD Classic

```
Command:
Command:

Command:
```

Key Principles

Key Principles in Module 39

1. The most powerful use of DesignCenter is being able to copy blocks, layers, dimensioning styles, linetypes, layouts, text styles and xrefs from one drawing to another. Using it will improve your overall drawing speed and production.

Module 40: Layouts – Part 2

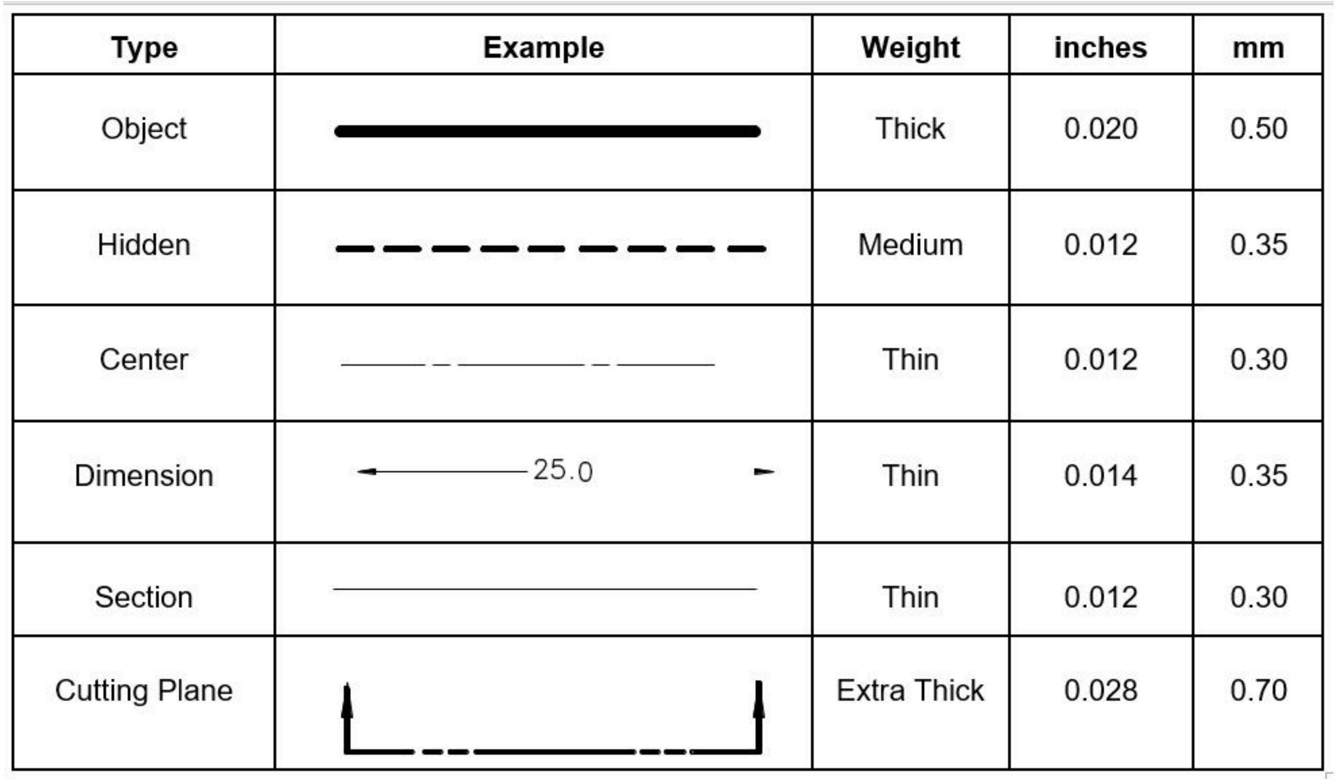
Learning Outcomes

When you have completed this module, you will be able to:

1. Describe lineweights and their use in a plotted drawing.
2. Describe and apply the object and color methods of assigning lineweights in a drawing.
3. Explain how to calculate and set customs viewport scakes.

Drafting Lesson: Lineweights

Industry standards have assigned a type, weight, and width to each drawing object on a plotted drawing. This makes the plotted drawings much easier to read and understand. It is important to follow those standards when printing a layout. The AutoCAD 2D book Lineweights chart, Figure 40-1, shows the basic linetypes and their assigned lineweight in both inches and millimeters.



Type	Example	Weight	inches	mm
Object		Thick	0.020	0.50
Hidden		Medium	0.012	0.35
Center		Thin	0.012	0.30
Dimension	25.0	Thin	0.014	0.35
Section		Thin	0.012	0.30
Cutting Plane		Extra Thick	0.028	0.70

Figure 40 -1
AutoCAD 2D book Lineweight Chart

Lineweights

Assigning and displaying lineweight is not important when creating and editing a drawing, but are very important when the layout of that drawing is printed. Lineweights are normally assigned by a company's standard. The AutoCAD 2D book Lineweight Chart is shown in Figure 40-1. The two methods of assigning lineweight are by object or by color.

Assigning Lineweights by Object

One method of assigning lineweights is to assign a lineweight to each drawing object. When the layout is plotted, the plotter simply prints the drawing object with its assigned lineweight. When lineweights are assigned using this method, the lineweight can be displayed in both the Graphics window and the paper print. The lineweights can be assigned to each drawing object individually or ByLayer. ByLayer is much easier and is the method practiced in most companies including the AutoCAD 2D book.

Assigning Lineweights by Color

Assigning the lineweight by color is the most popular method used in the CAD world. Using a company standard chart, as shown in Figure 40-1, lineweights are assigned to each color. When the layout is plotted, the plotter looks at the color of each drawing object and prints it with the lineweight assigned to that color. When lineweights are assigned using this method, the lineweight cannot be displayed in the Graphic window. They only appear on the paper print. The color can be assigned ByObject or ByLayer. ByLayer is used by most companies and used in throughout the AutoCAD 2D book.

WORKALONG: Assigning Lineweights

Step 1

Open the drawing: AutoCAD 2D Lab 32-1.

Step 2

Using the SAVEAS command, save the drawing with the name: AutoCAD 2D Workalong 40-1. (Figure Step 2)

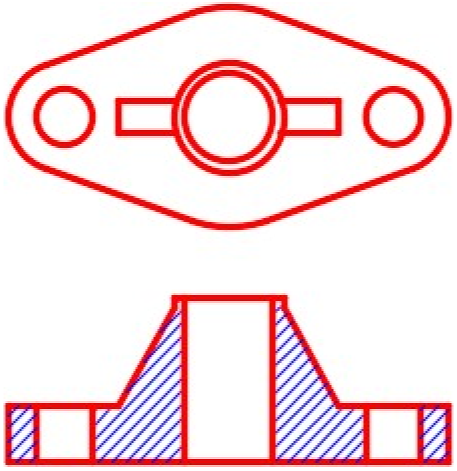


Figure Step 2

Step 3

Enable layout: Module Layout B. Create a viewport on layer: Viewport. Set the scale to 1.5:1 and lock the display. (Figure Step 3)

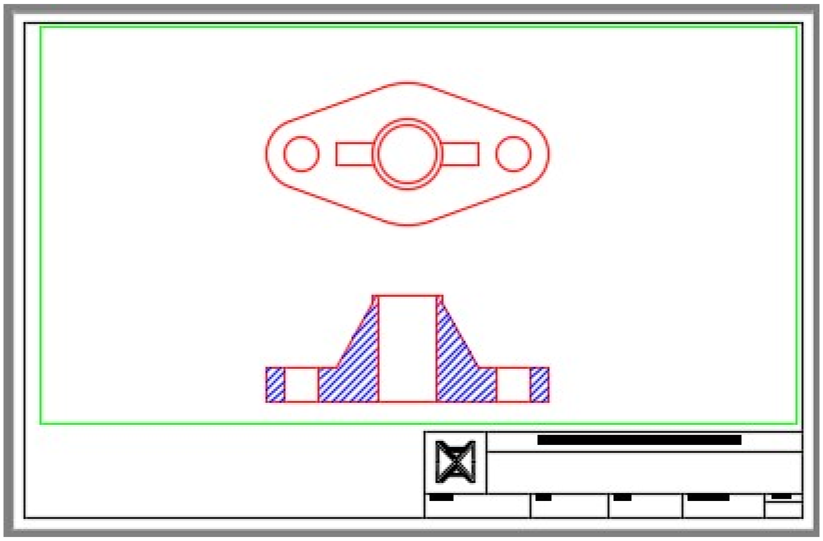


Figure Step 3

Step 4

In Paper space, on layer: Center Lines, draw the center lines on the views.

Step 5

Set the current dimensioning style to: 2D English. In Paper space, on layer: Dimensions, insert the dimensions shown on the figure. (Figure step 5A and 5B)

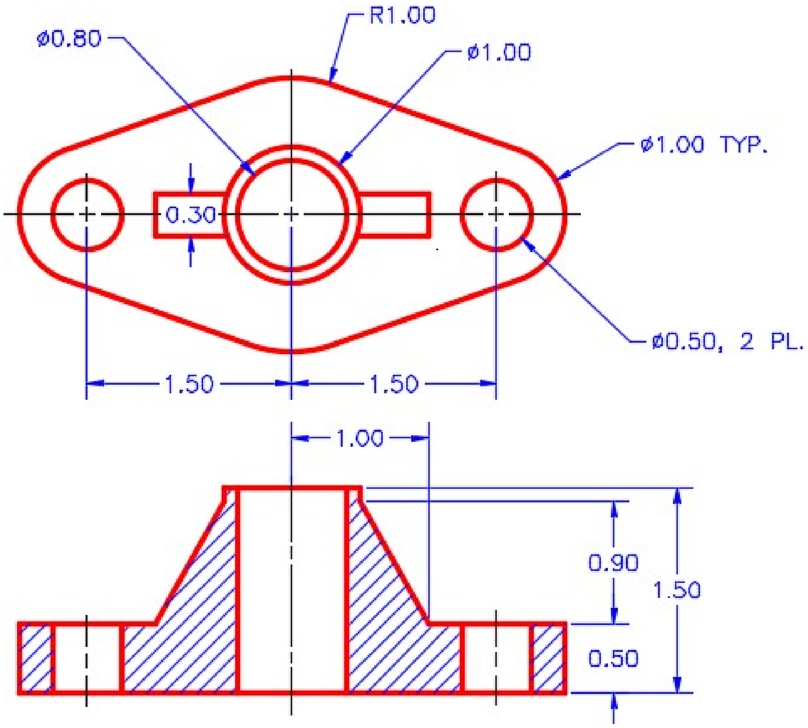

⌀0.80
R1.00
⌀1.00
⌀1.00 TYP.
0.30
⌀0.50, 2 PL.
1.50
1.50
1.00
0.90
1.50
0.50

Figure Step 5A

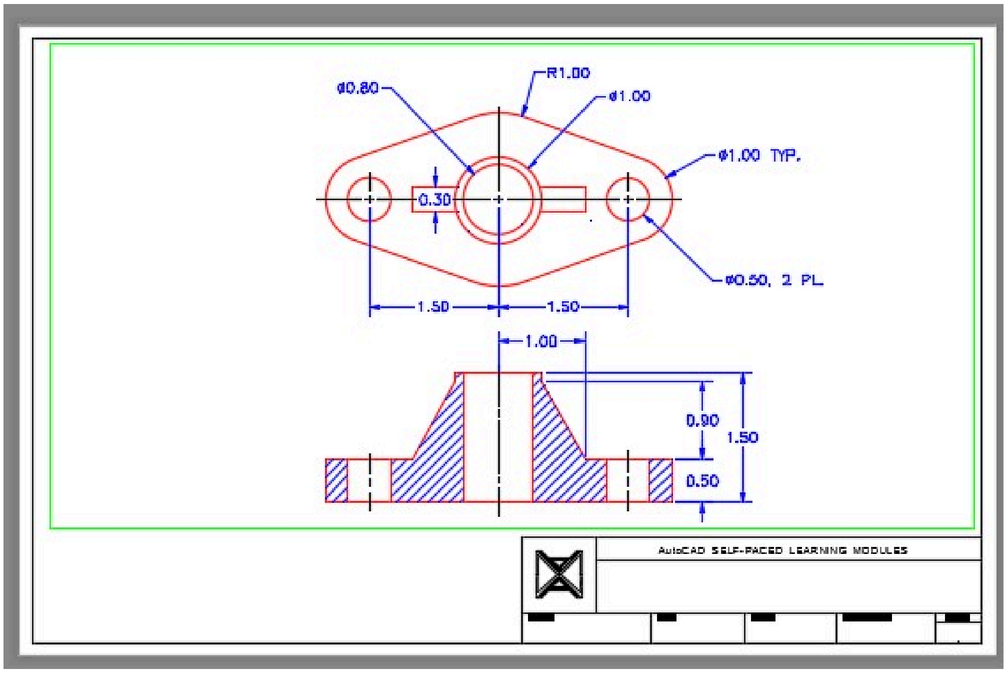
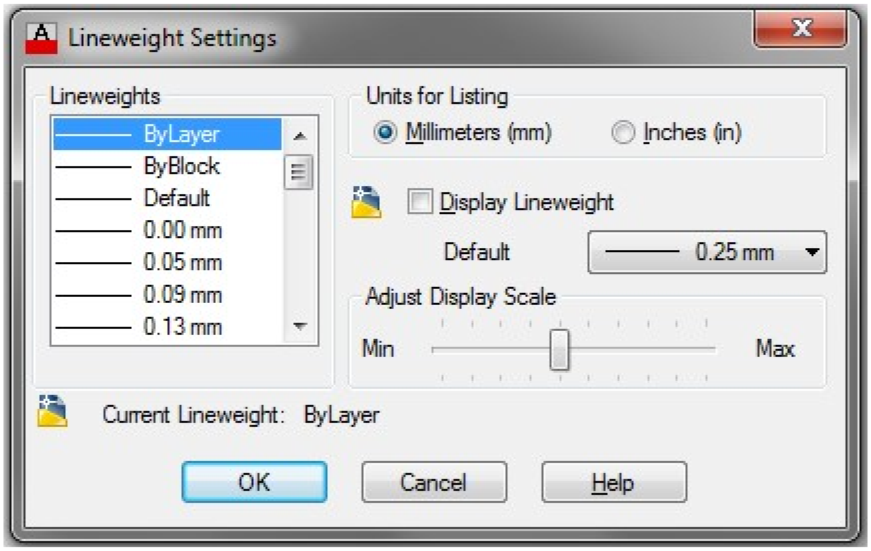

⌀0.80
R1.00
⌀1.00
⌀1.00 TYP.
0.30
⌀0.50, 2 PL.
1.50
1.50
1.00
0.90
1.50
0.50
AutoCAD SELF-PACED LEARNING MODULES

Figure Step 5B

Step 6

Enter the LINEWEIGHT command to open the Lineweight Settings dialogue box. Ensure that your dialogue box matches the settings in the figure. (Figure Step 6)



Lineweight Settings

Lineweights
- ByLayer
- ByBlock
- Default
- 0.00 mm
- 0.05 mm
- 0.09 mm
- 0.13 mm

Units for Listing
● Millimeters (mm) ○ Inches (in)

☐ Display Lineweight

Default: 0.25 mm

Adjust Display Scale: Min — Max

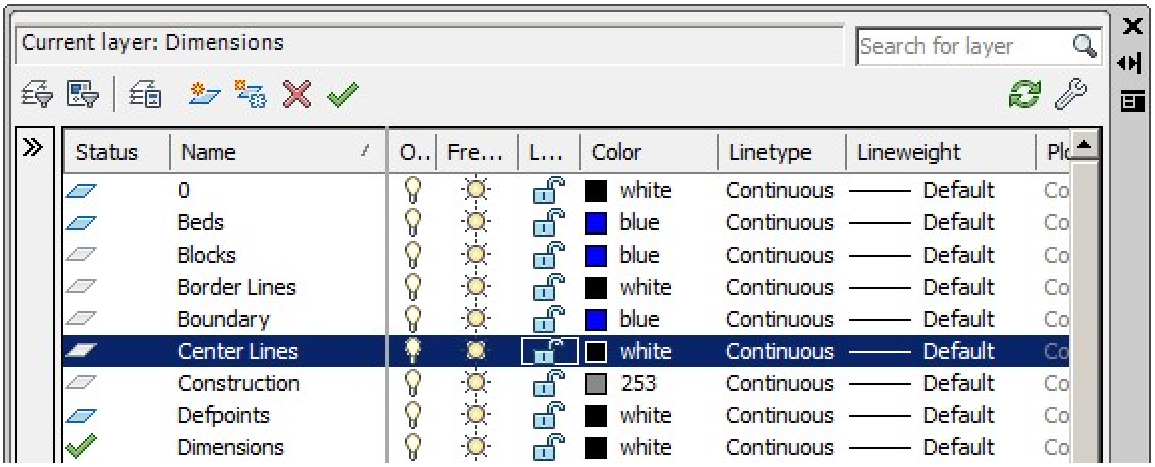
Current Lineweight: ByLayer

Figure Step 6

AUTHOR'S COMMENTS: Lineweight units are normally expressed in millimeters.

Step 7

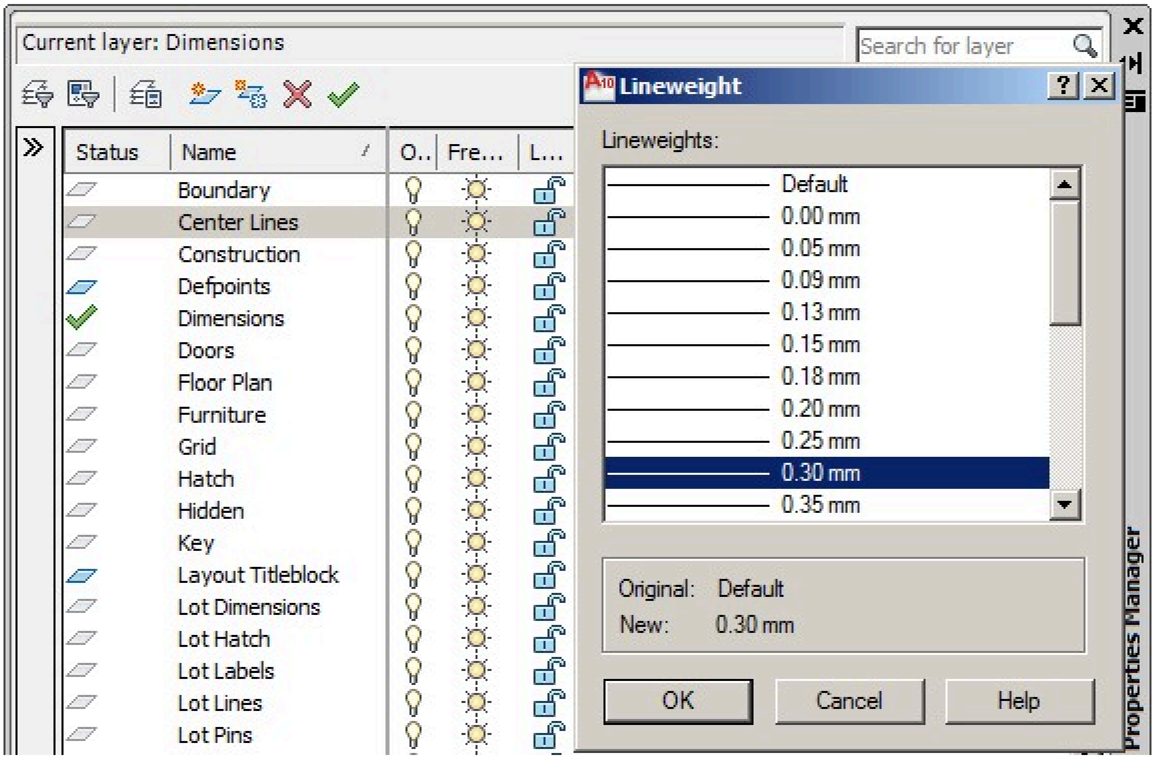
Open the Layer Properties Manager window. Select layer: Center Lines to highlight it. Click Default in the Lineweight column to open the Lineweight dialogue box. Select 0.30 mm. (Figure Step 7A, 7B, and 7C)



Current layer: Dimensions

Status	Name	O..	Fre...	L...	Color	Linetype	Lineweight	Pl
	0				■ white	Continuous	—— Default	Co
	Beds				■ blue	Continuous	—— Default	Co
	Blocks				■ blue	Continuous	—— Default	Co
	Border Lines				■ white	Continuous	—— Default	Co
	Boundary				■ blue	Continuous	—— Default	Co
	Center Lines				□ white	Continuous	—— Default	Co
	Construction				■ 253	Continuous	—— Default	Co
	Defpoints				■ white	Continuous	—— Default	Co
✓	Dimensions				■ white	Continuous	—— Default	Co

Figure Step 7A



Current layer: Dimensions

Status	Name
	Boundary
	Center Lines
	Construction
	Defpoints
✓	Dimensions
	Doors
	Floor Plan
	Furniture
	Grid
	Hatch
	Hidden
	Key
	Layout Titleblock
	Lot Dimensions
	Lot Hatch
	Lot Labels
	Lot Lines
	Lot Pins

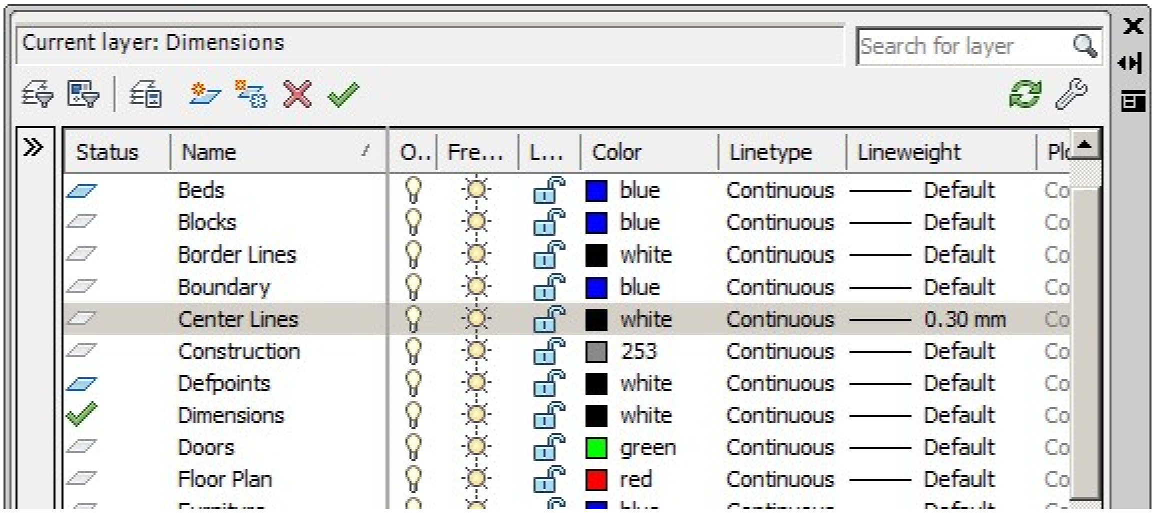
Lineweights:

Default
0.00 mm
0.05 mm
0.09 mm
0.13 mm
0.15 mm
0.18 mm
0.20 mm
0.25 mm
0.30 mm
0.35 mm

Original: Default
New: 0.30 mm

Search for layer
O..
Fre...
L...
Lineweight
OK
Cancel
Help
Properties Manager

Figure Step 7B



Current layer: Dimensions

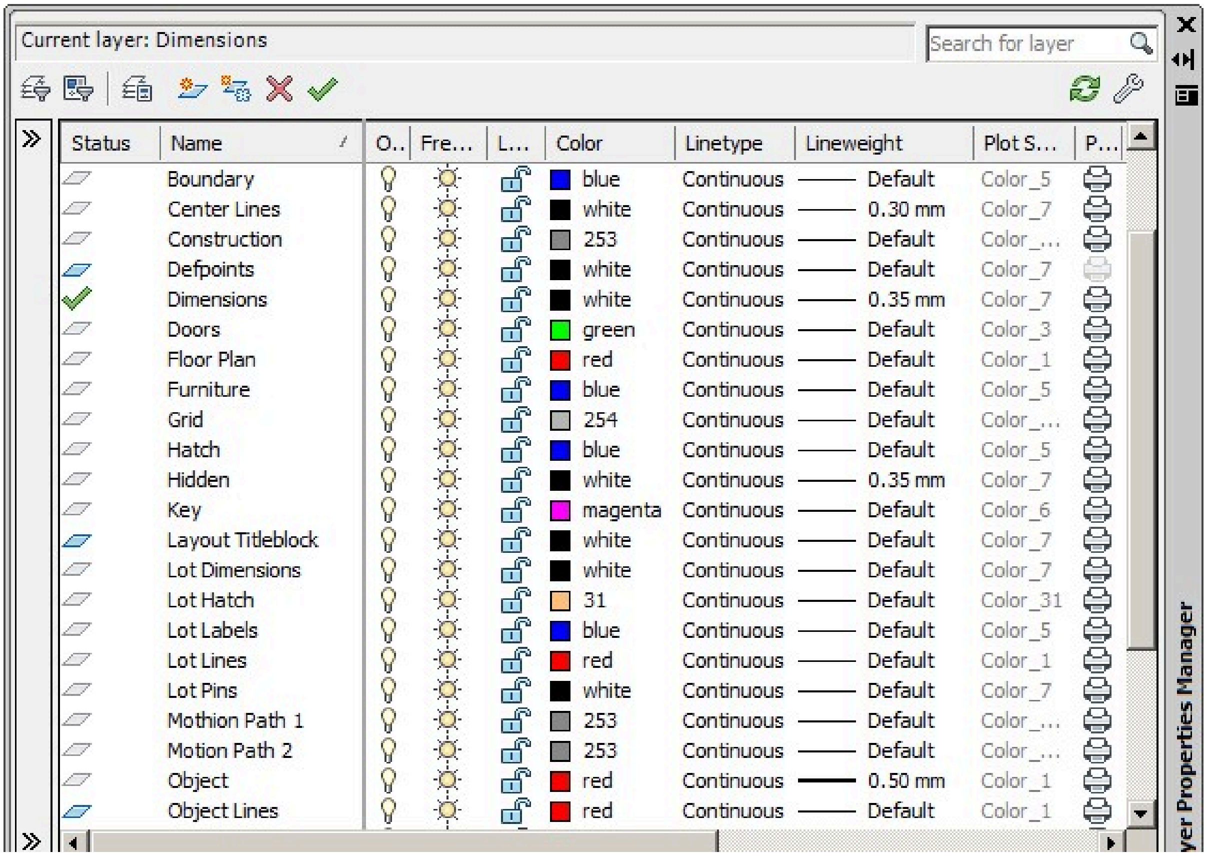
Status	Name	Color	Linetype	Lineweight
	Beds	blue	Continuous	Default
	Blocks	blue	Continuous	Default
	Border Lines	white	Continuous	Default
	Boundary	blue	Continuous	Default
	Center Lines	white	Continuous	0.30 mm
	Construction	253	Continuous	Default
	Defpoints	white	Continuous	Default
✓	Dimensions	white	Continuous	Default
	Doors	green	Continuous	Default
	Floor Plan	red	Continuous	Default

Search for layer
O..
Fre...
L...
Pl

Figure Step 7C

Step 8

Using what you just learned, change the lineweight of the layers: Dimensions, Hidden, and Object to the lineweights shown in the figure. (Figure Step 8)



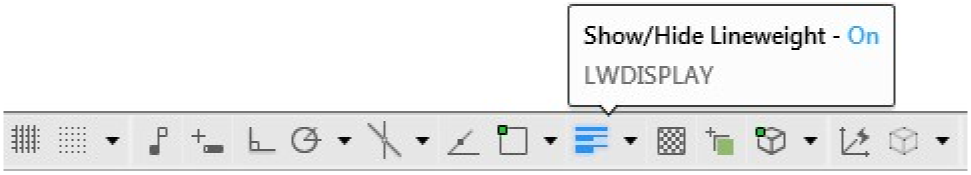
Current layer: Dimensions

Status	Name	O..	Fre...	L...	Color	Linetype	Lineweight	Plot S...	P...
	Boundary				blue	Continuous	Default	Color_5	
	Center Lines				white	Continuous	0.30 mm	Color_7	
	Construction				253	Continuous	Default	Color_...	
	Defpoints				white	Continuous	Default	Color_7	
✓	Dimensions				white	Continuous	0.35 mm	Color_7	
	Doors				green	Continuous	Default	Color_3	
	Floor Plan				red	Continuous	Default	Color_1	
	Furniture				blue	Continuous	Default	Color_5	
	Grid				254	Continuous	Default	Color_...	
	Hatch				blue	Continuous	Default	Color_5	
	Hidden				white	Continuous	0.35 mm	Color_7	
	Key				magenta	Continuous	Default	Color_6	
	Layout Titleblock				white	Continuous	Default	Color_7	
	Lot Dimensions				white	Continuous	Default	Color_7	
	Lot Hatch				31	Continuous	Default	Color_31	
	Lot Labels				blue	Continuous	Default	Color_5	
	Lot Lines				red	Continuous	Default	Color_1	
	Lot Pins				white	Continuous	Default	Color_7	
	Mothion Path 1				253	Continuous	Default	Color_...	
	Motion Path 2				253	Continuous	Default	Color_...	
	Object				red	Continuous	0.50 mm	Color_1	
	Object Lines				red	Continuous	Default	Color_1	

Figure Step 8

Step 9

To enable the display of lineweights, enable the LWT icon on the Status bar. (Figure Step 9)

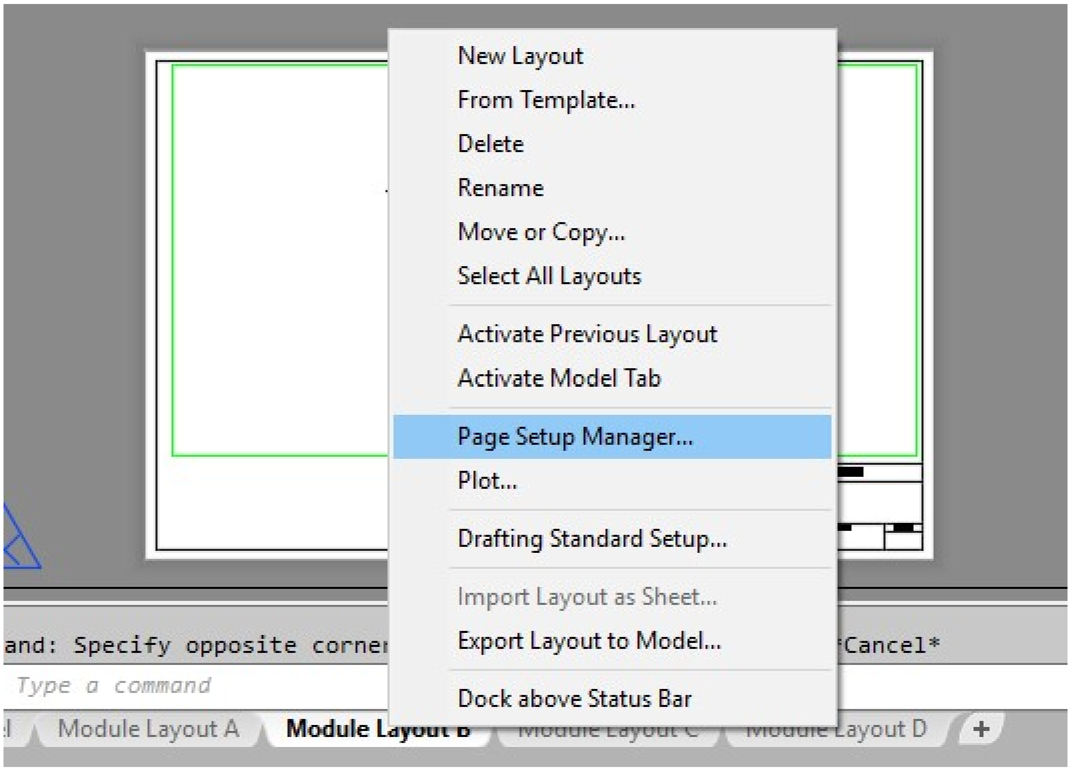


Show/Hide Lineweight - On
LWDISPLAY

Figure Step 9

Step 10

Enable layout: Module Layout B and move the cursor onto the tab for that layout. Right click the mouse. In the Right-click menu, select Page Setup Manager. (Figure Step 10)



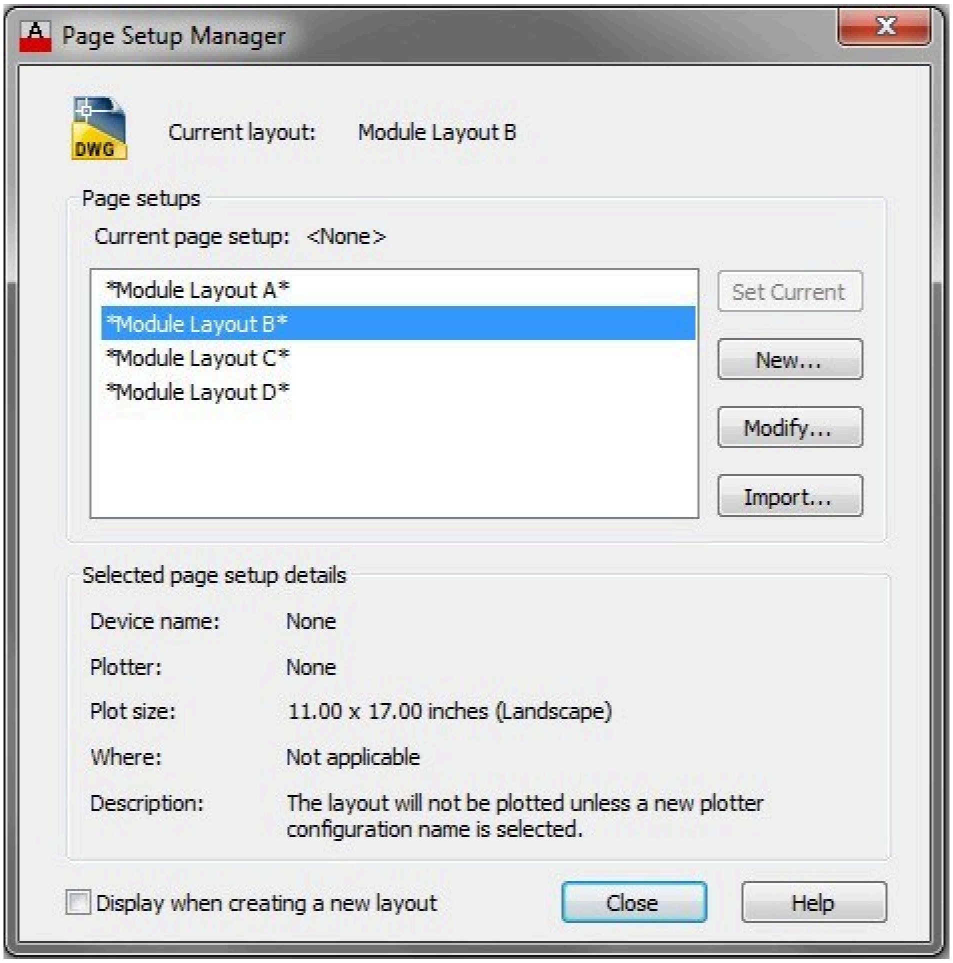
New Layout
From Template...
Delete
Rename
Move or Copy...
Select All Layouts
Activate Previous Layout
Activate Model Tab
Page Setup Manager...
Plot...
Drafting Standard Setup...
Import Layout as Sheet...
Export Layout to Model...
Dock above Status Bar
Module Layout A
Module Layout B
Module Layout C
Module Layout D
+
Type a command

```
and: Specify opposite corner                       *Cancel*
```

Figure Step 10

Step 11

In the Page Setup Manager dialogue box, select: Module Layout B and then click Modify. (Figure Step 11)



Current layout: Module Layout B

Page setups

Current page setup: <None>

Module Layout A
Module Layout B
Module Layout C
Module Layout D

Selected page setup details

Device name: None

Plotter: None

Plot size: 11.00 x 17.00 inches (Landscape)

Where: Not applicable

Description: The layout will not be plotted unless a new plotter configuration name is selected.

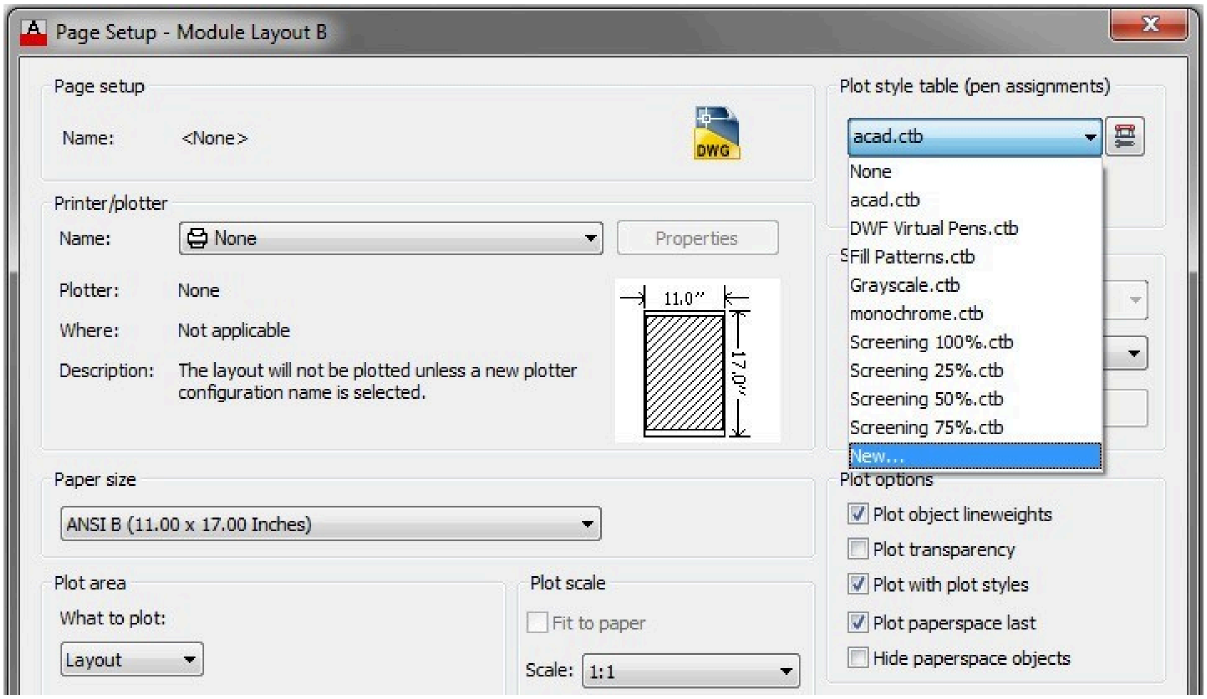
☐ Display when creating a new layout

Page Setup Manager
DWG
Set Current
New...
Modify...
Import...
Close
Help

Figure Step 11

Step 12

In the Pager Setup – Module Layout B dialogue box, pull down the Plot style table list and click New. (Figure Step 12)



Page Setup - Module Layout B

Page setup
Name: <None>

Printer/plotter
Name: None
Plotter: None
Where: Not applicable
Description: The layout will not be plotted unless a new plotter configuration name is selected.

Paper size
ANSI B (11.00 x 17.00 Inches)

Plot area
What to plot: Layout

Plot scale
☐ Fit to paper
Scale: 1:1

Plot style table (pen assignments)
acad.ctb

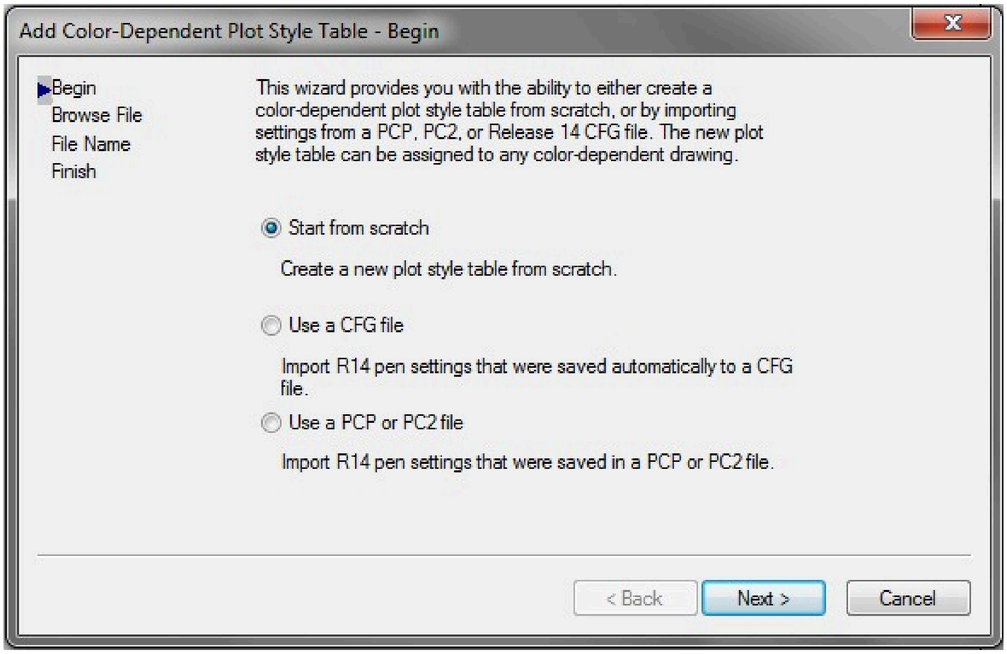
Plot options
☑ Plot object lineweights
☐ Plot transparency
☑ Plot with plot styles
☑ Plot paperspace last
☐ Hide paperspace objects

DWG
Properties
11.0″
17.0″
None
acad.ctb
DWF Virtual Pens.ctb
Fill Patterns.ctb
Grayscale.ctb
monochrome.ctb
Screening 100%.ctb
Screening 25%.ctb
Screening 50%.ctb
Screening 75%.ctb
New...

Figure Step 12

Step 13

In the Add Color Dependent Plot Style Table dialogue box, enable Start from scratch and click Next. (Figure Step 13)


Add Color-Dependent Plot Style Table - Begin
Begin
Browse File
File Name
Finish

This wizard provides you with the ability to either create a color-dependent plot style table from scratch, or by importing settings from a PCP, PC2, or Release 14 CFG file. The new plot style table can be assigned to any color-dependent drawing.

● Start from scratch

Create a new plot style table from scratch.

○ Use a CFG file

Import R14 pen settings that were saved automatically to a CFG file.

○ Use a PCP or PC2 file

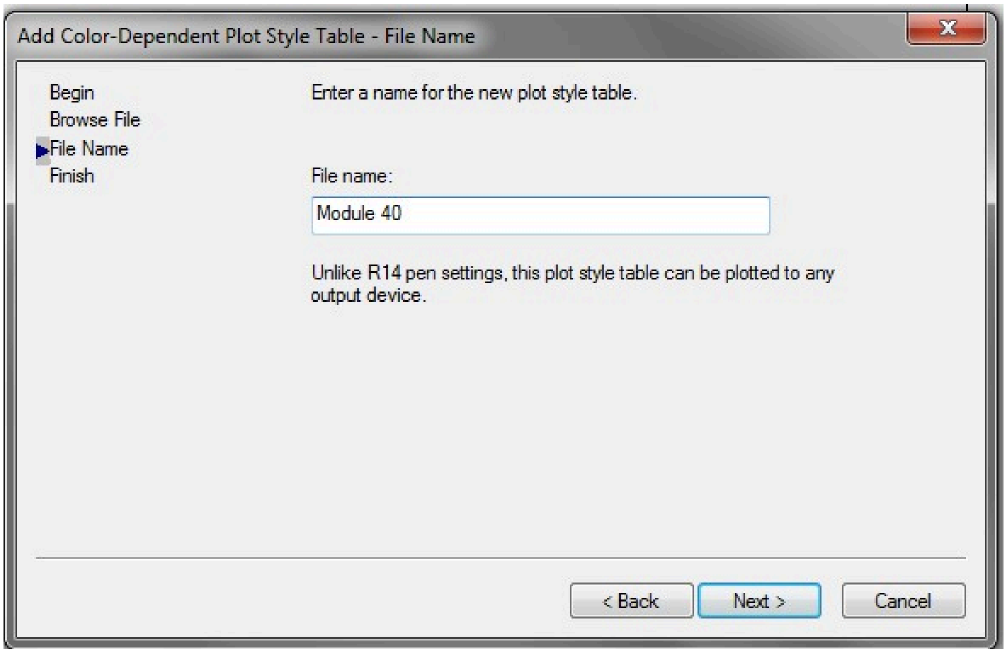
Import R14 pen settings that were saved in a PCP or PC2 file.

< Back
Next >
Cancel

Figure Step 13

Step 14

Enter the name: Module 40 and click Next. (Figure Step 14)


Add Color-Dependent Plot Style Table - File Name
Begin
Browse File
File Name
Finish

Enter a name for the new plot style table.

File name: Module 40

Unlike R14 pen settings, this plot style table can be plotted to any output device.

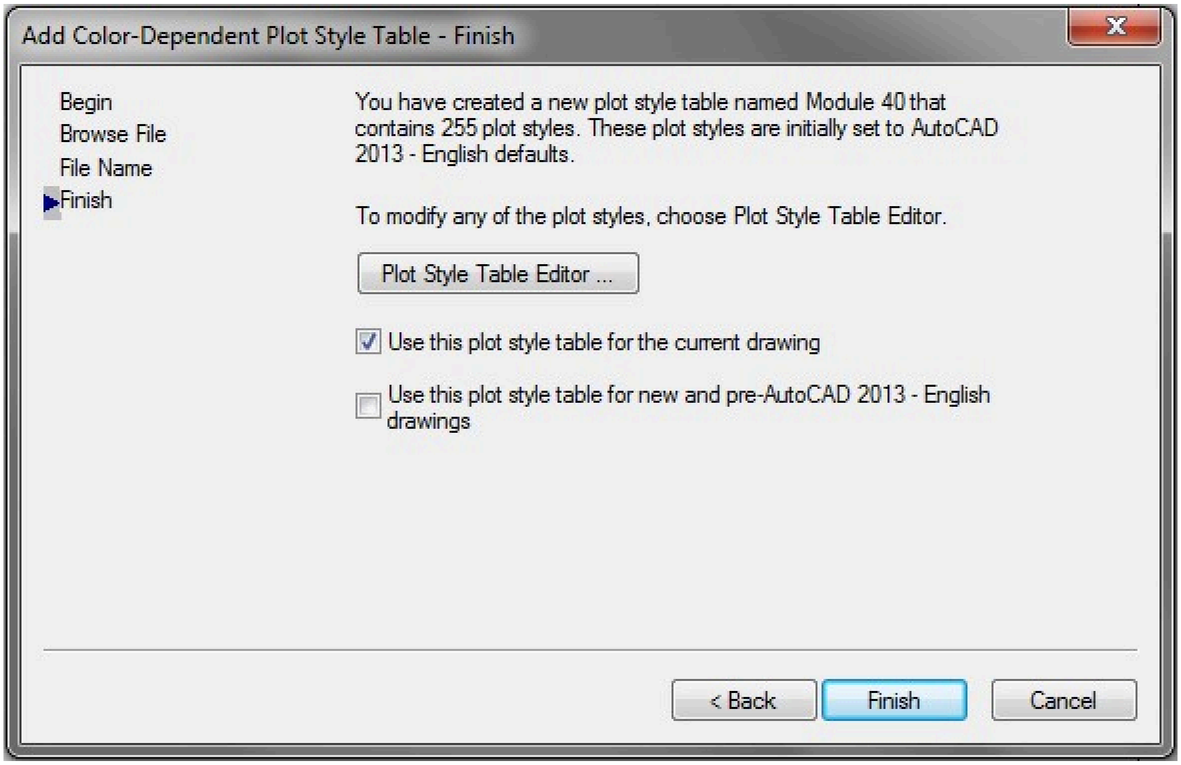
< Back
Next >
Cancel

Figure Step 14

AUTHOR'S COMMENTS: After configuring and saving the plot style, it can be used to plot other drawings.

Step 15

Click the Plot Style Table Editor box. (Figure Step 15)



Add Color-Dependent Plot Style Table - Finish

Begin
Browse File
File Name
Finish

You have created a new plot style table named Module 40 that contains 255 plot styles. These plot styles are initially set to AutoCAD 2013 - English defaults.

To modify any of the plot styles, choose Plot Style Table Editor.

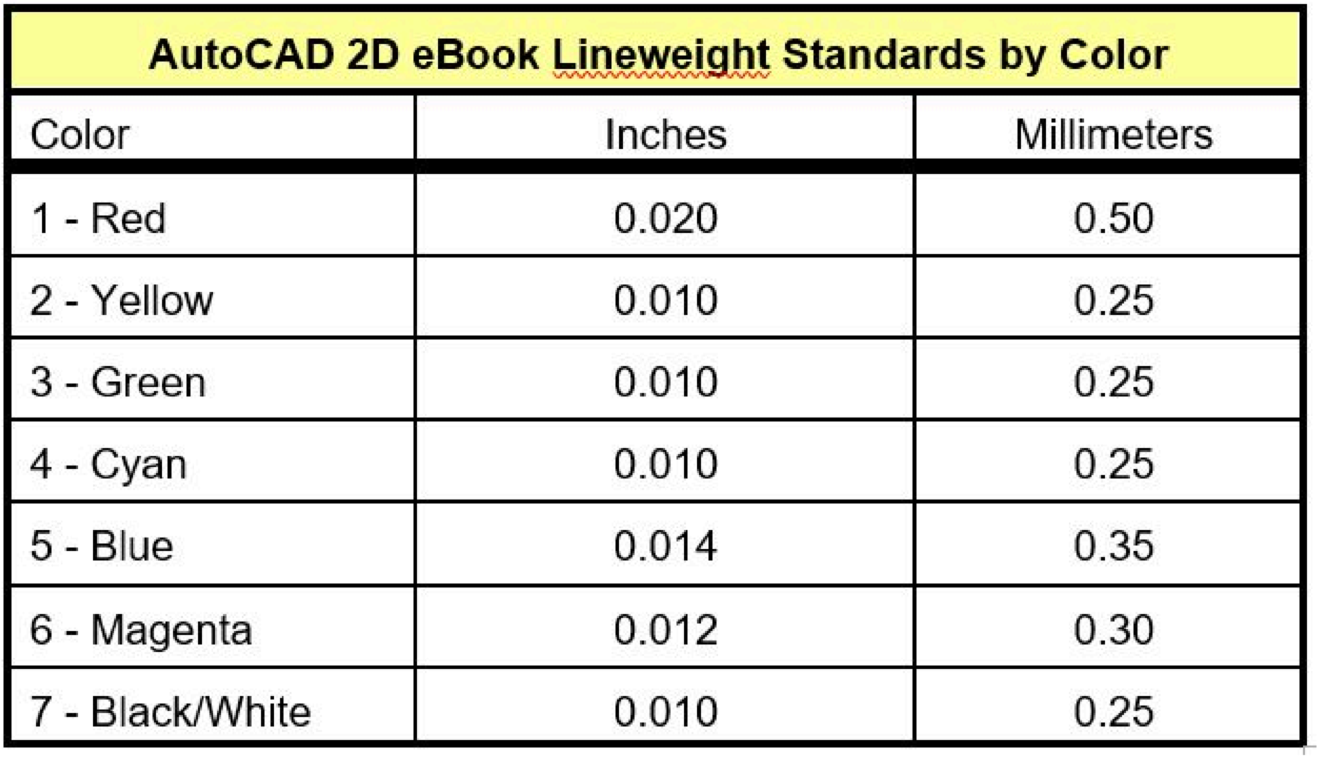
Plot Style Table Editor ...

☑ Use this plot style table for the current drawing

☐ Use this plot style table for new and pre-AutoCAD 2013 - English drawings

< Back
Finish
Cancel

Figure Step 15

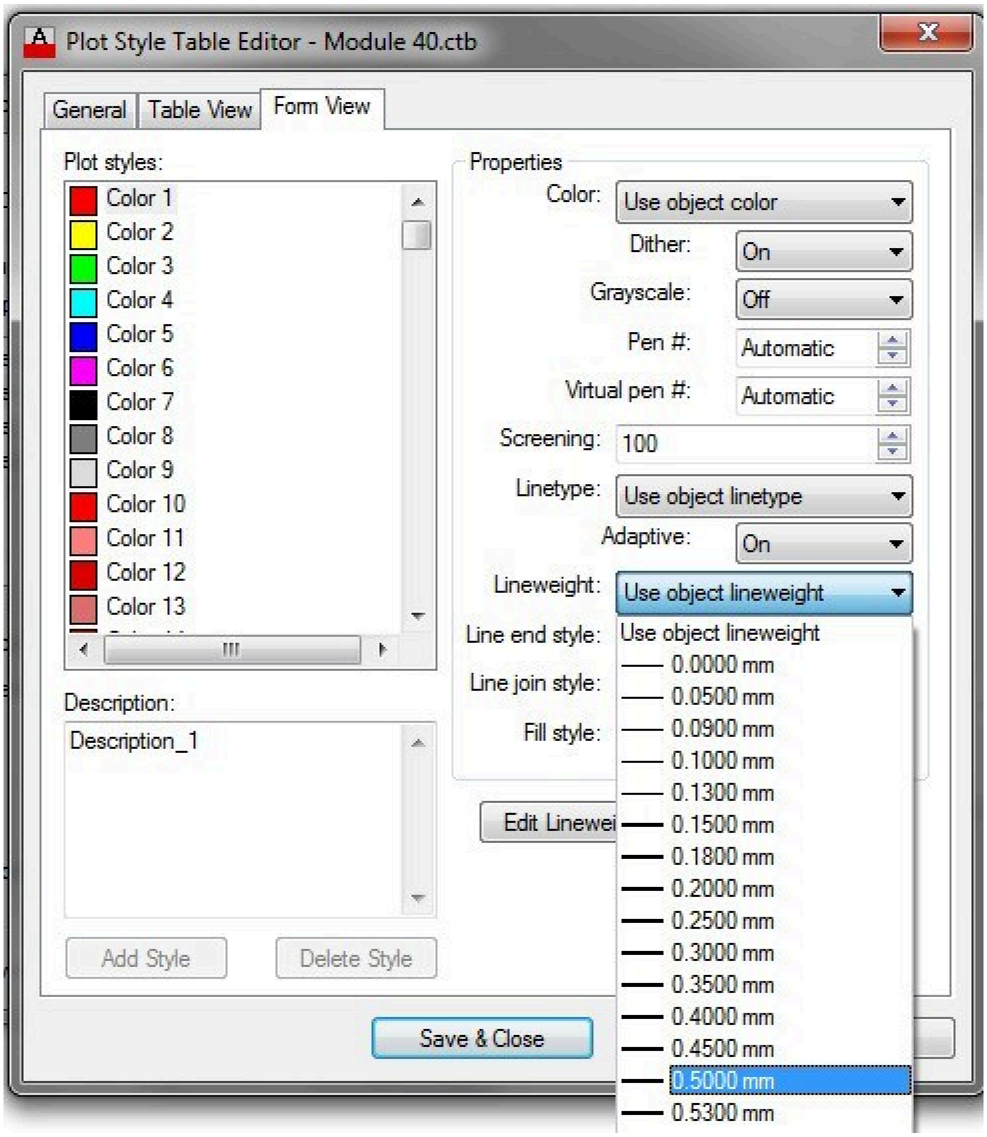


AutoCAD 2D eBook Lineweight Standards by Color		
Color	Inches	Millimeters
1 - Red	0.020	0.50
2 - Yellow	0.010	0.25
3 - Green	0.010	0.25
4 - Cyan	0.010	0.25
5 - Blue	0.014	0.35
6 - Magenta	0.012	0.30
7 - Black/White	0.010	0.25

Figure 40-2
AutoCAD 2D book Lineweight Standards by Color Chart

Step 16

In the Plot styles list, select Color 1 (Red) and then pull down the Lineweight list, select 0.5000 mm. (Figure Step 16)



Plot Style Table Editor - Module 40.ctb

Color: Use object color
Dither: On
Grayscale: Off
Pen #: Automatic
Virtual pen #: Automatic
Screening: 100
Linetype: Use object linetype
Adaptive: On
Lineweight: Use object lineweight
Description: Description_1

General
Table View
Form View
Plot styles:
Color 1
Color 2
Color 3
Color 4
Color 5
Color 6
Color 7
Color 8
Color 9
Color 10
Color 11
Color 12
Color 13
Description:
Add Style
Delete Style
Properties
Line end style:
Line join style:
Fill style:
Edit Linewei
Save & Close
Use object lineweight
0.0000 mm
0.0500 mm
0.0900 mm
0.1000 mm
0.1300 mm
0.1500 mm
0.1800 mm
0.2000 mm
0.2500 mm
0.3000 mm
0.3500 mm
0.4000 mm
0.4500 mm
0.5000 mm
0.5300 mm

Figure Step 16

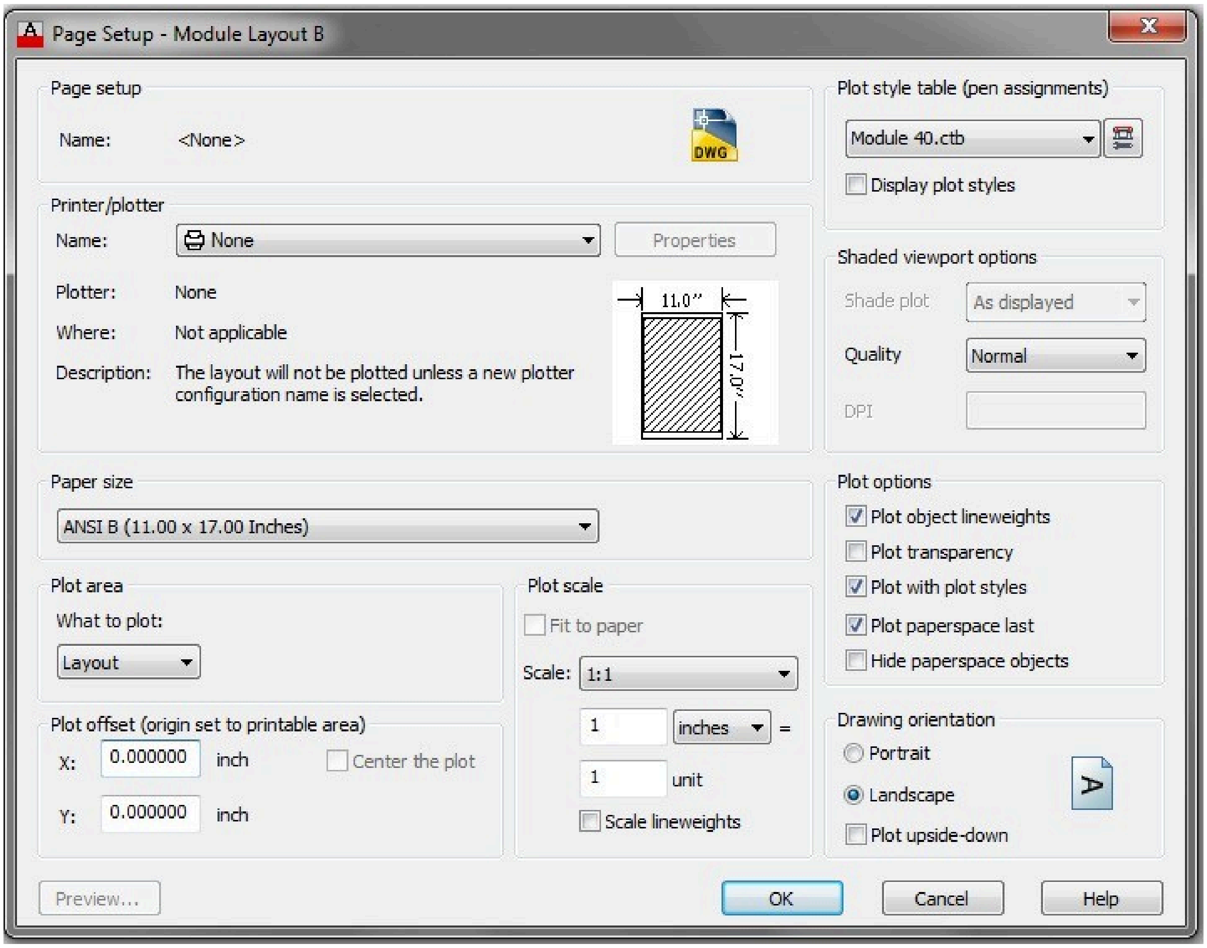
AUTHOR'S COMMENTS: All drawing objects that display red will plot at the line width of 0.5000 mm.

Step 17

Using the chart shown in Figure 40-2, set the six remaining colors.

Step 18

Click Save & Close. (Figure Step 18)



Page Setup - Module Layout B

Page setup

Name: <None>

Printer/plotter

Name: None

Plotter: None

Where: Not applicable

Description: The layout will not be plotted unless a new plotter configuration name is selected.

Paper size: ANSI B (11.00 x 17.00 Inches)

Plot area

What to plot: Layout

Plot offset (origin set to printable area)

X: 0.000000 inch

Y: 0.000000 inch

☐ Center the plot

Plot scale

☐ Fit to paper

Scale: 1:1

1 inches =

1 unit

☐ Scale lineweights

Plot style table (pen assignments): Module 40.ctb

☐ Display plot styles

Shaded viewport options

Shade plot: As displayed

Quality: Normal

DPI:

Plot options

☑ Plot object lineweights

☐ Plot transparency

☑ Plot with plot styles

☑ Plot paperspace last

☐ Hide paperspace objects

Drawing orientation

○ Portrait

● Landscape

☐ Plot upside-down

Figure Step 18

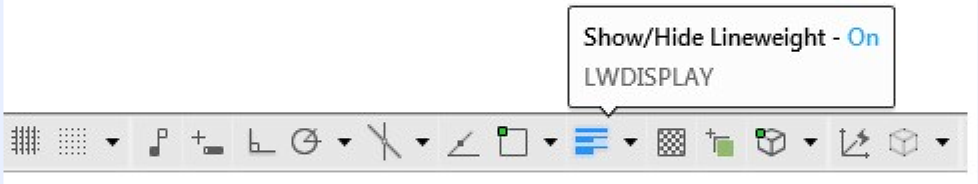
Step 19

Click OK to close the box.

Step 20

Save and close the drawing.

MUST KNOW:The display of the lineweights assigned to objects in the drawing is enabled or disabled by toggling the LT con in the Status bar.

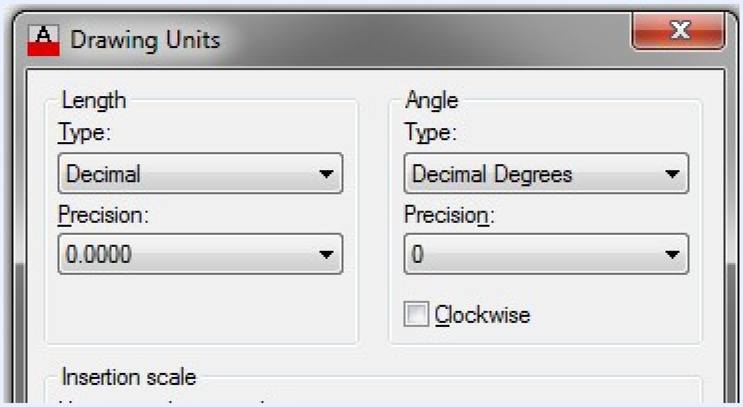


Show/Hide Lineweight - On

LWDISPLAY

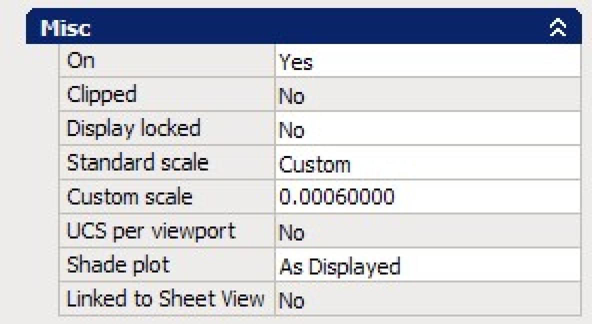
USER TIP: To copy an object(s) from Model space to Paper space or vise versa, use the principles of the copy and paste method that was covered in Module 20. Remember that you have to have Model space as the working space to select model objects and then change the working space to Paper before pasting theobject(s). You can also copy and paste objects from Paper space to Model space. The properties of the objects will be converted into the type of space they are being pasted into.

:**MUST KNOW:** When setting the custom scale property of a viewport, it is important that the length units precision is set to 8 decimal places first using the UNITS command. For example, to set the custom scale of 1:1500, divide 1 by 1500 =0.00066666. To see what could happen if the wrong precision setting is used see below



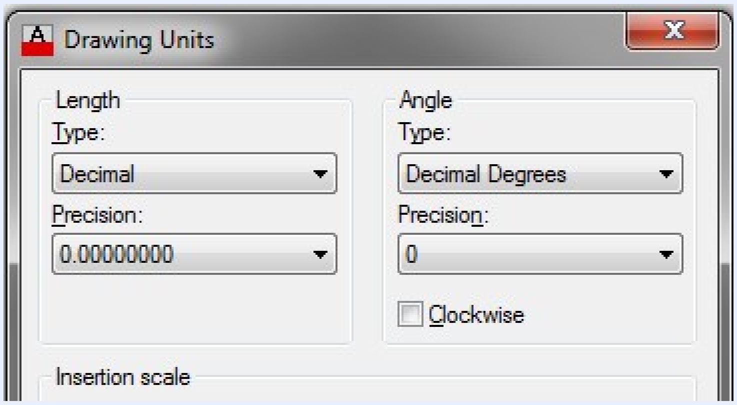
Length
Type: Decimal
Precision: 0.0000

Angle
Type: Decimal Degrees
Precision: 0
☐ Clockwise



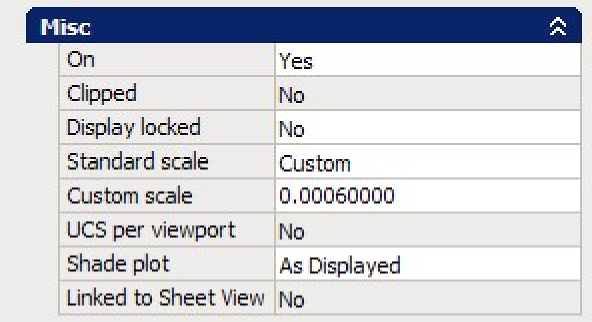
Misc	
On	Yes
Clipped	No
Display locked	No
Standard scale	Custom
Custom scale	0.00060000
UCS per viewport	No
Shade plot	As Displayed
Linked to Sheet View	No

The Same Custom Scale Displayed with 8 Decimal Places of Precision



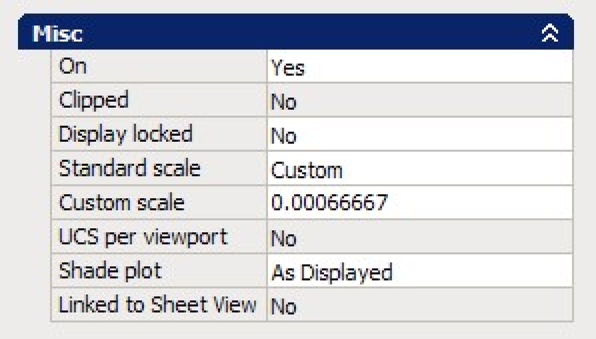
Length
Type: Decimal
Precision: 0.00000000

Angle
Type: Decimal Degrees
Precision: 0
☐ Clockwise



Misc	
On	Yes
Clipped	No
Display locked	No
Standard scale	Custom
Custom scale	0.00060000
UCS per viewport	No
Shade plot	As Displayed
Linked to Sheet View	No

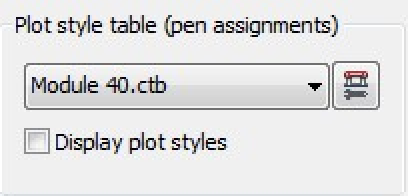
The Same Custom Scale Displayed with 8 Decimal Places of Precision



Misc	
On	Yes
Clipped	No
Display locked	No
Standard scale	Custom
Custom scale	0.00066667
UCS per viewport	No
Shade plot	As Displayed
Linked to Sheet View	No

Correct Custom Scale

MUST KNOW: Before plotting any layout, the user must choose between plotting it with object lineweights or with a plot style color table. This can be set in the Plot options area in the Page Setup dialogue box. When setting plot by plot style, ensure that the plot style table is set to the desired table.

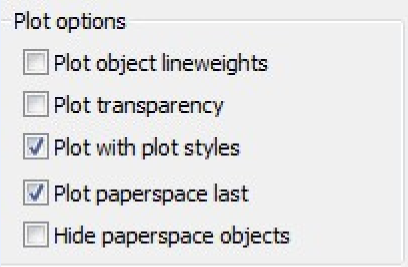


Plot style table (pen assignments)

Module 40.ctb

☐ Display plot styles

Setting the Plot Style



Plot options

☐ Plot object lineweights

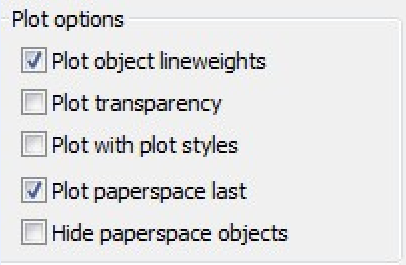
☐ Plot transparency

☑ Plot with plot styles

☑ Plot paperspace last

☐ Hide paperspace objects

Setting to Plot with the Assigned Plot Style



Plot options

☑ Plot object lineweights

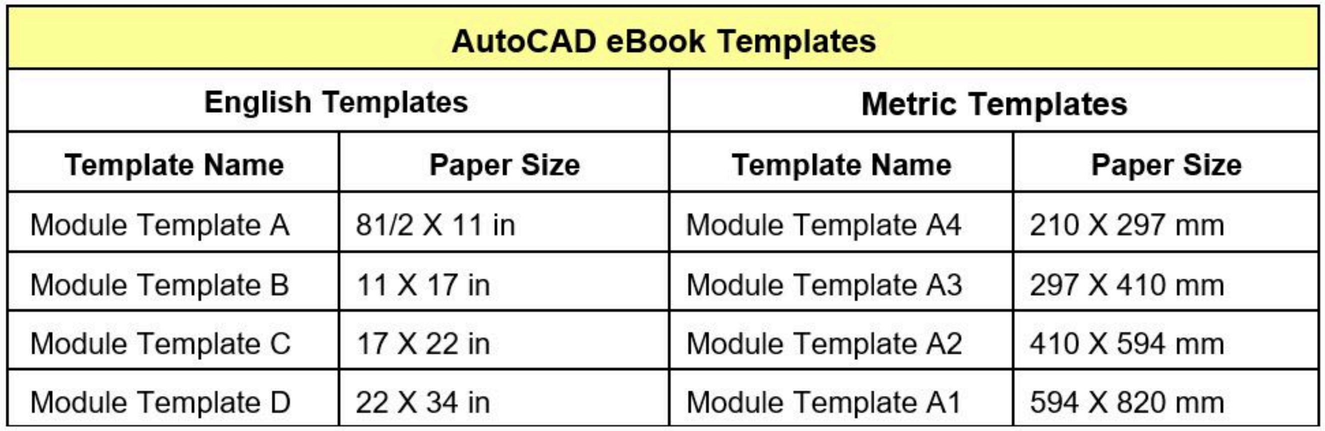
☐ Plot transparency

☐ Plot with plot styles

☑ Plot paperspace last

☐ Hide paperspace objects

Setting to Plot with Lineweight by Object



AutoCAD eBook Templates			
English Templates		**Metric Templates**	
Template Name	**Paper Size**	**Template Name**	**Paper Size**
Module Template A	81/2 X 11 in	Module Template A4	210 X 297 mm
Module Template B	11 X 17 in	Module Template A3	297 X 410 mm
Module Template C	17 X 22 in	Module Template A2	410 X 594 mm
Module Template D	22 X 34 in	Module Template A1	594 X 820 mm

Figure 40-3
Standard Drawing Paper Sizes

Geometry Lesson: Setting Custom Viewport Scales

When setting the scale for a viewport and the required scale not listed in the Standard scale property list, you must calculate a custom scale. The custom scale is simple to calculate. It is the scale factor or a ratio of the scale related to 1.

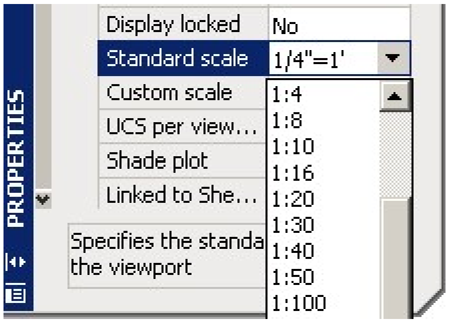
For example, to calculate the scale of:

1/4″ -= 1′-0″

1/4″=1′-0″ or 1/4″=12″ or 1=48 or 1:48

To get the scale factor in decimal format, divide: 1 by 48 = 0.0208

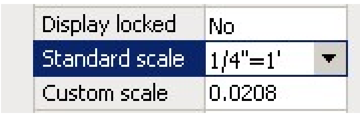
The custom scale would be set at 0.0208



Display locked No
Standard scale 1/4"=1'
Custom scale
UCS per view...
Shade plot
Linked to She...
Specifies the standa the viewport
1:4
1:8
1:10
1:16
1:20
1:30
1:40
1:50
1:100

Here are some additional examples:

3/4″ = 1′-0″ or 3/4″=12″ or 48 divided by 3 = 1:16 = 0.0625



Display locked No
Standard scale 1/4"=1'
Custom scale 0.0208

3/4″=1″	1 divided by 0.75	= 1.33333333
1:1.5	1 divided by 0.75	= 0.6666666
1:5	1 divided by 5	= 0.2000000
1:75	1 divided by 75	= 0.013333333
1:500	1 divided by 500	= 0.00200000
1:1500	1 divided by 1500	= 0.00066666
15:1	15 divided by 1	= 15

Key Principles

Key Principles in Module 40

1. There are two methods of assigning lineweight to an object. They are lineweight by color and lineweight by object.
2. The display of the lineweights assigned to objects in the drawing is enabled or disabled by toggling the LWT icon on the Status bar.
3. Before plotting a drawing, choose between plotting it with object lineweights or with a plot style color table.
4. When setting the custom scale for a viewport, ensure that the precision is set to 8 decimal places in the Units dialogue box.

Module 41: Competency Test 8 Open Book

Learning Outcomes

When you have completed this module, you will be able to:

1. Within a two hour time limit, complete a written exam and the lab exercise without the aid of a key.

The AutoCAD 2D book was written with competency based modules. What that means is that you have not completed each module until you have mastered it. The Competency Test module contains multiple choice questions and a comprehensive lab exercise to test your mastery of the set of modules that you completed. There are no answers or keys supplied in a Competency Test module since it is meant to be checked by your instructor. If there are any parts of this module that you have trouble completing, you should go back and reread the module or modules containing the information that you are having trouble with. If necessary, redo as many lab exercises required until you fully understand the material.

If you are Completing this book:

- Without the Aid of an Instructor, complete the written test and the lab exercise.
- In a Classroom with an Instructor, the instructor will give instructions on what to do after this module has been completed.

Multiple Choice Questions

Select the BEST answer.

1. What snap mode is used to snap to a point located on an object that is the closest to the current location of the cursor?
    A. Node
    B. Point
    C. Nearest
    D. Closet
    E. On
2. What AutoCAD window is used to copy blocks from one drawing to another?
    A. Layer Property Manager

   - B. Properties
   - C. DesignCenter
   - D. External Reference
   - E. There is no window to do this.

3. When associative dimensions are inserted, they insert points at the dimension’s location. What are these points called?
   - A. Definition Points
   - B. Dimension Points
   - C. Extension Points
   - D. Insert Points
   - E. Assoc Points
4. On what layer are the points in question 3 inserted on?
   - A. The current layer
   - B. Layer Dimpoints
   - C. Layer 0
   - D. Layer Defpoints
   - E. Layer Dimension
5. Which one of the following best describes one difference between an associative and an exploded dimension?
   - A. Exploded dimensions are inserted as blocks.
   - B. Associative dimensions are inserted as xrefs.
   - C. Exploded dimensions are inserted as xrefs.
   - D. Associative dimensions are inserted as blocks.
   - E. Associative dimensions are inserted as individual objects.
6. Which one of the following cannot be dragged and dropped from one drawing to another using DesignCenter?
   - A. text style
   - B. dimensioning style
   - C. plot styles
   - D. layouts
   - E. layers
7. If the scale of a viewport is set 1:800, what number is entered for the custom scale?
   - A. 800
   - B. 0.008

   C. 0.0125
   D. 0.00125
   E. 0.125

8. What characters are used in the Text Override property to represent the actual dimension measured by AutoCAD?
   A. < >
   B. ” “
   C. { }
   D. [ ]
   E. # #

9. Which one of the following statements is true?
   A. If a lineweight is not assigned to a layer, the layer is automatically set to zero lineweight.
   B. Color is used to assign lineweights to a drawing when plotted.
   C. Lineweights cannot be displayed in a drawing.
   D. Only layers can be used to assign lineweights in a drawing.
   E. Lineweights can only be assigned with layers.

10. Which one of the following can be used to copy an object from model space to paper space?
   A. Copy and Paste
   B. COPY command
   C. COPYMODEL command
   D. MVIEW command
   E. This cannot be done.

Lab Exercise 41-1 OPEN BOOK

Drawing Name	**Template**	**Units**
AutoCAD 2D Lab 41-1	N/A	Millimeters

Step 1

Open drawing: AutoCAD 2D Lab 34-1. (Figure Step 1)

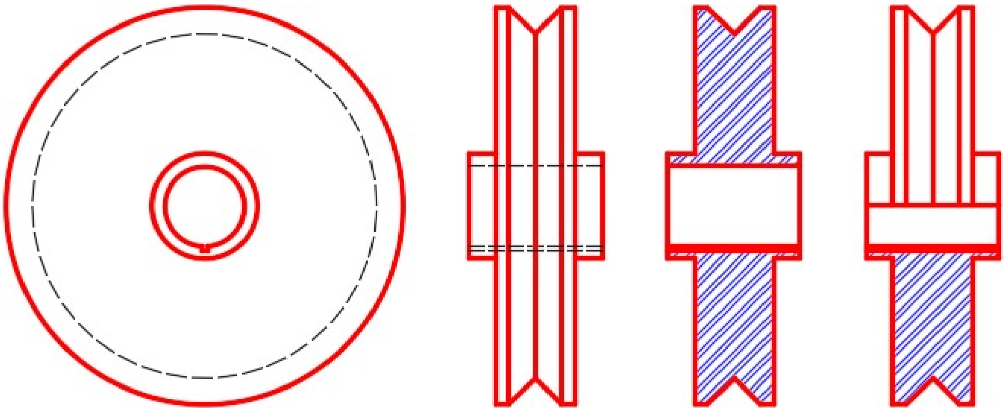


Figure Step 1

Step 2

Using the SAVEAS command, save the drawing with the name: AutoCAD 2D Lab 41-1.

Step 3

In the layout: Module Layout A3, on layer: Viewport, create two viewports as shown in the figure. Set the scale as follows and lock their display. (Figure Step 3)

Left Viewport Scale: 1:1.5

Right Viewport Scale: 2.5:1

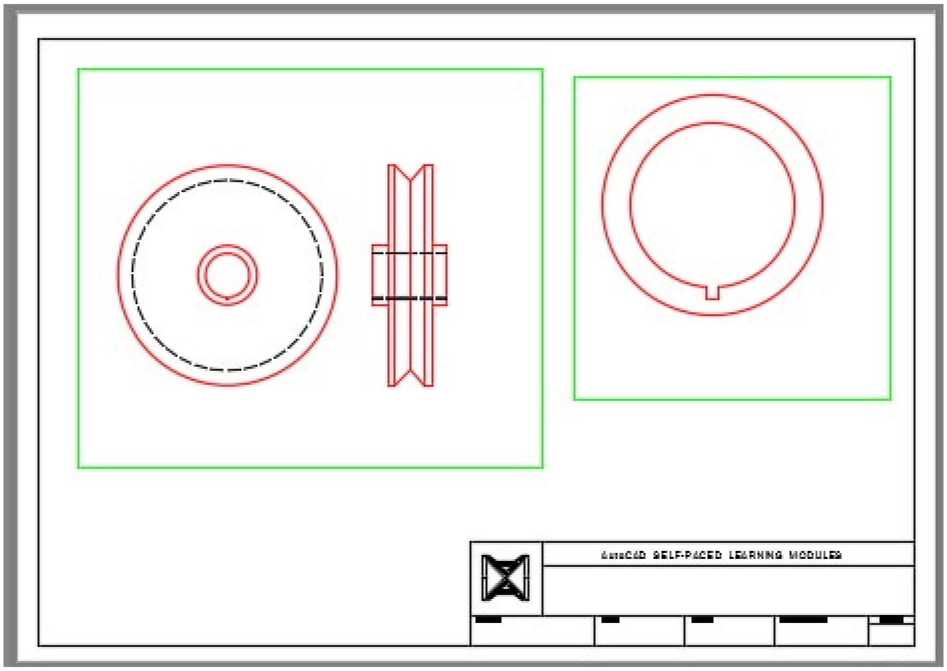
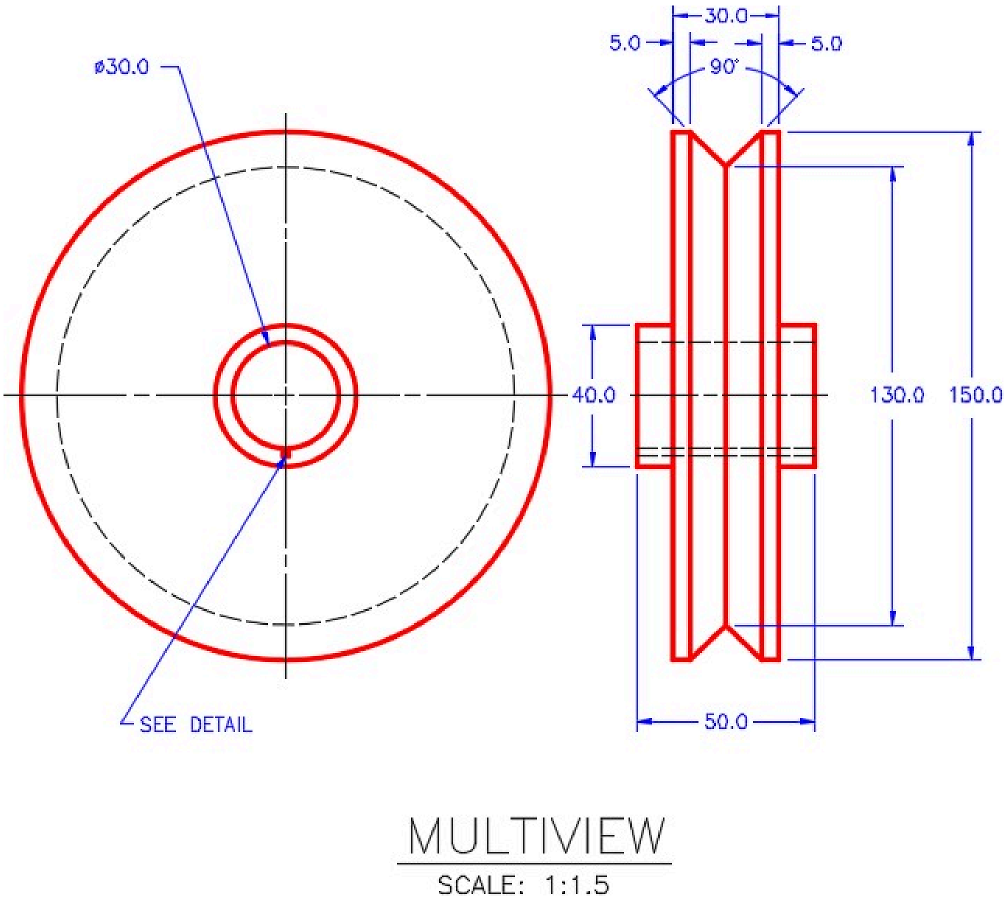

AutoCAD SELF-PACED LEARNING MODULES

Figure Step 3

Step 4

Using Figure Step 4:

- A In Paper space, on layer: Center Lines, add the center
- B In Paper space, on layer: Dimensions, add the dimensions in each
- C In Paper space, on layer: Text, add the title and scale of the views. Use your own text style and text height. (Figure Step 4)


ø30.0
SEE DETAIL
30.0
5.0
5.0
90°
40.0
130.0
150.0
50.0
MULTIVIEW
SCALE: 1:1.5

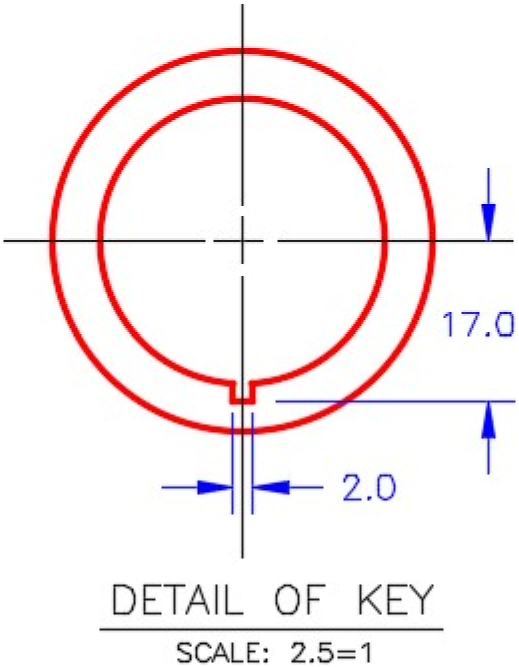

17.0
2.0
DETAIL OF KEY
SCALE: 2.5=1

Figure Step 4

AUTHOR’S COMMENTS: All dimensions and text MUST be in Paper space. Ensure that you snap to model space objects when inserting the dimensions.

Step 5

Turn Layer: Viewport off and fill in the titleblock. (Figure Step 5)

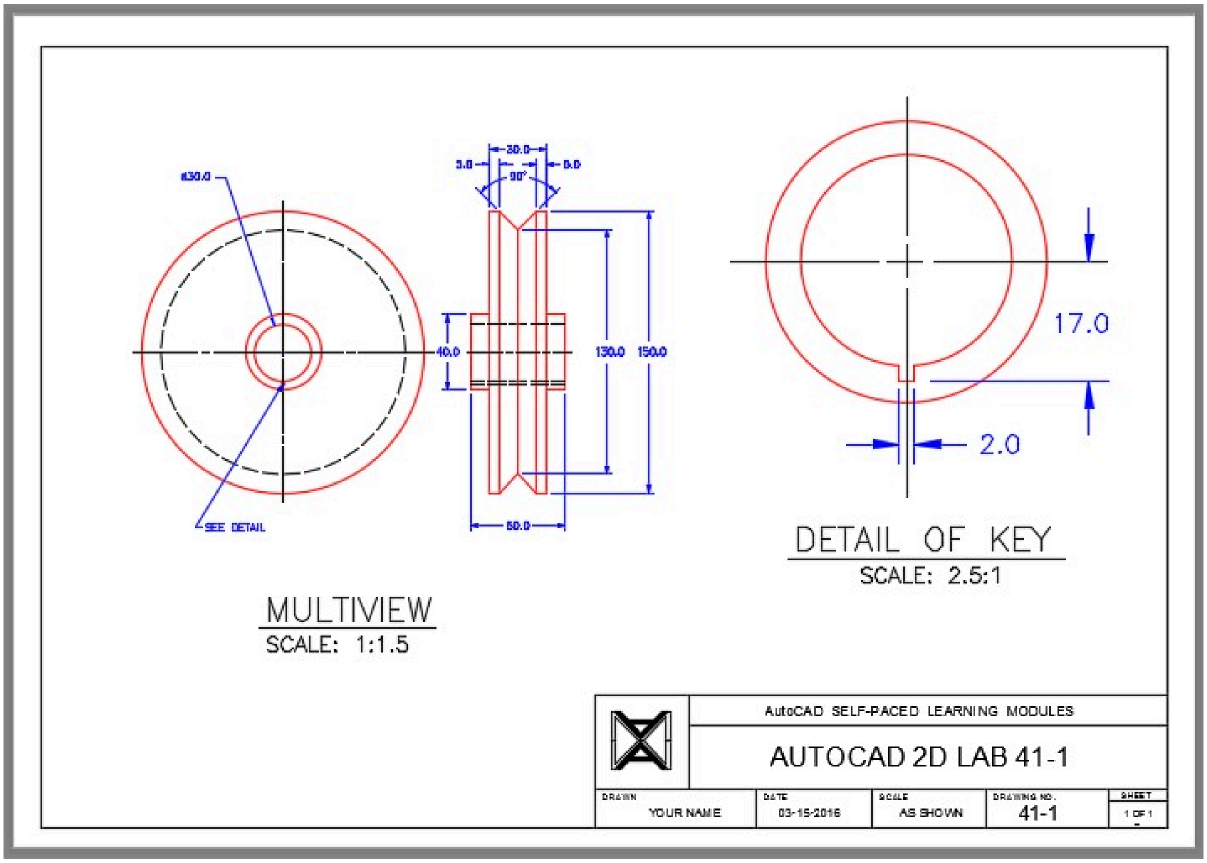

ø130.0
SEE DETAIL
30.0
3.0
0.0
90°
40.0
130.0
150.0
60.0
17.0
2.0
DETAIL OF KEY
SCALE: 2.5:1
MULTIVIEW
SCALE: 1:1.5
AutoCAD SELF-PACED LEARNING MODULES
AUTOCAD 2D LAB 41-1
DRAWN
YOUR NAME
DATE
03-15-2016
SCALE
AS SHOWN
DRAWING NO.
41-1
SHEET
1 OF 1

Figure Step 5

Step 6

Change to Model space. Your figure should match the figure. (Figure Step 6)

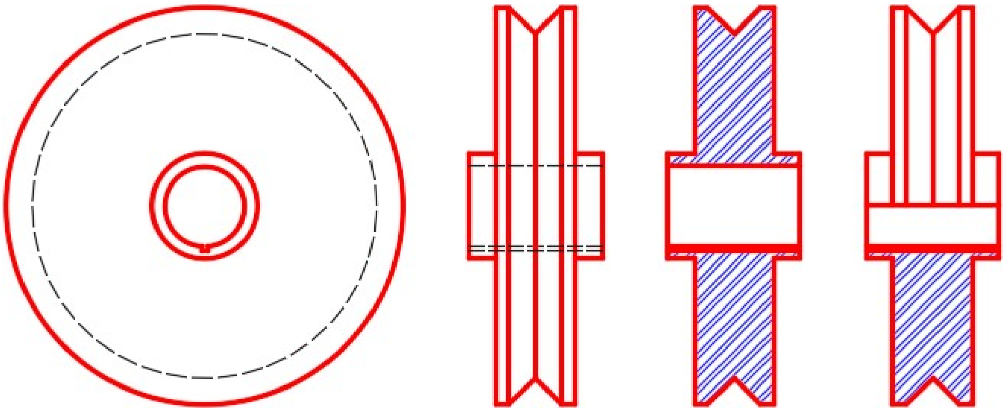


Figure Step 6

Step 7

Save and close the drawing.

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AUTHOR’S COMMENTS: If using a digital copy, you can also search for specific terms. See For Students: How to Access and Use this Textbook for instructions on how to search digital versions of this book.

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Versioning History

This page provides a record of edits and changes made to this book since its initial publication. Whenever edits or updates are made in the text, we provide a record and description of those changes here. If the change is minor, the version number increases by 0.01. If the edits involve substantial updates, the version number increases to the next full number.

The files posted by this book always reflect the most recent version. If you find an error in this book, please fill out the Report an Error form.

Version	**Date**	**Change**	**Details**
1.00	November 3, 2021	Book published.	