Introduction to Drafting and AutoCAD 3D

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Wally Baumback

BCCAMPUS
VICTORIA, B.C.

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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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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.
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    - 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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AutoCAD 3D Self-paced course by Wally Baumback was funded by BCcampus Open Education.

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This book was produced using the following styles: AutoCAD 3D Style Sheet [Word file]

Introduction

Learning Outcomes

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

1. List the required software required to complete the AutoCAD 3D book.
2. Describe the book's philosophy and explain how to use the book to learn how to apply the AutoCAD commands and features to draw 3D drawings.

Software Requirements

AutoCAD Software Required

The AutoCAD 3D book was written to be used with the Windows version of AutoCAD 2015, 2016, 2017, 2018, and 2019. You must have AutoCAD 2015, 2016, 2017, 2018, or 2019 software running on your computer to complete the lab exercises in this book.

Files Required for the AutoCAD 3D book

To complete the lab exercises in this book, you must load the two template files that accompany this course onto your hard disk drive. "Configuring Your AutoCAD Software" will instruct you how to configure them and Module 2 will teach you how to use them.

About the Book

Philosophy of the AutoCAD Books

The AutoCAD books were written as a tool to guide and teach you to master AutoCAD. No two students learn at the same pace, therefore the books were written with competency-based modules. The competency-based modules are bite-size pieces that allows 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

The AutoCAD books were written in five parts, 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 the Windows operating system and are proficient at drawing 2D drawing objects using AutoCAD and are familiar with all commands and features used in the AutoCAD 2D book.

The AutoCAD 3D book

The modules in the AutoCAD 3D book were written in a very logical step-by-step order. The following information will help you get the most from this book.

The book Modules

This modules in the book were written as bite-size pieces in a precise order and contain the information that is required to learn AutoCAD 3D. Using AutoCAD is a never ending learning process and you will continue learning long after completing this book.

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.

Book Structure and Components

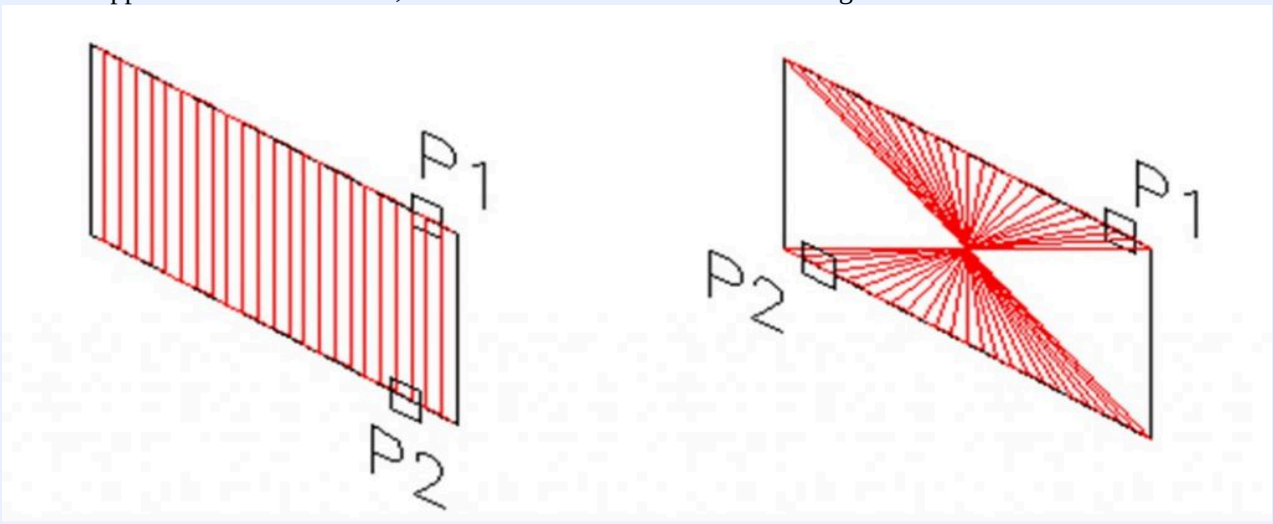
Each module may contain Must Know's, User Tips, Workalongs, Lessons, Key Principles and Lab Exercises.

Must Know's

Most modules contain user *Must Know's*. 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. i.e.

(Do not attempt to understand the following Must Know, it is here for an example only.)

MUST KNOW: When using the RULESURF command, ensure you select the objects closest to matching ends. If opposite ends are selected, the mesh will twist as shown in the figure below.



User Tips

User Tips are in the modules to help you to complete drawings faster and more efficiently. 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 doing your labs exercises. Don't memorize them as you will not be tested on them. You can re-read them anytime you wish. They are there to help you work smarter, not harder. For example:

(Do not attempt to understand the following User Tip, it is here for an example only.)

USER TIP: Before you start drawing any model, you must first decide which view is the best to start with. Usually it is the view with the most difficult contour. The more practice you get modeling the easier it will be to select the best view to start your model.

Workalongs

A Workalong is an exercise you complete by working along with the modules, tutorial style, to complete the practice session that follows. Completing the workalongs in this manner will help you understand how the commands you are being taught in that module work by actually using them in AutoCAD. This will prepare you so that when you are completing the drawings in the lab exercises at the end of each module, you should be able to complete them without any assistance. For example:

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

WORK ALONG: Polar Arraying in 3D

Step 1

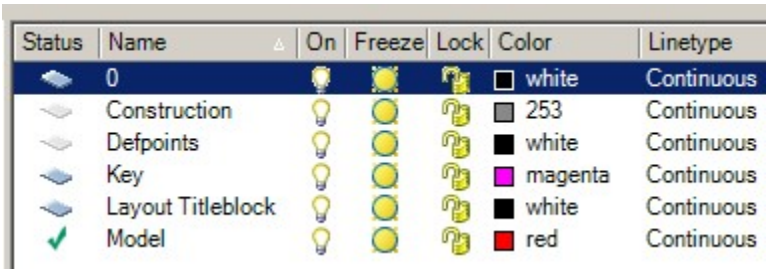
Using the NEW command, start a new drawing using the template 3D Layout English.

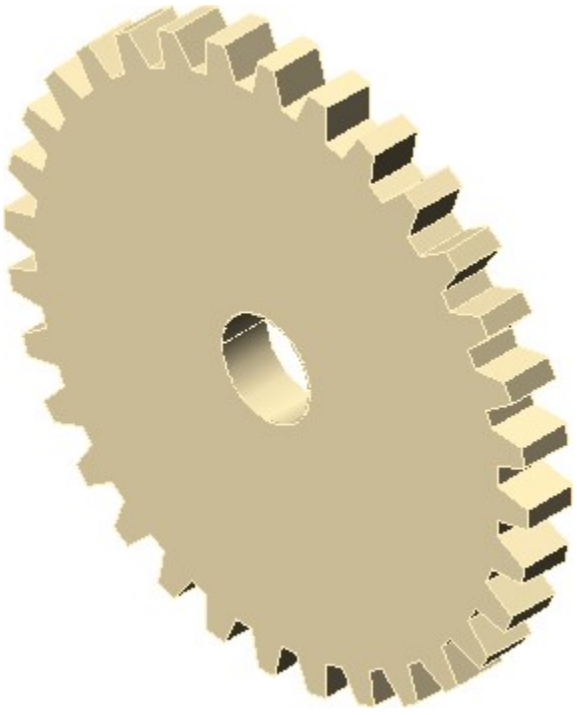
Step 2

Save and name the drawing AutoCAD 3D Workalong 09-2.

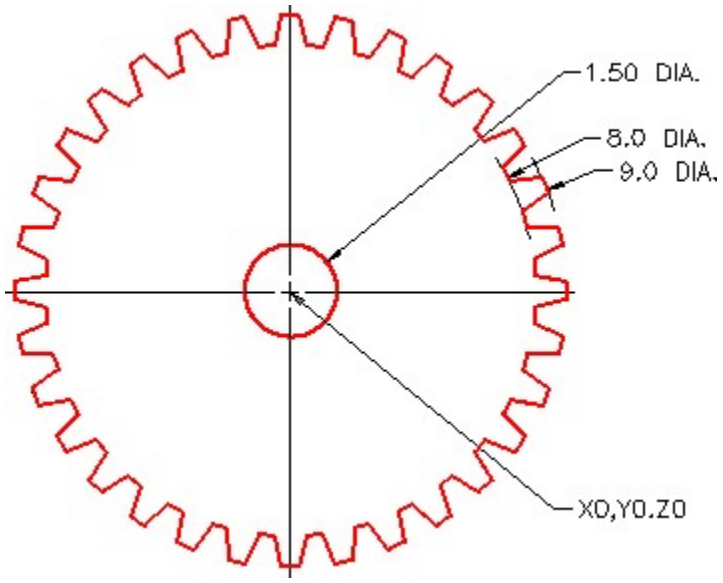
Step 3

Create the layers Construction and Model as shown in Figure Step 3.





Solid Model – SE Isometric View



Multiview Drawing – Gear is 0.5 Units Thick

Geometry and Drafting Lessons

Some modules contain drafting and/or geometry lessons. They are included to teach students 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 the theory in the lesson, skip it and go on to the next topic in the module. If you don't know it, study it. This theory is not part of the course and you will not be tested on it.

However, knowing it will help you complete the lab exercises. For example:

(Do not try to understand the following 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 centre point. The centre point is a single XY coordinate. Study the drawings below for a complete description of the characteristics of a circle.

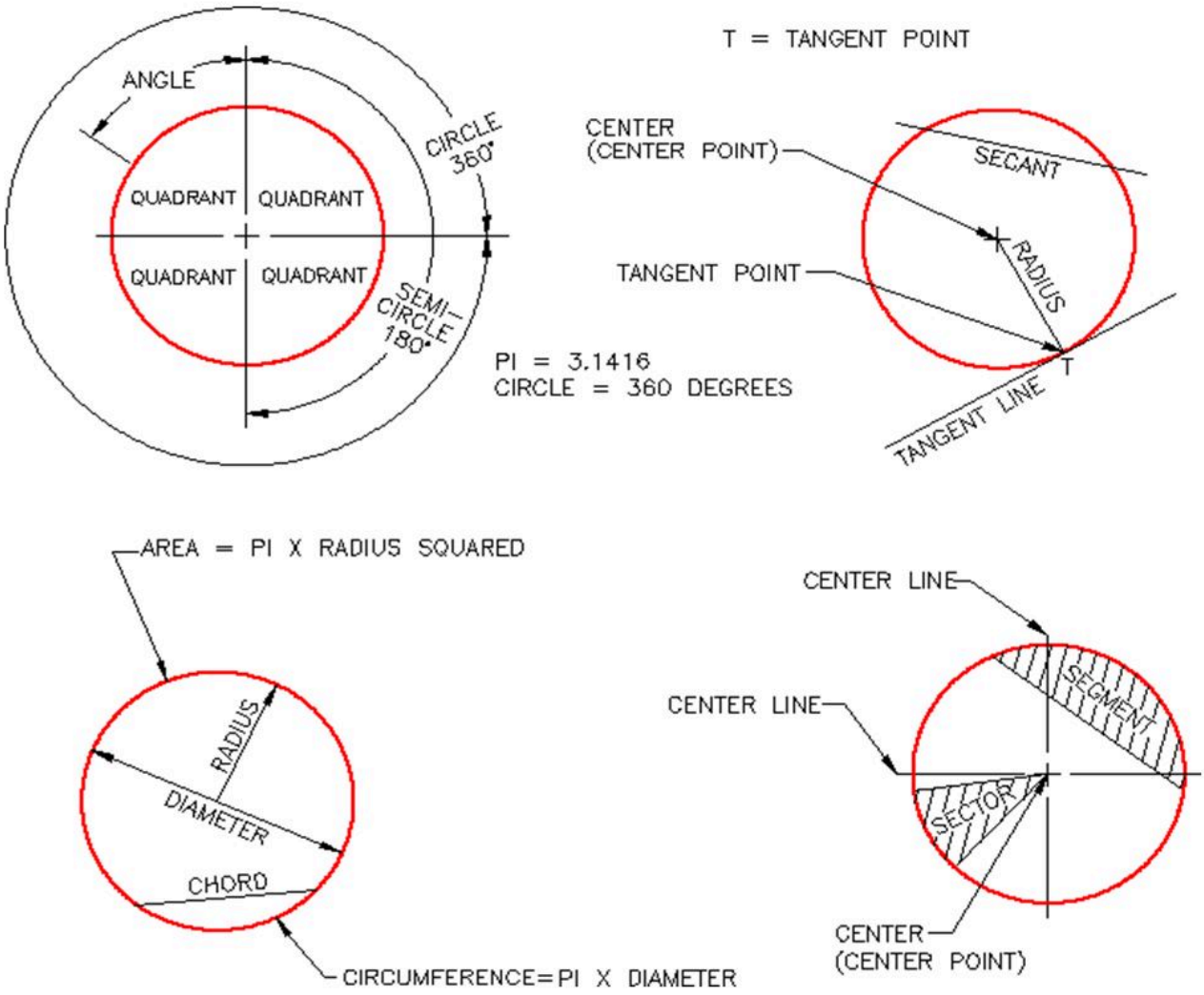


Figure 7-1 Circle Geometry

Key Principles

Each module contain a list of key principles. The *Key Principles* are principles that you should have learned and understand in the module. It is important that you understand each one of these principles as you will be required to use them in future AutoCAD work. For example:

(Do not try to learn the following Key Principles now, it is here for an example only.)

Key Principles in Module 8

1. The TRIM and EXTEND commands should have the following default settings when working in 3D: Projection=None, Edge=Extend
2. The ROTATE command, used for a 2D rotate, uses a point to rotate For a 3D rotate using the ROTATE3D command, a line (2 points) must be specify to rotate around or the X, Y or Z axis.

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.

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 modules to help the user understand the material.

Words in 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. i.e.

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

Words and Numbers that are in Bold Font

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

Command: **3DMESH**

Enter size of mesh in M direction: **5** Enter size of mesh in N direction: **4** Specify location for vertex (0, 0): **0,0,0.2**

Specify location for vertex (0, 1): **0,1,0.5**

Specify location for vertex (0, 2): **0,2,0.4**

Command Names

Command names are always in uppercase. For example:

To construct a fillet, you 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, then use the FILLET command ‘.

Picking Objects or Locations on the Drawing

To help you 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. i.e.

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

WORK ALONG: Using the 3DFACE and AI_MESH Commands

Step 15

Using the AI-MESH command shown below, insert two additional surfaces. See Figure Step 15A and 15B.

Command: **AI_MESH**

(Press F3 and enable Endpoint osnap mode. See Step 14.)

Specify first corner point of mesh: *(end)* **P5**

Specify second corner point of mesh: *(end)* **P6**

Specify third corner point of mesh: *(end)* **P7**

Specify fourth corner point of mesh: *(end)* **P8**

Enter mesh size in the M direction: **6**

Enter mesh size in the N direction: **8**

Command: **AI_MESH**

(Press F3 and enable Endpoint osnap mode. See Step 14.)

Specify first corner point of mesh: *(end)* **P9**

Specify second corner point of mesh:*(end)* **P10**

Specify third corner point of mesh: *(end)* **P11**

Specify fourth corner point of mesh: *(end)* **P12**

Enter mesh size in the M direction: **6**

Enter mesh size in the N direction: **8**

Command:

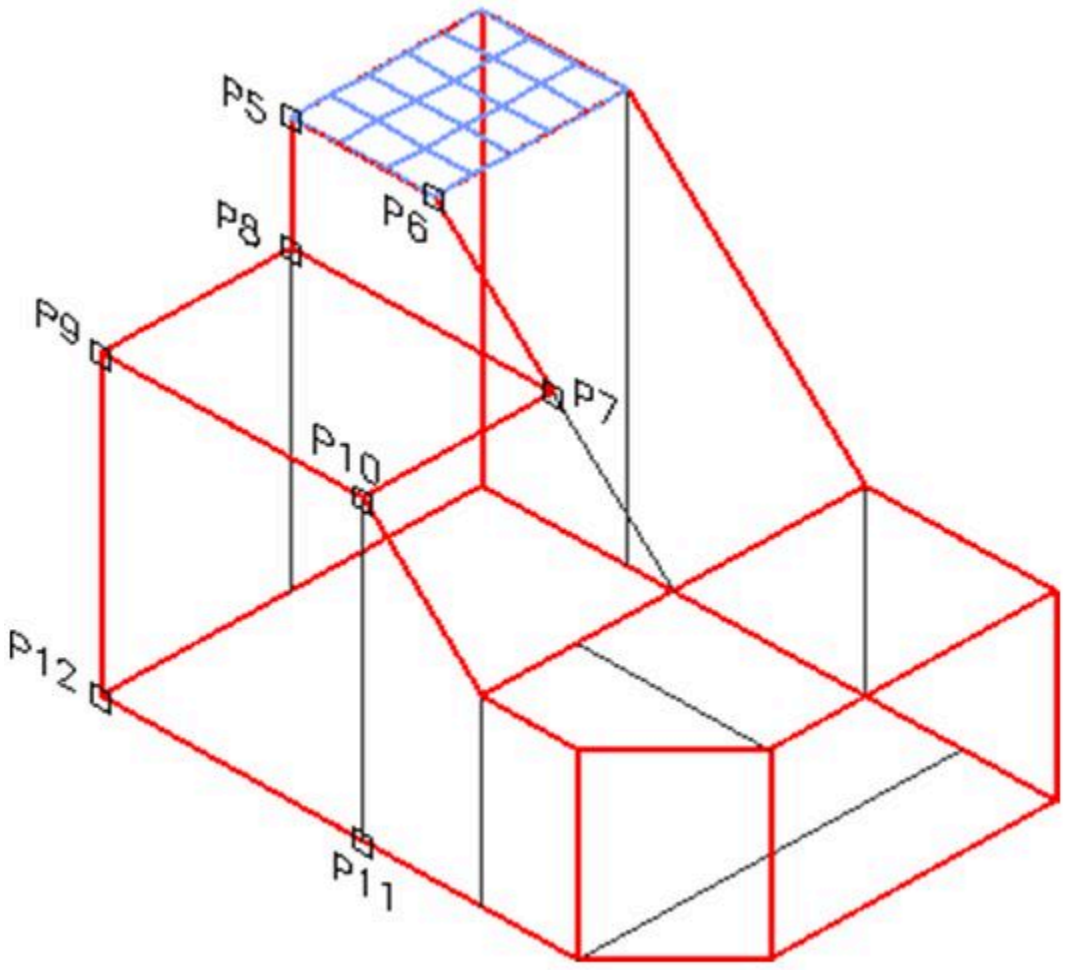


Figure Step 15A

Author's Comments

A comment by the author to help explain what is happening in a command is in italics inside brackets. For example:

Command: **REVSURF**

Current wire frame density: SURFTAB1=64 SURFTAB2=24

Select object to revolve

(Select the closed polyline.)

Select object that defines the axis of revolution:

(Select the axis line.)

Specify start angle <0>:

(Accept the default of 0.)

Specify included angle (+=ccw, -=cw) <360>:

(Accept the default.)

Command:

There are also author's comments in the workalong or lab exercise to help you understand a principle or to complete a task. For example:

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

Key Principles

Key Principles in Module 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 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
4. All lab exercises have a time limit allocated to 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 memorized 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

Configuring Your AutoCAD Software

Learning Outcomes

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

1. Configure your AutoCAD software for the AutoCAD 3D book.

WORK ALONG: Configuring Your AutoCAD Software

Step 1

Enter the OPTIONS command to open the Options dialogue box.

Step 2

In the Options dialogue box, enable the Profiles tab. (Figure Step 2)

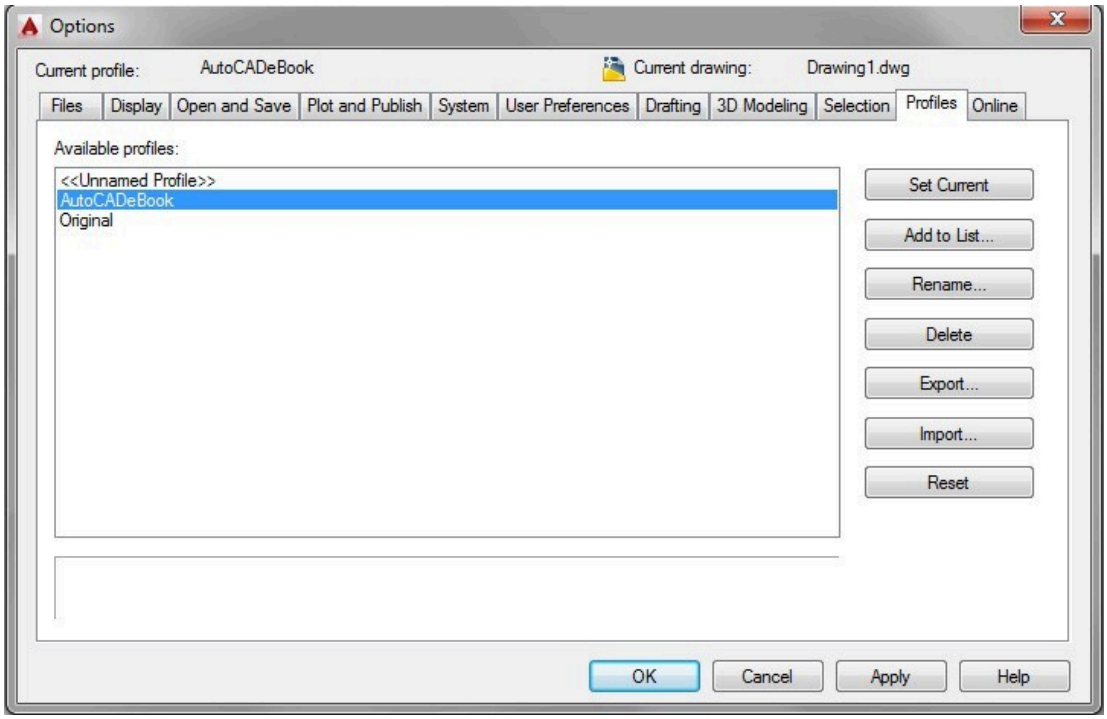


Figure Step 2

AUTHOR'S COMMENTS: Do not be concerned if your dialogue box does not match the figure exactly.

Step 3

Click the Add to List button. This will open the Add Profile dialogue box. Enter the Profile name: Original 3D. (Figure Step 3A and 3B)

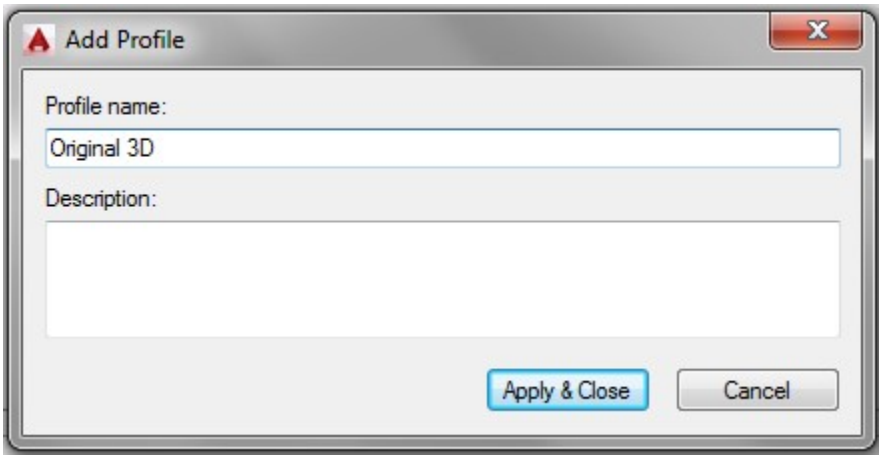


Figure Step 3A

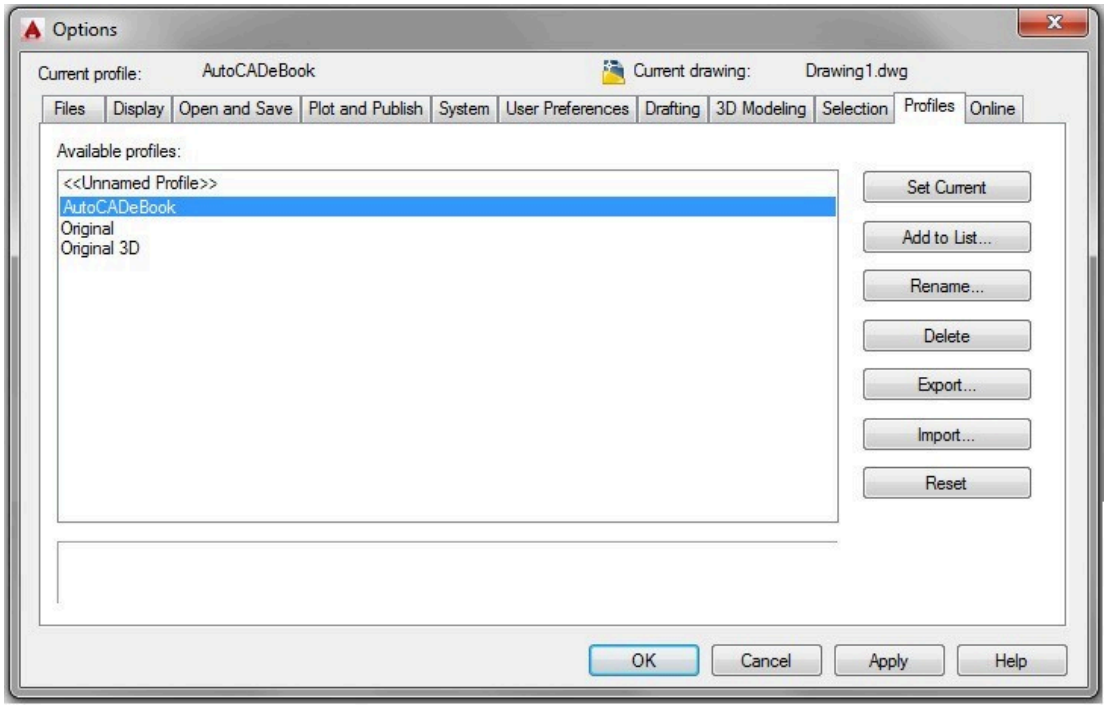


Figure Step 3B

AUTHOR'S COMMENTS: I am having you create a profile named ' Original 3D ' 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 profile to restore the

original configuration.

Step 4

Using what you learned in Step 3, create a new profile named: AutoCAD 3D. (Figure Step 4)

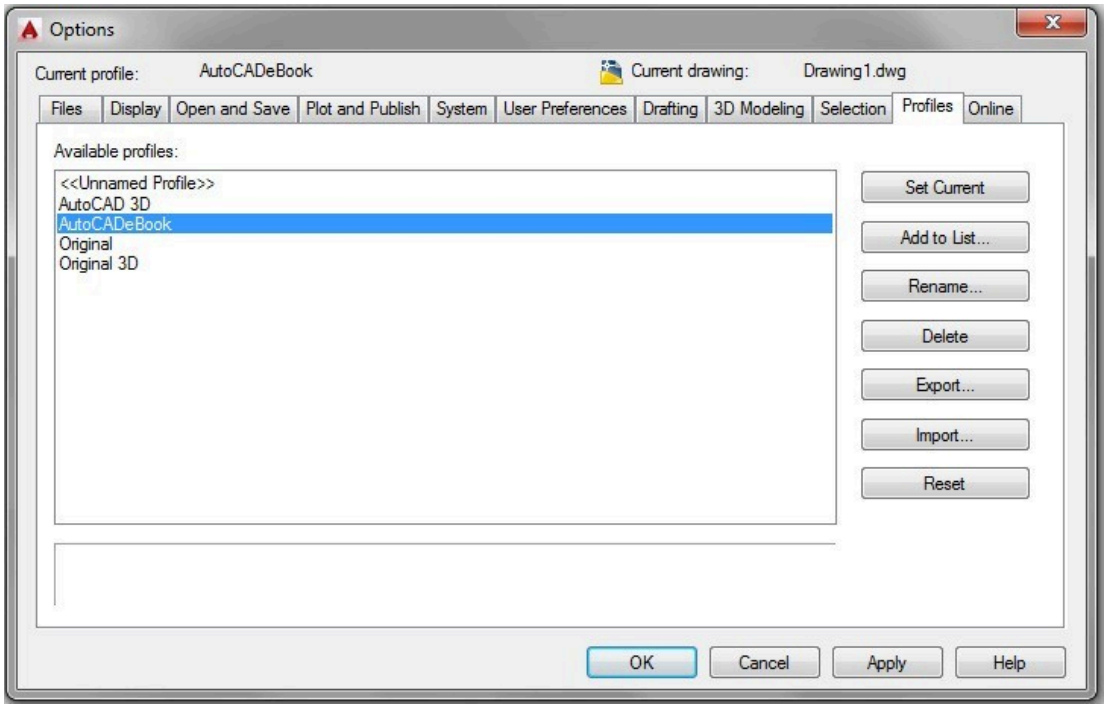


Figure Step 4

Step 5

Click the profile: AutoCAD 3D to highlight it. While it is selected, click the Set Current button. Ensure that the Current profile is: AutoCAD 3D as shown across the top of the dialogue box. (Figure Step 5)

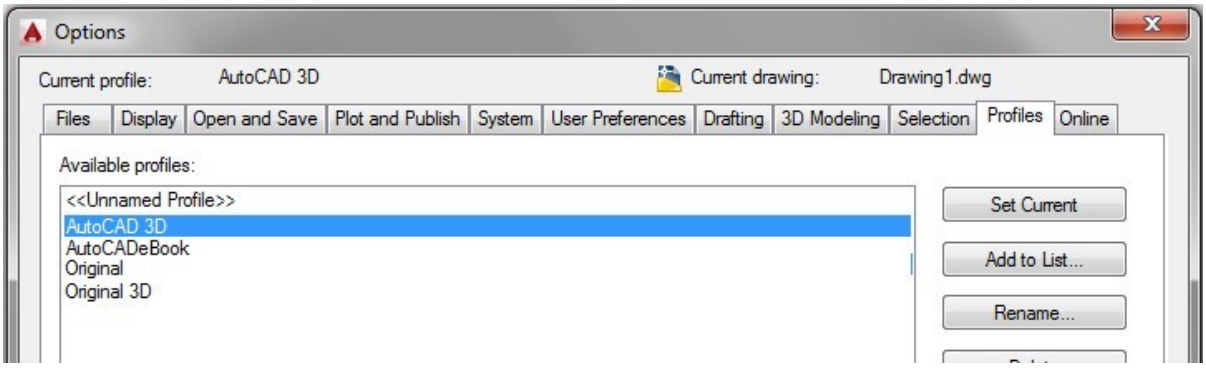


Figure Step 5

Step 6

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 6)

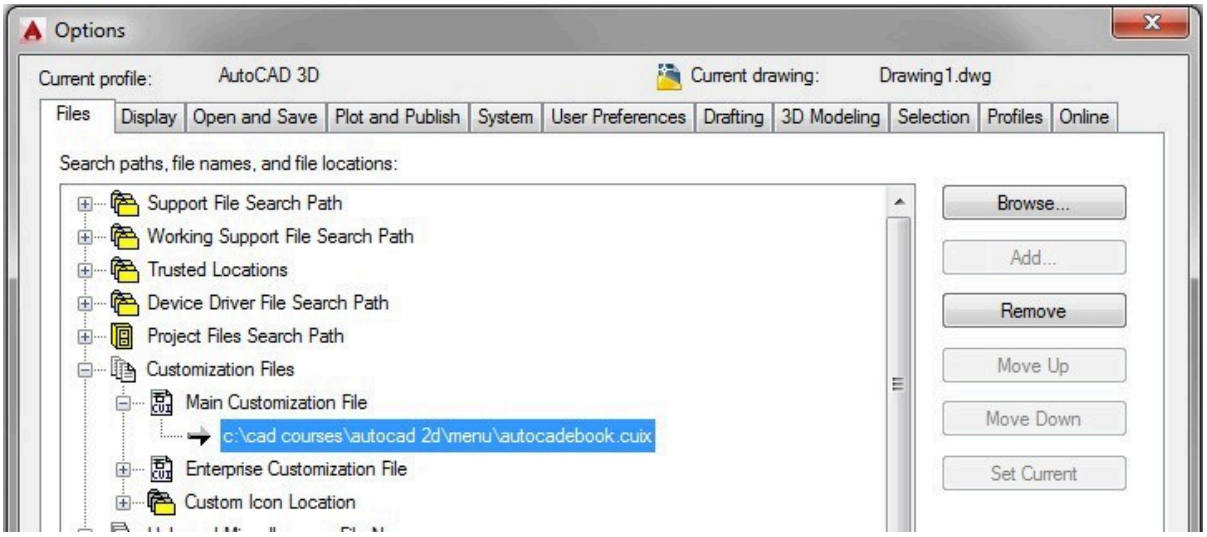


Figure Step 6

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

Step 7

Step 6 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 7)

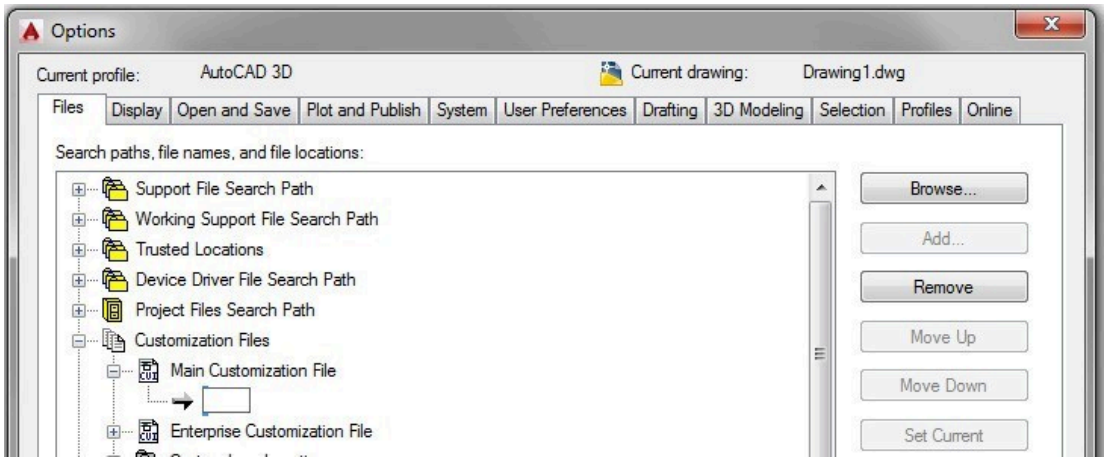


Figure Step 7

Step 8

In the Select a file dialogue box, pull down the Look in: box and find C drive. Click the folder: CAD Courses. (Figure Step 8)

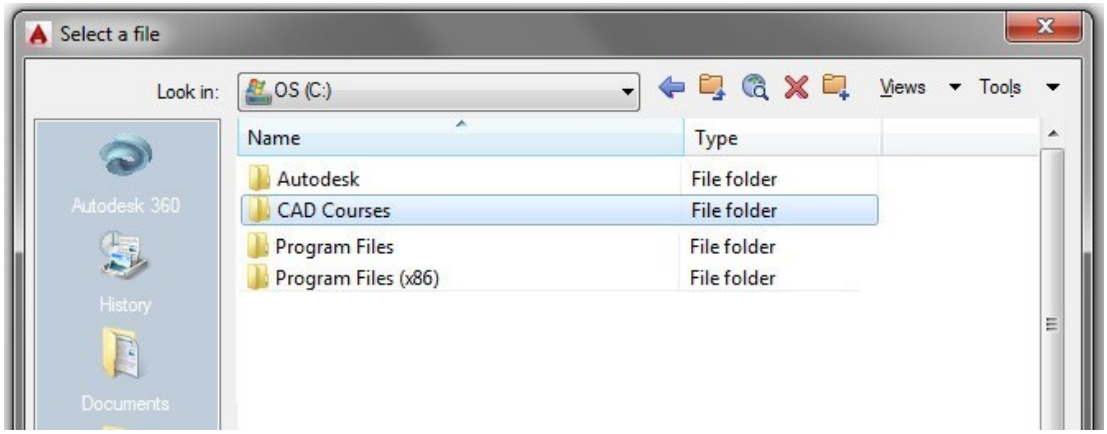


Figure Step 8

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 9

Under the folder : CAD Courses, select the folder: AutoCAD 3D. Under the folder: AutoCAD 3D, select the folder: Menu. In the folder: Menu, select the file: AutoCADbook.CUIX and then click Open. (Figure Step 9)

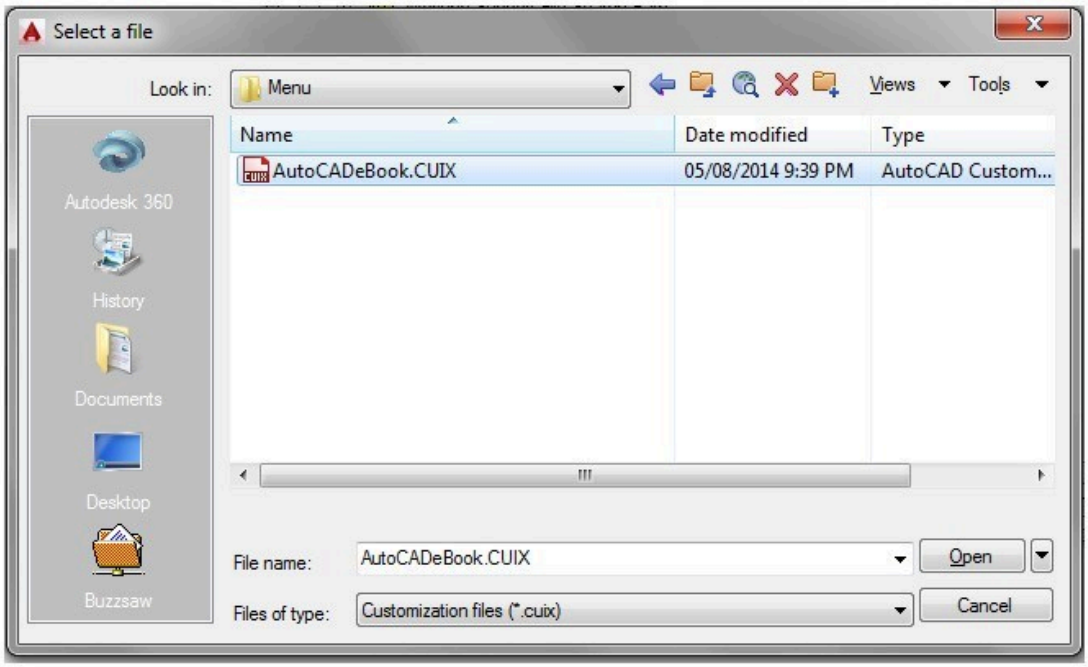


Figure Step 9

Step 10

You should now see the file location:

C:\CAD Course\AutoCAD 3D\Menus\AutoCADbook.CUIX. (Figure Step 10A and 10B)

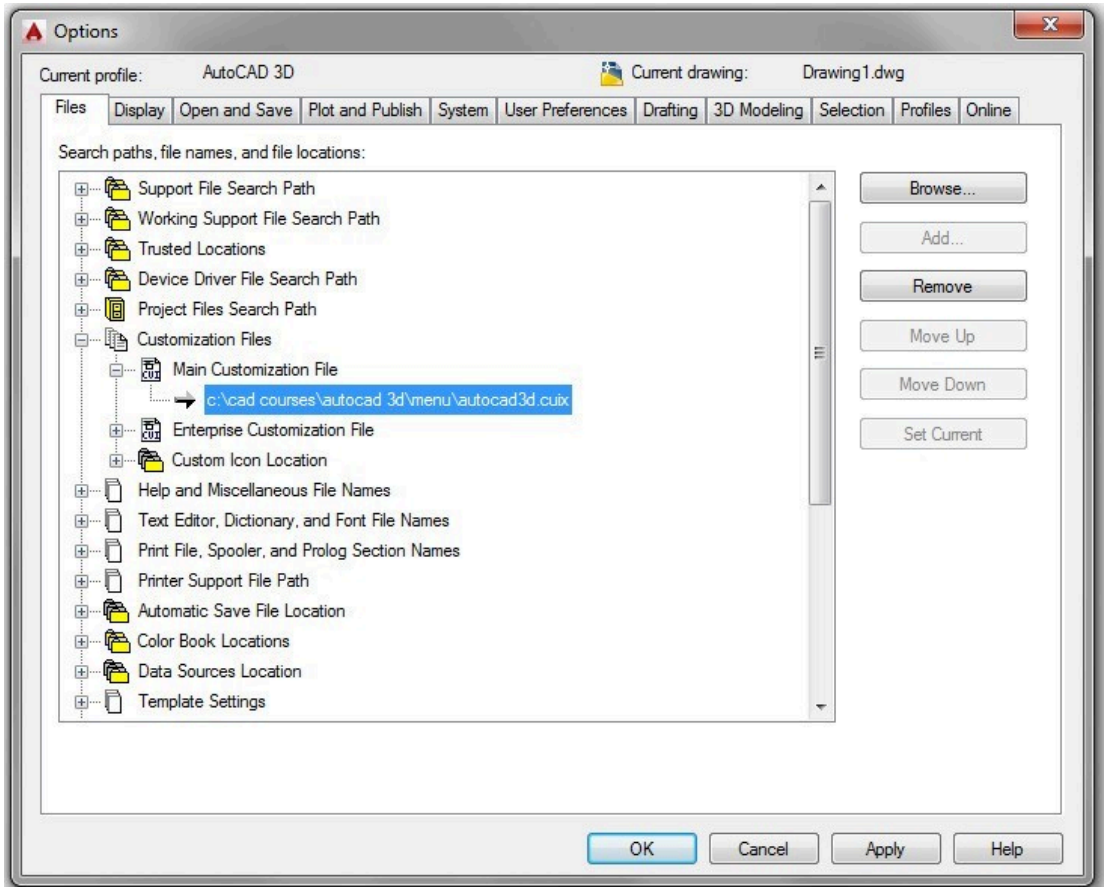


Figure Step 10A

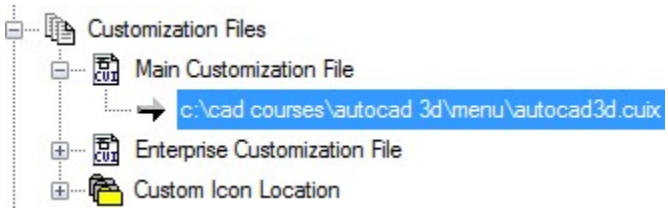


Figure Step 10B

Step 11

Click the small arrow in the Workplace Switching icon on the Status bar located along the bottom of the Graphic window. (Figure Step 11)

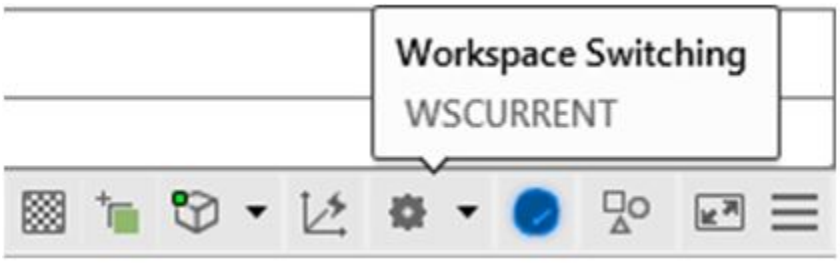


Figure Step 11

Step 12

In the Workspace pull-up menu, click Workspace Settings… This will open the Workspace Settings dialogue box. (Figure Step 12)

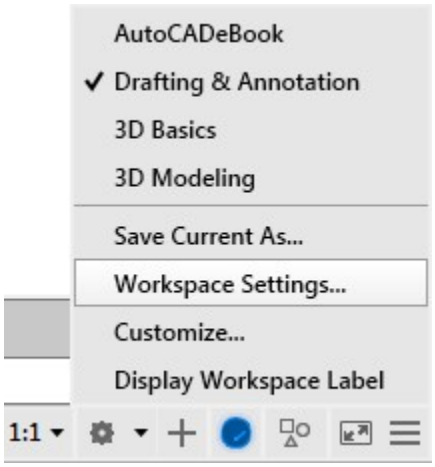


Figure Step 12

AUTHOR'S COMMENTS: AutoCAD has two menu systems. They are the Toolbar menus and the Ribbon menus. You must use one or the other but you cannot use both at the same time. You can however, switch back and forth between them. Along with using either the Toolbar or Ribbon menus, you must add the Pull-down menu and use it in conjunction with either menu. 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 the following steps and skip Step 13 (Ribbon menus) and Step 14 (Ribbon menus). If you intend to use Ribbon menus to do this book, skip to Step 13 (Ribbon menus).

Step 13 (Toolbar menus)

In the Workspace Settings dialogue box, set My Workspace to the workspace: AutoCADbook. Enable Automatically save workspace changes by clicking the radio button. Click OK to close the box. (Figure Step 13)

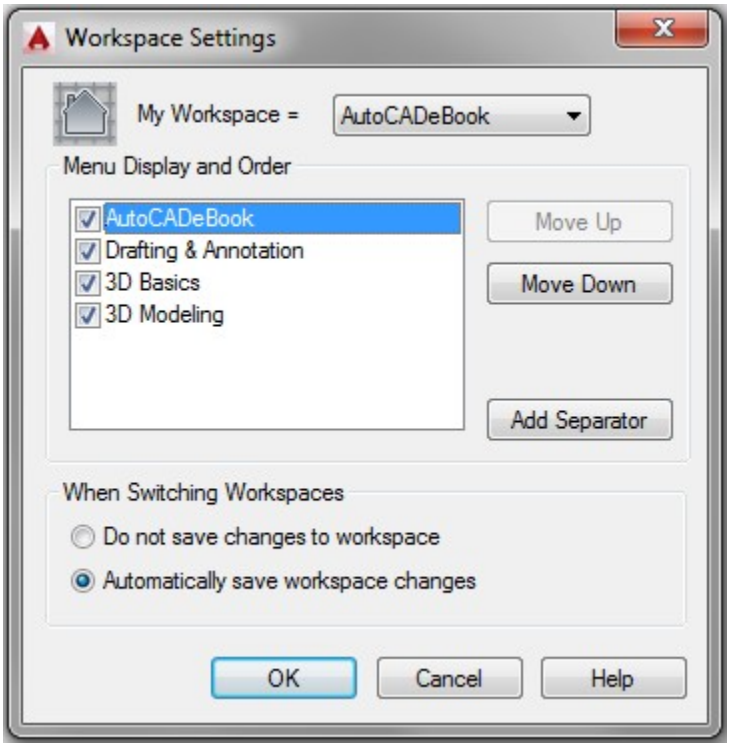


Figure Step 13

Step 14 (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 14)

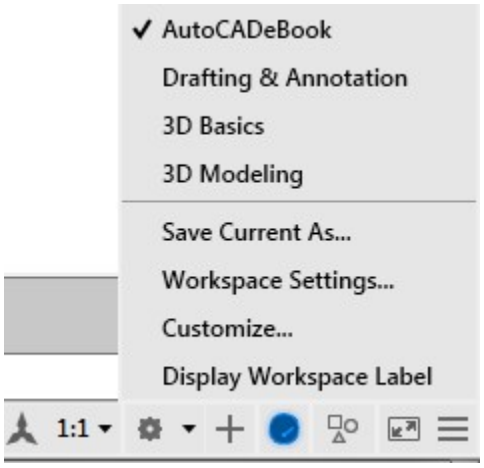


Figure Step 14

If you intend to use Ribbon menus to do this book, complete the following steps. If you intend to use Toolbar menus to do this book, skip to Step 15.

Step 13 (Ribbon menus)

In the Workspace Settings dialogue box, set My Workspace to the workspace: 3D Modeling. Enable Automatically save workspace changes by clicking the radio button. Click OK to close the box (Figure Step 13)

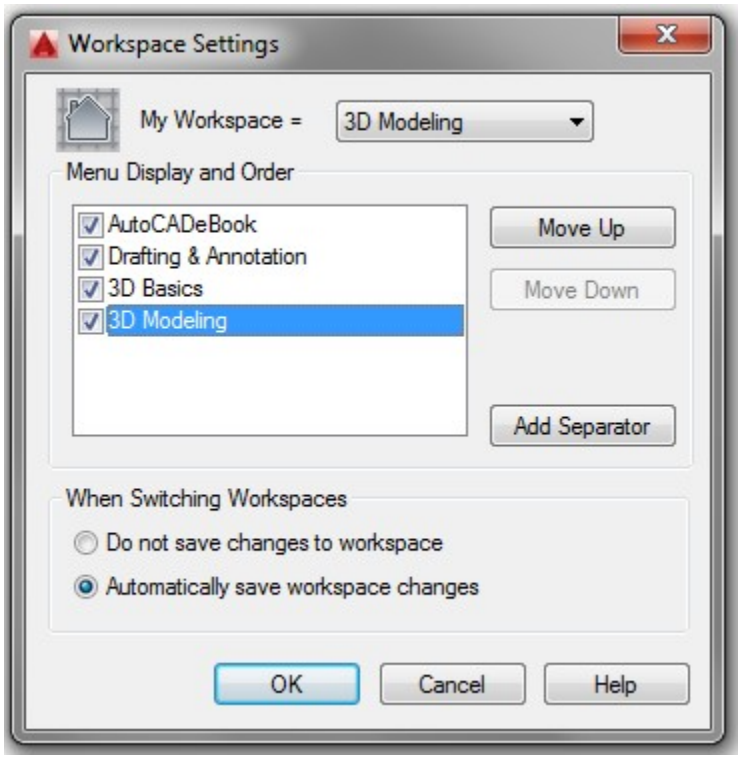


Figure Step 13

Step 14 (Ribbon menus)

Click the small arrow in the Workplace Switching icon again on the Status bar. Click the workspace: 3D Modeling to set it as the current workspace. Make sure that the check mark is beside the workspace: 3D Modeling as shown in the figure. (Figure Step 14)

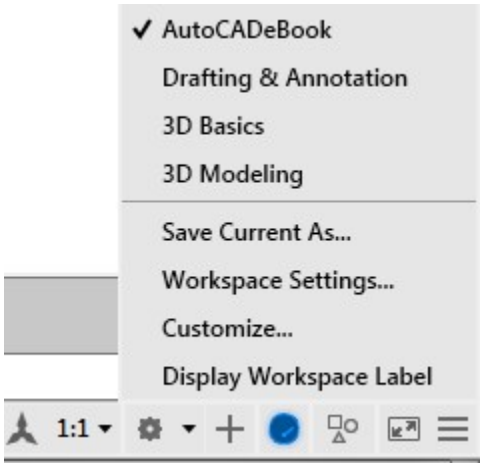


Figure Step 14

Step 15

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 15)

Figure Step 15

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

Step 16

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

Enter new value for MENUBAR <0>: **1**

Command:

Step 17

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

Enter new value for NAVBARDISPLAY <1>: **0**

Command:

Step 18

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

Enter new value for VPCONTROL <ON>: **OFF**

Command:

If you are using AutoCAD 2015 execute the following command:

Step 19 (2015)

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

Enter new value for NEWTABMODE <1>: **0**

Command:

If you are using AutoCAD 2016, 2017, or 2018 execute the following command:

Step 19 (2016, 2017, and 2018)

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

Command: **STARTMODE**

Enter new value for STARTMODE <1>: **0**

Command:

Step 20

Click the item Tools located in the top line of the in the Pull-down menu. In the Pull- down menu, click Options. (Figure Step 20)

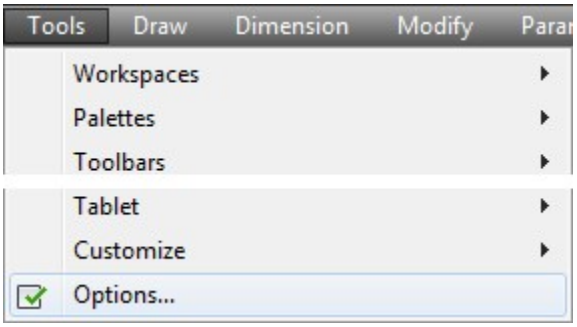
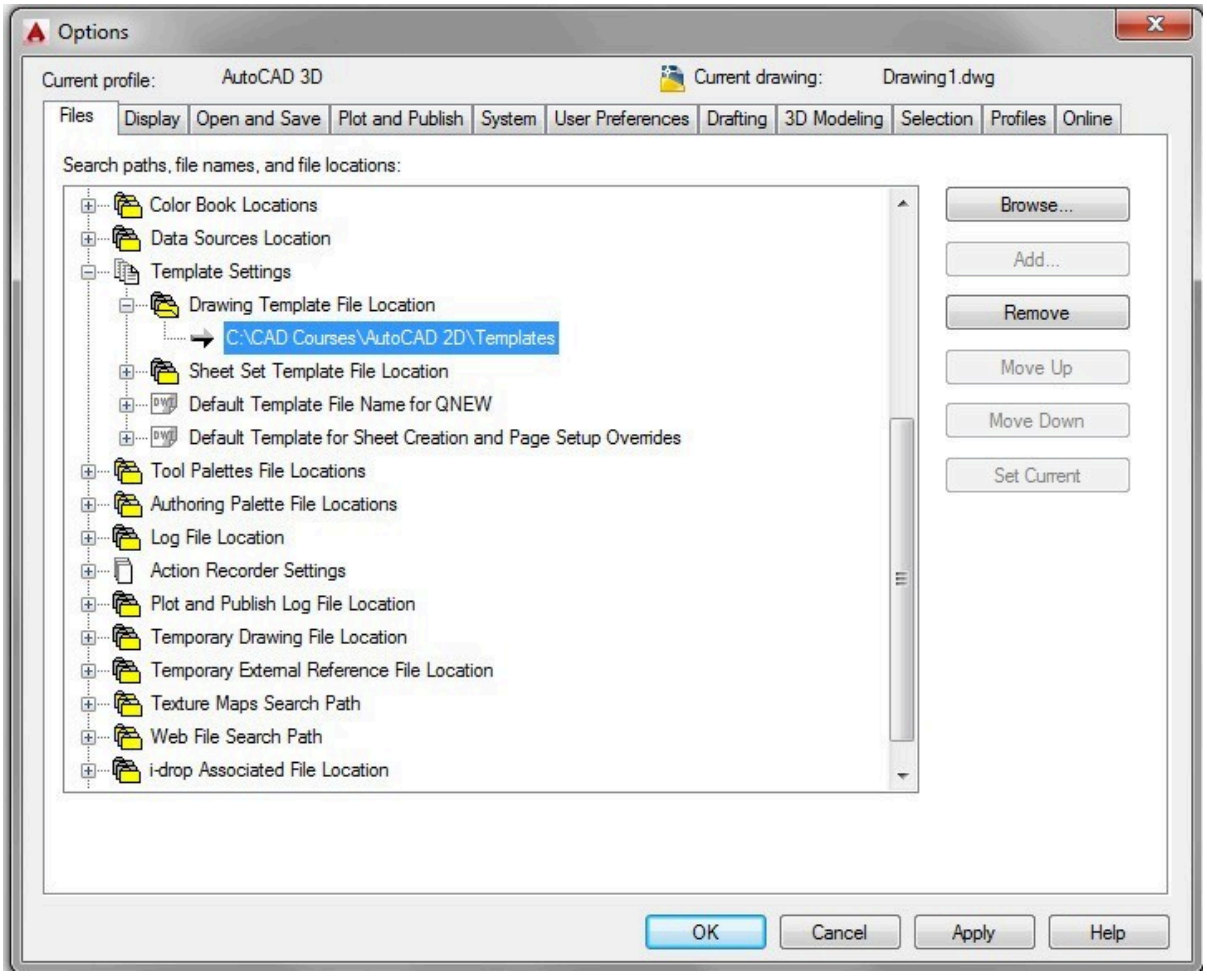


Figure Step 20

Step 21

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 21)



Step 22

It will display an empty box as shown in the figure. (Figure Step 22).

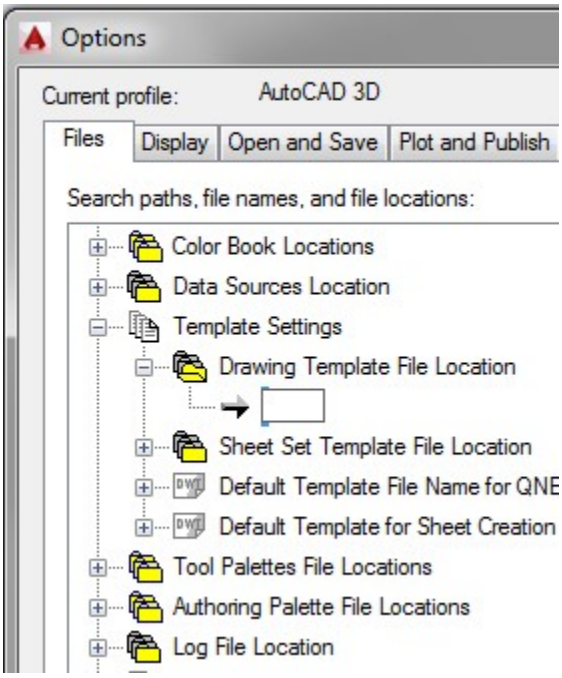


Figure Step 22

Step 23

Click the Browse button. In the Browse for Folders dialogue box, locate the folder: (Figure Step 23)

CAD Courses/ AutoCAD 3D/ Templates

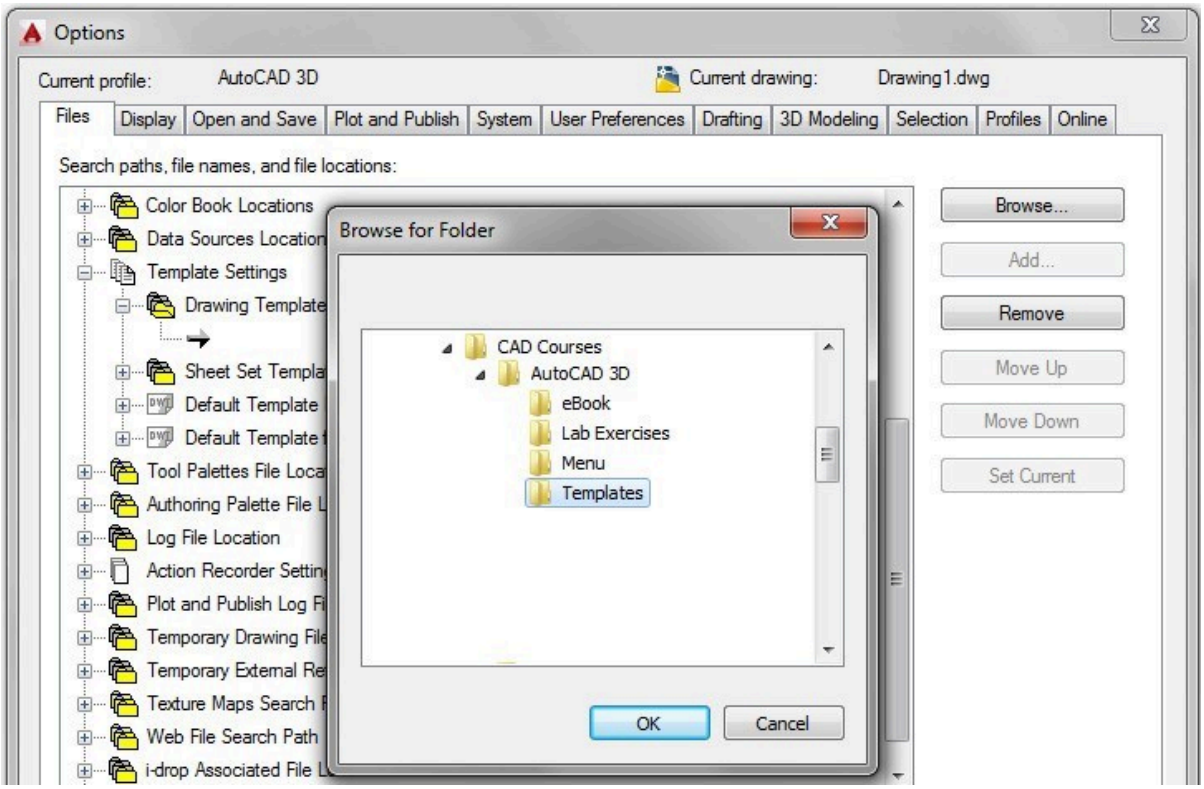


Figure Step 23

Step 24

Click OK and the new location will appear as shown in the figure. (Figure Step 24A and 24B)

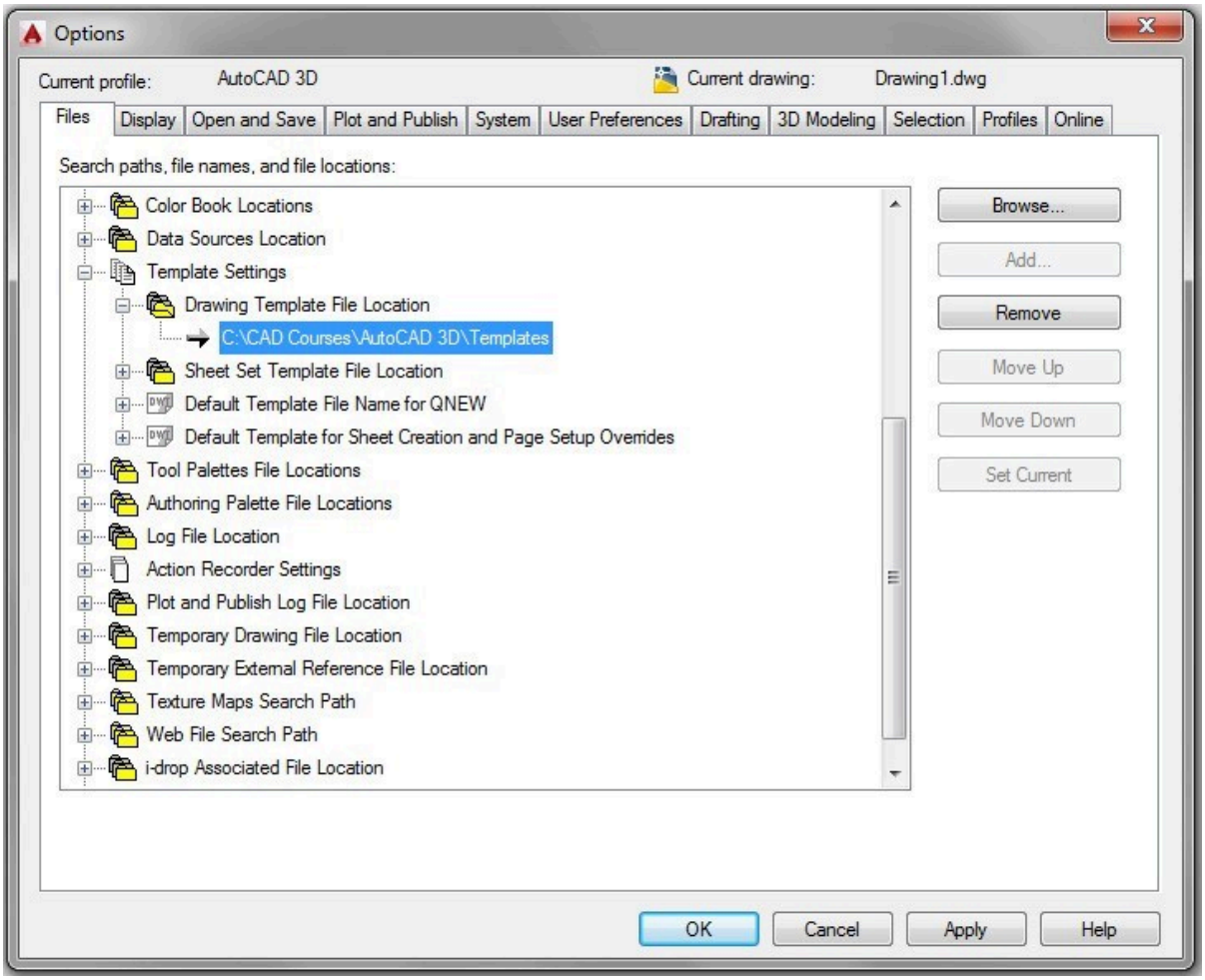


Figure Step 24A

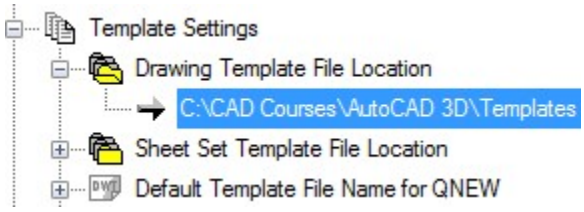


Figure Step 24B

Step 25

Enable the display of the Display tab. (Figure Step 25)

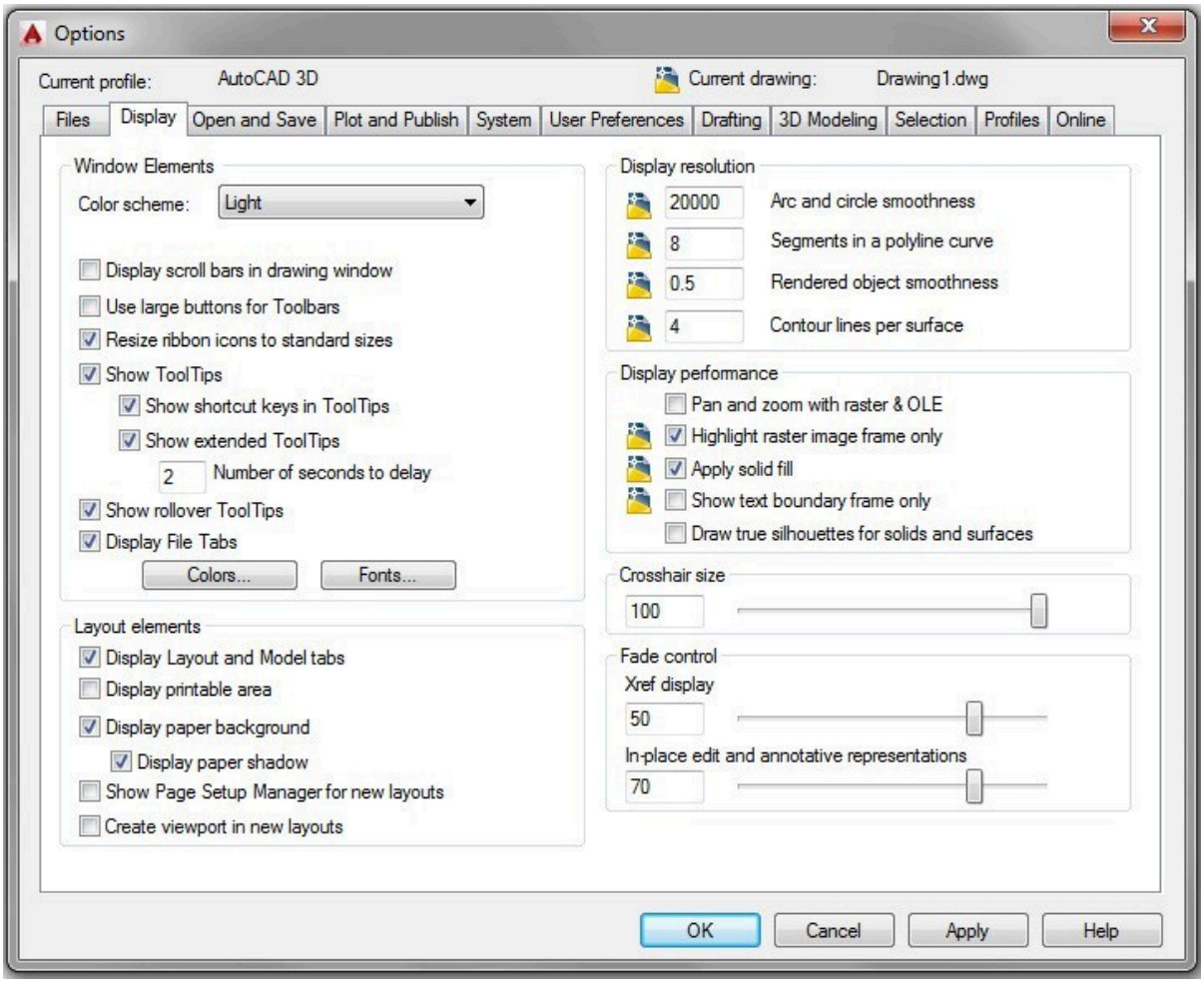


Figure Step 25

Step 26

In the Display resolution area, set the Arc and circle smoothness to 20000. (Figure Step 26)

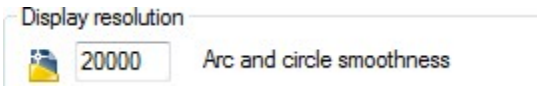


Figure Step 26

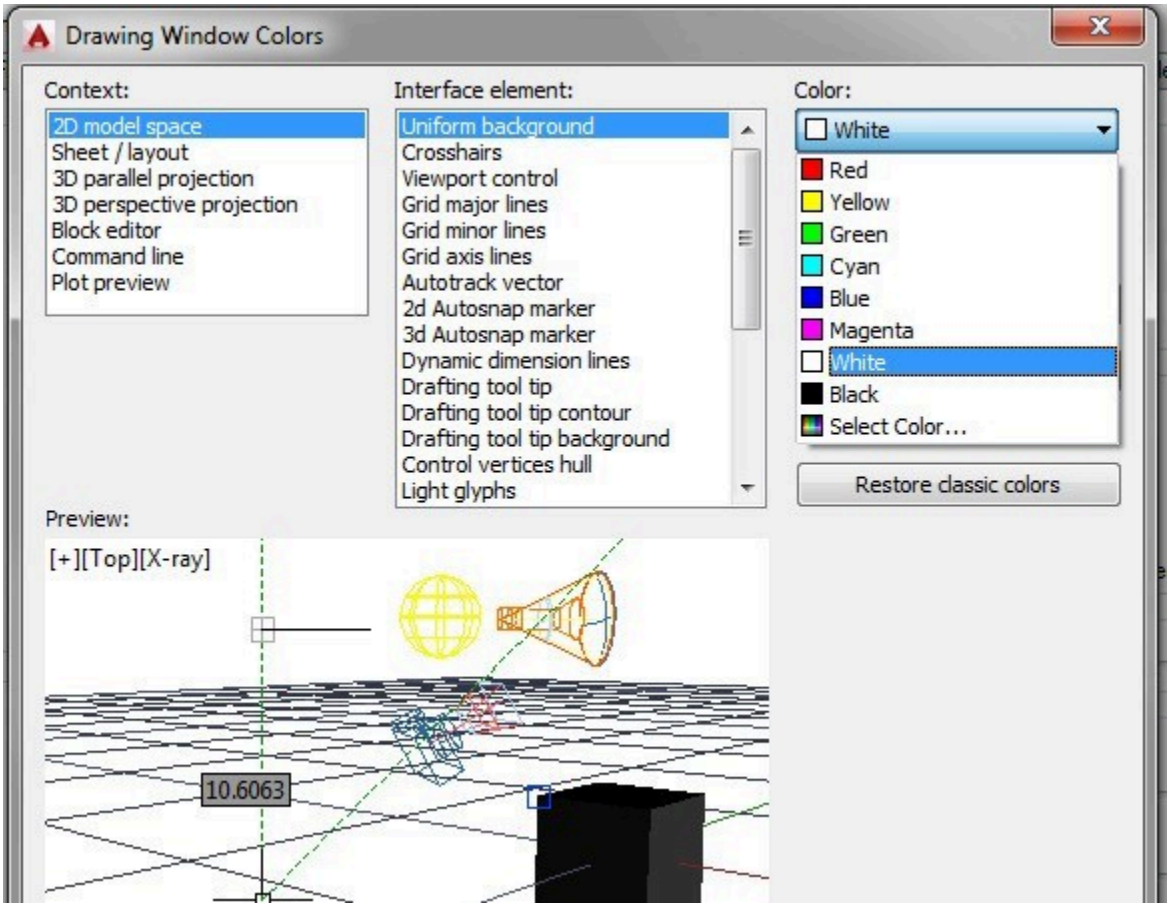
Step 27

Set the Crosshair size to 100. (Figure Step 27)

Figure Step 27

Step 28

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 28)



Step 29

Highlight 3D parallel projection, Uniform background and then select the color White. (Figure Step 29)

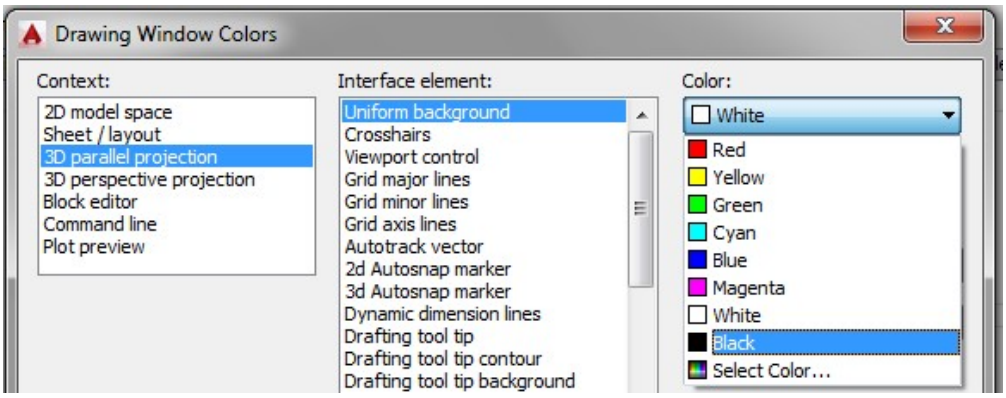


Figure Step 29

Step 30

Enable the User Preferences tab. (Figure Step 30)

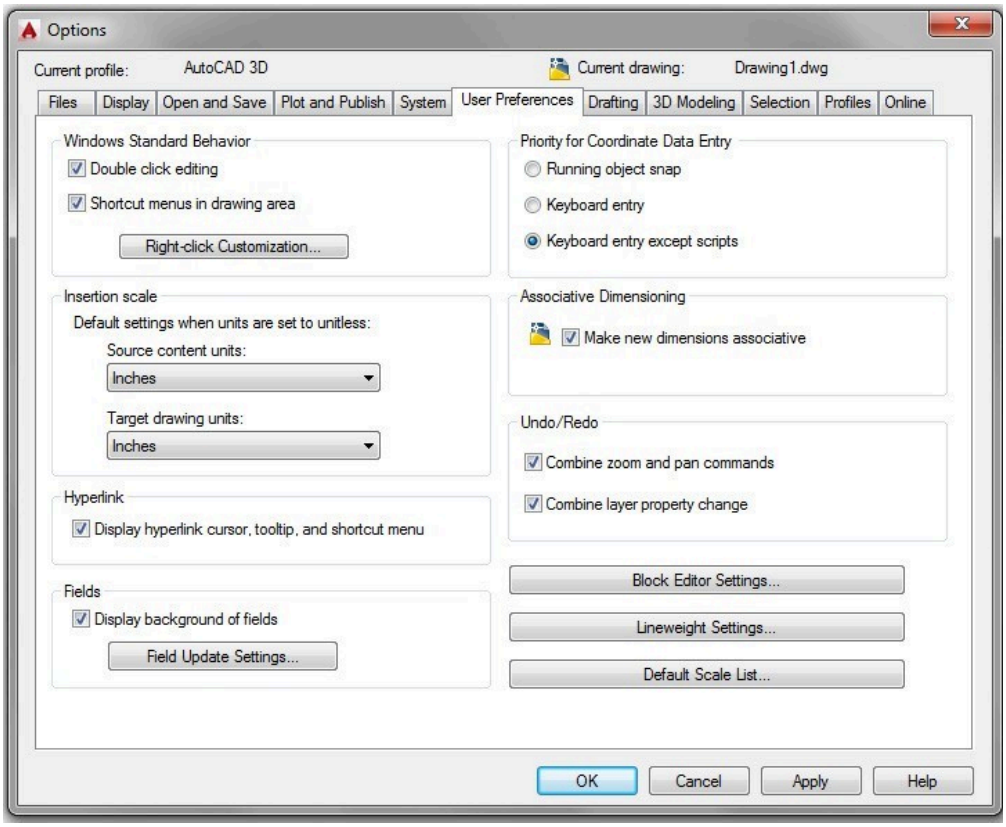


Figure Step 30

Step 31

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 31A and 31B)

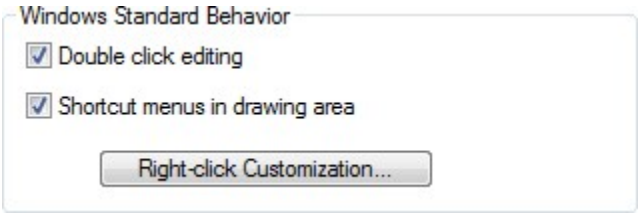


Figure Step 31A

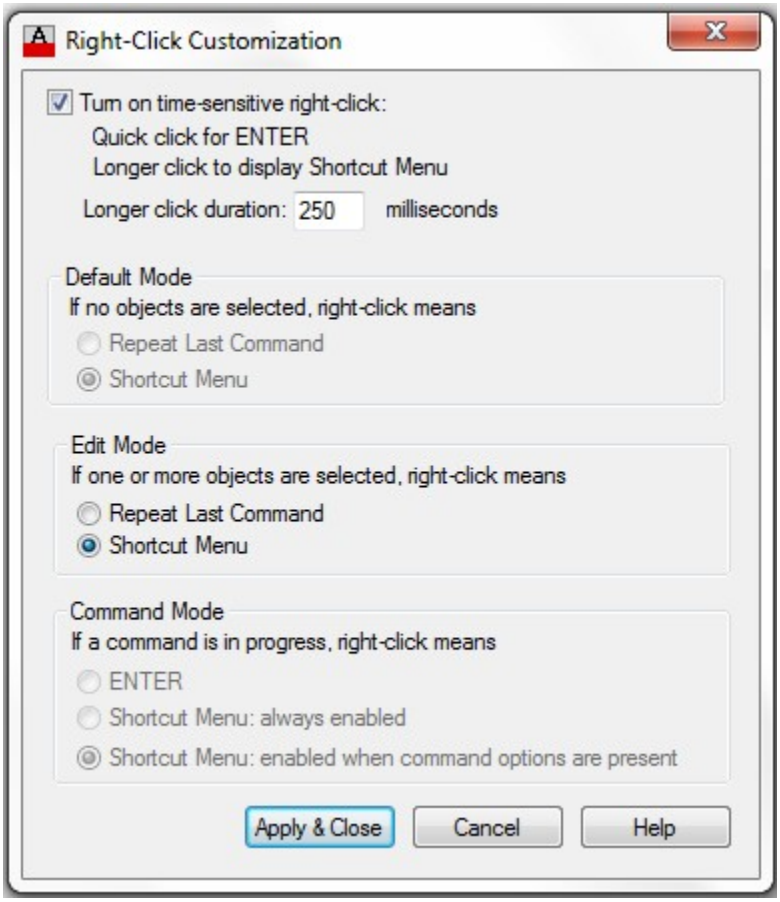


Figure Step 31B

Step 32

Click Apply and Close.

Step 33

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 33A and 33B)

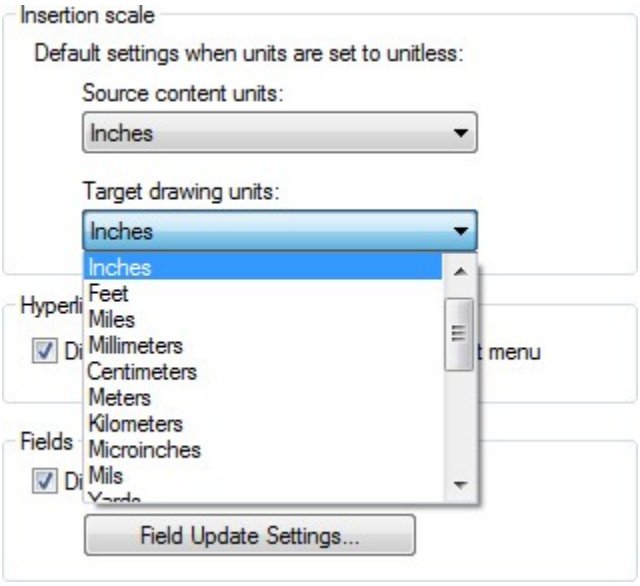


Figure Step 33A

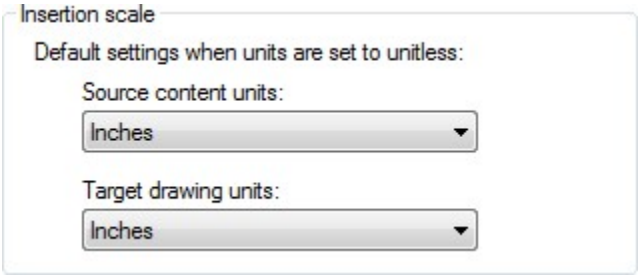


Figure Step 33B

Step 34

Enable the Drafting tab. (Figure Step 34)

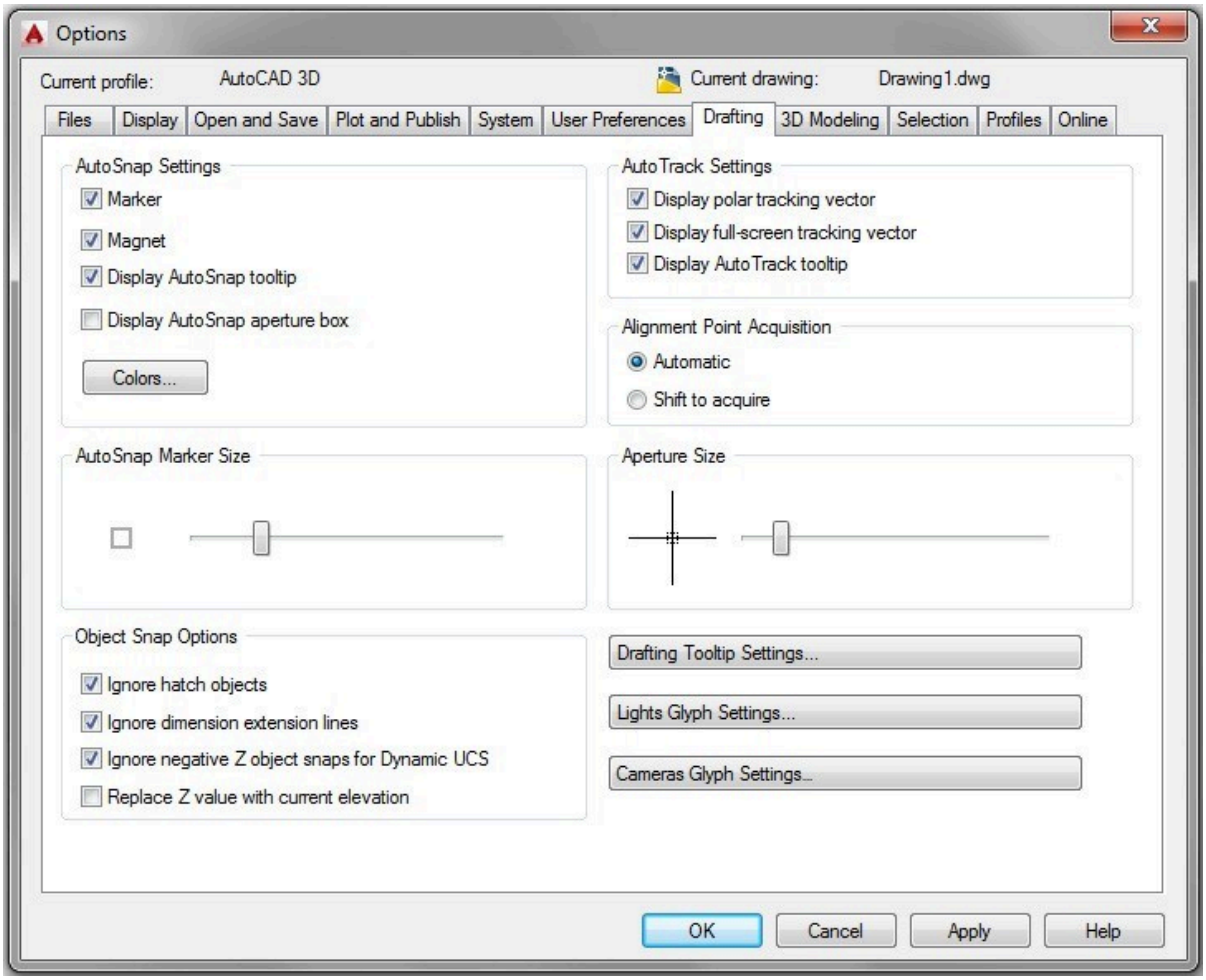


Figure Step 34

Step 35

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 35A, 35B, and 35C)

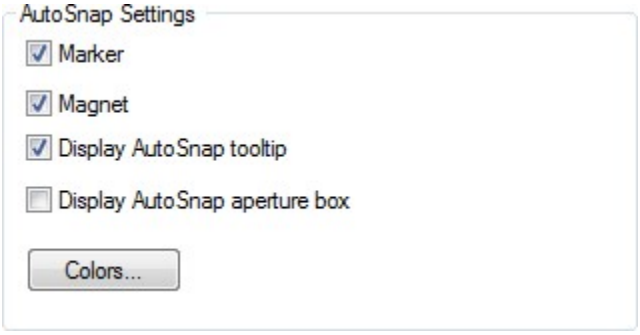


Figure Step 35A

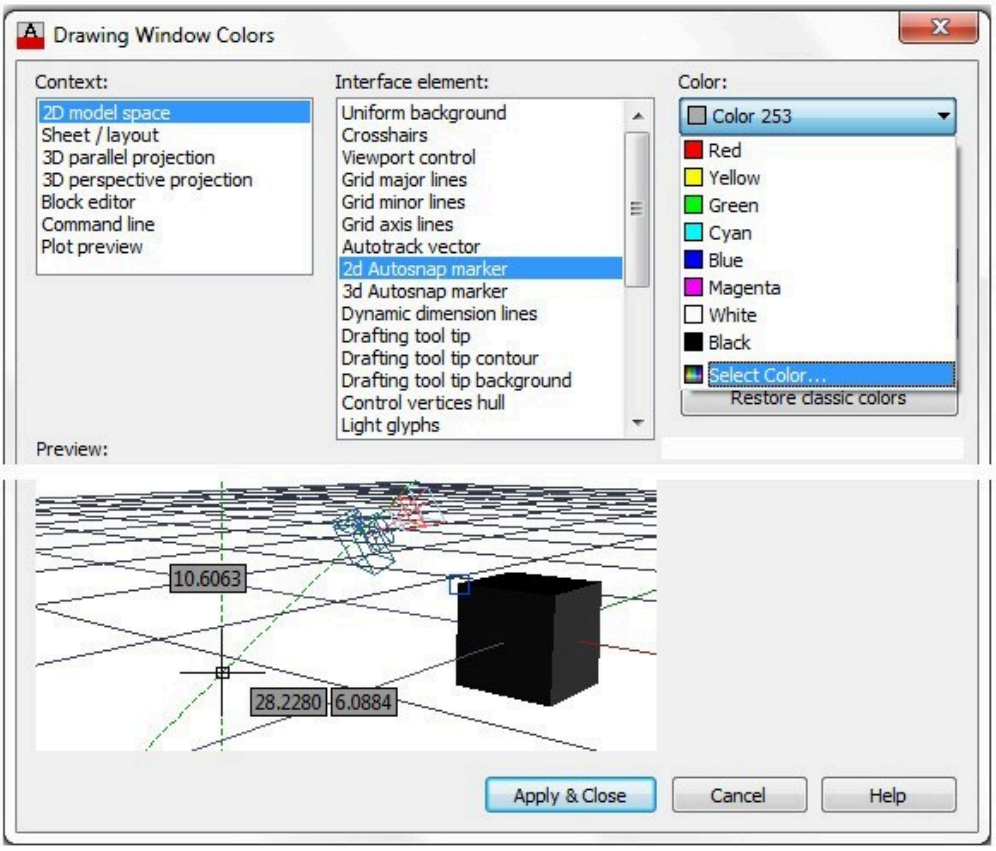


Figure Step 35B

Figure Step 35C

Step 36

Click OK and then click Apply and Close.

Step 37

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 37)

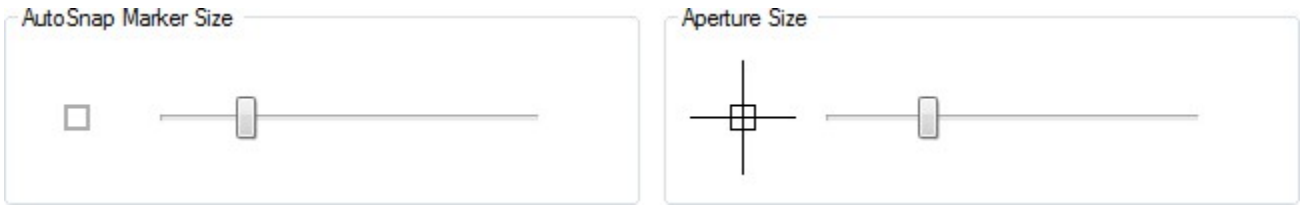


Figure Step 37

Step 38

Enable the 3D Modeling tab. (Figure Step 38)

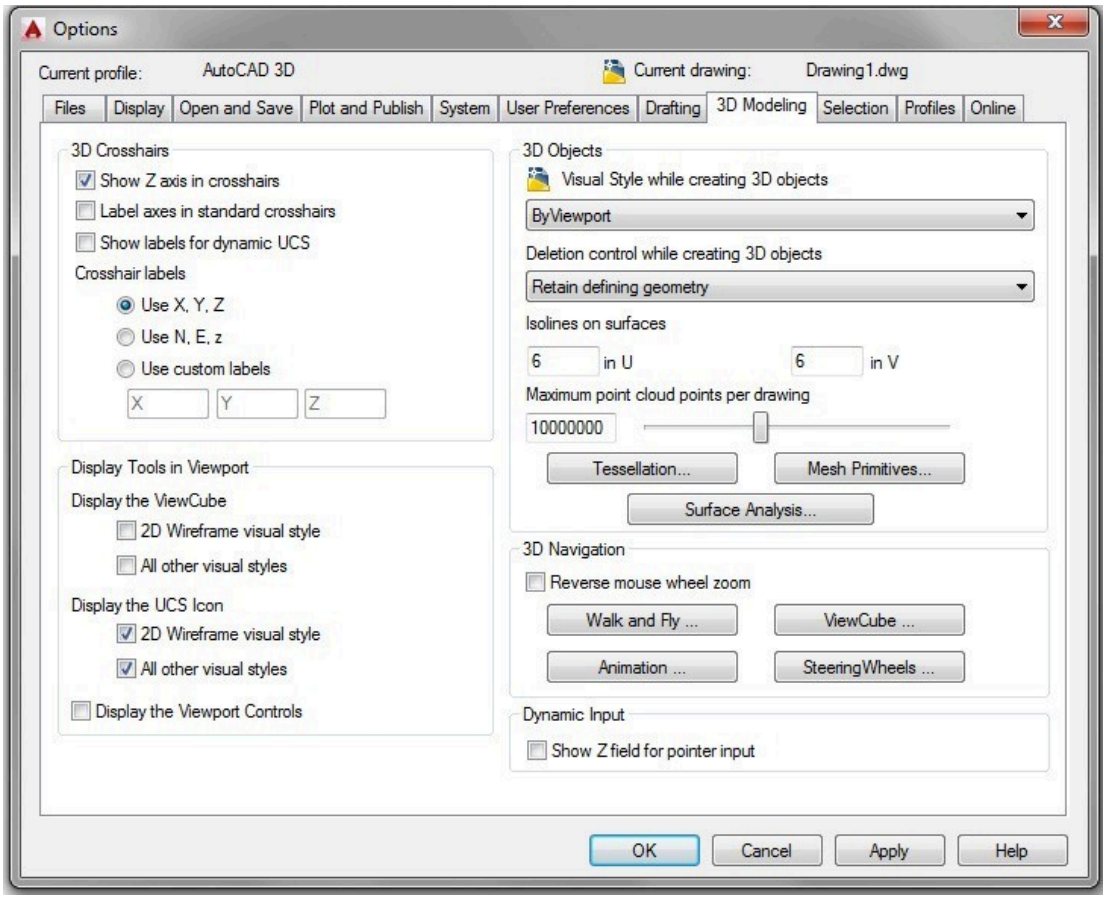


Figure Step 38

Step 39

In the Display Tools in Viewport area, set it as shown in the figure. (Figure Step 39)

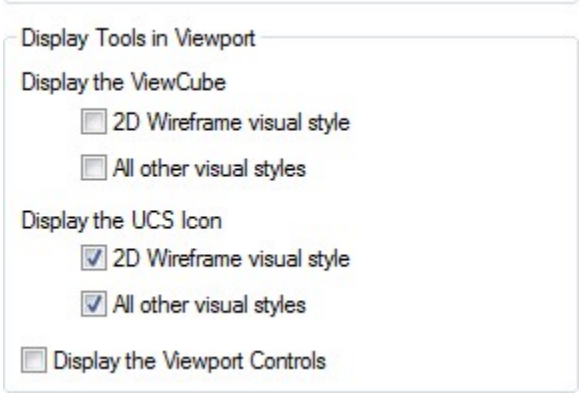


Figure Step 39

Step 40

Enable the Selection tab. (Figure Step 40)

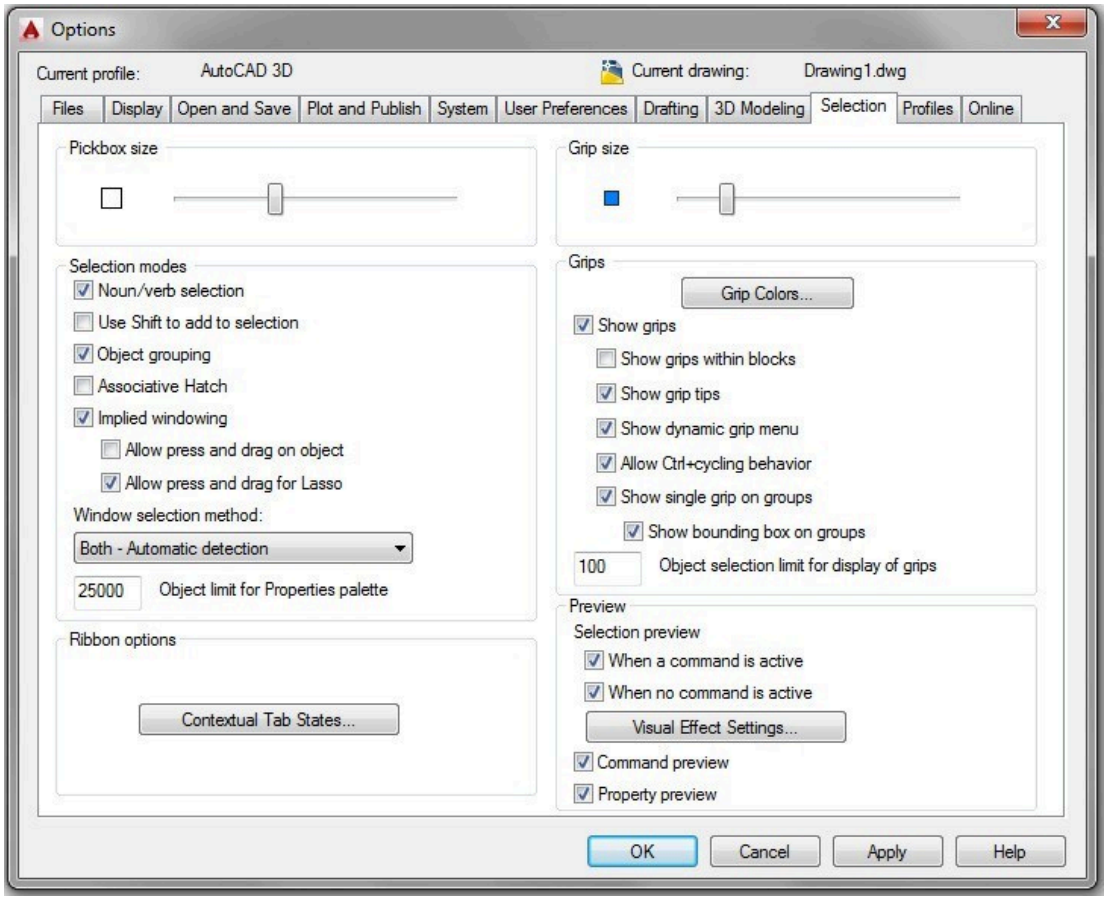


Figure Step 40

Step 41

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 41)

Figure Step 41

Step 42

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

Step 43

Go to Module 1.

Part 1

Module 1: Viewing 3D Models – Part 1

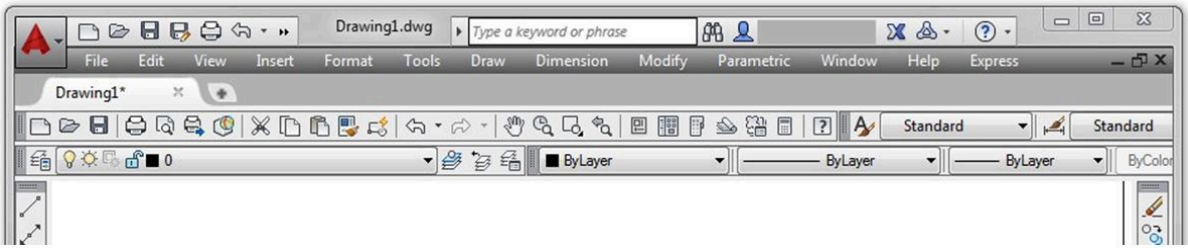
Learning Outcomes

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

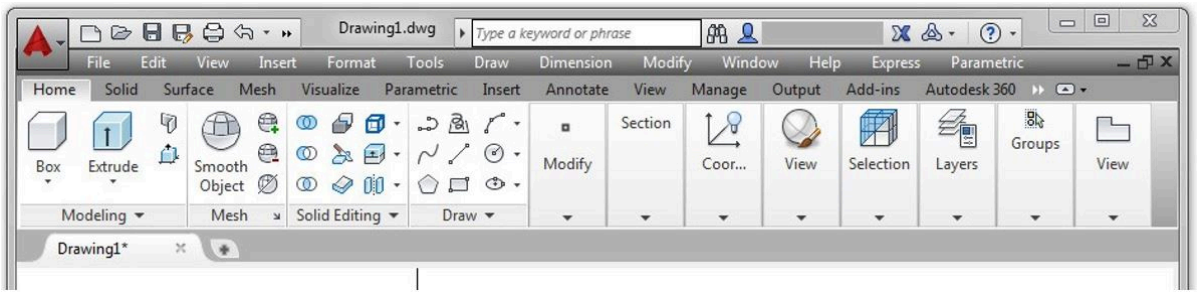
1. Describe AutoCAD's Ribbon and Toolbar menu systems and set one to used for this book.
2. Describe a wireframe model and explain how it is viewed in 3D.
3. Describe the UCS icon and explain how to configure and display it.
4. Apply the VIEW, 3DORBIT, 3DFORBIT, 3DCORBIT, and UCSICON commands.

AutoCAD's Menu Systems

AutoCAD has two menu systems that allows you to communicate with it. The two menu systems are Toolbar menu and Ribbon menu. See Figure 1-1. You can, and should, add the Pull-down menu to either the Toolbar or the Ribbon menu. You can use either menu system when completing labs exercises in this book. You can switch between them, if you wish.



Toolbar Menus



Ribbon Menus Figure 1-1 AutoCAD Menu Systems

WORK ALONG: Setting AutoCAD's Interface Using Toolbar Menus

If you are using Ribbon menus, skip to WORK ALONG: Setting AutoCAD's Interface Using Ribbon Menus

Step 1

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

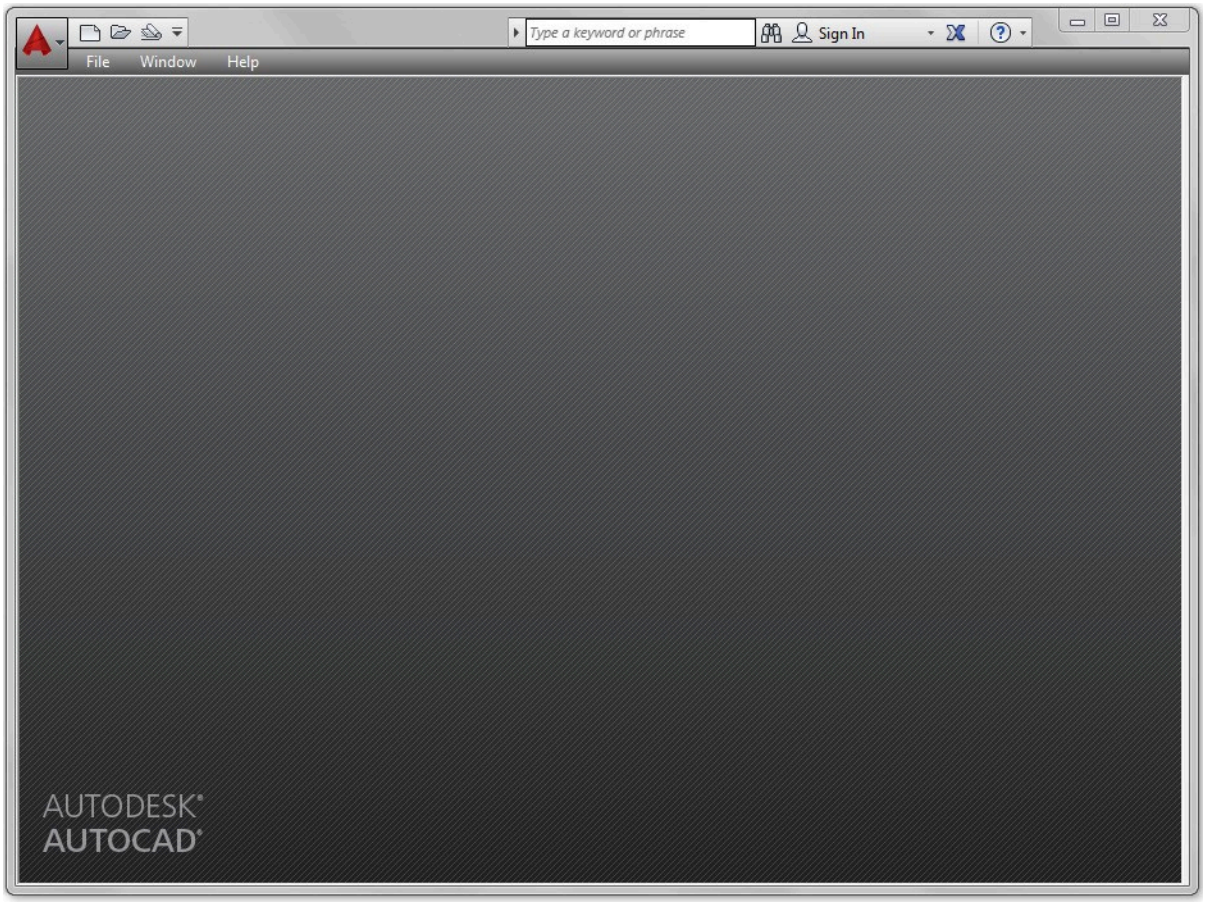


Figure Step 1

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

Step 2

Click the New icon. This will open the Select Template dialogue box. (Figure Step 2)

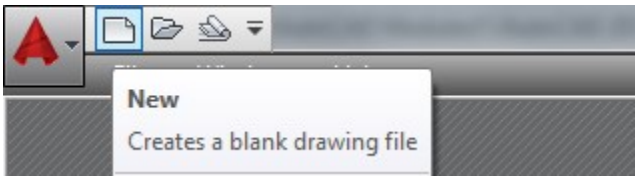


Figure Step 2

Step 3

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

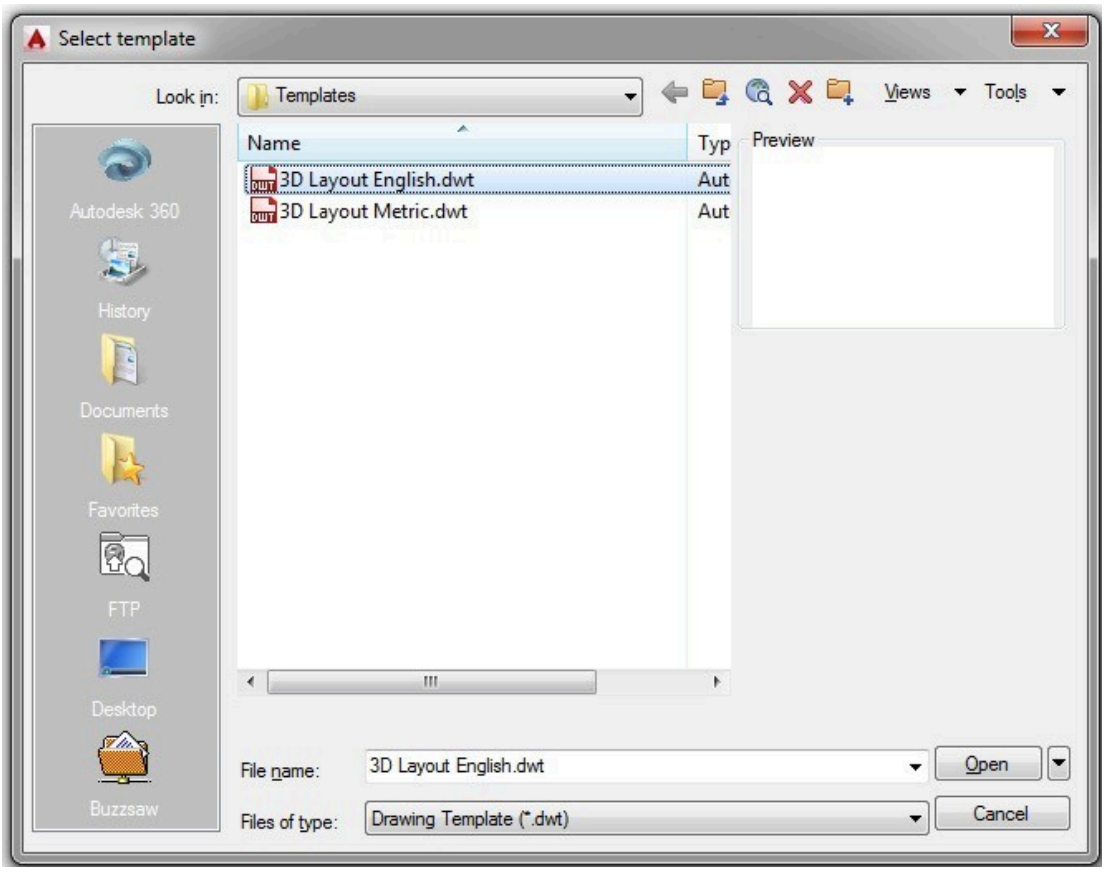


Figure Step 3

AUTHOR'S COMMENTS: The two templates are part of the AutoCAD 3D book and must be used to complete all workalongs and lab exercises.

Step 4

Click the small arrow in the Workplace Switching icon 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 4A and 4B)

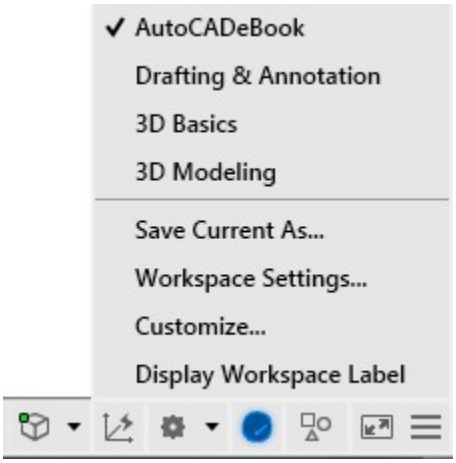


Figure Step 4A

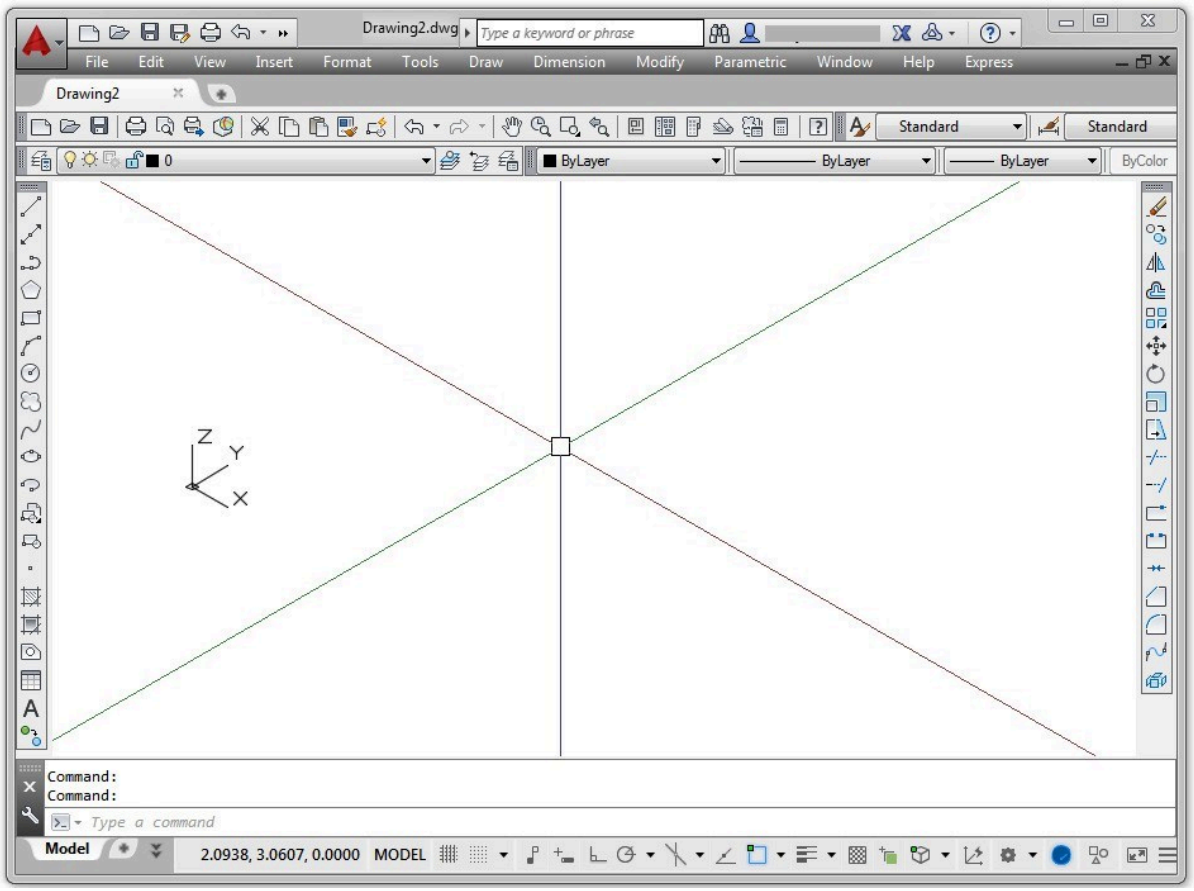


Figure Step 4B

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

Step 5

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

Figure Step 5

AUTHOR'S COMMENTS: Unless you are instructed otherwise, the Model tab should be enabled at all times while working on workalongs and lab exercises in the AutoCAD 3D book.

Step 6

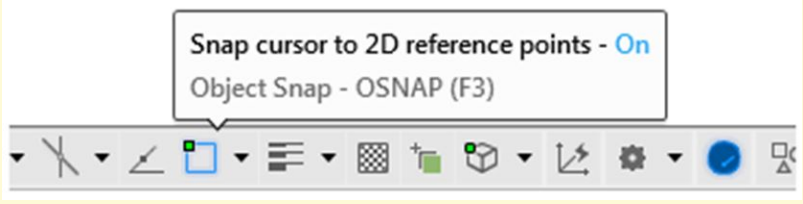
Disable all features on the Status bar by clicking any that display with a blue background as shown in Figure Step 6A. All features should display with a gray background as shown in Figure Step 6B. (Figure Step 6A and 6B)

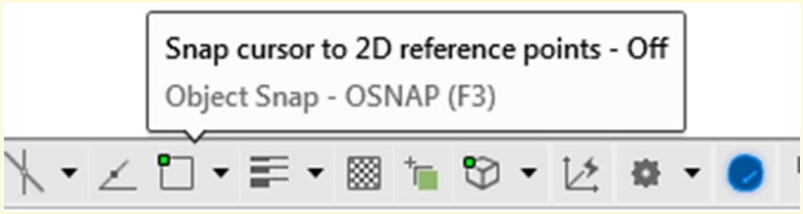
Figure Step 6A

Figure Step 6B

AUTHOR'S COMMENTS: **Important** Unless otherwise instructed, each time you start a workalong or lab exercise while working on the AutoCAD 3D book, all features on the Status bar should be disabled as shown in Figure Step 6B.

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 below.





Step 7

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: Layers, Orbit, Properties, Standard, Styles, UCS, UCS II, View, and Visual Styles are the only ones enabled. Click on the name to toggle the display of the toolbar. (Figure Step 7)

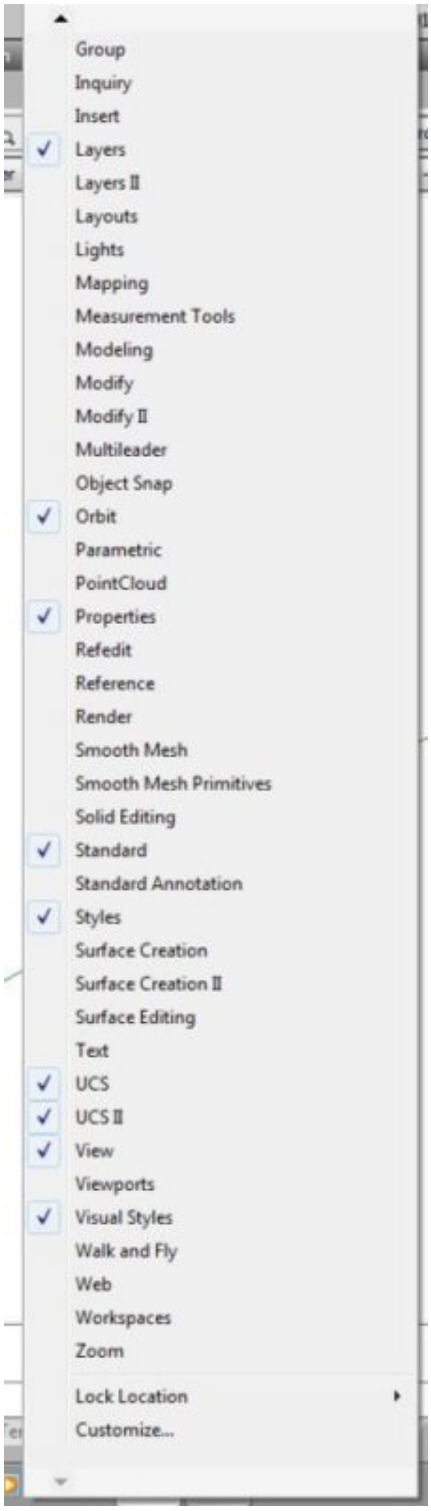


Figure Step 7

Step 8

Your Graphic screen should match the figure very closely. (Figure Step 8)

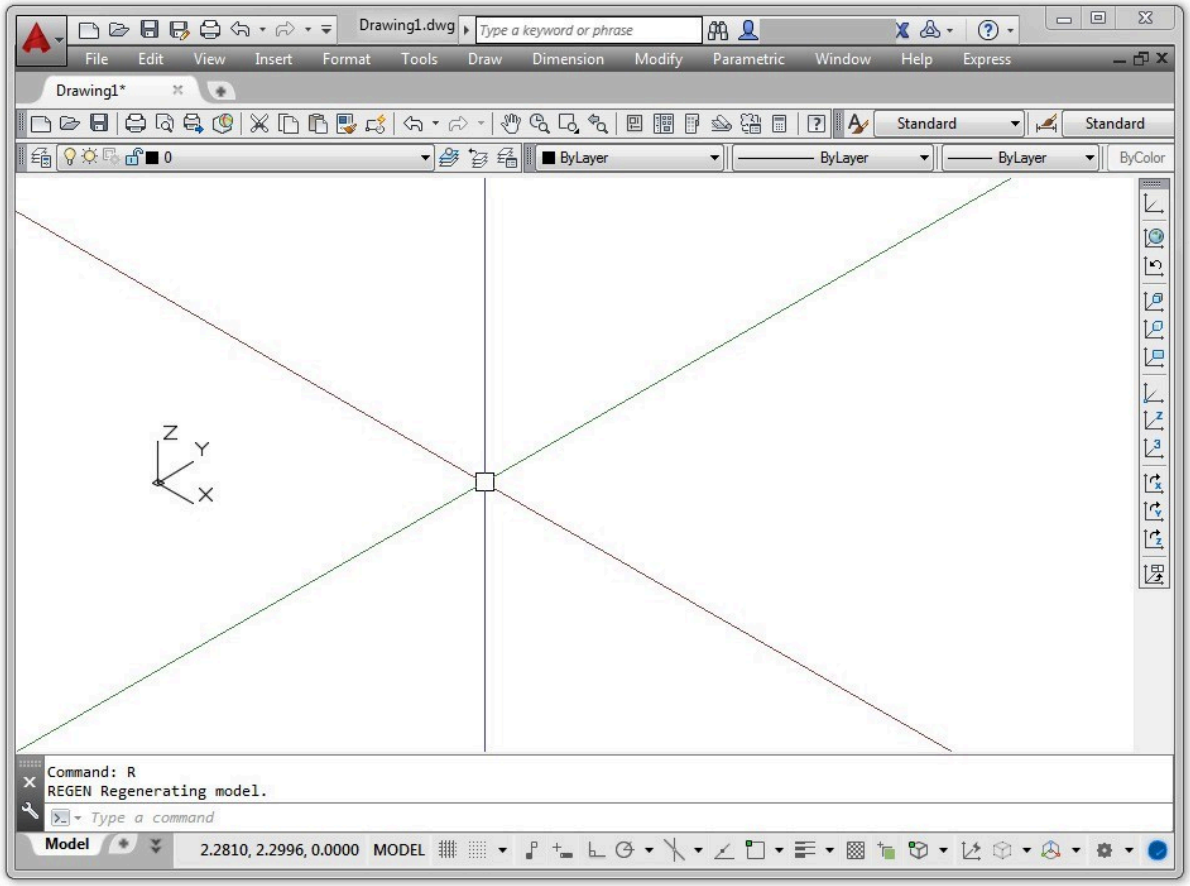


Figure Step 8

Step 9

Enter the MENUBAR command. If is set to 0, set it to 1, as shown below.

Command: **MENUBAR**

Enter new value for MENUBAR <0>: 1

Command:

AUTHOR'S COMMENTS: The MENUBAR system variable controls the display of the pull-down menu.

Step 10

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 10)

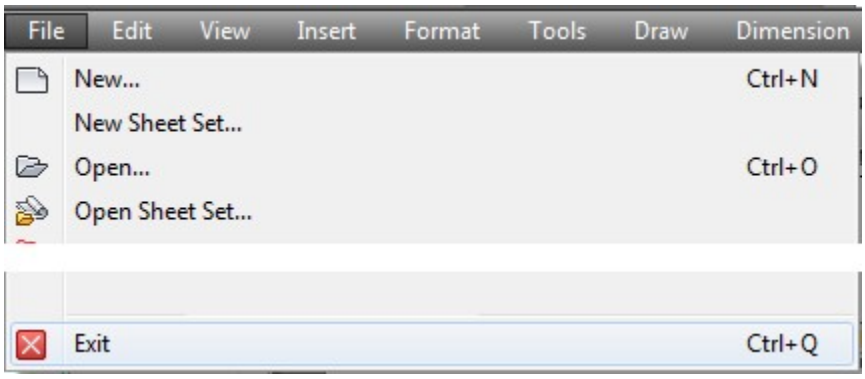


Figure Step 10

Step 11

Skip to Viewing in 3D

WORK ALONG: Setting AutoCAD's Interface Using Ribbon Menus

If you are using Toolbar menus, go to Viewing in 3D

Step 1

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

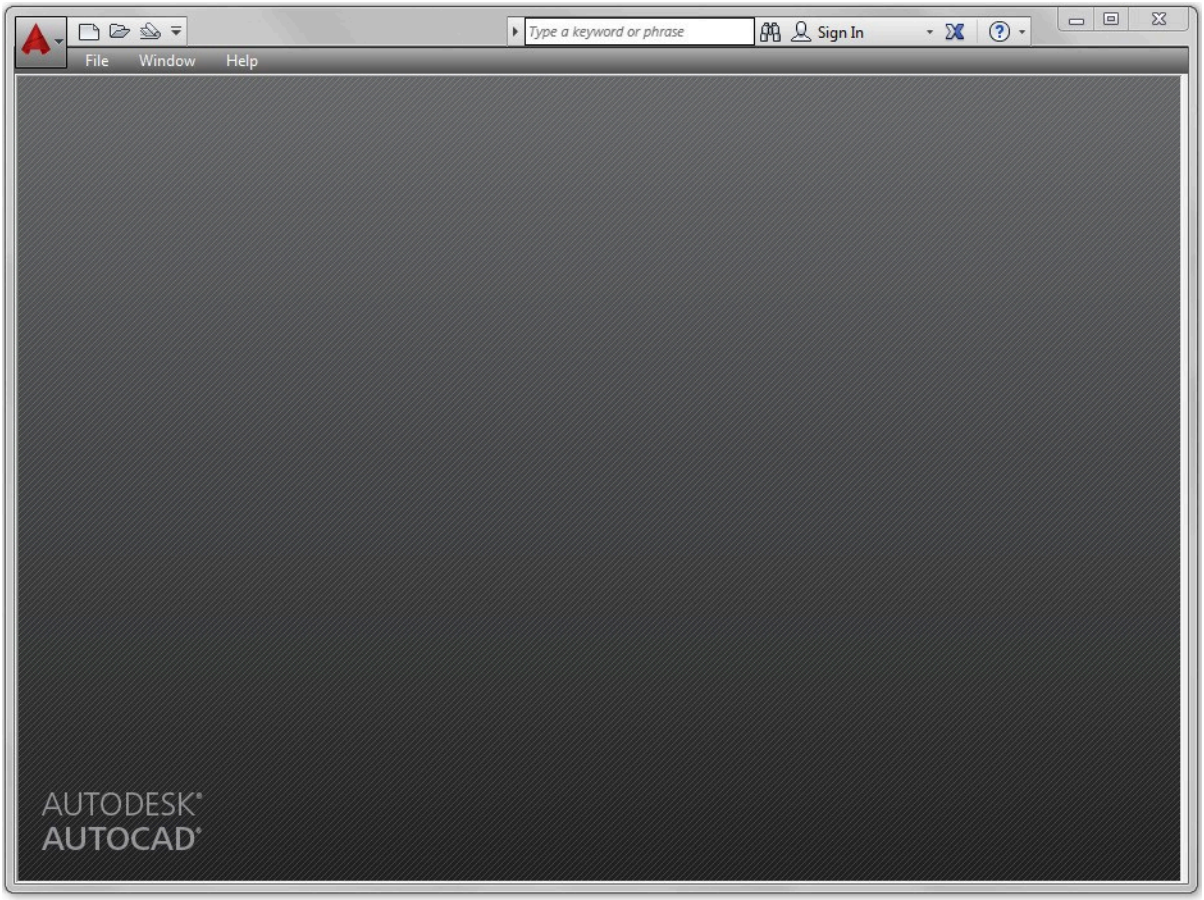


Figure Step 1

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

Step 2

Click the New icon. This will open the Select Template dialogue box. (Figure Step 2)

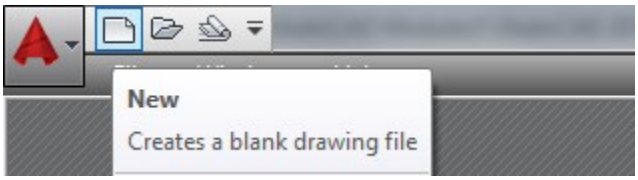


Figure Step 2

Step 3

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

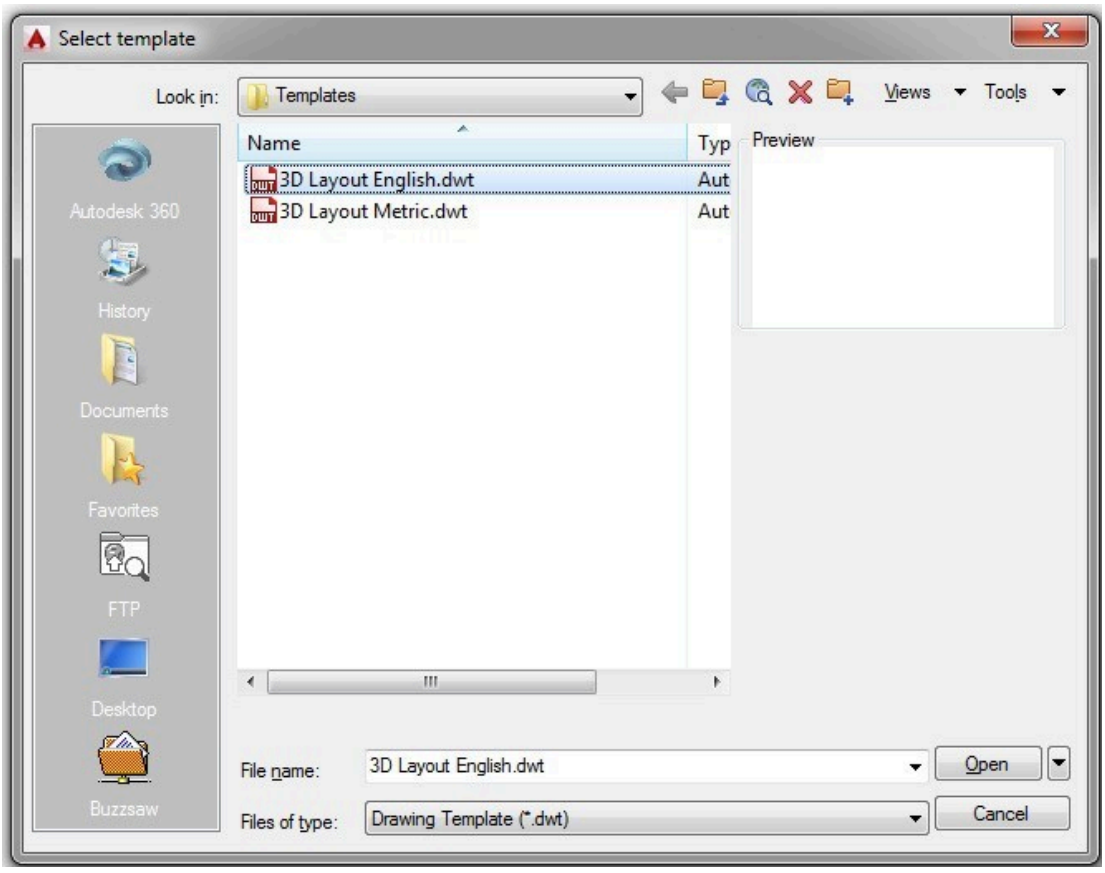


Figure Step 3

AUTHOR'S COMMENTS: The two templates are part of the AutoCAD 3D book and must be used to complete all workalongs and lab exercises.

Step 4

Click the small arrow in the Workplace Switching icon on the Status bar. Click the workspace: 3D Modeling to set it as the current workspace. Ensure that the check mark is beside the workspace: 3D Modeling as shown in the figure. (Figure Step 4A and 4B)

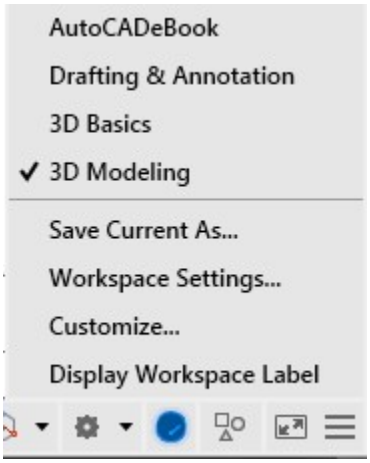


Figure Step 4A

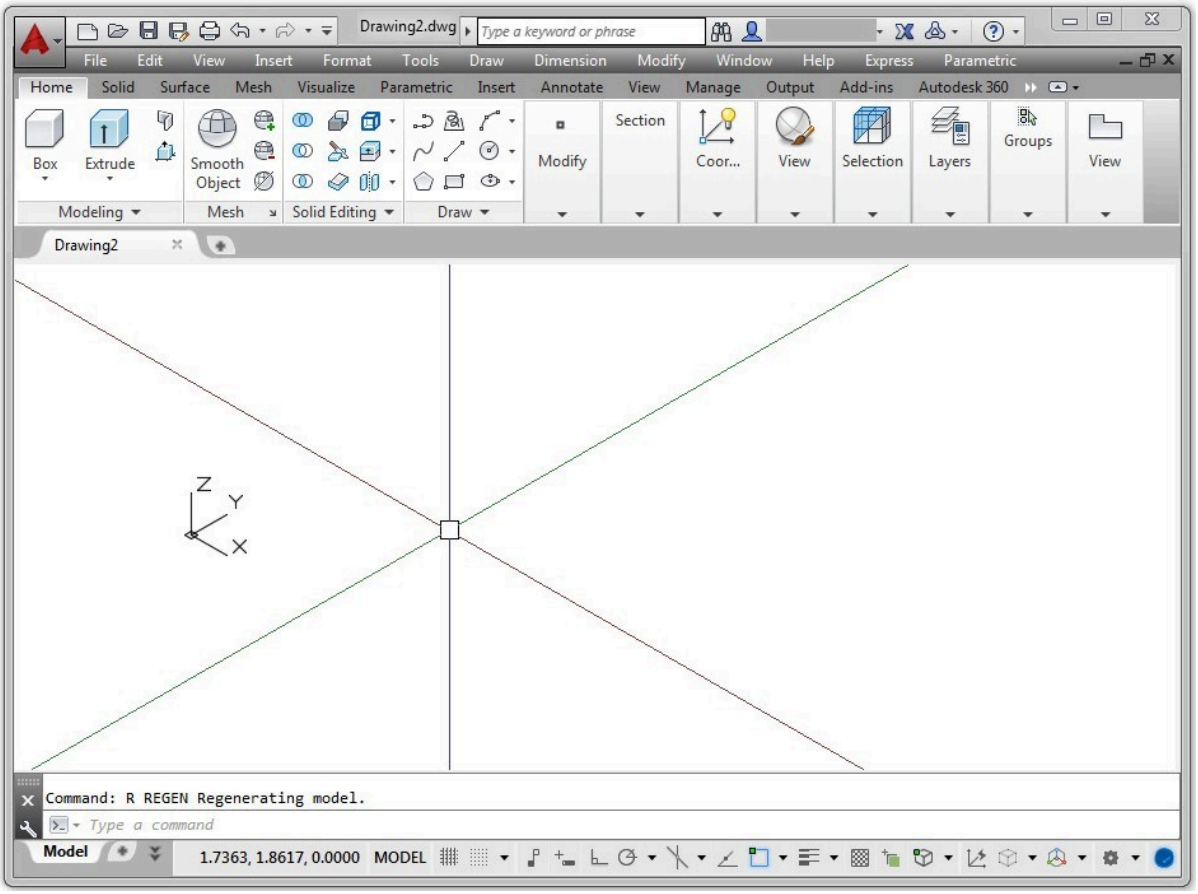


Figure Step 4B

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

Step 5

Enter the MENUBAR command and if is set to 0, set it to 1, as shown below. Command: **MENUBAR**

Enter new value for MENUBAR <0>: 1

Command:

AUTHOR'S COMMENTS: The MENUBAR system variable controls the display of the pull-down menu.

Step 6

Check to ensure that the tab Model is enabled. If it is not enabled, click it with the left mouse button. (Figure Step 6)

Figure Step 6

AUTHOR'S COMMENTS: Unless you are instructed otherwise, the Model tab should be enabled at all times while working on workalongs and lab exercises in the AutoCAD 3D book.

Step 7

Disable all features on the Status bar by clicking any that display with a blue background as shown in Figure Step 7A. All features should display with a gray background as shown in Figure Step 7B. (Figure Step 7A and 7B)

Figure Step 7A

Figure Step 7B

AUTHOR'S COMMENTS: **Important** Unless otherwise instructed, each time you start a workalong or lab exercise while working on the AutoCAD 3D book, all features on the Status bar should be disabled as shown in Figure Step 7B.

Step 8

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 8)

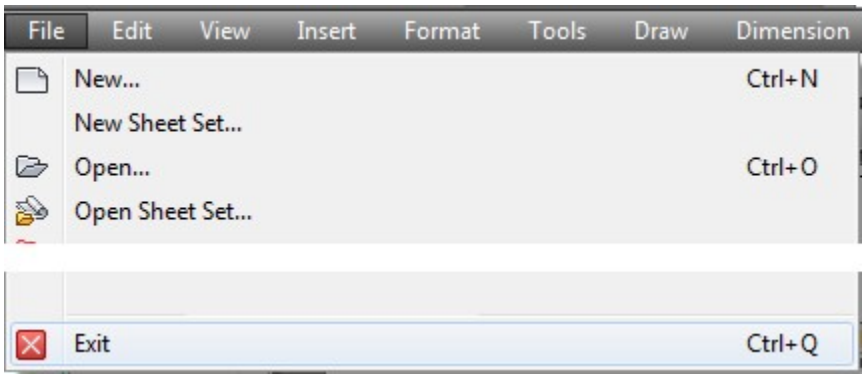
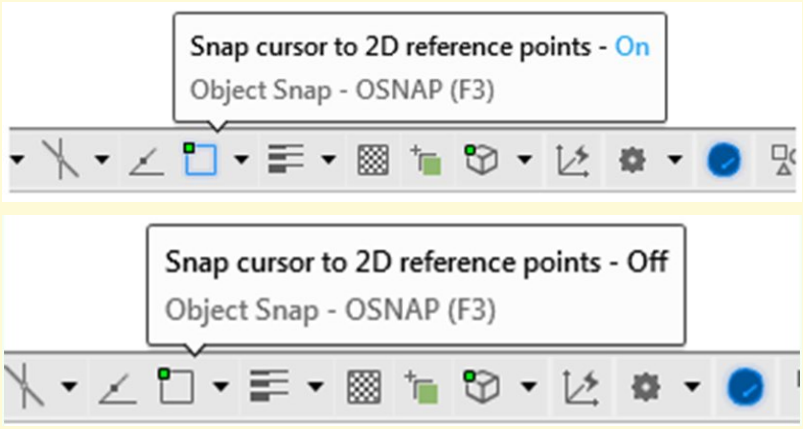


Figure Step 8

USER TIP: You can easily check to see if a feature on the Status bar is enabled or disabled by holding yourgraphic 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 on the right.



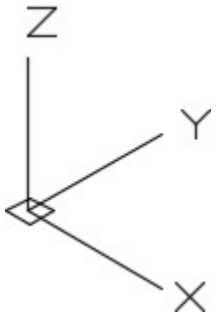
Viewing in 3D

While working in 2D (two dimensions), the model was always viewed looking down from the top. The only viewing adjustment made, when looking at the model, was the distance (zoom) and the location (pan). There was no need to change the viewing angle. Working in 3D (three dimensions) is quite different. While the model can still be zoomed and panned, you can change the viewing direction and angle of the model.

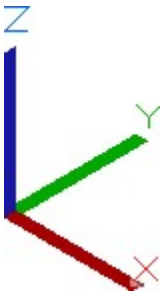
Throughout the AutoCAD 2D book, the objects being drawn and modified were called the drawing objects even though they were actually model objects on the top view of a model. The reason this was done was to keep it simple and allow you to concentrate on learning to draw and modify geometry. In the AutoCAD 3D book, all objects that are drawn in model space will be referred to as model objects and all objects drawn in paper space will be called drawing objects.

UCS Icon

The *User Coordinate System (UCS) icon* displays the location and orientation of the current coordinate system. Its location, which is controlled by you, is at either the current coordinate system origin point or in the lower-left corner of the current viewport. When constructing a model, it is best to locate it at the origin point as taught in Modules 3 and 4. Figure 1-2 shows the two different icons as they would appear in a 3D viewing orientation.



The 2D Wireframe UCS Icon



The 3D Wireframe UCS Icon
Figure 1-2
Coordinate System Icons

A Wireframe Model

A *model* is the 3 dimensional object drawn at full size and located accurately in relation to model space origin or X0Y0Z0 in the World UCS. A *wireframe model* is a real-world 3D object represented by lines, circles, arcs, and/or plines located along the edge of the model. See Figure 1-3. A wireframe model is hollow in the centre. That is the reason that lines and arcs on the opposite side of the model can be seen. Think of it as the skeleton of an object. The same model when surfaced or constructed as a solid model will appear as shown in Figure 1-4.

Surfacing a wireframe model and constructing a solid model are covered in Modules 12 to 30.

Wireframe models can be used to view the model from any vantage point, generate standard views in a drawing, generate exploded and perspective views, plus can be used as a skeleton when constructing the surfaced model. A wireframe model cannot be shaded or rendered. A wireframe model can also be used to analyze spatial relationships, including the shortest distance between corners and edges and checking for interferences.

It is important to plan, organize, and construct your model using layers and colors. This will allow you to more easily visualize a complex model and differentiate between objects in various views.

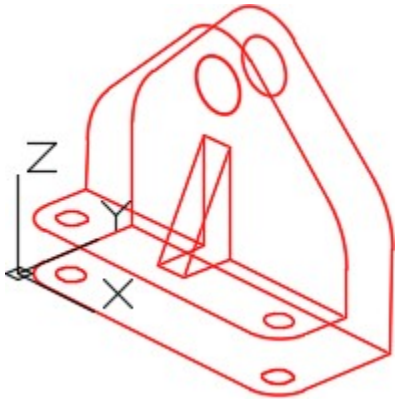


Figure 1-3
Wireframe Model

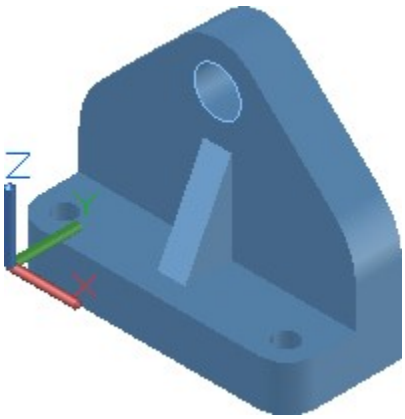


Figure 1-4
A Surfaced or Solid Model

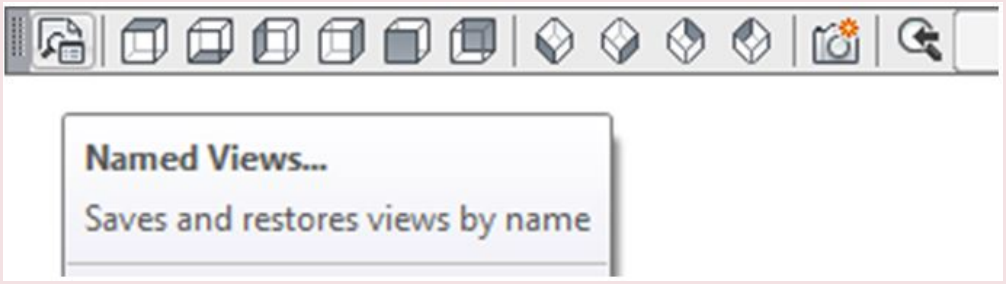
A View

A *view* is an area displayed by the viewing angle and direction and the location of the target. The *current view* is the view of the model that is being displayed in the Graphic window. There are many ways of setting the current view of the model and all of them will be covered throughout the AutoCAD 3D book. In this module, using the VIEW command is taught to either select a preset view or set and name user defined views. The basic principles of the orbit commands are also covered.

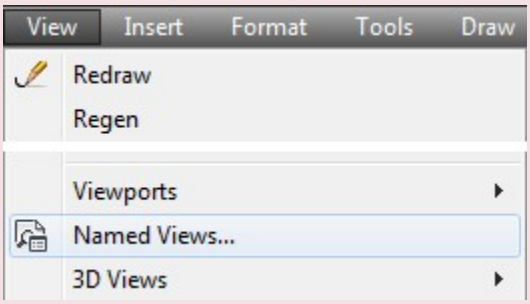
AutoCAD Command: VIEW

The VIEW command is used to display the preset views or to define, name, and display user defined views of the model.

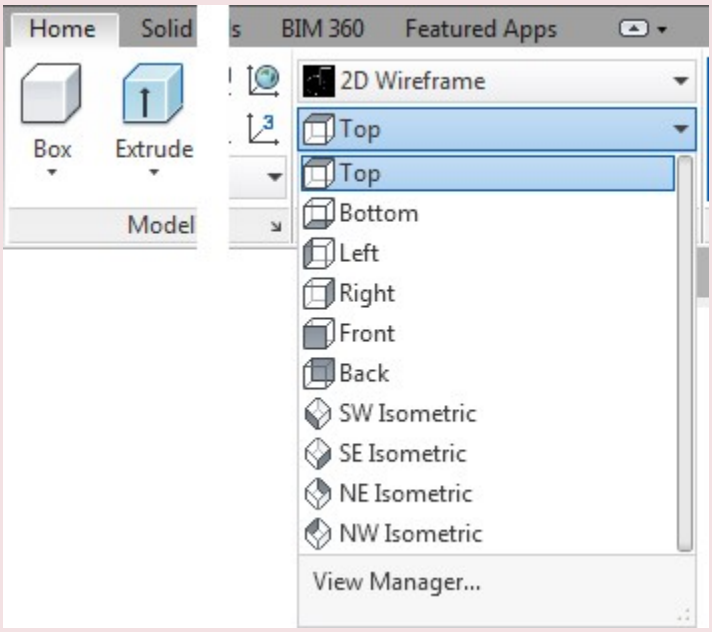
Shortcut: none



View Toolbar



View Pull-down

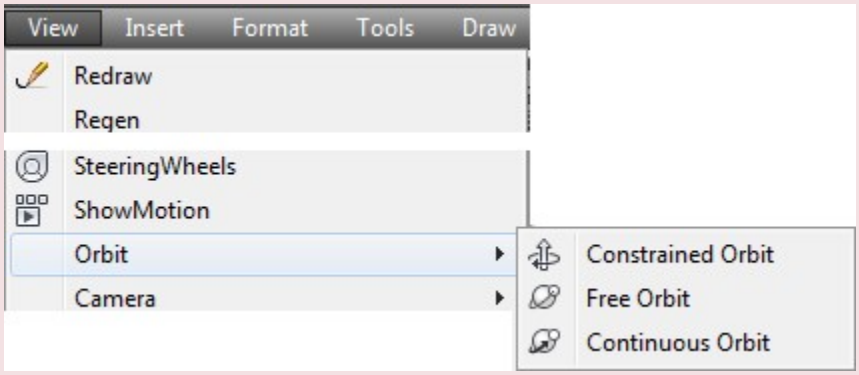


Home Ribbon

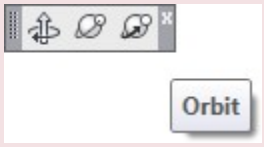
AutoCAD Commands: 3DORBIT, 3DFORBIT, 3DCORBIT

The 3DORBIT, 3DFORBIT, and 3DCORBIT commands allows the user to dynamically orbit the model changing the viewing angle, viewing direction, and target of the view.

Shortcut: none



View Pull-down

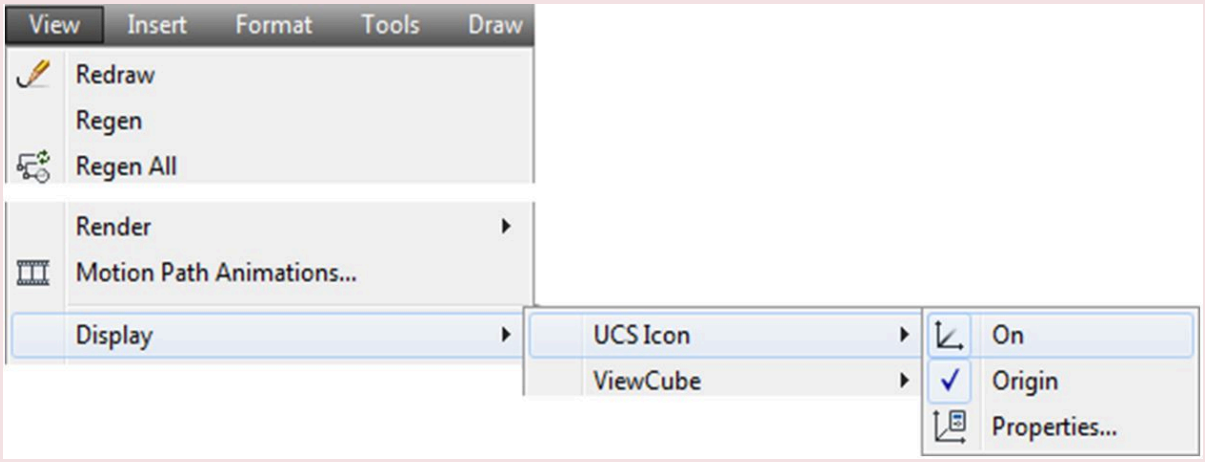


Orbit Toolbar

AutoCAD Commands: UCSICON

The UCSICON command is used to configure and control the display of the UCS icon.

Shortcut: none



View Pull-down

WORK ALONG: Using the UCSICON, VIEW, and 3DFORBIT Commands

Step 1

Using the NEW command, start a new drawing using the template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 01-1.

Step 3

In the View pull-down, select Display – UCS Icon – Properties. Ensure that both On and Origin enabled as shown in the figure. This will open the UCS Icon dialogue box. Ensure that the settings are the same in your dialogue box as they are in the figure. (Figure Step 3A and 3B)

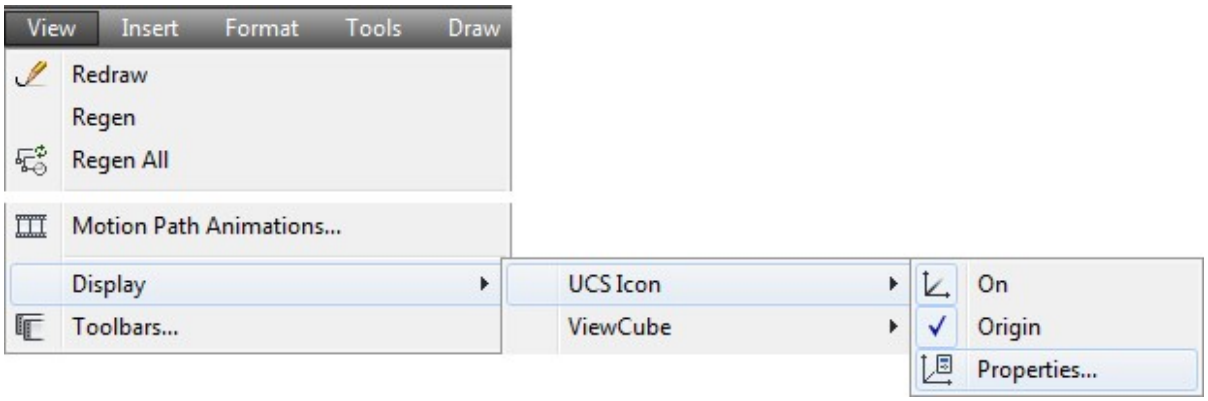


Figure Step 3A

AUTHOR'S COMMENTS: When Origin is enabled, the UCS icon will display at the current X0Y0Z0 location.

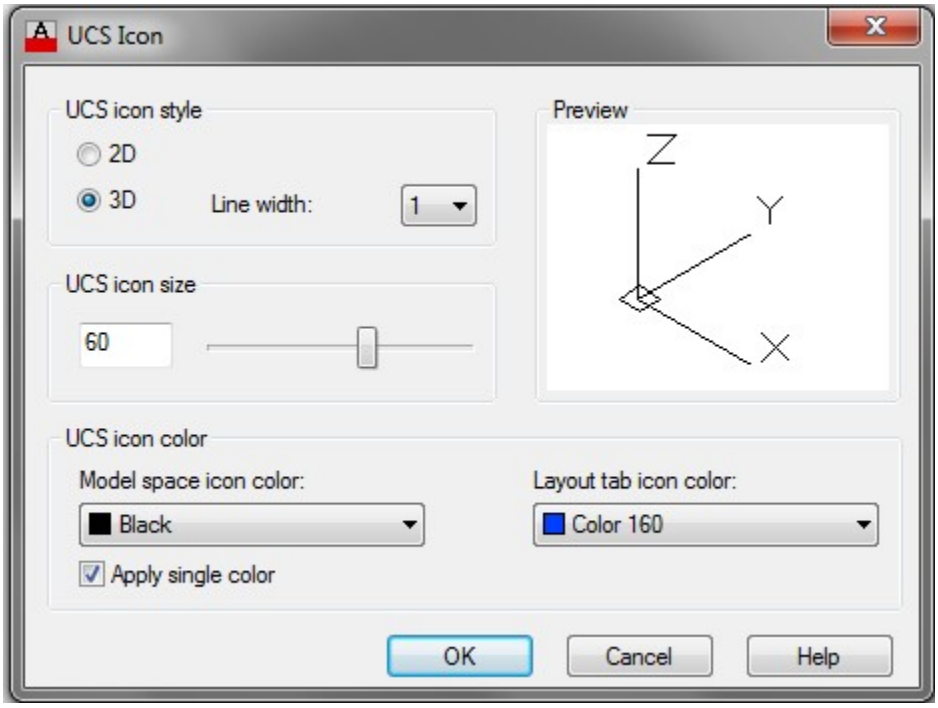


Figure Step 3B

Step 4

Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Inches. Using the INSERT command, insert the block: AutoCAD 3D Lab 01-1 at the coordinates 0,0,0. Explode the block that you just inserted. Your model should appear as shown in the figure. (Figure Step 4)

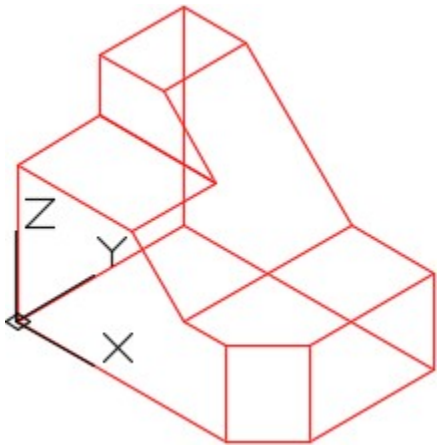


Figure Step 4

Step 5

If you are using Toolbar menus, use the View toolbar or if you are using Ribbon menus, use the View section of the Home tab to complete Steps 7 and 8. (Figure Step 5A and 5B)

Figure Step 5A

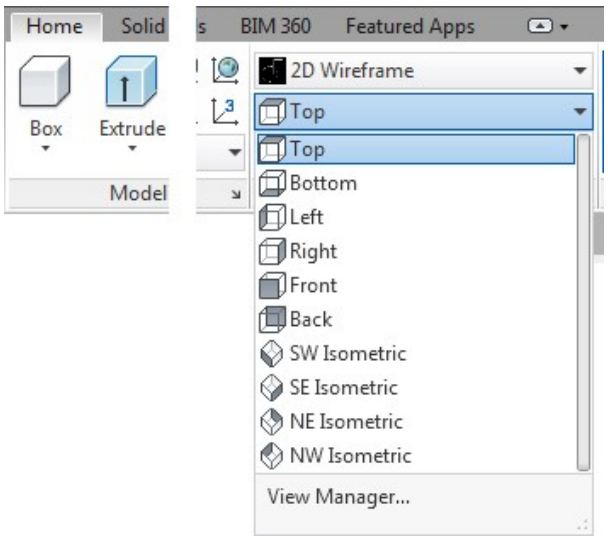


Figure Step 5B

Step 6

Enter the system variable UCSORTHO, as shown below, and set it to 0.

Command: **UCSORTHO**

Enter new value for UCSORTHO <1>: **0**

Command:

AUTHOR'S COMMENTS: This is an important step. Once it is set, it will keep that setting on your computer until you change it. It should be set to 0 while you are working on the AutoCAD 3D book. You can check its setting by entering the system variable again as shown below.

Step 7

Click the Top view icon. Note how the model is now being viewed from the top as shown in the figure. Continue to change the view to display the ten predefined views. Note how the view of the model changes each time you click an icon. (Figure Step 7A, 7B, 7C, 7D, 7E, 7F, 7G, 7H, 7I, and 7J)

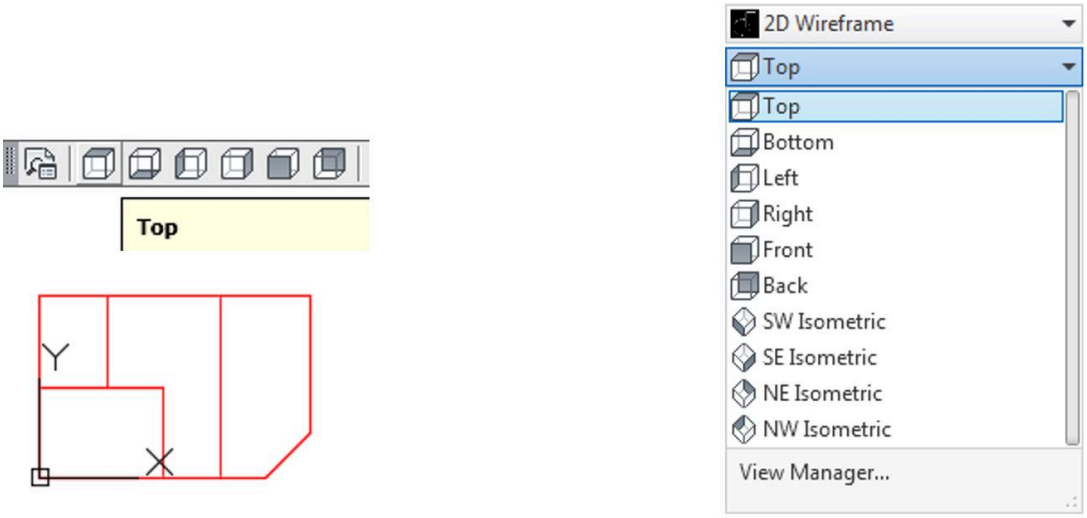


Figure Step 7A

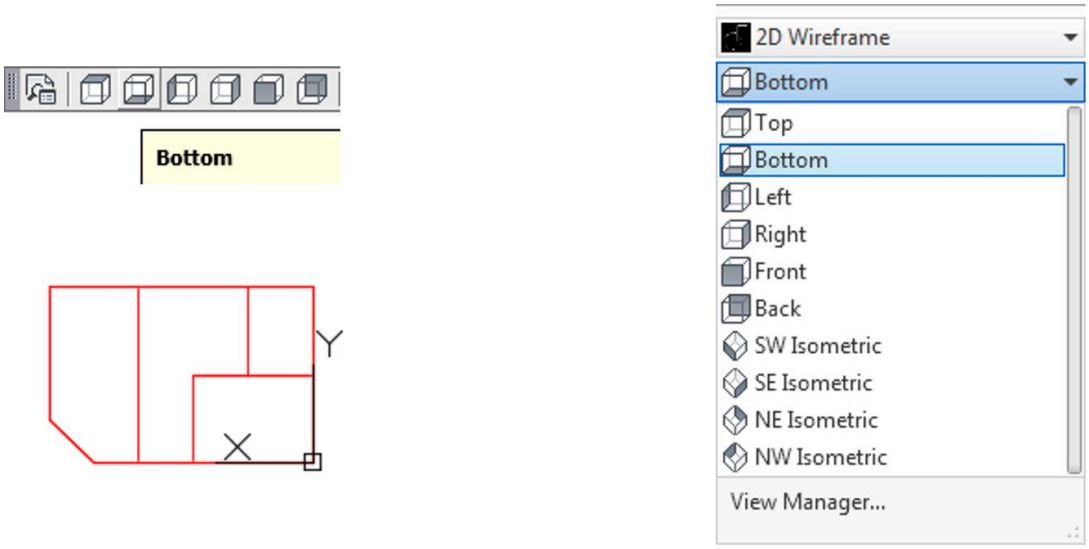
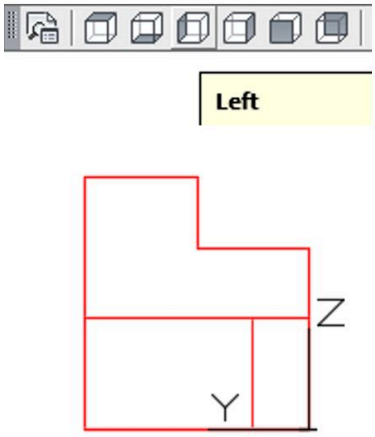


Figure Step 7B



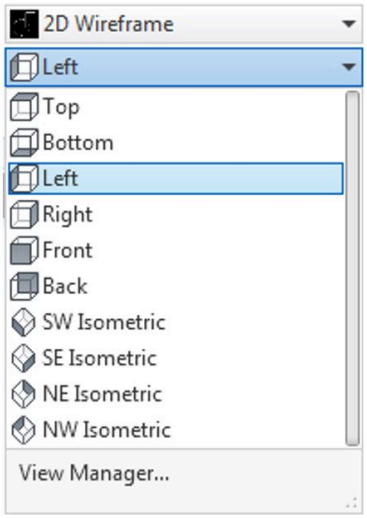
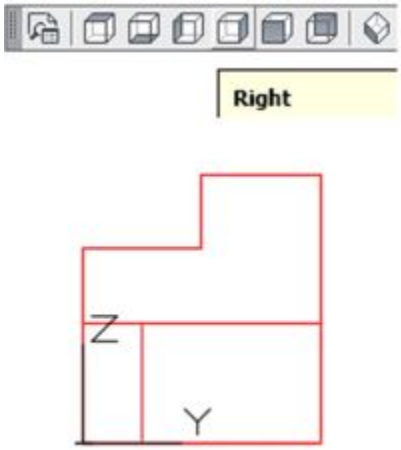


Figure Step 7C



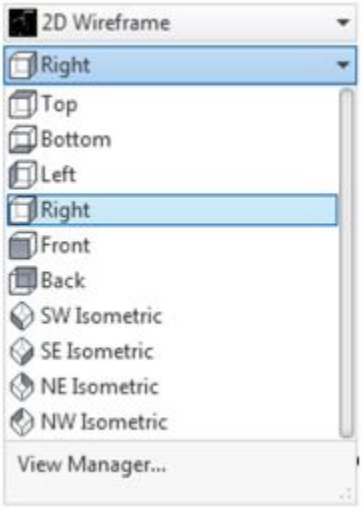
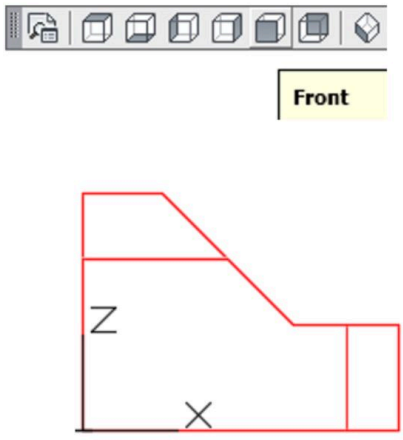


Figure Step 7D



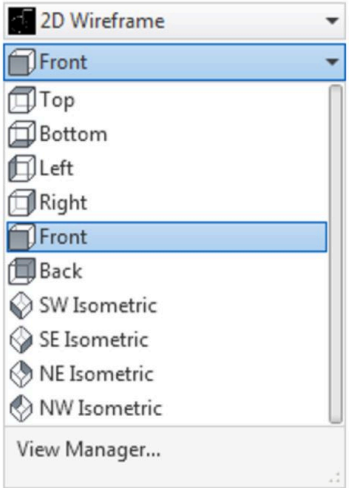
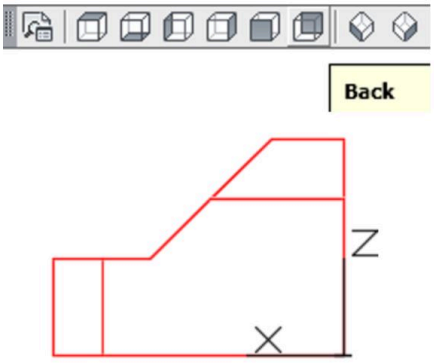


Figure Step 7E



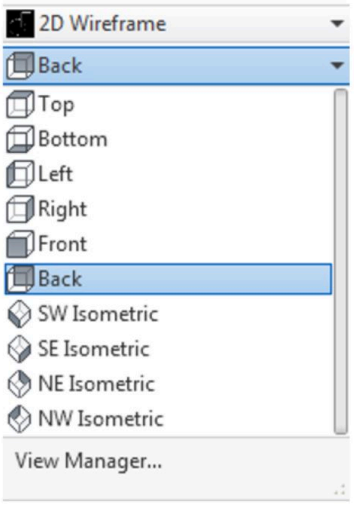
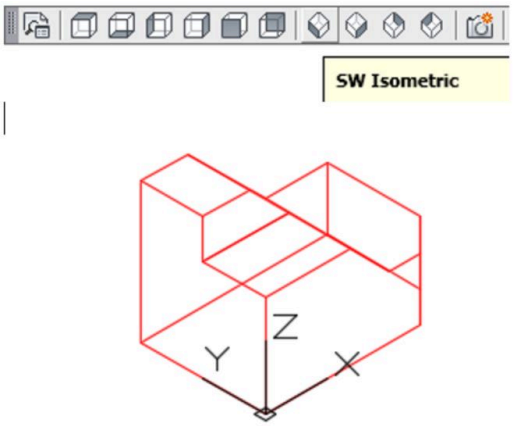


Figure Step 7F



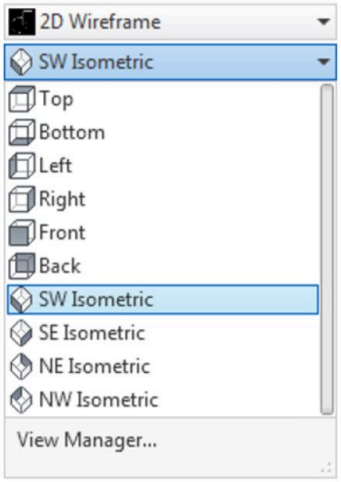
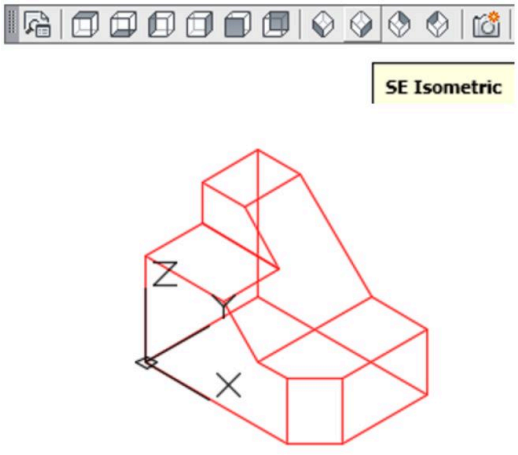


Figure Step 7G



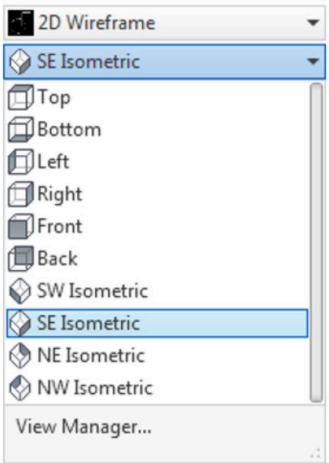
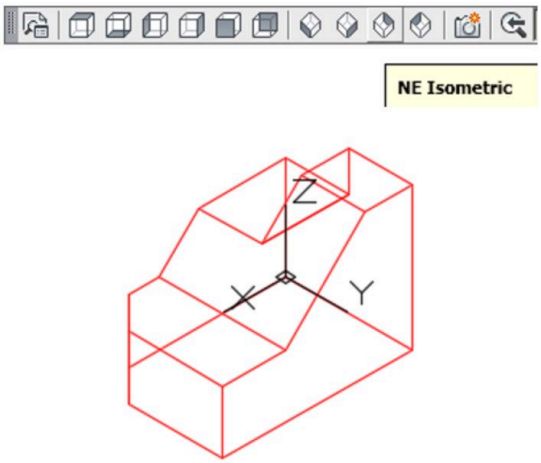


Figure Step 7H



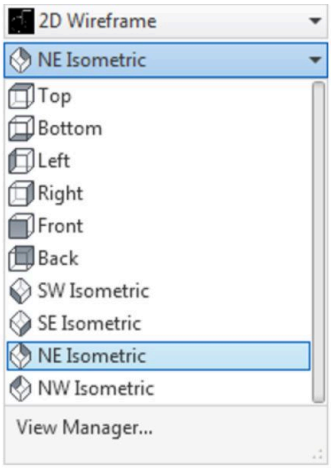
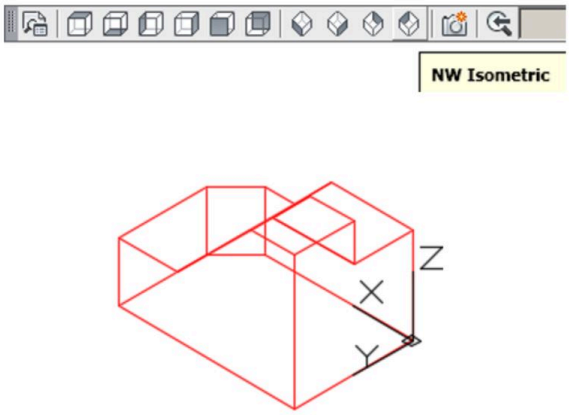


Figure Step 7I



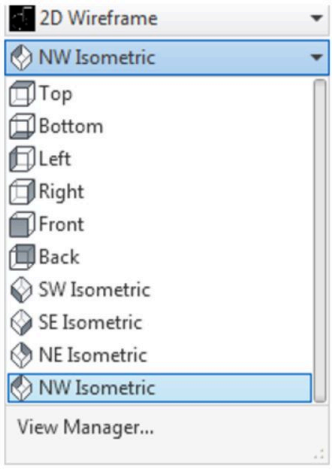


Figure Step 7J

Step 8

Change the current view to SE Isometric as shown in the figure. (Figure Step 8)

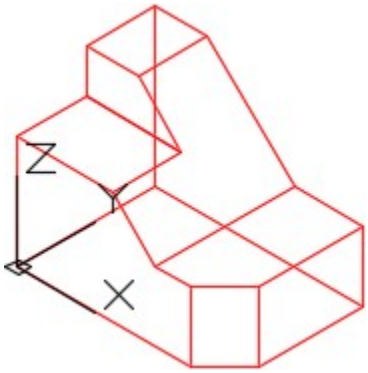


Figure Step 8

AUTHOR'S COMMENTS: The SE Isometric view is the Home view used in the AutoCAD 3D book and is the view that you can return to anytime you require to a mental view of the model.

AUTHOR'S COMMENTS: The 3DFORBIT command activates and displays the arcball. The *arcball* allows you to dynamically orbit the model changing the viewing angle, viewing direction and target. The arcball displays a small circle at each quadrant. See Figure 1-5

Depending on the location of the cursor when the left mouse button is held down, the model will orbit around a different axis. If the cursor is located:

1. outside the arcball the Z
2. on the upper or lower circles the X
3. on the left or right circles, the Y axis
4. inside the arcball, the XYZ

In this module, only the basic principles of the 3DFORBIT command are covered. Play with this command and practice orbiting the model around each of the axis. After entering the command, move the cursor to the desired location press and hold down the left mouse button and move the cursor. Press Esc to exit the command. Any change can be reversed with the U command.

Be careful when orbiting the model. It is very easy to over-orbit it and lose your mental view of the model. Orbit it slowly and deliberately keeping a mental view of the model throughout. If you lose the mental view of the model, you can always exit the orbit and change the view to SE Isometric and then start the orbit again. The 3DFORBIT command will be covered in greater detail as you work your way through the AutoCAD 3D book.

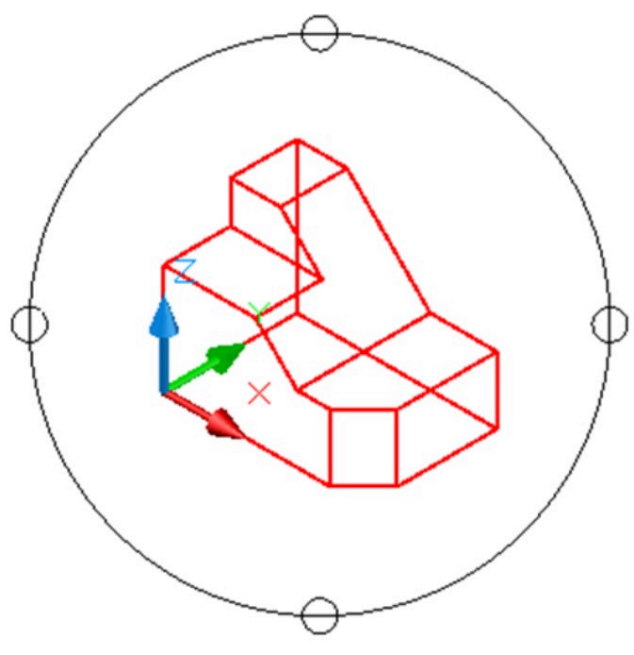


Figure 1-5
The Arcball

MUST KNOW: The SE Isometric view is the Home view used in the AutoCAD 3D book. It is the view that you can return to anytime you require to a mental view of the model.

Step 9

Enter the 3DFORBIT command using either the toolbar or the pull-down menu. It will display the arcball around the model. Zoom and pan the model to locate it in the centre of the arcball. (Figure Step 9A, 9B, and 9C)

Figure Step 9A

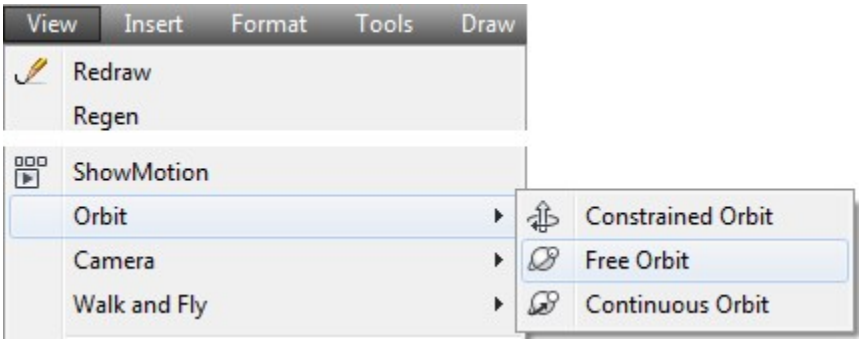


Figure Step 9B

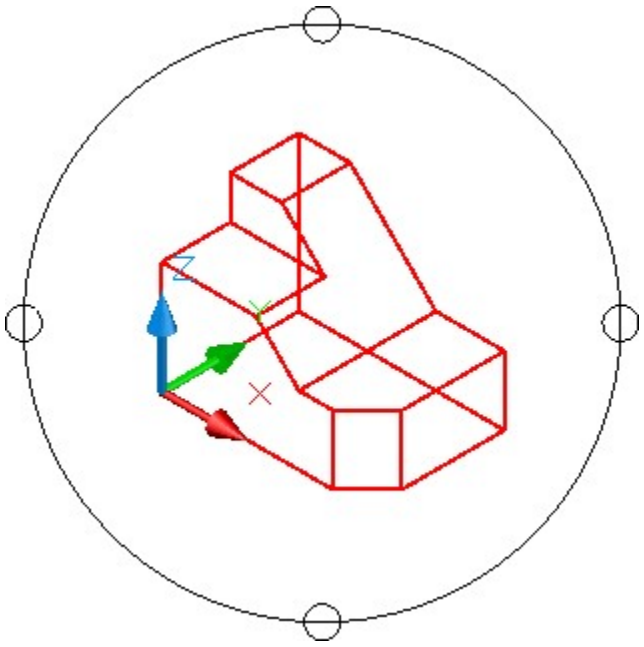


Figure Step 9C

AUTHOR'S COMMENTS: If you have a wheel on your mouse, you can use it to zoom and pan the model to centre it in the arcball.

Step 10

Move the cursor onto the small circle on the left side. Press and hold down the left mouse button and while holding it down, move the cursor slowly to the right. until your model appears similar to the figure. You do not have to move it very far. (Figure Step 10)

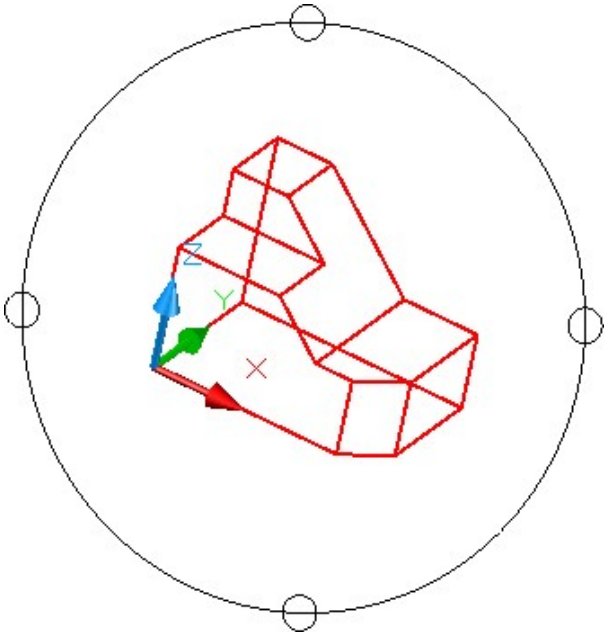


Figure Step 10

Step 11

Move the cursor onto the small circle on the bottom. Press and hold down the left mouse button and while holding it down, move the slowly up until your model appears as shown in the figure. (Figure Step 11)

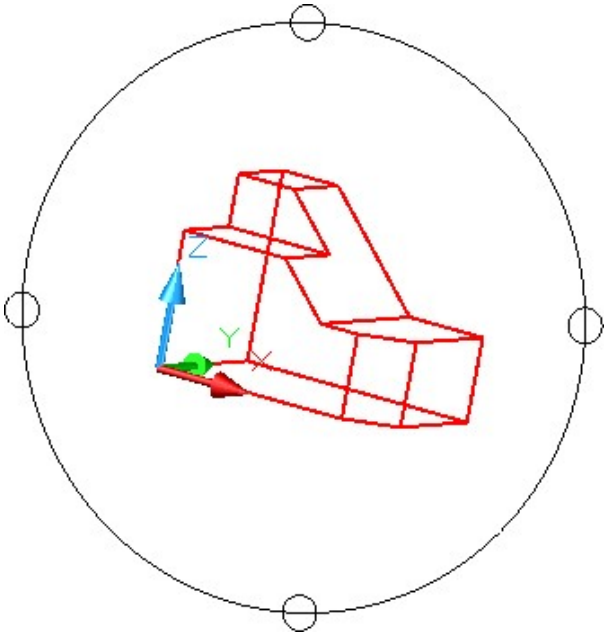


Figure Step 11

Step 12

Move the cursor inside the arcball. Press and hold down the left mouse button and while holding it down, move it until the model appears as shown in the figure. (Figure Step 12)

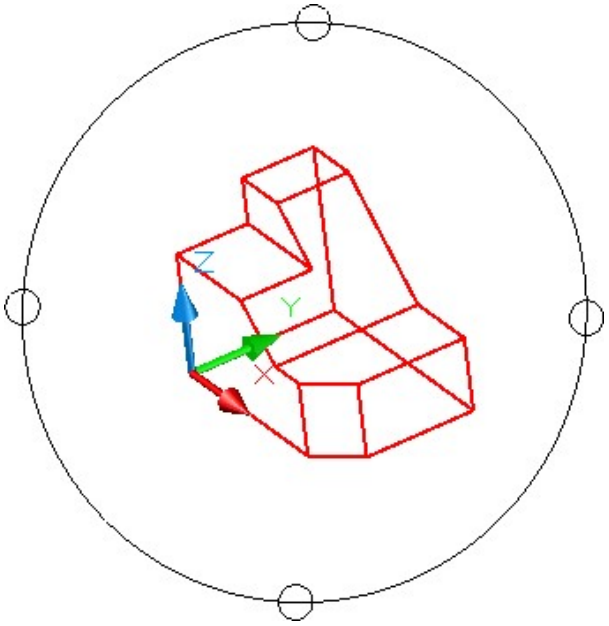


Figure Step 12

Step 13

Press Esc to exit the command. Change the current view to SE Isometric. (Figure Step 13)

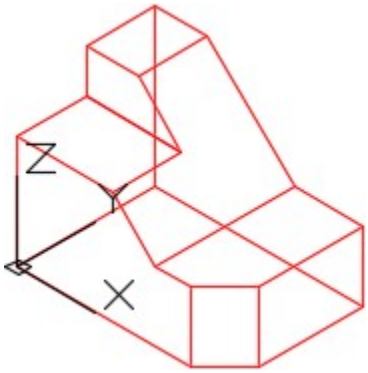


Figure Step 13

Step 14

Using 3DFORBIT, orbit the model to appear similar to the figure. It does not have to match exactly. (Figure Step 14)

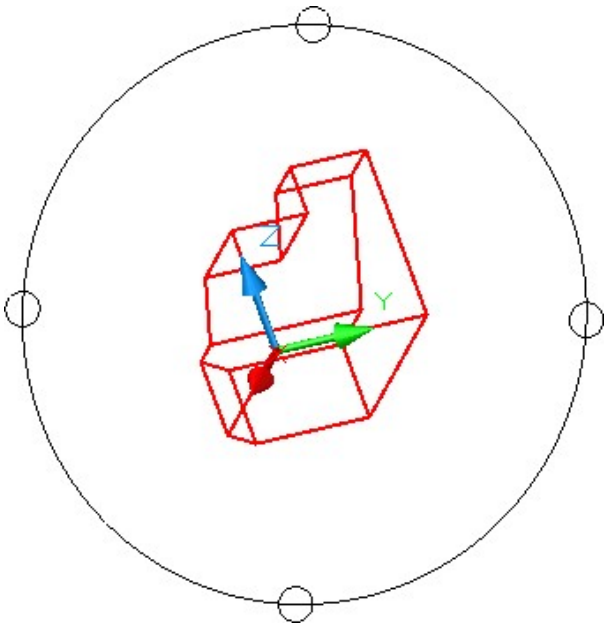


Figure Step 14

Step 15

Enter the VIEW command using either the toolbar or ribbon. This will open the View Manager dialogue box. (Figure Step 15A and 15B)

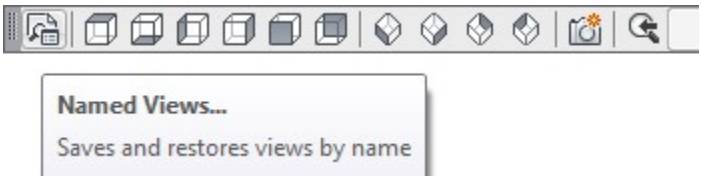


Figure Step 15A

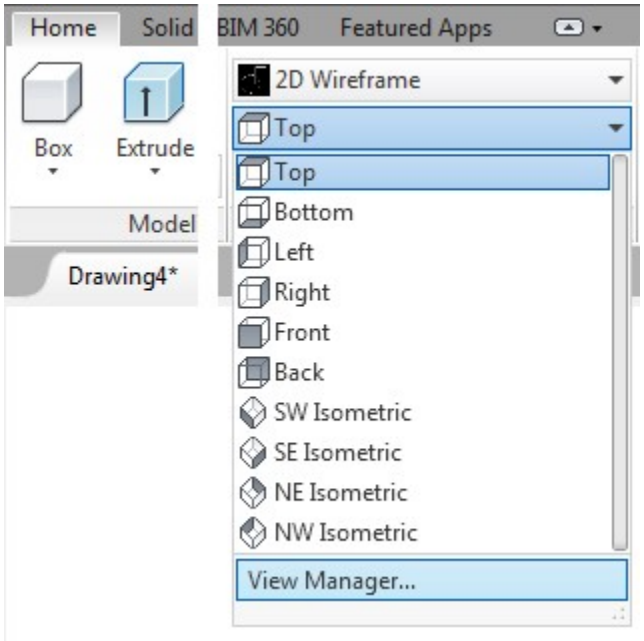


Figure Step 15B

AUTHOR'S COMMENTS: Views of the model can be named and saved for future use. The views can be easily restored as required.

Step 16

In the Views box, click Model Views to highlight it as shown in the figure and then click the New icon. This will open the New View dialogue box. In the View name box, enter the name: Module 1 View. Click OK to see the view name as shown in the figure. Click OK to close the dialogue box. (Figure Step 16A, 16B, and 16C)

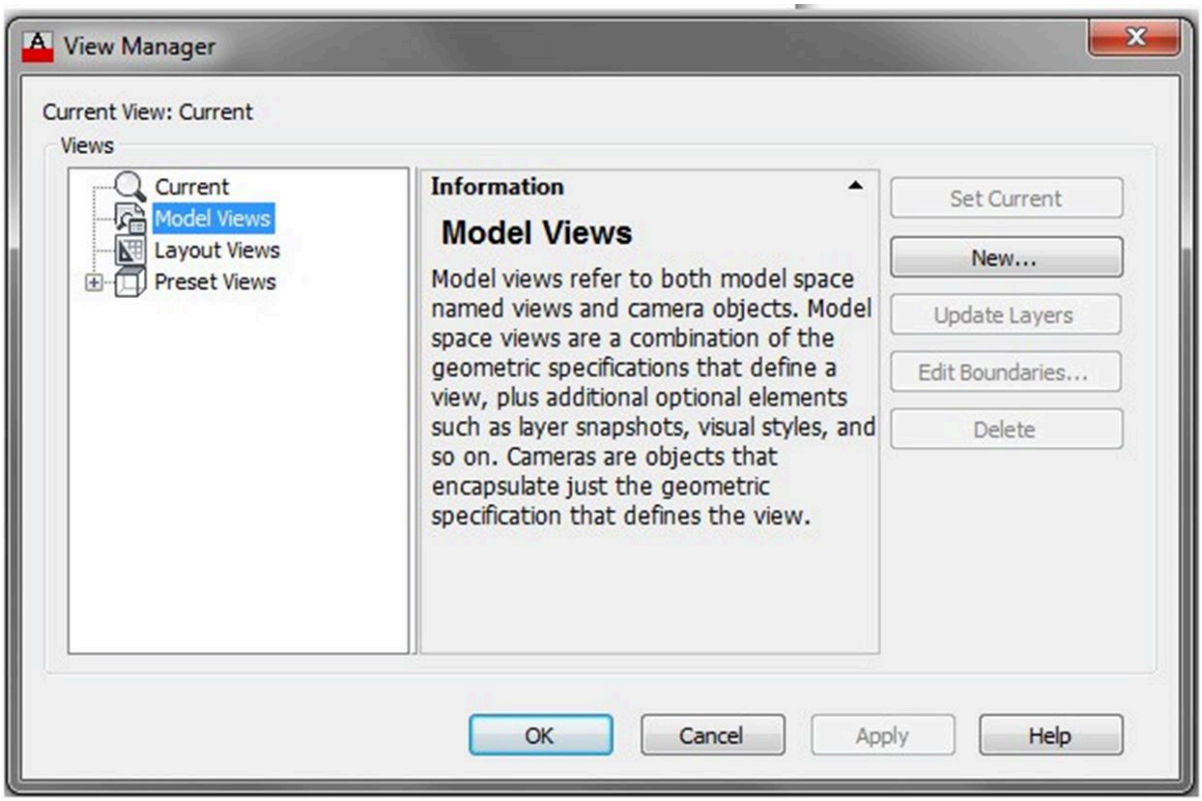
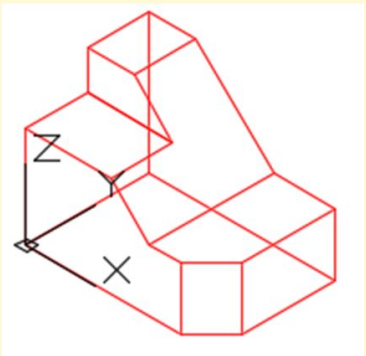


Figure Step 16A

USER TIP: When constructing 3D wireframe models, it is best to view the model in a position that best allows geometry to be inserted and edited. When first beginning to construct 3D models, it is best to start with the SE Isometric view (the Home view) and then slightly orbit it with one of the orbit commands. Do not over-orbit it and lose your mental view of the model. At times, wireframe models can be hard to visualize since they can be visualized inside out. This is a skill that can be learned but in the beginning it is best to keep a mental view of the model.



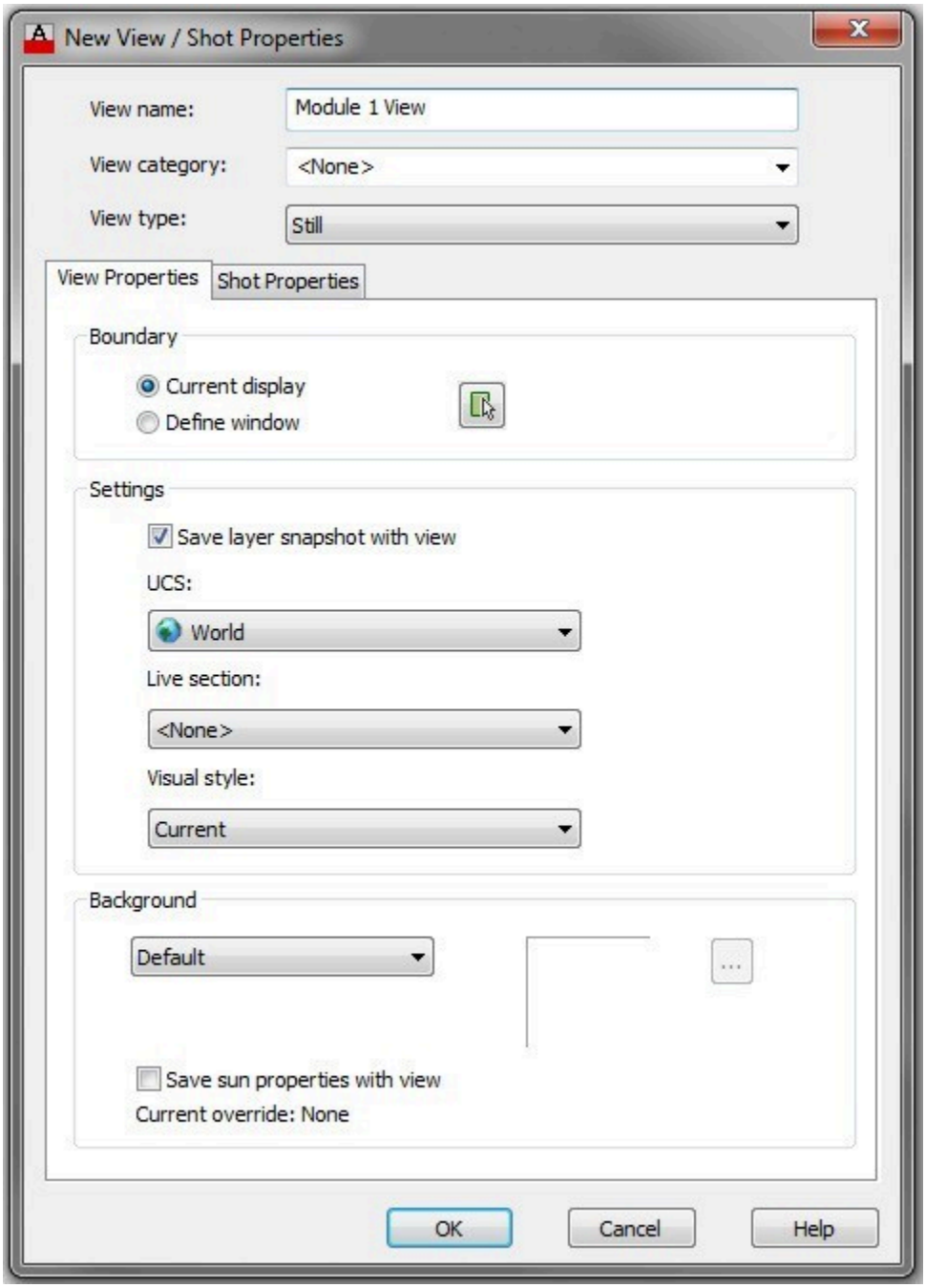


Figure Step 16B

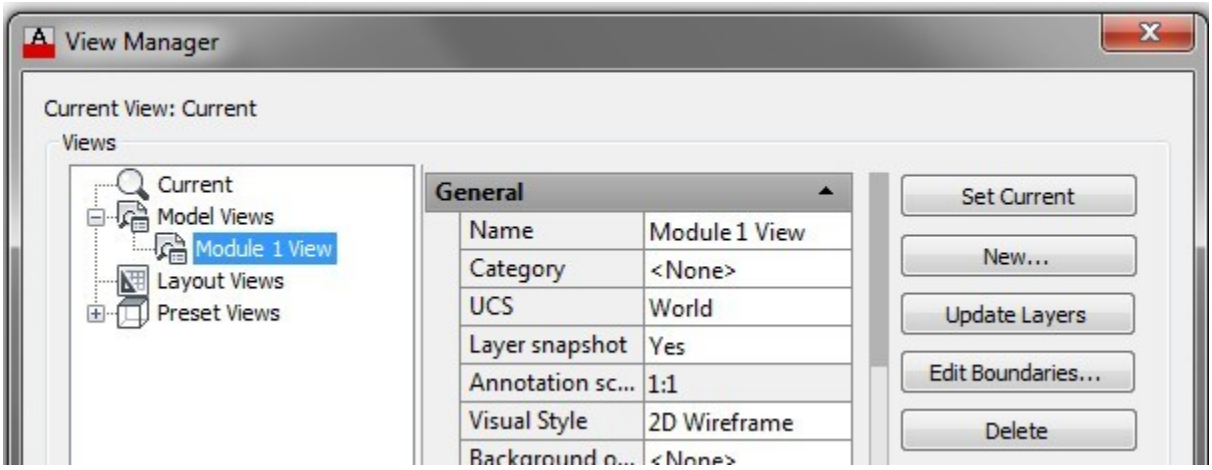


Figure Step 16C

Step 17

Change the current view to SE Isometric. (Figure Step 17)

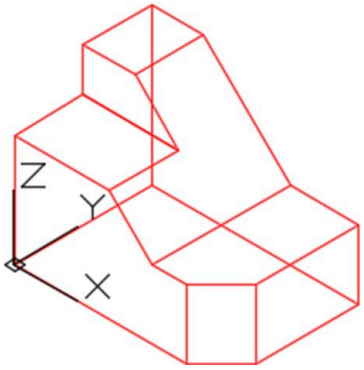


Figure Step 17

Step 18

Save and close the drawing.

MUST KNOW: Be careful when orbiting the model with the arcball. It is very easy to over-orbit and lose your mental view of the model. Orbit it slowly and deliberately keeping a mental view of the model throughout. If you lose the mental view of the model, exit the orbit command and change the view to SE Isometric. Then start the orbit command over again.

Key Principles

Key Principles in Module 1

1. A wireframe model is a real-world 3D object with lines, circles, arcs, and/or plines located along the edge of the A wireframe model is a hollow object.
2. A model is drawn in model space. It is a 3 dimensional object drawn at full size and located accurately in relation to model space origin X0Y0Z0 in the World UCS.
3. A view is simply the way you are looking at the model. It is the model displayed by the viewing angle, viewing direction, and the location of the target.
4. The SE Isometric view is the home view used to keep a mental view of the model.
5. Do not over-orbit the model using the orbit It best to retain a good mental view of the model at all times during model construction.

Module 2: Checking the Accuracy of Models

Learning Outcomes

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

1. Describe how the models drawn in lab exercises are checked for accuracy.

Checking the Accuracy of Models

It is extremely important in AutoCAD to draw with 100% accuracy. This cannot be stressed enough as it is essential that each model be positioned and sized perfectly. Models constructed in the lab exercises, in the AutoCAD 3D book, can be checked for accuracy by overlaying it with a *key*. As a beginner, this helps you know that the models are drawn accurately and shows you any part of the model that is drawn incorrectly. As you become more experienced, you will not require keys to check your models.

WORK ALONG: Checking the Accuracy of Models

Step 1

Using the NEW command, start a new drawing using the template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Lab 02-1.

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

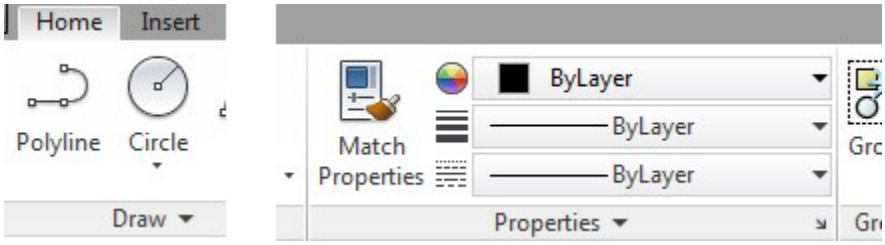
Ensure that all features on the Status bar are disable by clicking any that are enabled. Your Status bar should match the figure. (Figure Step 4)

Figure Step 4

Step 5

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

Properties Toolbar



Home Ribbon

Step 6

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

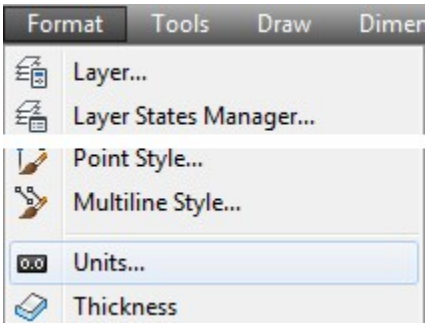


Figure Step 6A
Format Pull-down

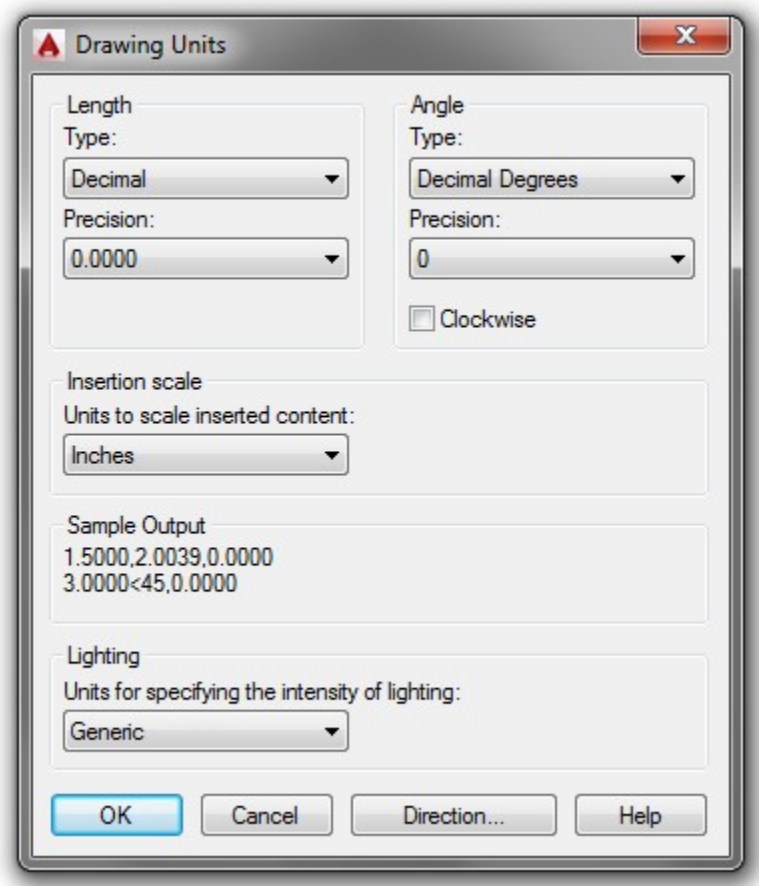


Figure Step 6B

Step 7

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

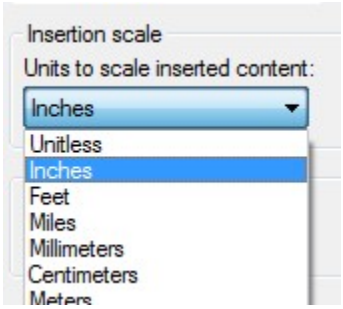


Figure Step 7

Step 8

Click the Insert pull-down and select Block. This will open the Insert dialogue box. (Figure Step 8)

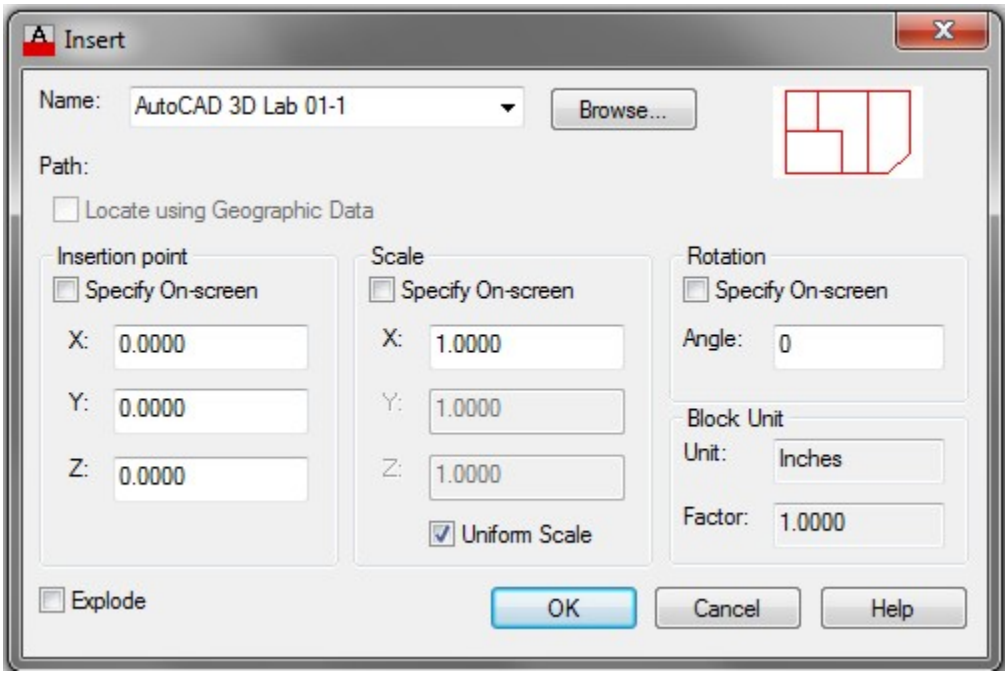


Figure Step 8

Step 9

Click the small triangle at the end of the Name box. It will pull-down the list of the block names contained in the drawing file. (Figure Step 9)

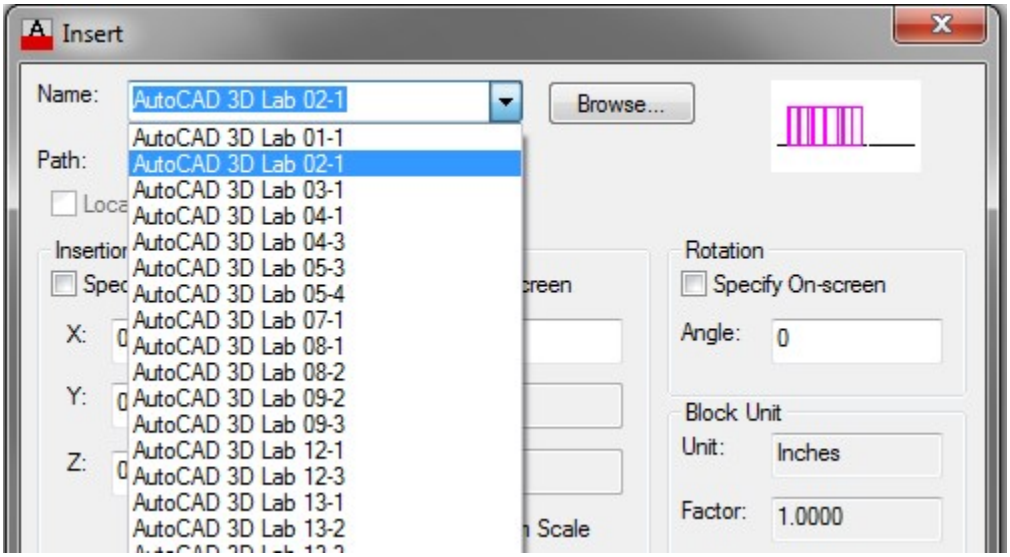


Figure Step 9

Step 10

Click the block: AutoCAD 3D Lab 02-1. It will highlight as shown in the figure. The name you select is always the name of the drawing for the lab exercise you are working on. (Figure Step 10)

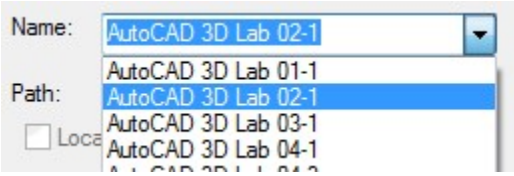


Figure Step 10

Step 11

Ensure that the three Specify On-screen boxes are disabled. (Figure Step 11A and 11B)

Figure Step 11A

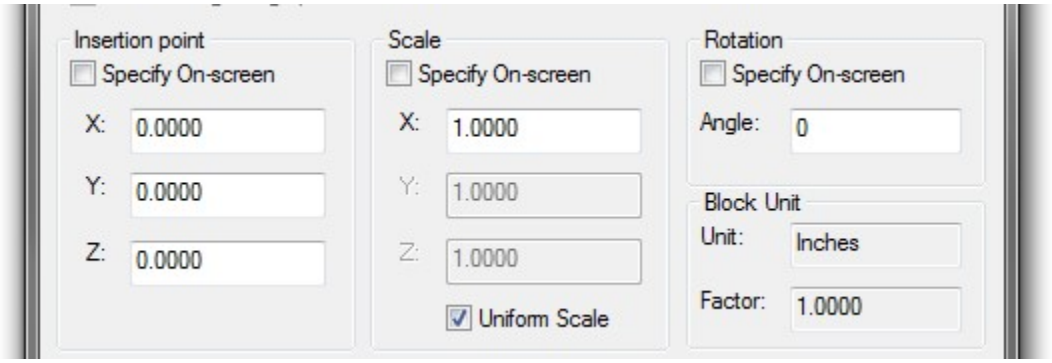


Figure Step 11B

Step 12

Click the OK button.

Step 13

This will insert a magenta colored overlay key on your model. If there double objects or places where the users model and the magenta model don't match, the users model is incorrect. If only one model displays, even though it may share the colors magenta and red, the model is accurate. The magenta key resides on layer: Key. If the model is incorrect, turn layer: Key off and correct the model before going on to the next lab exercise or module. After your model is corrected, check it again by turning layer: Key on. (Figure Step 13)

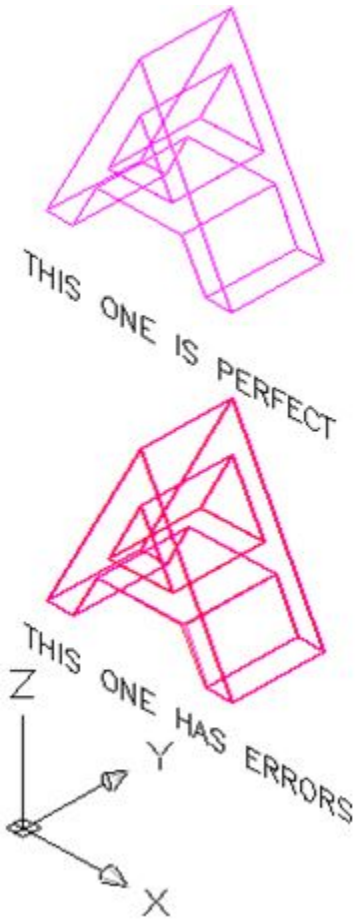


Figure Step 13

AUTHOR'S COMMENTS: This is just a demo, therefore, you are only learning the process. You will be checking your first model for accuracy in Module 3.

Step 14

Save and close the drawing.

Key Principles

Key Principles in Module 2

1. AutoCAD models must be drawn with 100% accuracy.
2. Most models that you will be drawing in the lab exercises in the AutoCAD 3D book can be checked for accuracy using an overlay key.
3. The name of the key that you select to check your model is always the name of the drawing for the lab exercise for that model.
4. After the key is inserted and only one model displays, the model is 100% accurate.

Module 3: World Coordinate System

Learning Outcomes

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

1. Describe the World Coordinate System, the UCS icon, and the right-hand rule.
2. Draw 3D models with the UCS located at the World Coordinate System.

The World Coordinate System

AutoCAD has two distinct three-dimensional coordinate systems: the World Coordinate System (WCS) and the User Coordinate System (UCS). The *World Coordinate System* is permanently located at the absolute coordinates X0Y0Z0. It is a fixed coordinate system which can never be moved. The WCS is normally not used to construct models. You will be using it in this module to construct models as learning tool only.

The *User Coordinate System* (UCS) is the coordinate system that is used to construct 3D models. The UCS can be placed exactly at the WCS or at any location in 3D space. In this module, model will be constructed with the UCS located at the WCS. It is essential to be able to locate and orientate the UCS to construct most models. This is taught in Module 4 and 5.

The UCS Icon

Figure 3-1 shows the UCS icon and the positive X, Y and Z directions indicated by the UCS icon. When constructing models, it is very important to know which direction is positive and negative on all three axis.

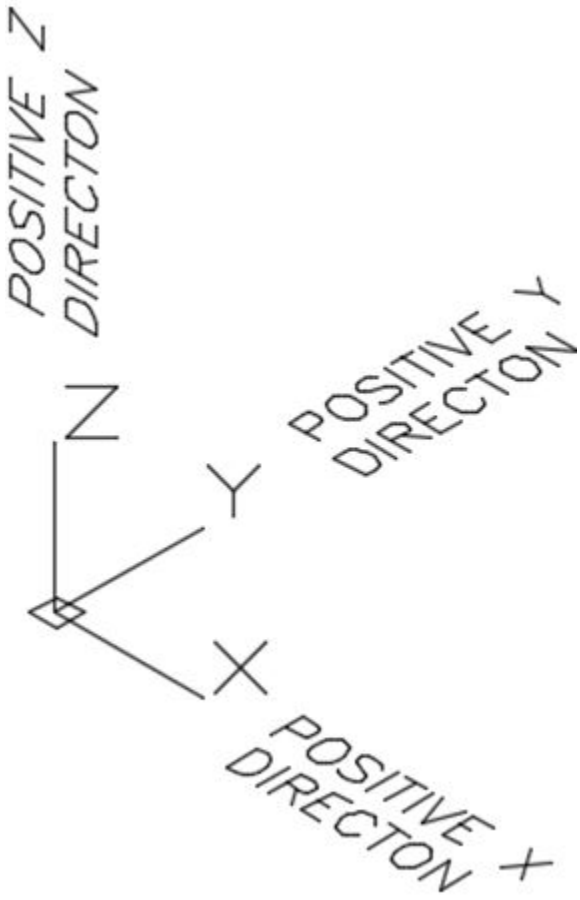


Figure 3-1
UCS Icon Coordinate Directions

When the UCS is located at the World Coordinate System, it will display a small square at the origin as can be seen in Figure 3-2. If it is located at any other location, it will display as shown in Figure 3-3.

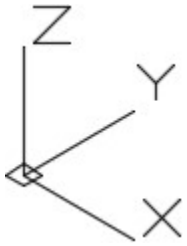


Figure 3-2
UCS at the World Coordinate System

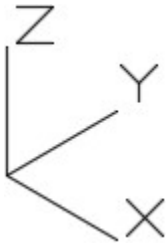


Figure 3-3
UCS Not at the World Coordinate System

The Right-Hand Rule

Even though the 3D UCS icon indicates the positive Z direction, it is important to know how it is obtained. AutoCAD uses the *right-hand rule* to find the positive Z direction. See Figure 3-4. To use the right-hand rule, you must first know the positive X and Y directions of the current UCS.

Using your right hand, point your thumb in the direction of the positive X axis. Extend your index finger in the direction of the positive Y axis. The middle finger indicates the direction of the positive Z axis. By rotating your hand, the X, Y and Z axes will rotate to change the UCS location and orientation. It is important to be able to visualize how and where to move the UCS as you construct more complicated models.

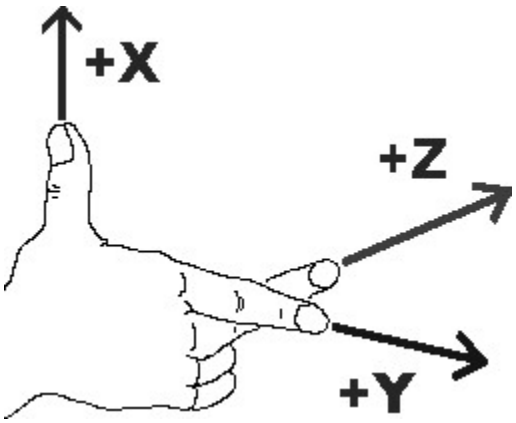


Figure 3-4
The Right-hand Rule

Drawing with theZ Coordinate

While drawing in 2D, you only had to worry about entering the X and Y coordinates. Since the Z coordinate was omitted, AutoCAD used the default value of zero. When drawing in 3D, you must add the Z value in some coordinate input. For example, to enter the coordinates X2Y3Z4, enter 2,3,4 if it is an absolute coordinate and @2,3,4 if it is a relative coordinate.

Absolute X0Y0Z0

The absolute coordinate 0,0,0 is the origin of the World Coordinate System. This is the same location used in 2D when only 0,0 was entered. This is an important coordinate location as everything drawn in model space relates back to this location. Keep this in mind when drawing all future models. It is especially important when constructing models that relate to real world locations. For example, when drawing a map, X0Y0Z0 is located at the equator and your model must be drawn in relation to that location.

USER TIP: Similar to working in 2D, it is important to save the objects drawn for construction purposes. In all lab exercises, daw all construction objects on layer Construction and do not delete them when the model is complete. After completing the model, freeze layer Construction. When required, simply thaw layer Construction to display the construction objects.

WORK ALONG: Drawing 3D Wireframe Models with the UCS Located at the WCS

Step 1

Using the NEW command, start a new drawing using the template: 3D Layout English.

Step 2

Save the drawing with the name: AutoCAD 3D Workalong 03-1. (Figure Step 2)

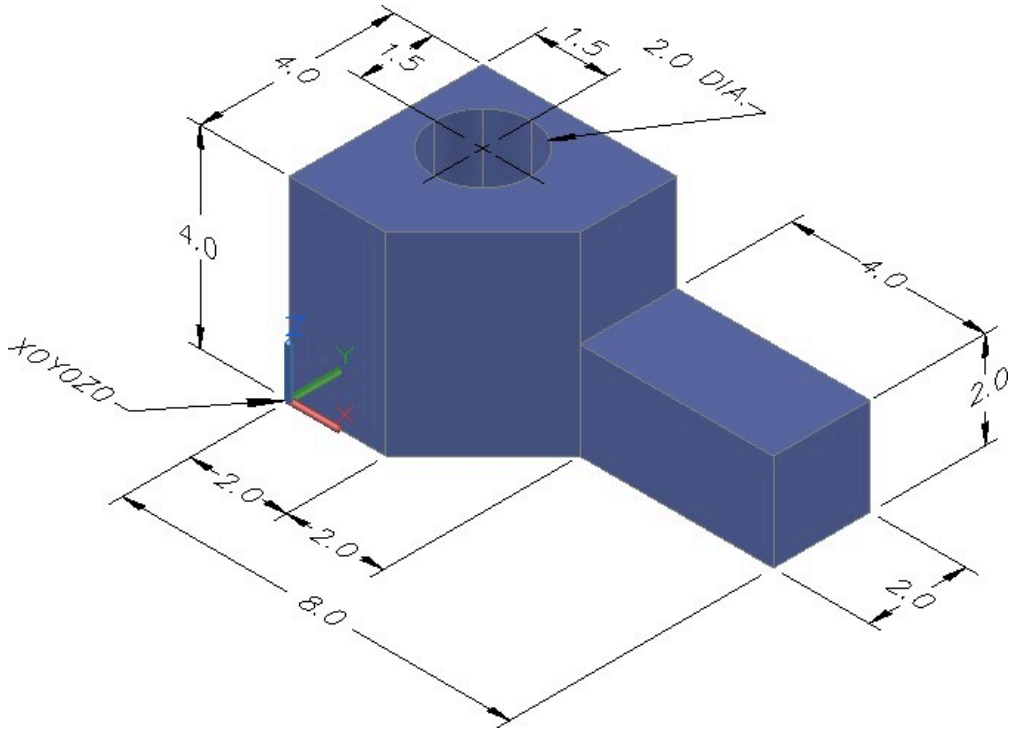


Figure Step 2
Dimensioned Solid Model

AUTHOR'S COMMENTS: Constructing models with the UCS located at the WCS is not the best method of constructing models. As a learning tool for this module, it simplifies model construction. This will help you when you draw your first few models.

Step 3

Set the current view to SE Isometric. (Figure Step 3)

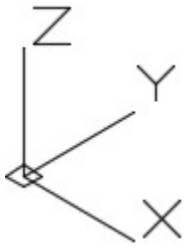


Figure Step 3

Step 4

Ensure that the UCS Icon is enabled (On) and the Origin is enabled as shown in Module 1 WORKALONG: Using the UCSICON, VIEW, and 3DFORBIT Commands.

Step 5

Set layer: Model as the current layer. Enter the LINE command, as shown below, to draw the lines to start the construction of the model. (Figure Step 5)

Command: **LINE**

Specify first point: **0,0,0**

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

(Since you are working at Z zero, you can omit the Z value.)

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

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

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

Specify next point or [Close/Undo]: **@-8,0**

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

Command:

(Draw the object that is located on the XY axis as you did in 2D.)

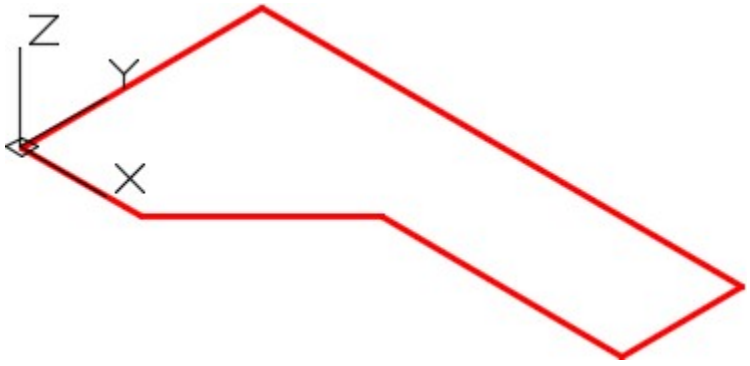


Figure Step 5

Step 6

Using the OFFSET command, offset the two lines 1.5 units as shown in the figure. Change the layer properties of the two lines to layer: Construction. (Figure Step 6)

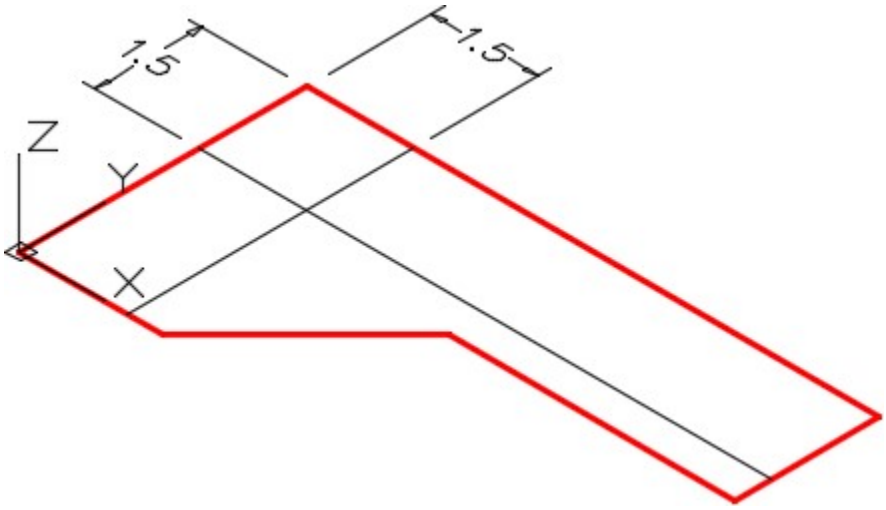


Figure Step 6

Step 7

Use the CIRCLE command to draw a 2 diameter circle with its centre located at the intersection of the two construction lines. (Figure Step 7)

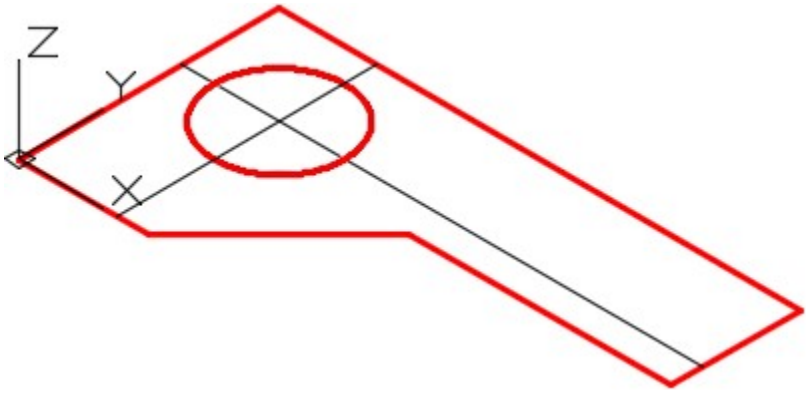


Figure Step 7

Step 8

Freeze layer: Construction. Enter the COPY command, as shown below, and copy the three lines and the circle 4 units in the positive Z direction. (Figure Step 8).

Command: **COPY**

Select objects:

(Select the three lines and the circle as shown in Figure Step 8.)

Select objects:

Specify base point or displacement, or [Multiple]: **0,0,0**

Specify second point of displacement or <use first point as displacement>: **@0,0,4**

(Copy the 4 objects 4 units in the positive Z direction. Looking at the UCS or using the right- hand rule will indicated if it is a positive or negative direction.)

Command:

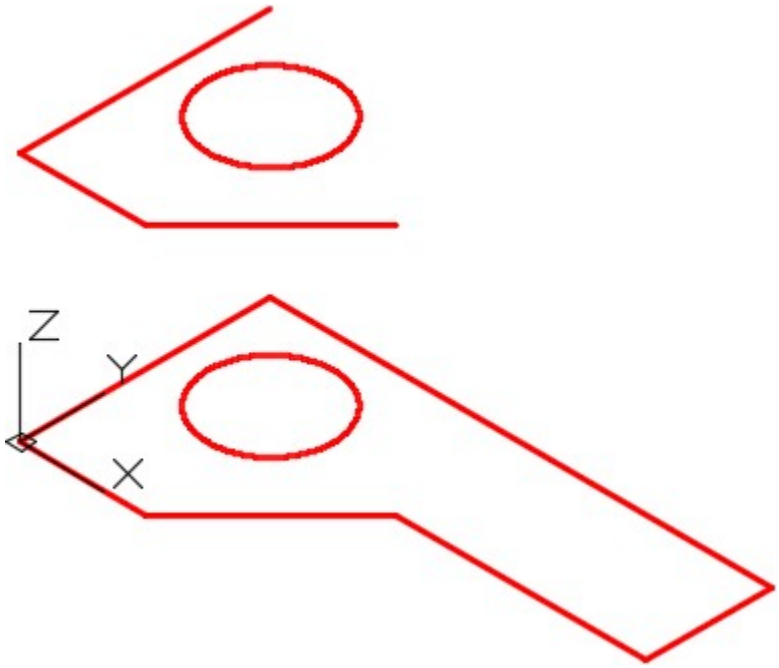


Figure Step 8

Step 9

Using the 3DFORBIT command, orbit the model slightly. (Figure Step 9)

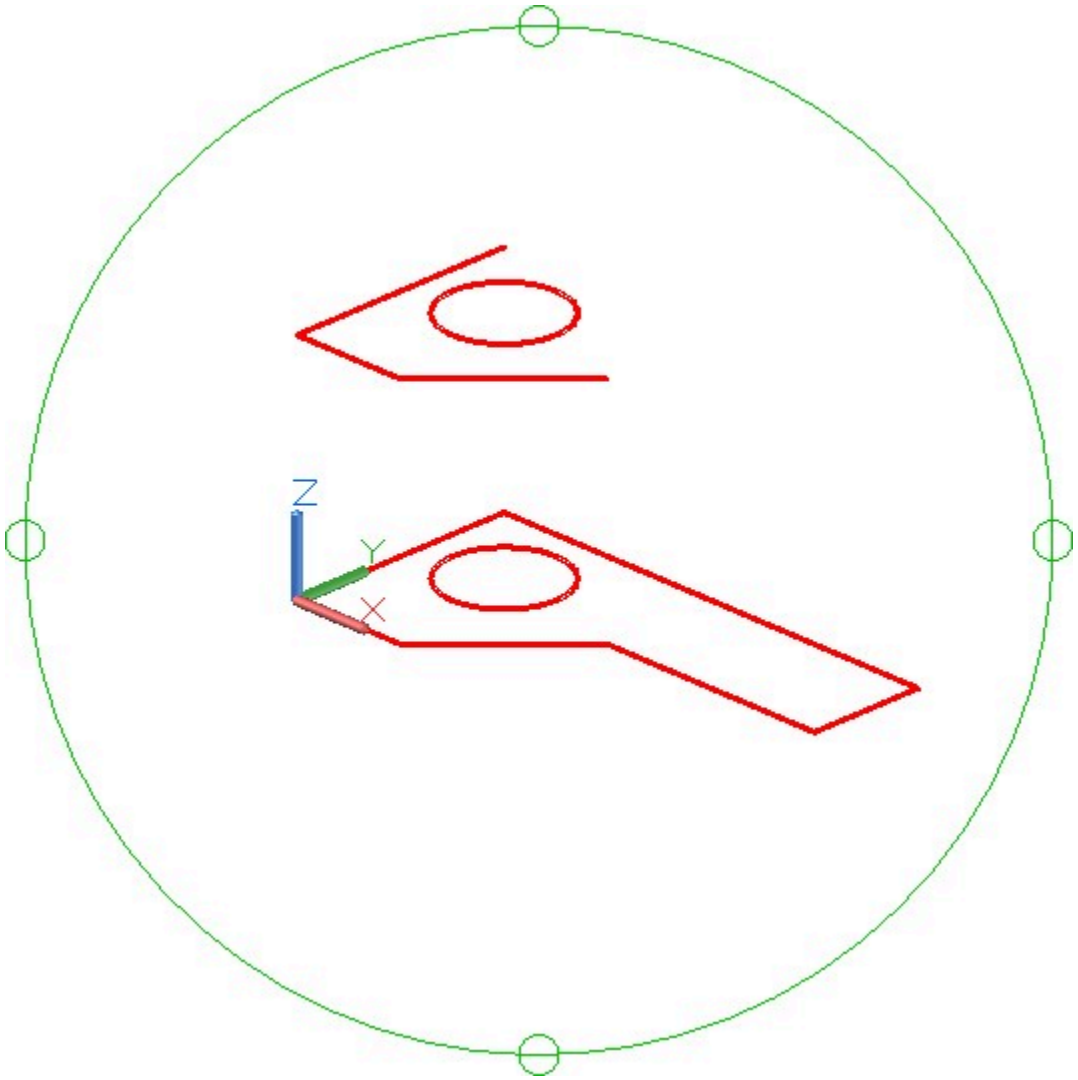


Figure Step 9

Step 10

Using the COPY command, copy the 2 lines 2 units in the positive Z directions. (Figure Step 10)

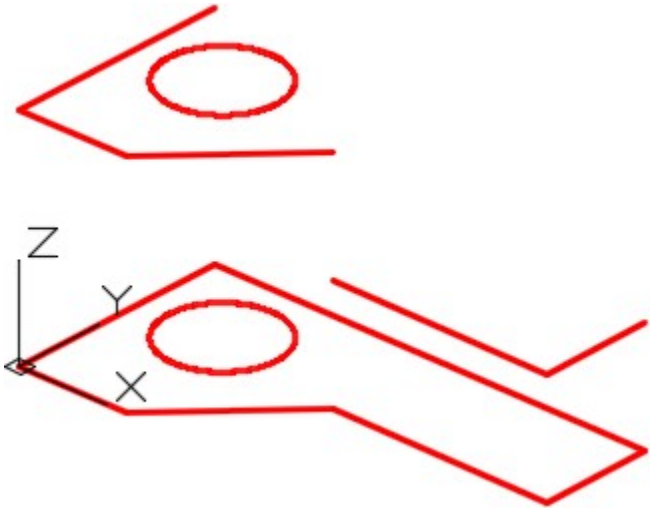


Figure Step 10

Step 11

Use the LINE command to draw six lines between the ends of the existing lines. Ensure that you snap to the ends of the lines. (Figure Step 11)

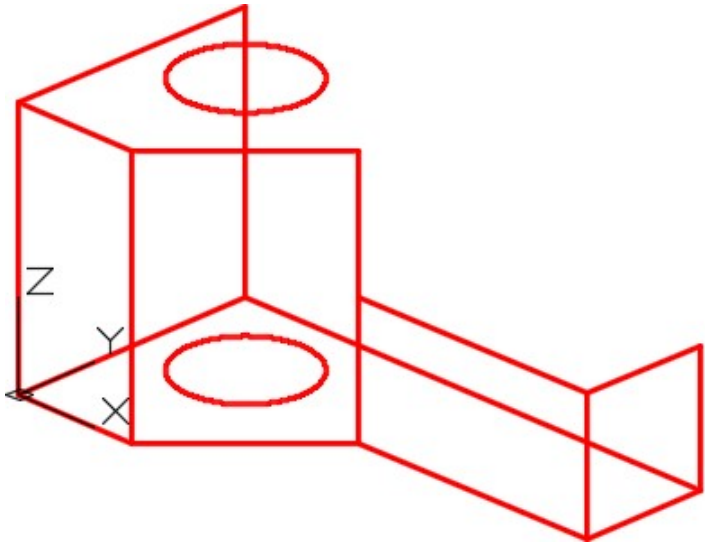


Figure Step 11

Step 12

Using the COPY command, copy the short line located at the right end of the model two times. Ensure that you use snap mode to locate the lines exactly. (Figure Step 12)

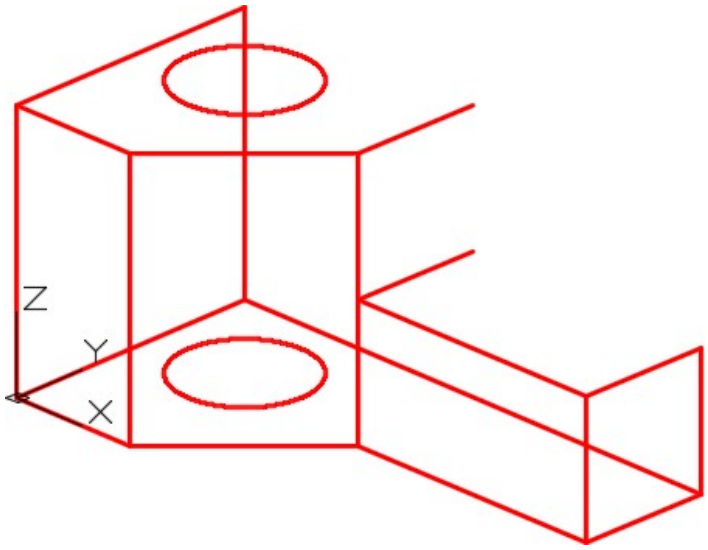


Figure Step 12

Step 13

Draw three lines between the ends of the existing lines. Ensure to snap to the ends of the existing lines. (Figure Step 13)

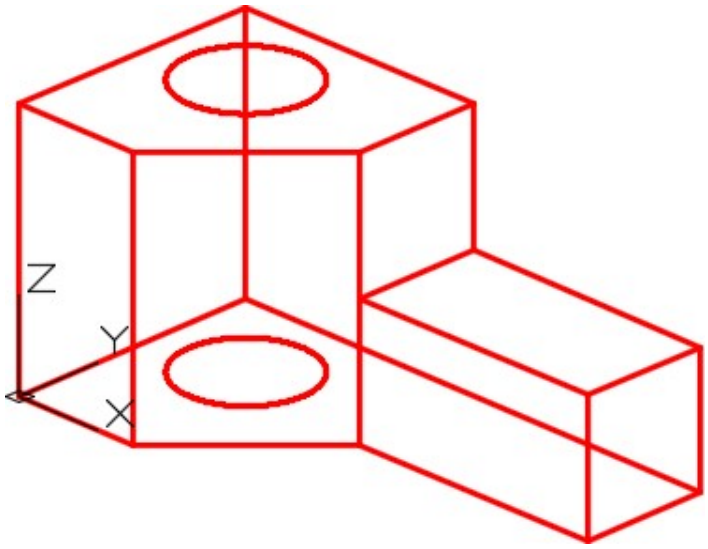


Figure Step 13

Step 14

Using the VIEW command, save the current view with the name: Working Isometric. (Figure Step 14)

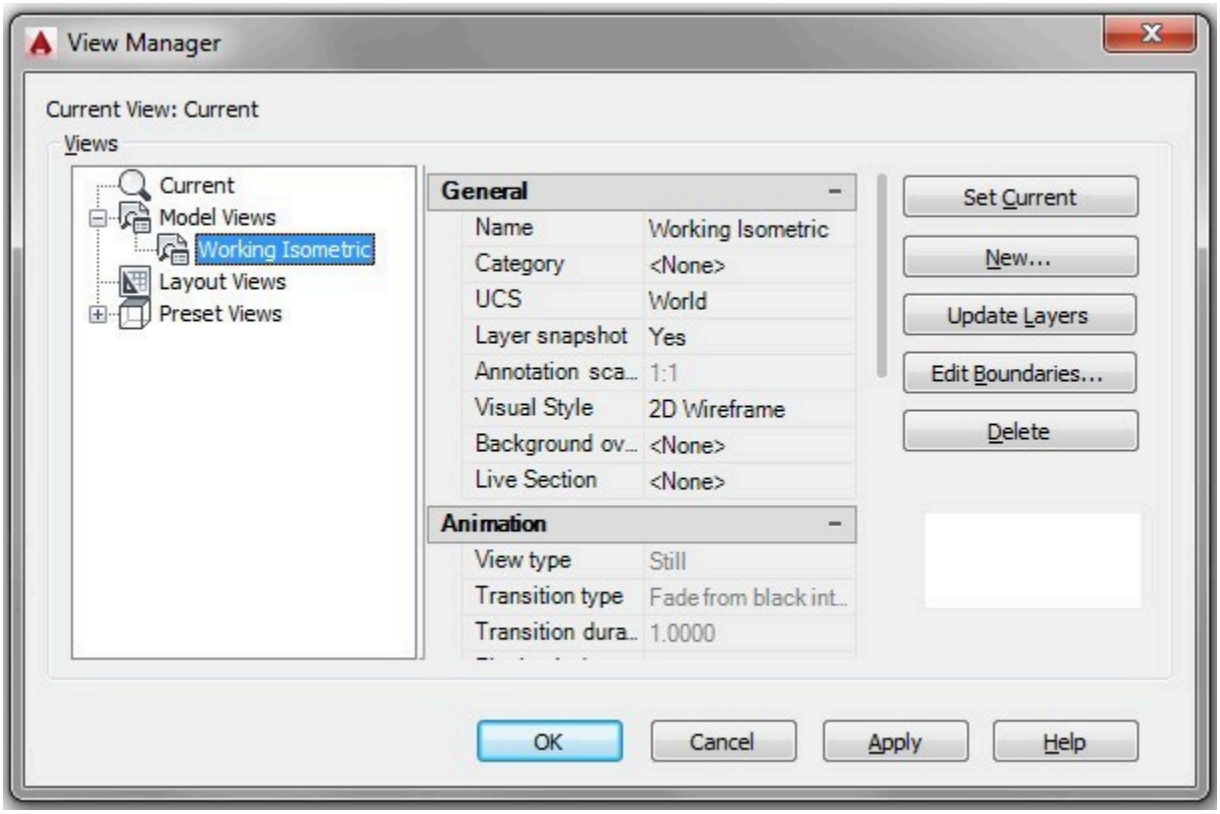


Figure Step 14

Step 15

Change the current view to SE Isometric. (Figure Step 15)

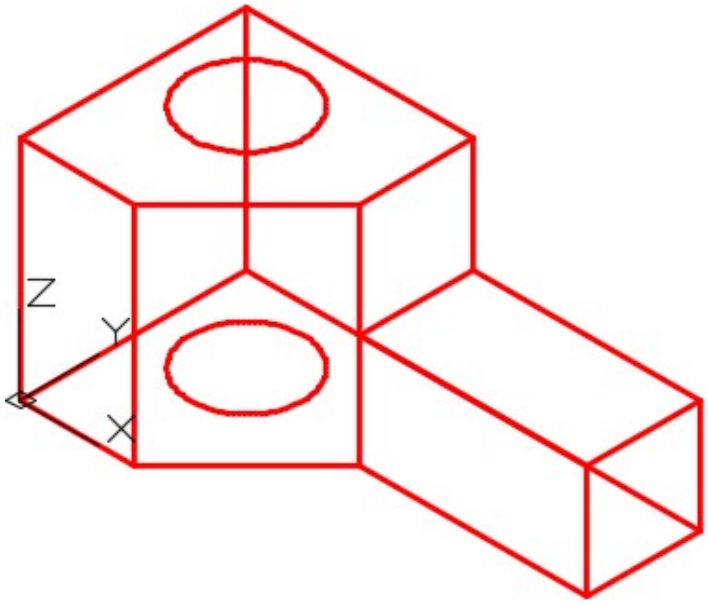


Figure Step 15

Step 16

Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Inches. Insert the key: AutoCAD 3D Workalong 03-1. It will overlay the model with a magenta model. (Figure Step 16)

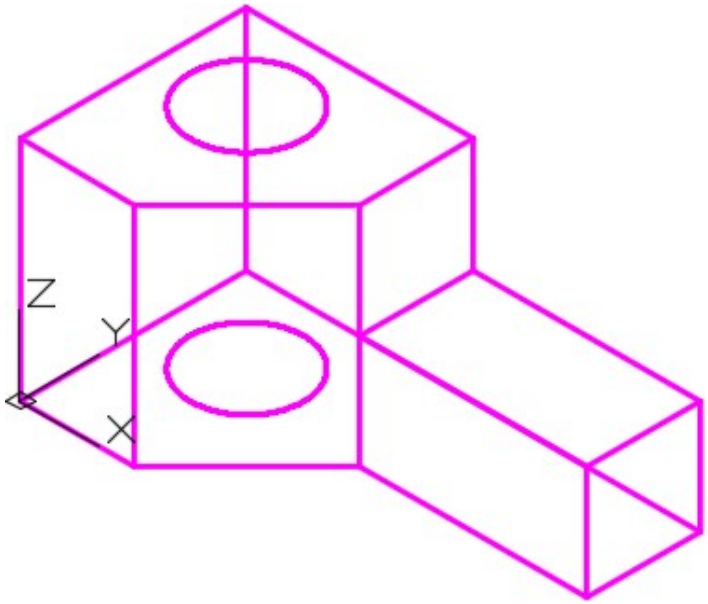


Figure Step 16

AUTHOR'S COMMENTS: If you require help doing this, redo Module 2.

Step 17

Save and close the drawing.

WORK ALONG: Drawing 3D Wireframe Models With the UCS Located at the WCS

Step 1

Using the NEW command, start a new drawing using the template: 3D Layout Metric.

Step 2

Save the drawing with the name: AutoCAD 3D Workalong 03-2. (Figure Step 2)

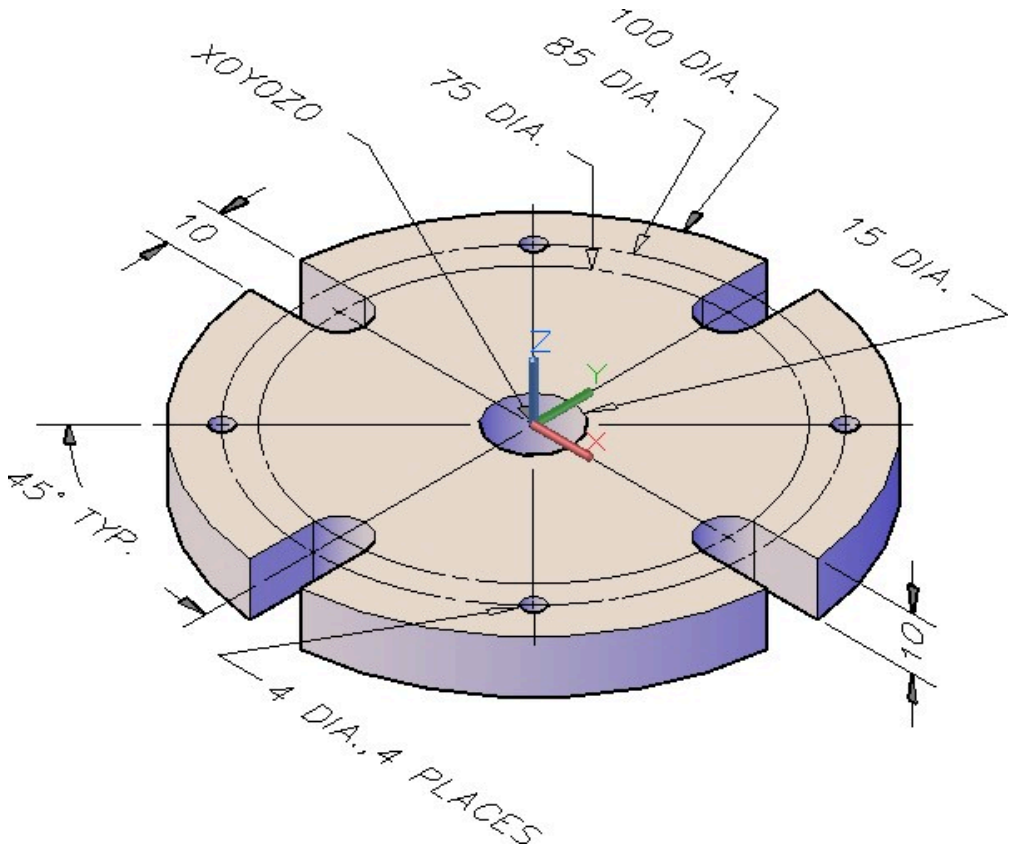


Figure Step 2
Dimensioned Solid Model

MUST KNOW: AutoCAD has two distinct three-dimensional coordinate systems: the World Coordinate System (WCS) and the User Coordinate System (UCS). The World Coordinate System is located at the absolute coordinates X0Y0Z0. It is a fixed coordinate system and cannot be moved. The WCS is normally not used to construct models. The UCS the coordinate system that is used to construct 3D models. The UCS can be placed exactly at the WCS or at any location in 3D space.

Step 3

Set the current view to SE Isometric. (Figure Step 3)

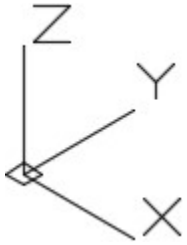


Figure Step 3

Step 4

Ensure that the UCS Icon is enabled (On) and the Origin is enabled as shown in Module 1, WORKALONG: Using the UCSICON, VIEW, and 3DFORBIT Commands.

Step 5

Set layer: Model as the current layer. Draw the shape of the top of the object. Use the ARRAY command to speed up the construction. (Figure Step 5)

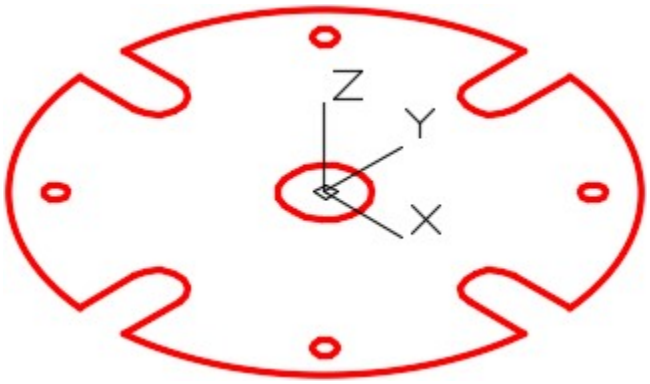


Figure Step 5

Step 6

Using the COPY command, as shown below, copy all of the objects 10 units in the negative Z direction. (Figure Step 6)

Command: **COPY**

Select objects:

(Select all objects.)

Select objects:

Specify base point or displacement, or [Multiple]: **0,0,0**

Specify second point of displacement or <use first point as displacement>: **@0,0,-10**

Command:

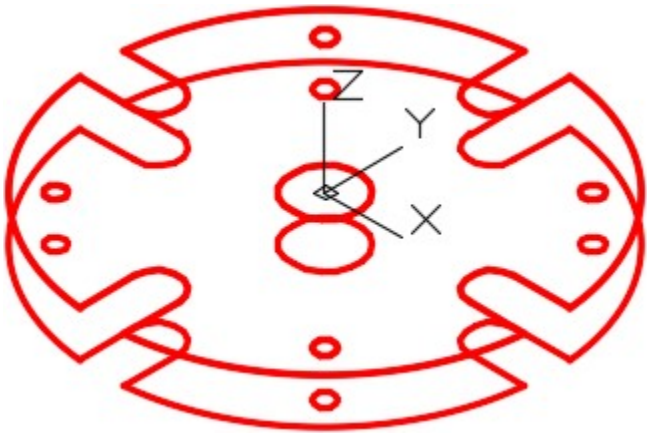


Figure Step 6

Step 7

Using the 3DFORBIT command, orbit the model slightly until it appears similar to the figure. (Figure Step 7)

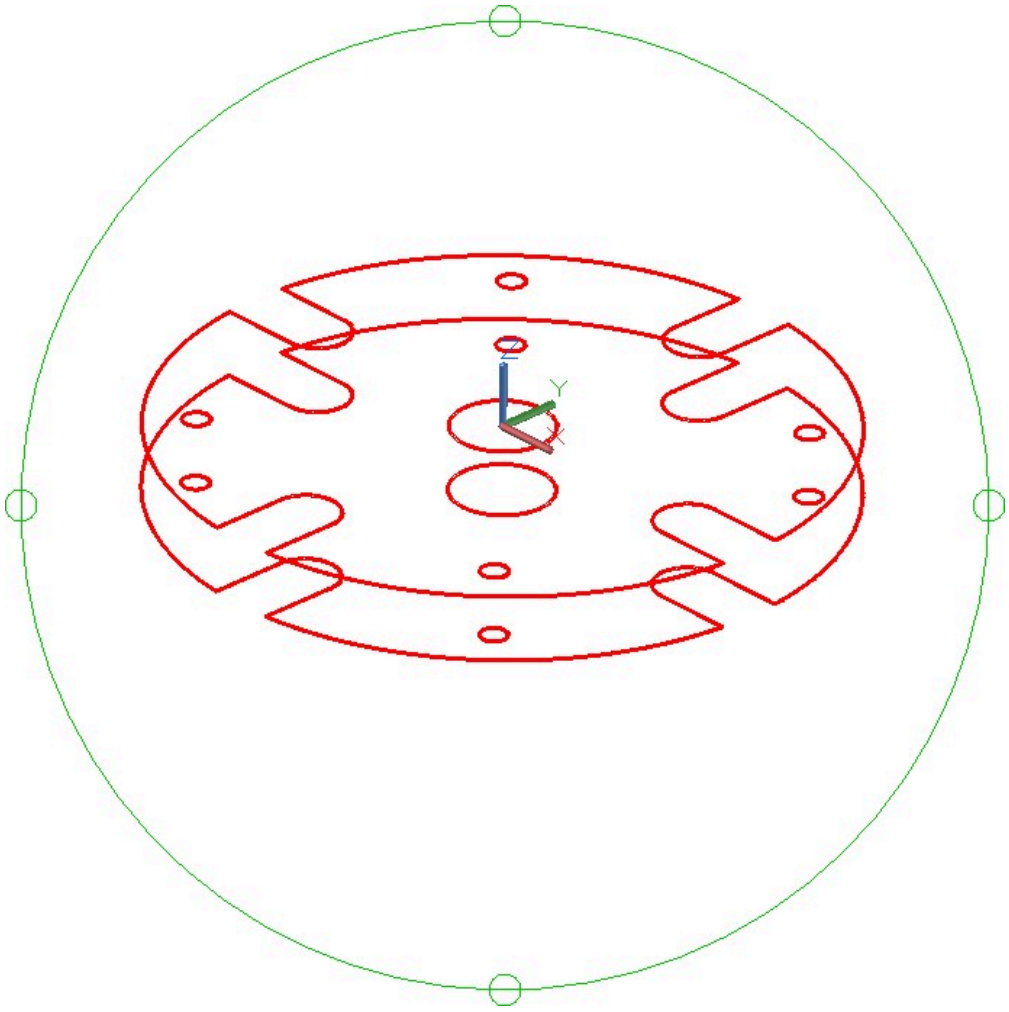


Figure Step 7

Step 8

Draw the vertical lines by snapping to the endpoints to complete the wireframe model. (Figure Step 8)

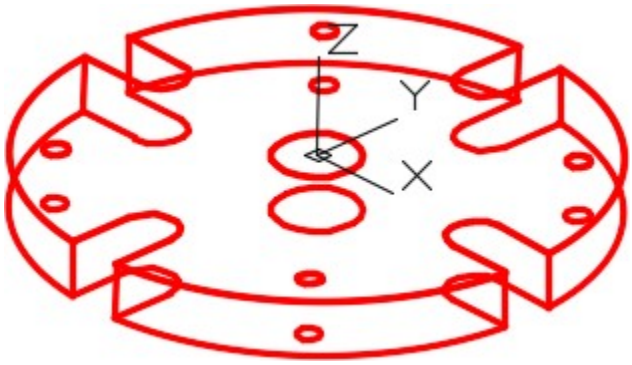


Figure Step 8

Step 9

Using the VIEW command, save the current view with the name: Working Isometric. (Figure Step 9)

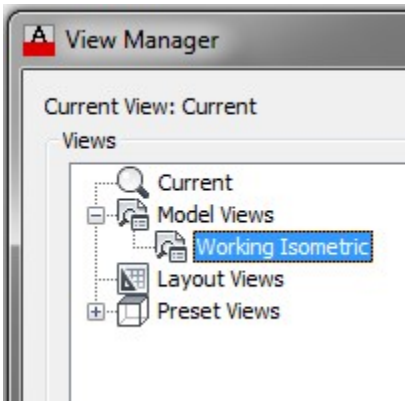


Figure Step 9

Step 10

Change the view to SE Isometric. (Figure Step 10)

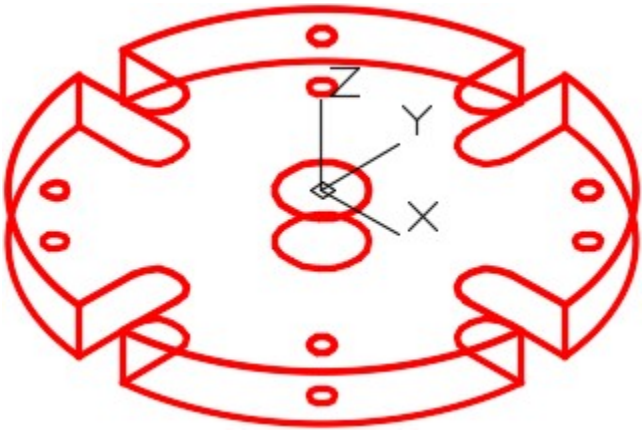


Figure Step 10

Step 11

Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Millimeters. Insert the key:

AutoCAD 3D Workalong 03-2. It will overlay the model with a magenta model. (Figure Step 11)

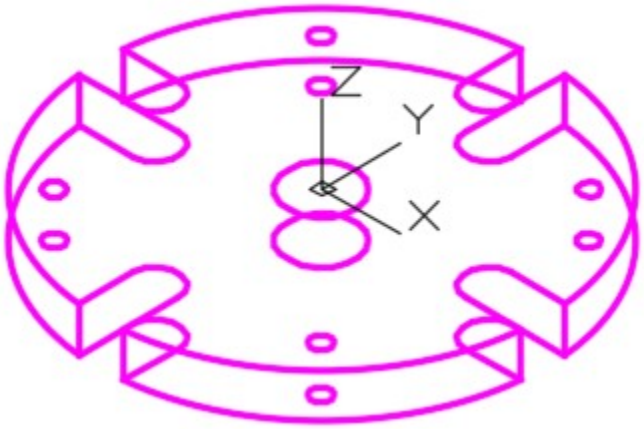


Figure Step 11

Step 12

Save and close the drawing.

Key Principles

Key Principles in Module 3

1. AutoCAD has two distinct three dimensional coordinate systems: the World Coordinate System (WCS) and the User Coordinate System (UCS).
2. The World Coordinate System is permanently located at the absolute coordinates It is a fixed coordinate system which can never be moved. The WCS is normally not used to construct models.
3. The User Coordinate System is the coordinate system that is normally used to construct 3D The UCS can be placed exactly at the WCS or at any location in 3D space.
4. AutoCAD uses the right-hand rule to find the positive Z To use the right-hand rule, you must first know the positive X and Y directions of the current UCS. Using your right hand, point the thumb in the direction of the positive X axis. Extend your index finger in the direction of the positive Y axis. Your middle finger indicates the direction of the positive Z axis.
5. The absolute coordinate 0,0,0 is the origin of the world coordinate This is the same point used when drawing in 2D when 0,0 was entered. This is an important coordinate location as everything drawn in model space relates back to this location.

Lab Exercise 3-1

Time allowed: 45 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 03-1	3D Layout English	Inches

Step 1

Save and name the drawing: AutoCAD 3D Lab 03-1.

Step 2

Draw all construction objects on layer: Construction and model objects on layer: Model.

Step 3

Draw a wireframe model of the object shown in the figure. (Figure Step 3A and 3B)

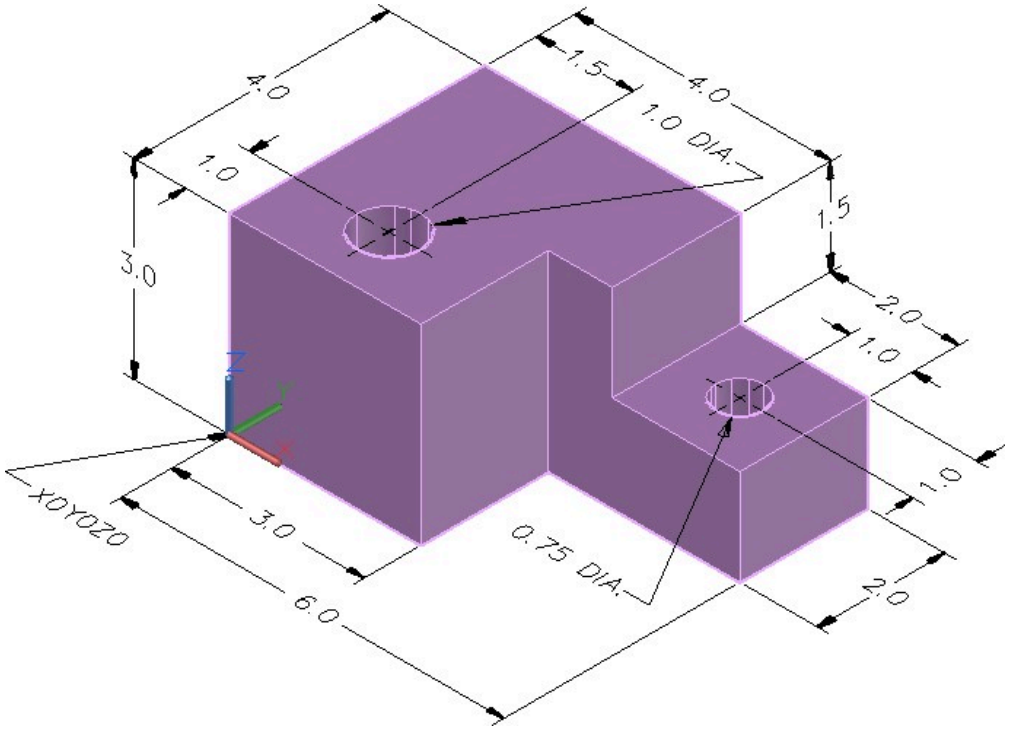


Figure Step 3A
Dimensioned Solid Model

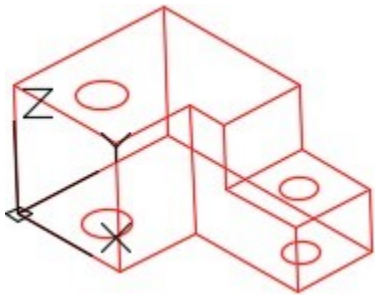


Figure Step 3B Completed Wireframe Model SE Isometric View

Step 4

Start your model with the current view SE Isometric. If required, orbit it slightly with 3DFORBIT to help the line of sight.

Step 5

Save the isometric working view with the name: Working Isometric.

Step 6

When complete, freeze layer: Construction.

Step 7

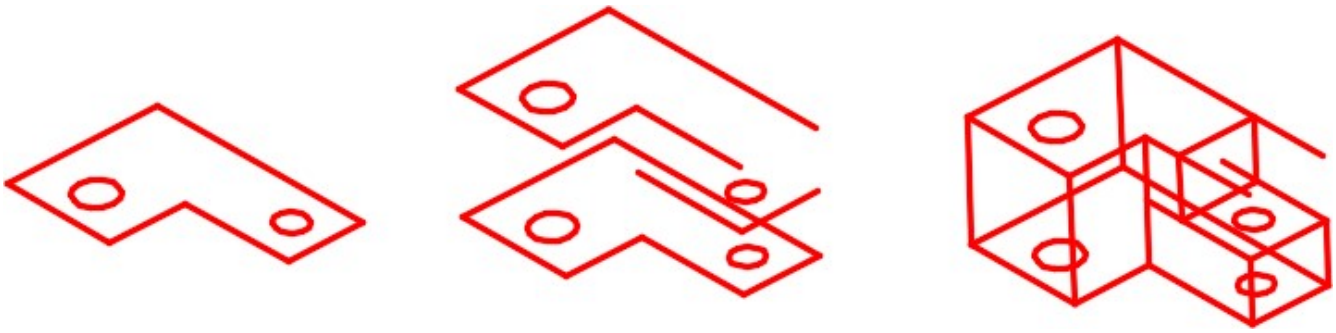
Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Inches.

Step 8

Check your drawing with the key. The key name is the same as the drawing name.

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.

Hint 1



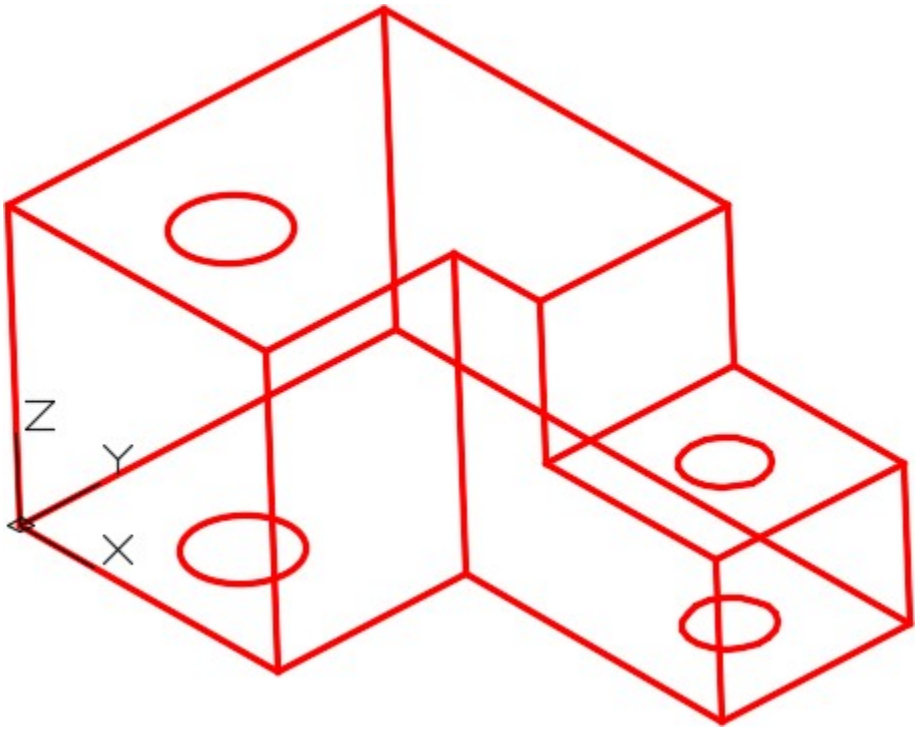


Figure Hint 1
Construction Steps

Lab Exercise 3-2

Time allowed: 45 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 03-2	3D Layout English	Millimeters

Step 1

Save the drawing with the name: AutoCAD 3D Lab 03-2.

Step 2

Draw all construction objects on layer: Construction and all model objects on layer: Model.

Step 3

Draw a wireframe model of the object shown in the figure. (Figure Step 3A, 3B, 3C, and 3D)

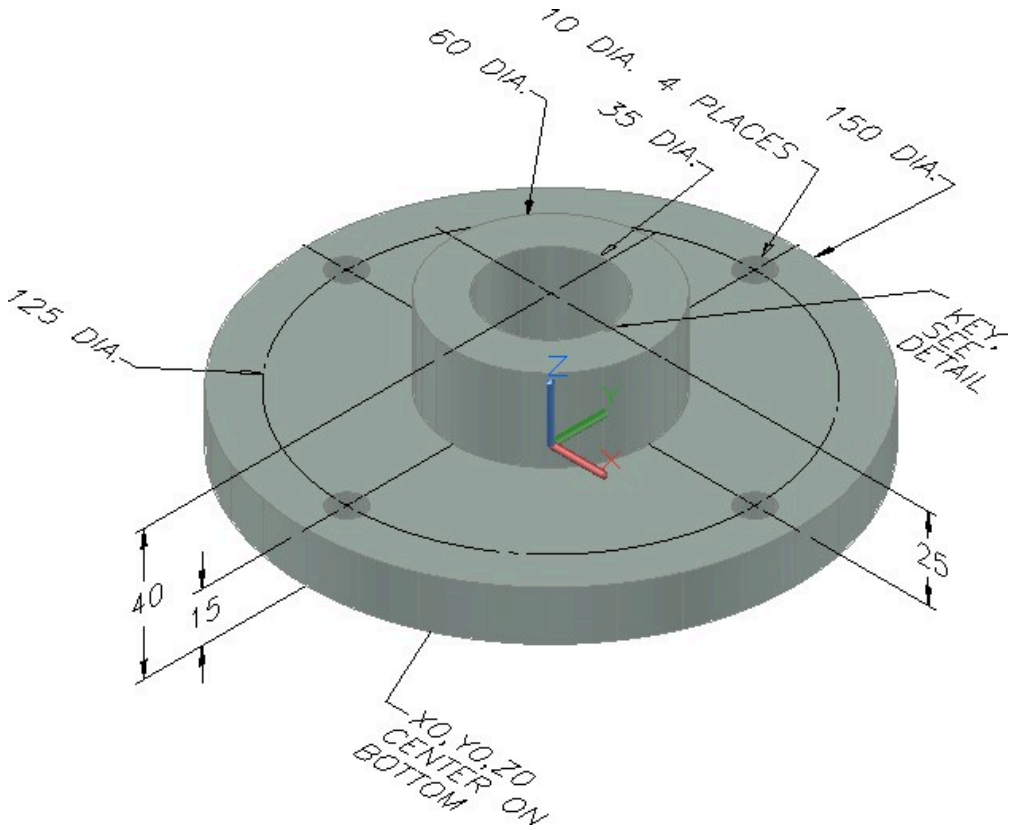


Figure Step 3A
Dimensioned Solid Model

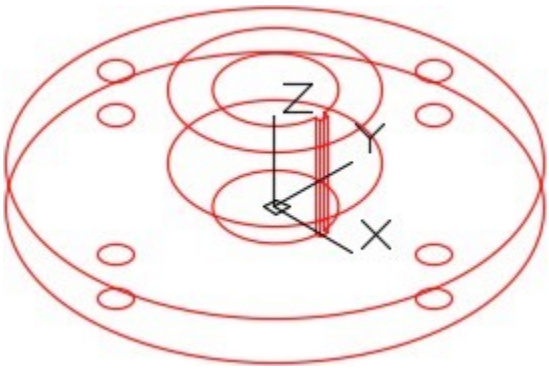


Figure Step 3B Completed Wireframe Model SE
Isometric View

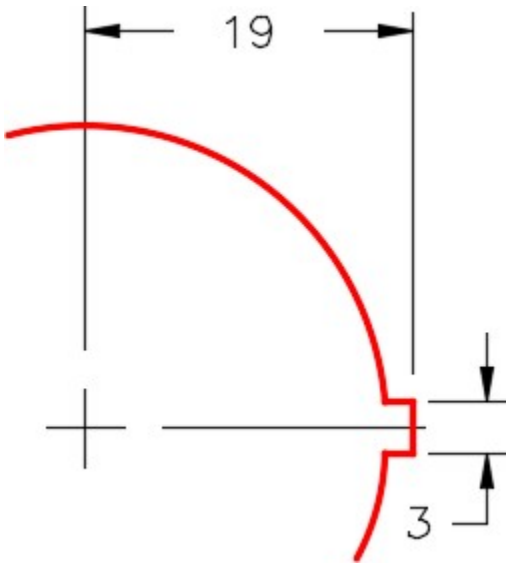


Figure Step 3C Key Detail

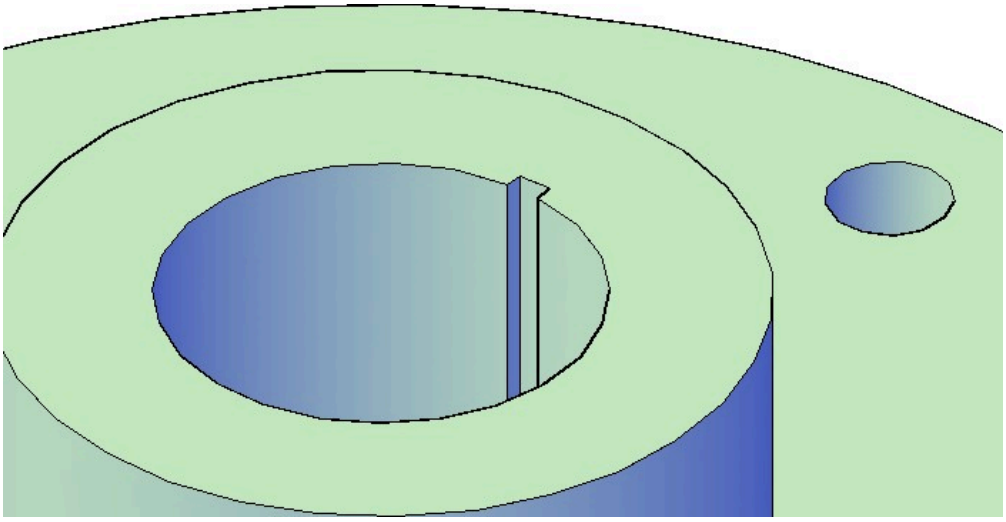


Figure Step 3D View of Keyway – Orbited

Step 4

Start your model with the current view SE Isometric. If required, orbit it slightly with 3DFORBIT to help the line of sight.

Step 5

Save the isometric working view with the name: Working Isometric.

Step 6

When complete, freeze layer: Construction.

Step 7

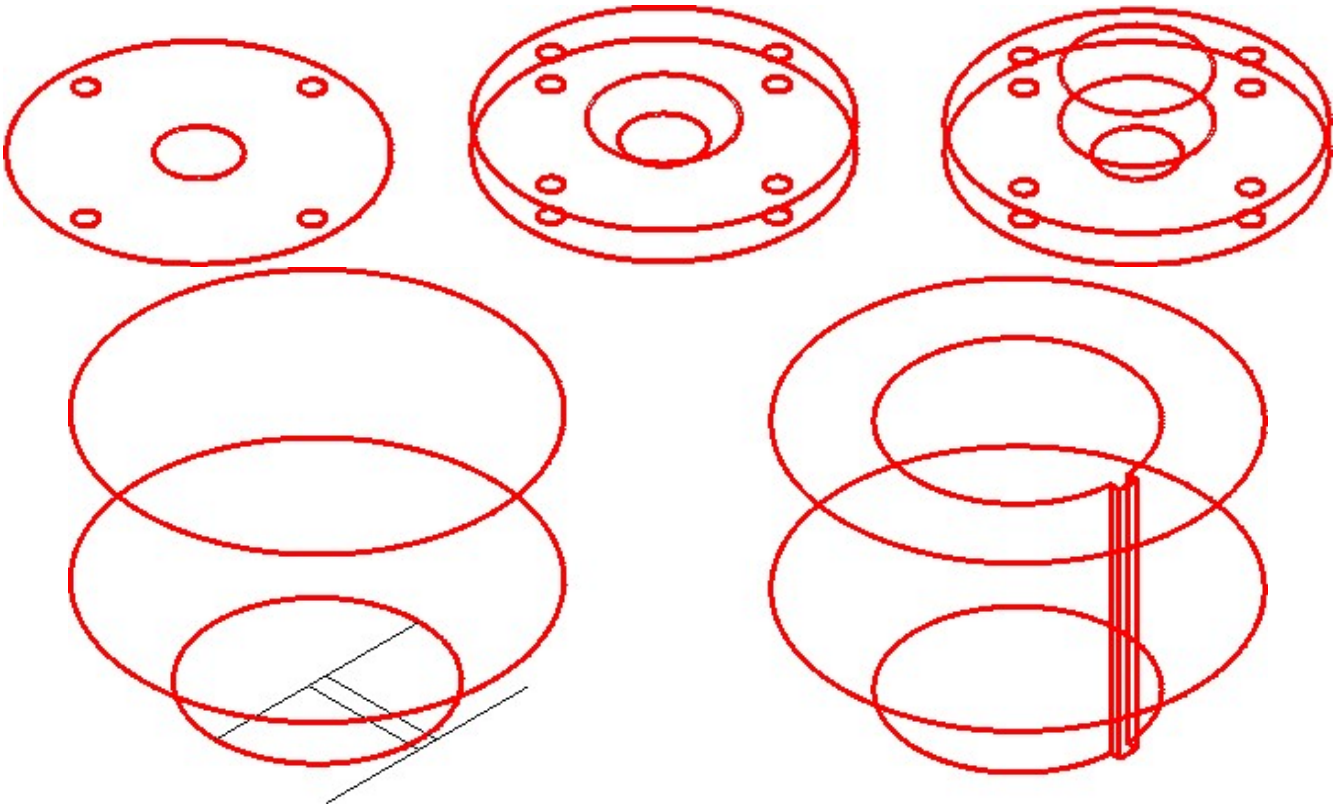
Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Millimeters.

Step 8

Check your drawing with the key. The key name is the same as the drawing name.

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.

Hint 1



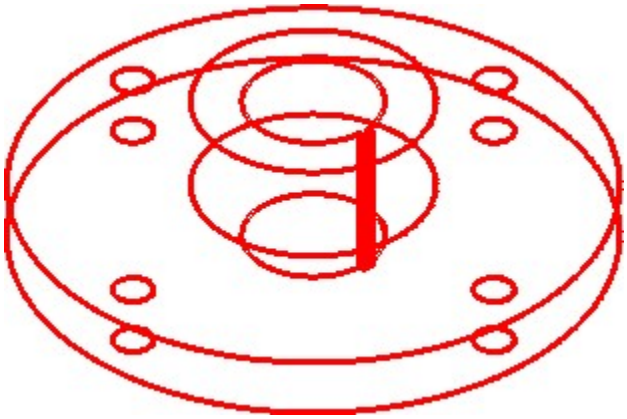


Figure Hint 1 Construction Steps

Module 4: User Coordinate System – Part 1

Learning Outcomes

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

1. Describe the User Coordinate System and the User Construction Plane.
2. Apply the UCSMAN command.
3. Draw 3D models using the User Coordinate System located at the World or at the predefined orthographic UCS locations only.

IMPORTANT – PLEASE READ BEFORE COMPLETING THIS MODULE

AutoCAD 2015 has a bug using the LINE command in 3D. The LINE command works properly when either the World or Top is the current UCS. When any other UCS is the current UCS, the cursor of the LINE command will display the new line in the wrong location. It locates the line in the correct location after you execute the LINE command. It displays correctly when snapping to existing objects and only appears wrong when using coordinates or drawing lines freehand.

A good workaround is to use the PLINE command rather than the LINE command, as it works correctly. After you complete the PLINE command, you can explode the pline to create a line object.

This bug is only in AutoCAD 2015. The LINE command in AutoCAD 2016, 2017 and 2018 works in all UCS locations.

The User Coordinate System

The User Coordinate System (UCS) is the coordinate system used to construct 3D models. It can be located at the WCS or at any location or orientation in 3D space. Being able to locate and orientate the UCS anywhere in 3D space is the secret of 3D modeling.

The User Construction Plane

If you draw an imaginary line from the X axis to the Y axis on the user coordinate system, it forms a imaginary triangular plane as shown in Figure 4-1. When you locate the UCS onto the 3D model, as shown in Figure 4-2, you can see this imaginary triangular plane laying on the surface. Next, picture the plane expanded to fill the whole surface it is located on. See Figure 4-3. This surface is called the *User Construction Plane* (UCP).

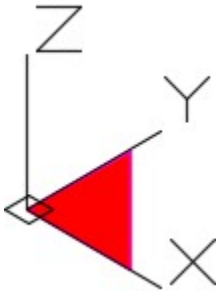


Figure 4-1
User Construction Plane

The easiest method of drawing 3D models is to locate and orientate the user construction plane to lie on the surface you currently want to insert or modify object(s) on. Relocate the location of the UCS and continue your model construction until you complete the model. You can use the Z axis to copy the object(s) and draw at a depth parallel to the UCP.

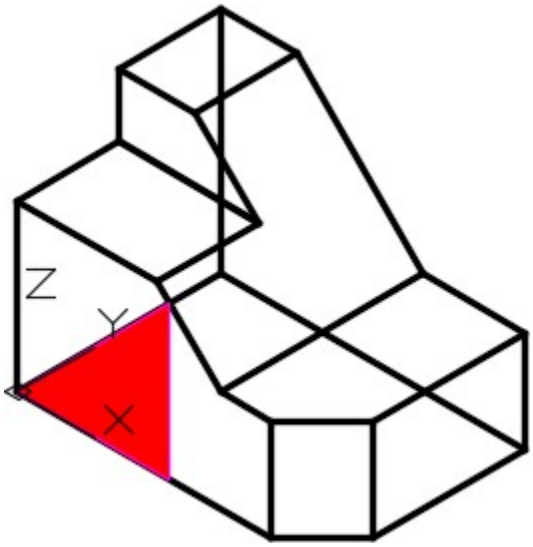


Figure 4-2
User Construction Plane on the WCS

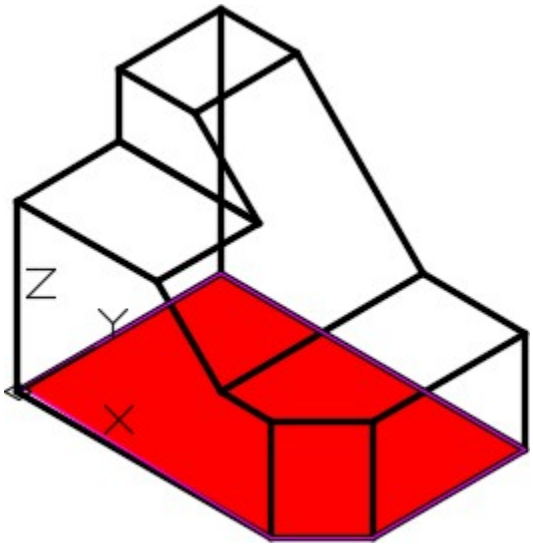


Figure 4-3
UCS Plane expanded to cover the Whole Surface

By relocating and orientating the UCS to different locations, you can draw or modify objects on any surface you want. Figure 4-4 shows the user construction plane located on the right side of the model and Figure 4-5 shows it located on the inclined surface.

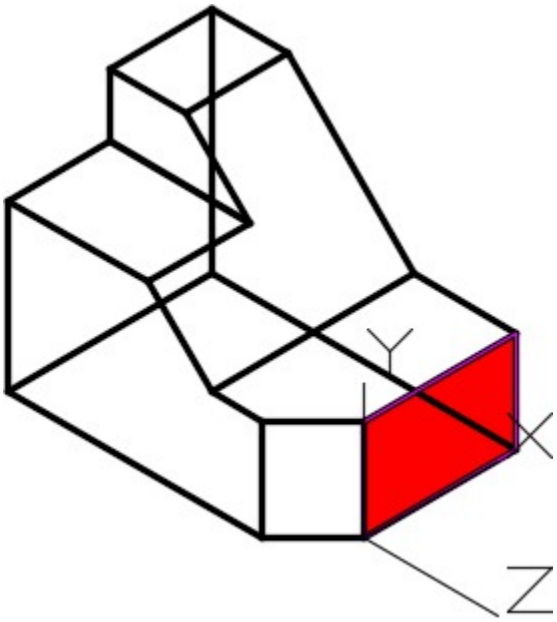


Figure 4-4
UCP located on the Right Side

In this module, all model construction will be drawn in the with the UCS located at the World or in one of the preset orthographic locations only. In Module 5, locating the UCS anywhere in model space will be taught.

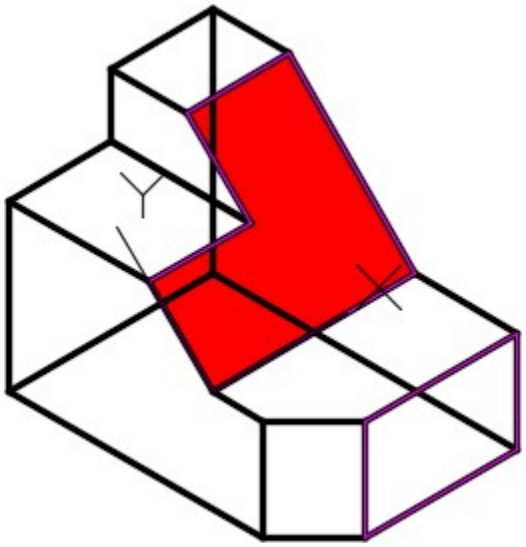
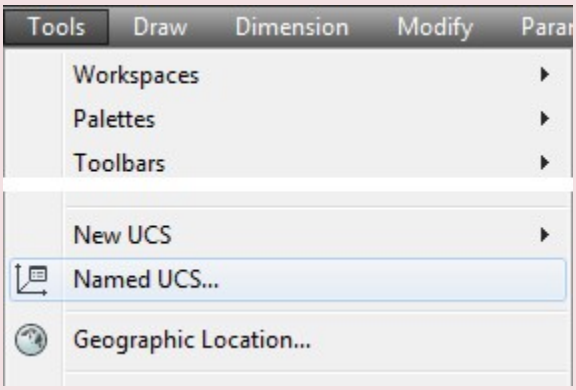


Figure 4-5
UCP located on an Inclined Surface

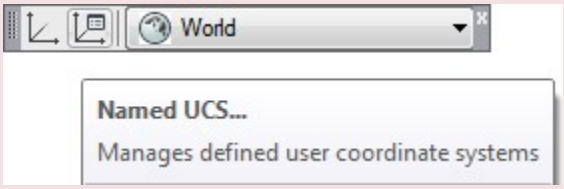
AutoCAD Command: UCSMAN

The UCSMAN command is used to locate, orientate, and manage the UCS.

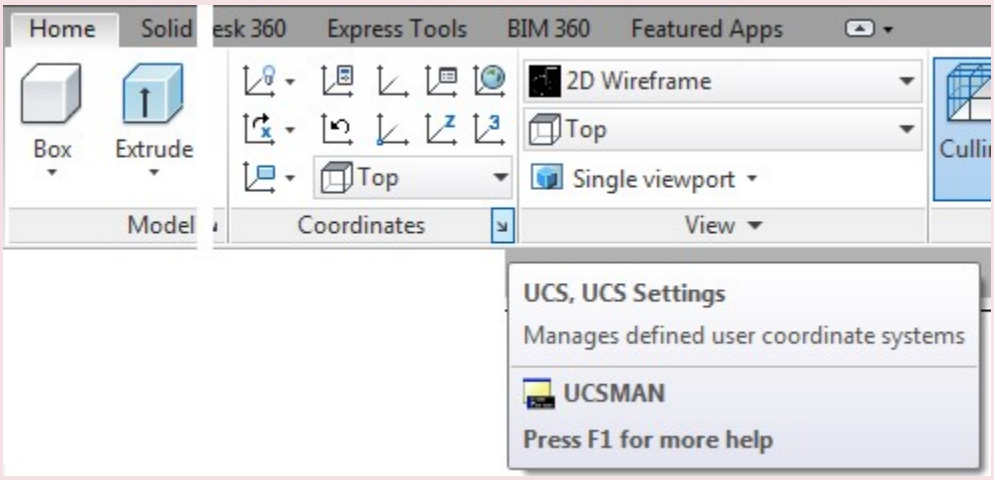
Shortcut: none



Tools Pull-down



UCS II Toolbar



Home Ribbon

WORK ALONG: Moving the UCS to the Preset Locations

Step 1

Using the NEW command, start a new drawing using the template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 04-1. Save it in the folder: CAD Courses/ AutoCAD 3D/Lab Exercises

Step 3

Set the current UCS to World. Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Inches. Using the INSERT command, insert the block: AutoCAD 3D Lab 01-1 at the coordinates 0,0,0. Explode the block.

Step 4

Set the current view to SE Isometric. Your wireframe model should appear as shown in the figure. (Figure Step 4)

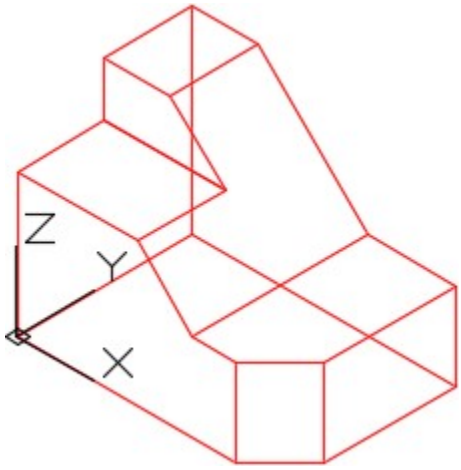


Figure Step 4

Step 5

Enter the UCSMAN command. It will open the UCS dialogue box. Enable the Settings tab and ensure that the settings in the dialogue box matches the figure. (Figure Step 5)

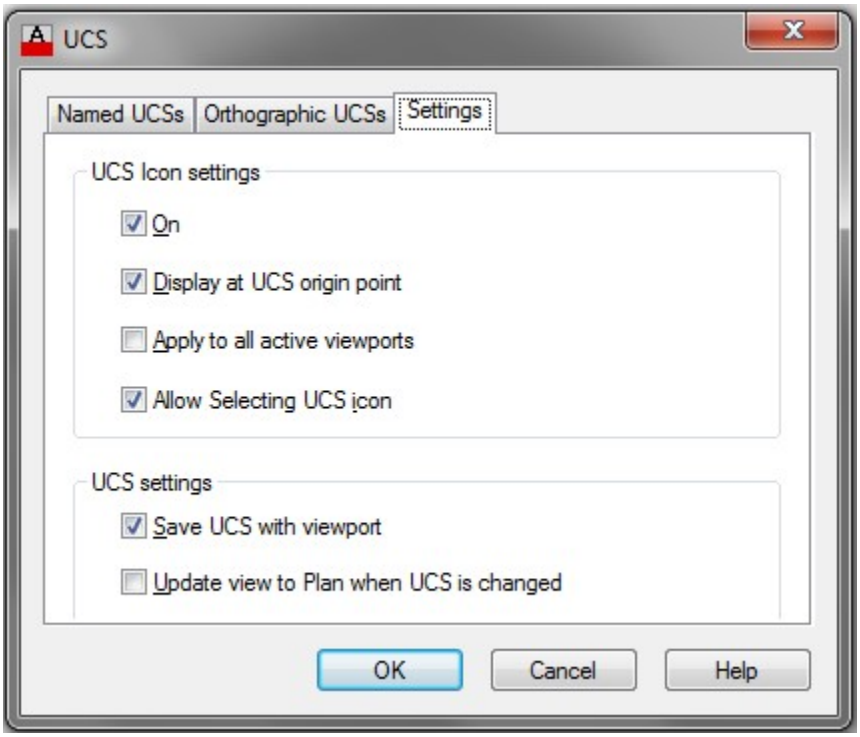


Figure Step 5

Step 6

Enable the Orthographic UCS's tab. Select Front and then click the Set Current button. (Figure Step 6)

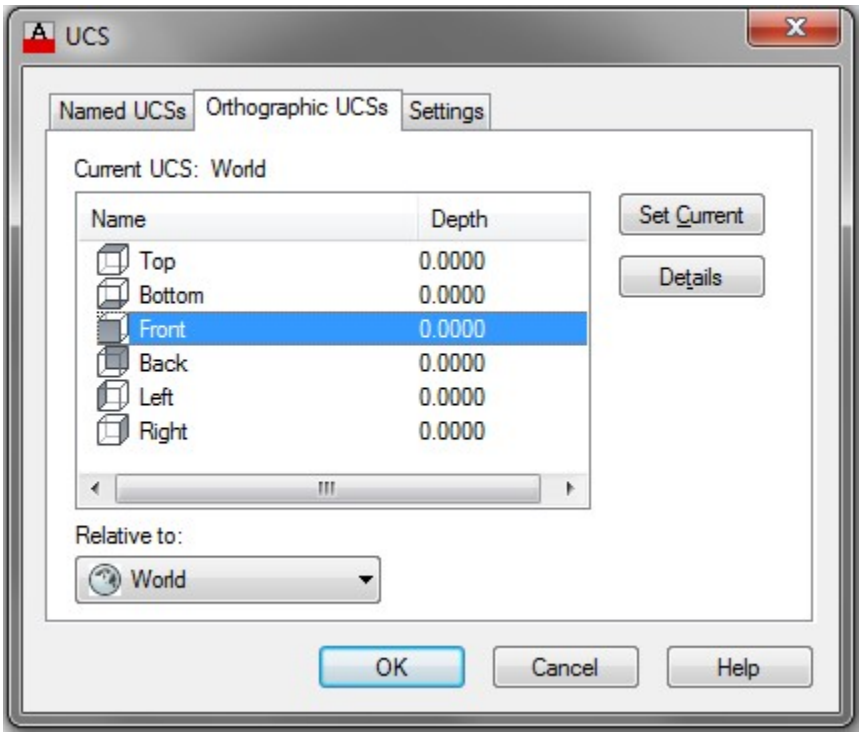


Figure Step 6

Step 7

The wireframe model should now appear as shown in the figure. Notice how the XY axis lies on the front plane of the model. (Figure Step 7)

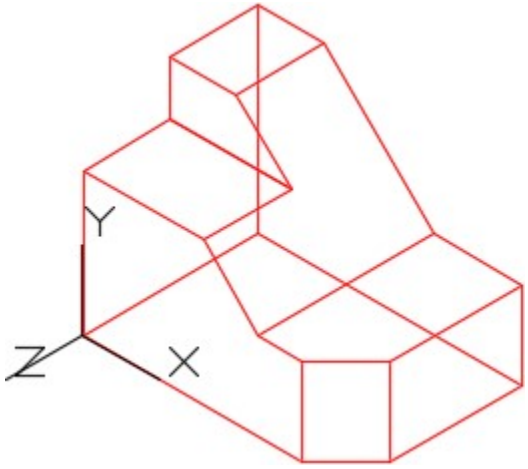


Figure Step 7

Step 8

Using either the UCS II toolbar or the Home tab ribbon. (Figure Step 8A and 8B)

Figure Step 8A

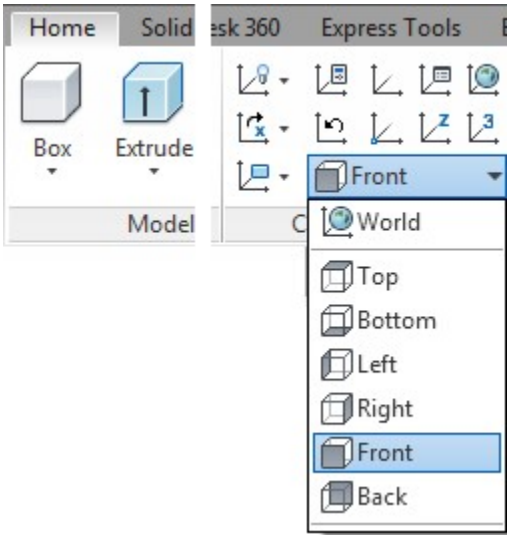


Figure Step 8B

Step 9

Set the current UCS location to Right and the UCS icon will move onto the right side of the model. Note that the XY plane is now located on the right side plane of the wireframe model. (Figure Step 9A and 9B)

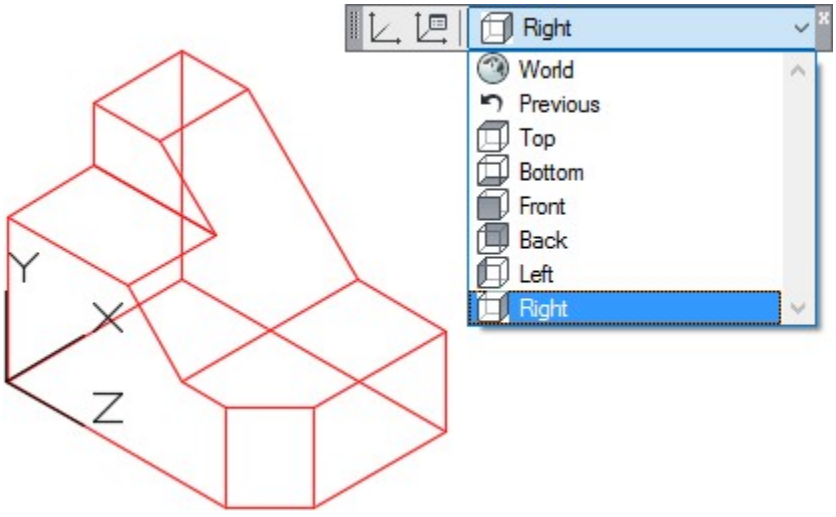


Figure Step 9A

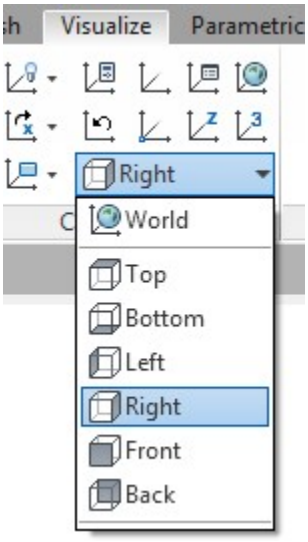


Figure Step 9B

MUST KNOW: To draw in 3D, it is important to visualize a user construction plane located on the X and Y axis. The easiest method of drawing 3D models is to locate and orientate the user construction plane on the surface you are currently modifying. Relocate the location of the UCS and continue model construction until the model is complete.

Step 10

Click the UCS command icon as shown in the figures. With osnap enabled, snap to the end of the line. Note that the UCS icon has now moved to the end of the line that you just selected but it remains located on the right side plane. (Figure Step 10A, 10B, 10C, and 10D)

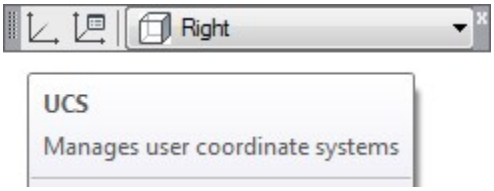


Figure Step 10A

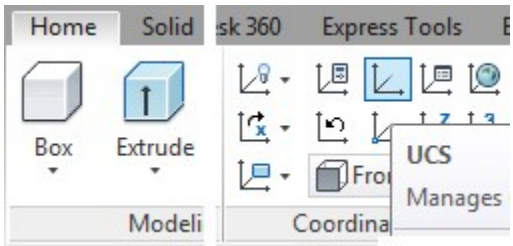


Figure Step 10B

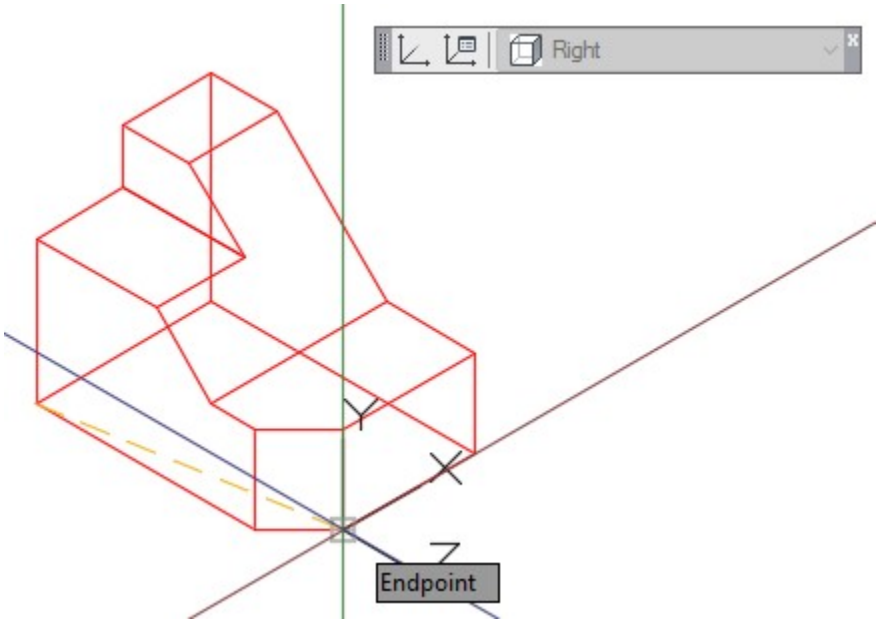


Figure Step 10C

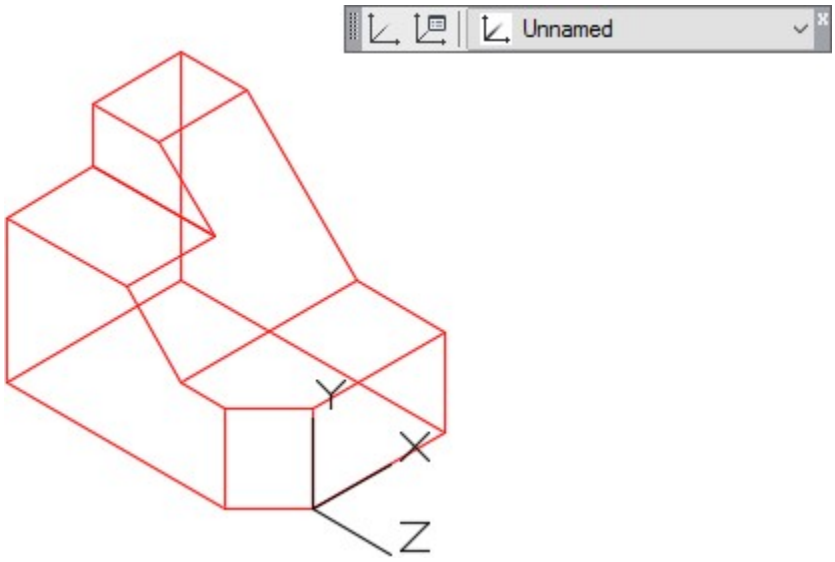


Figure Step 10D

Step 11

Click Top from the pull-down list. The UCS icon will move onto the top plane of the model. In this case, it locates itself on the WCS. (Figure Step 11)

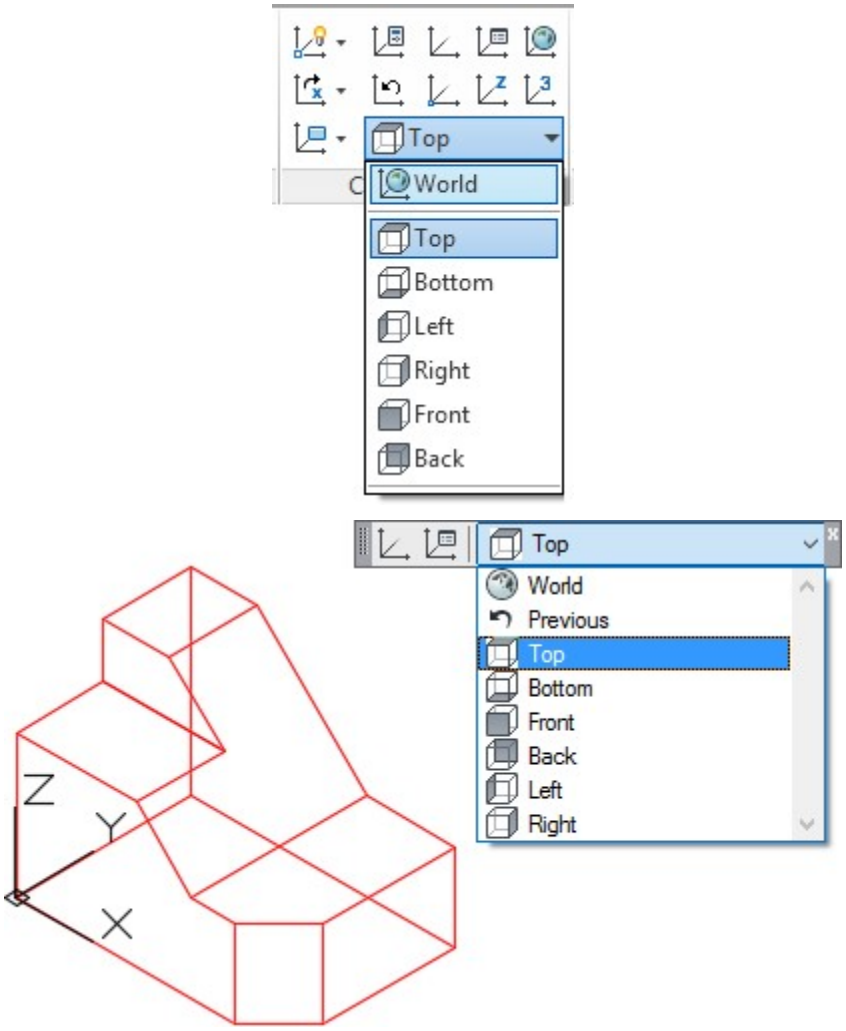


Figure Step 11

Step 12

Using the UCS command, move the UCS to corner as shown in the figure. Ensure that you snap to the end of the line. Note how the name of the UCS will display ‘ Unnamed ‘. (Figure Step 12)

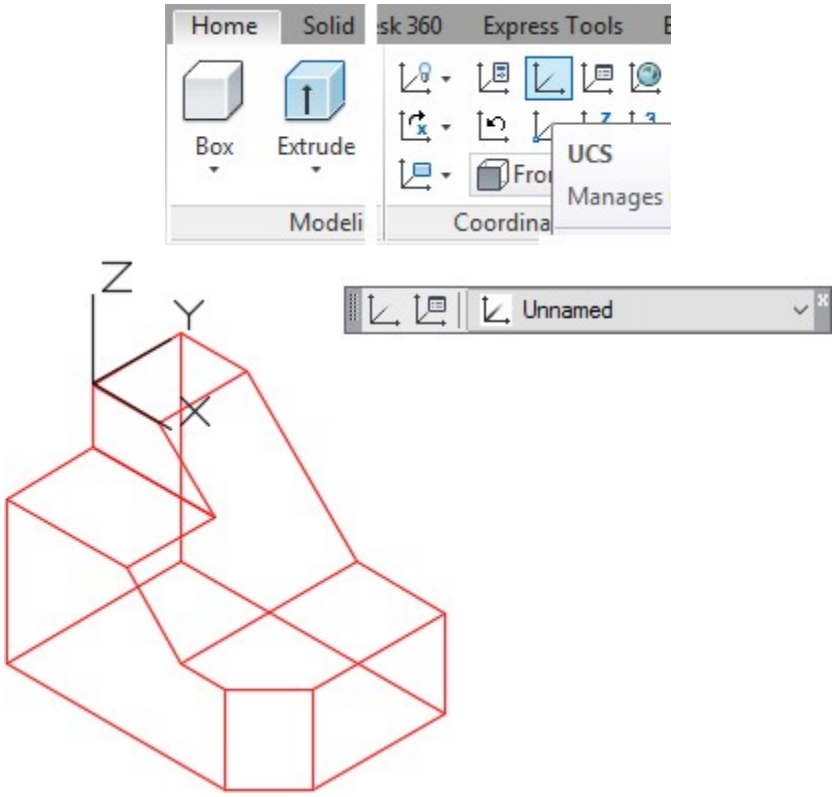


Figure Step 12

Step 13

Click World from the pull-down list. The UCS icon will move onto the WCS. (Figure Step 13)

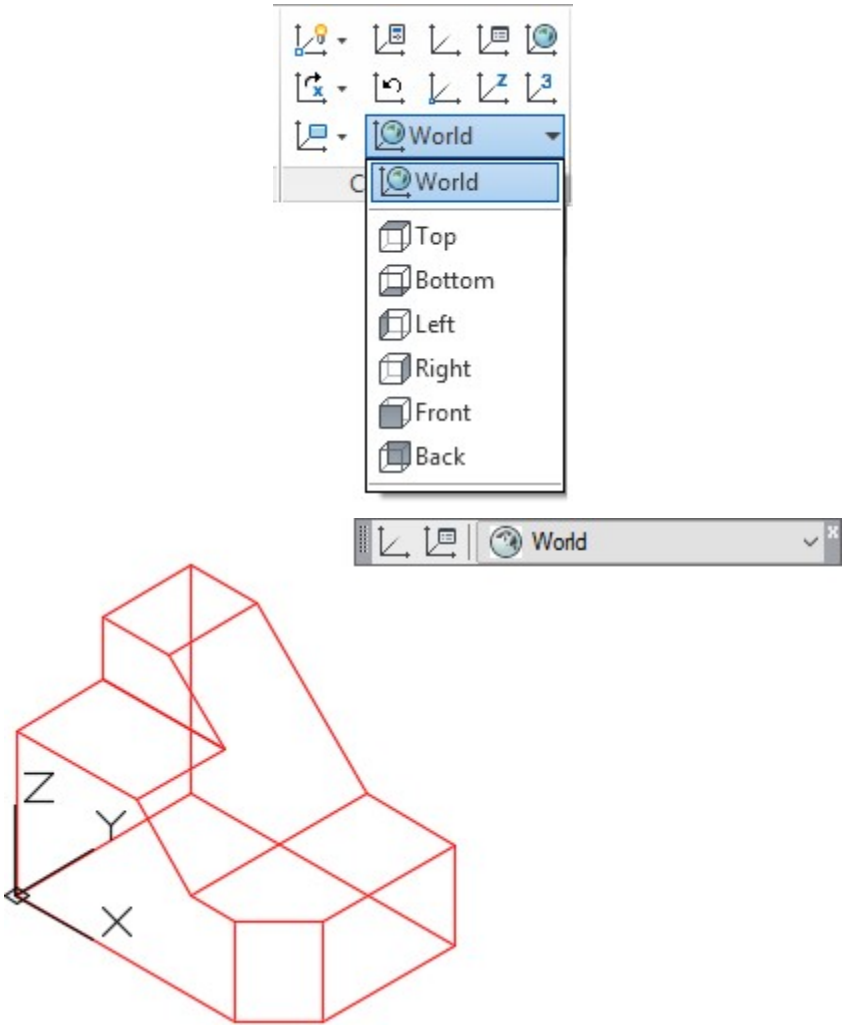


Figure Step 13

Step 14

Save and close the drawing.

WORK ALONG: Drawing 3D Wireframe Models Using the UCS

Step 1

Using the NEW command, start a new drawing using the template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 04-2. (Figure Step 2A and 2B)

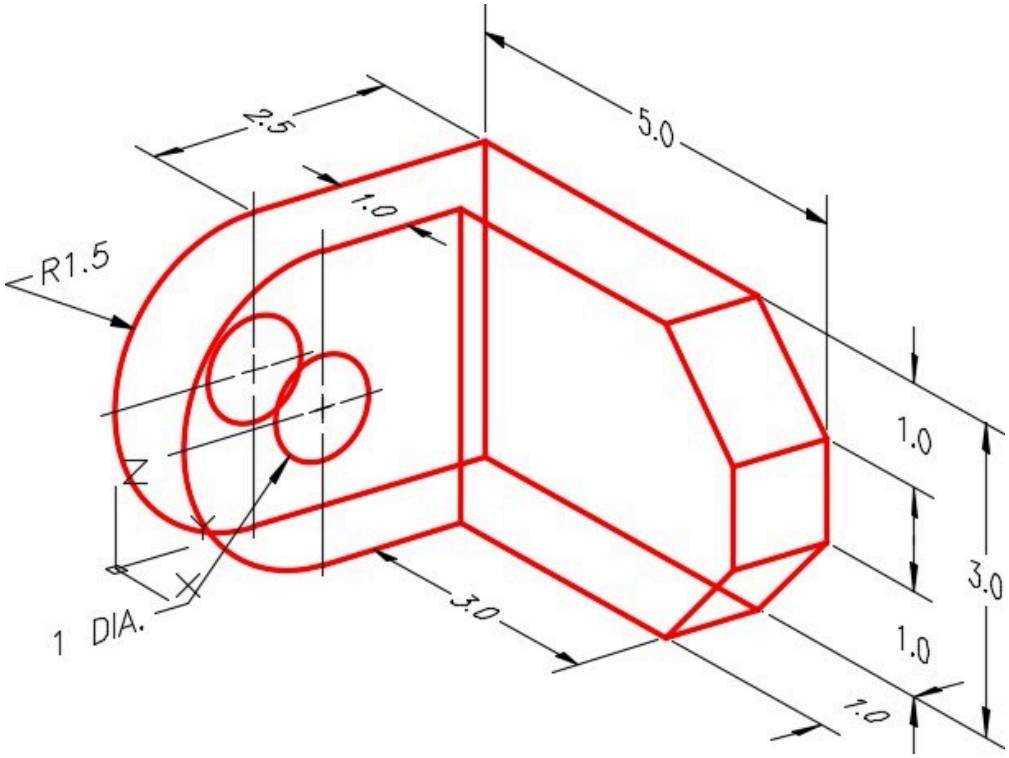


Figure Step 2A
Dimension Wireframe Model

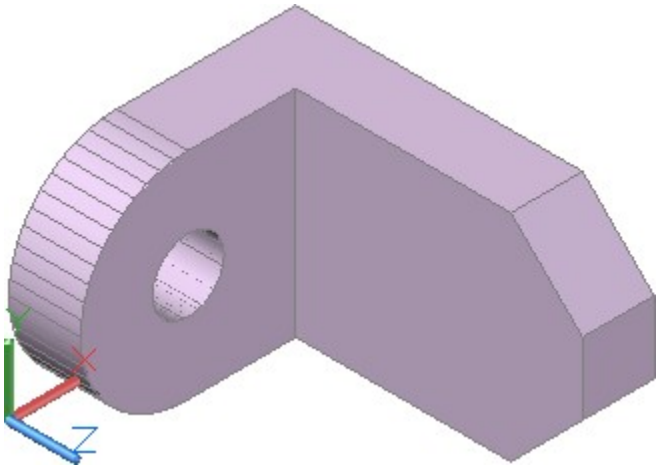


Figure Step 2B
Solid Model SE Isometric View

Step 3

Ensure that the current UCS is set to World and the current view is set to SE Isometric.

Step 4

Set layer: Model as the current layer.

Step 5

Using the LINE command, draw the lines to outline the Top view. (Figure Step 5)

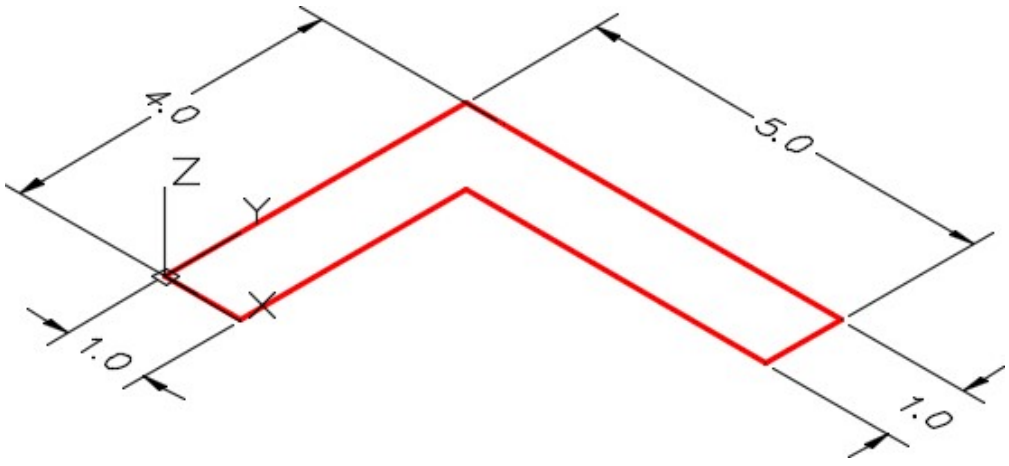


Figure Step 5

Step 6

Copy all of the lines 3 units in the positive Z direction. (Figure Step 6)

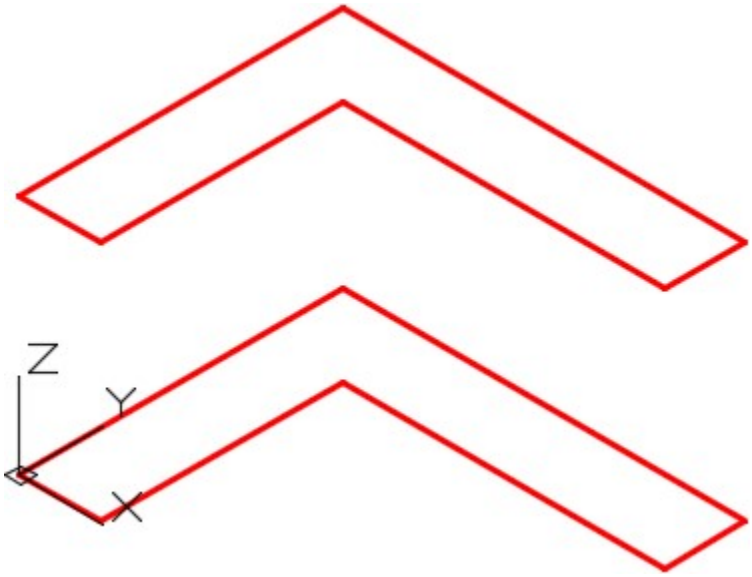


Figure Step 6

Step 7

Using the 3DFORBIT command, orbit the model slightly. (Figure Step 7)

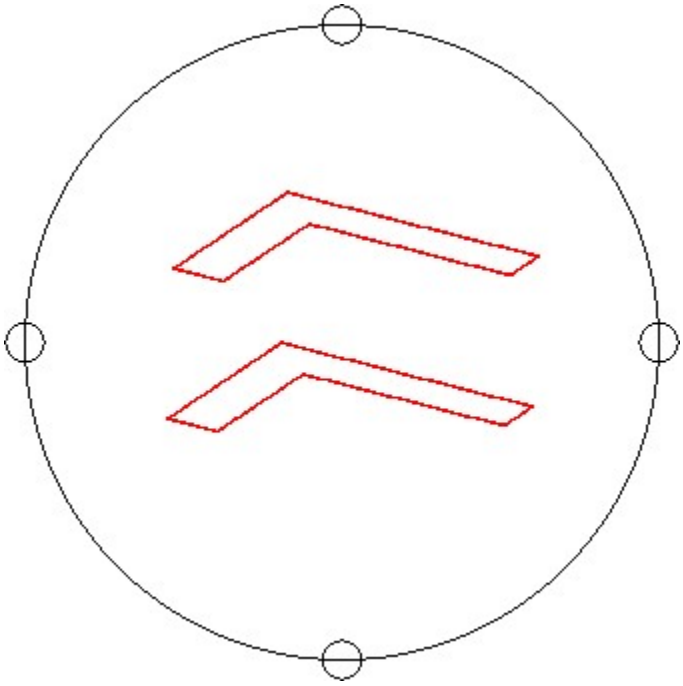


Figure Step 7

AUTHOR'S COMMENTS: I orbited the model so that the corner does not line up and the lines in the front do not cover the lines in the back.

Step 8

Using the LINE command, draw the 6 vertical lines. Ensure that you enable object snap and snap to endpoints of the existing lines. (Figure Step 8)

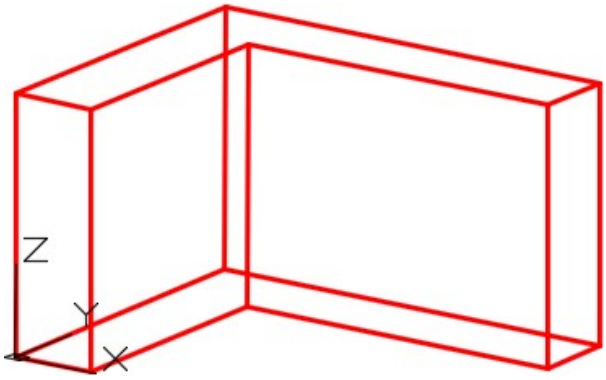


Figure Step 8

Step 9

Set the current UCS to Right and then locate the icon by snapping to the corner of the model as shown in the figure. (Figure Step 9)

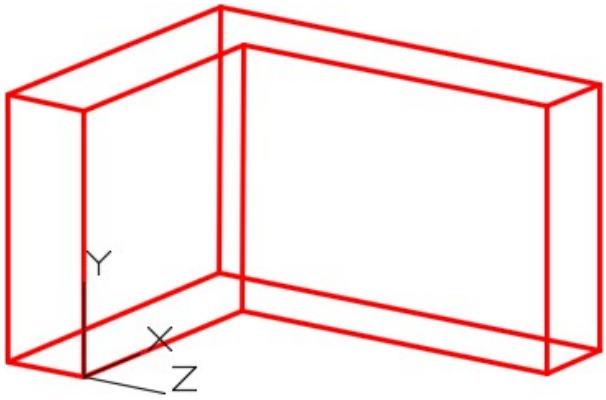


Figure Step 9

Step 10

Draw the necessary construction lines using the OFFSET command. Change their layer to layer: Construction. Draw the arc and the circle. (Figure Step 10)

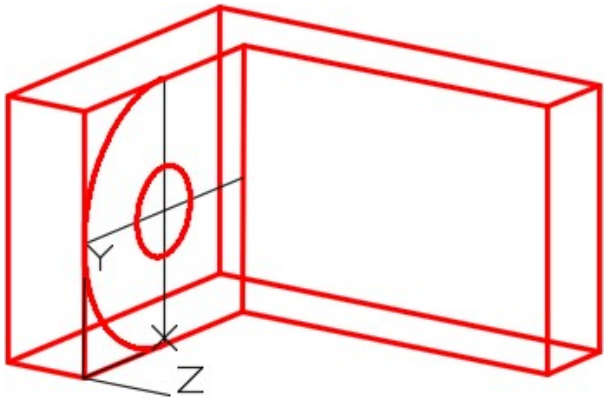


Figure Step 10

Step 11

Copy the arc and the circle in the negative Z direction. (Figure Step 11)

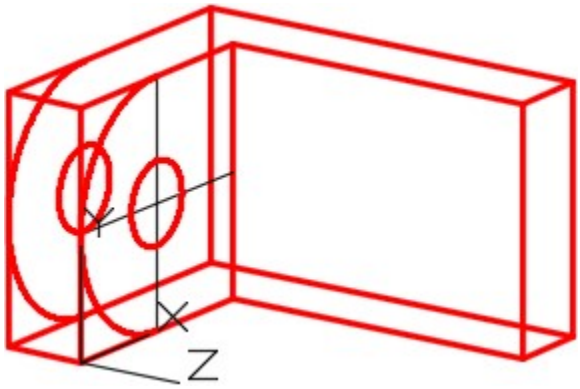


Figure Step 11

Step 12

Trim or delete necessary lines. (Figure Step 12)

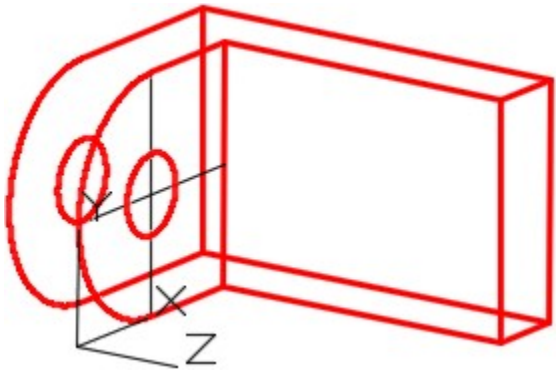


Figure Step 12

Step 13

Set the current UCS to Front and then locate it to the corner as shown in the figure. (Figure Step 13)

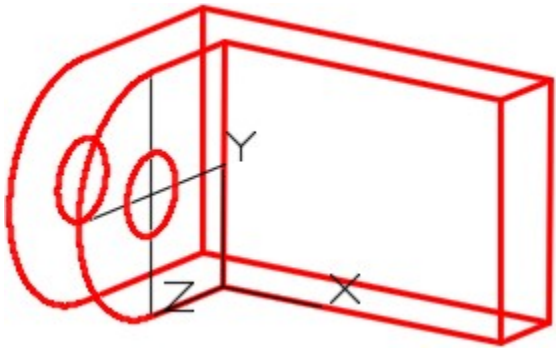


Figure Step 13

Step 14

Draw the necessary construction lines using the OFFSET command. Change their layer to layer: Construction. Draw the object lines as shown in the figure. (Figure Step 14)

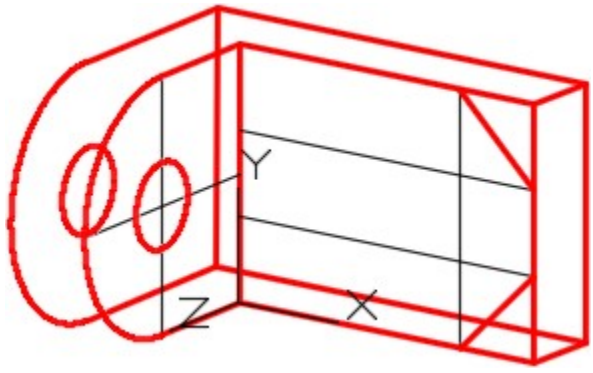


Figure Step 14

Step 15

Copy the lines in the negative Z direction and add the necessary object lines. (Figure Step 15)

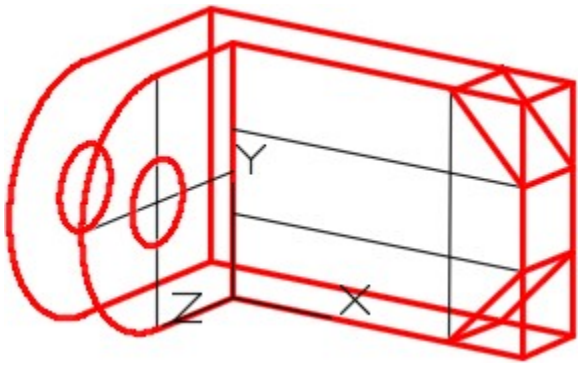


Figure Step 15

Step 16

Trim or delete the necessary lines and freeze layer: Construction to complete the wireframe model. (Figure Step 16)

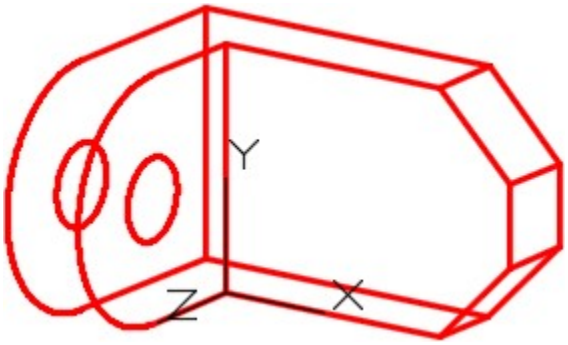


Figure Step 16

Step 17

Change the current view to SE Isometric. (Figure Step 17)

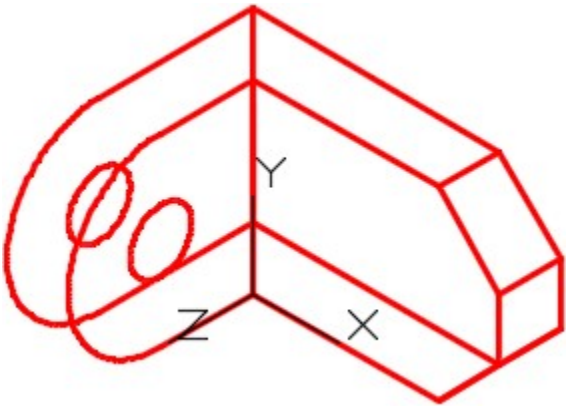


Figure Step 17

Step 18

Set the current UCS to World. (Figure Step 18)

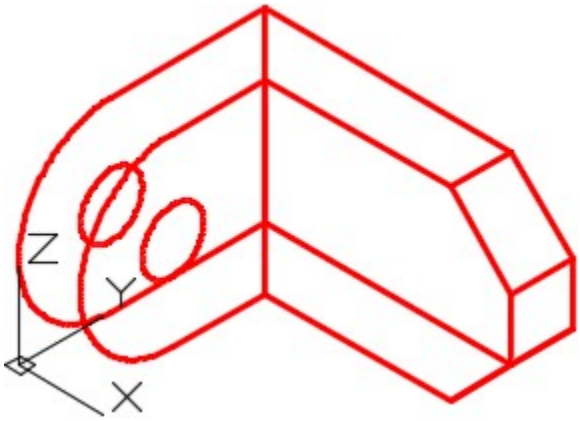


Figure Step 18

AUTHOR'S COMMENTS: The UCS must always be located on the World when inserting a key.

Step 19

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

Insert the key: AutoCAD 3D Workalong 04-2. It will overlay the model with a magenta wireframe model. (Figure Step 19)

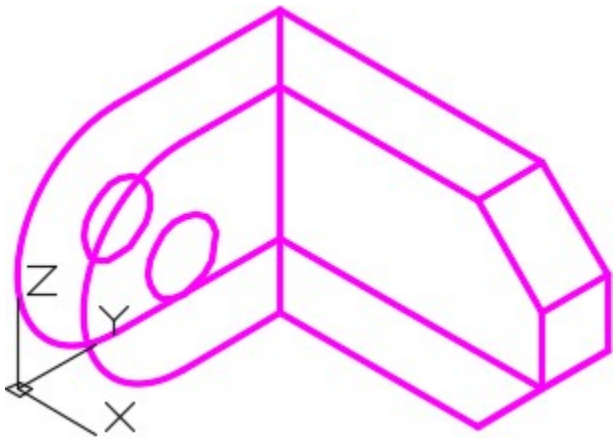
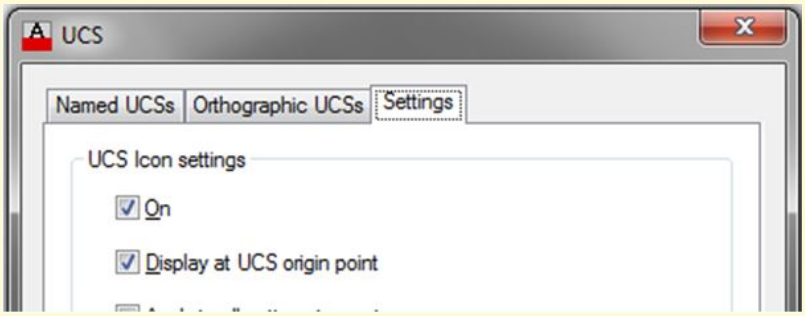


Figure Step 19

USER TIP: It is best to display the UCS icon at its origin point when you are constructing 3D wireframe models. See the Settings tab in the UCS dialogue box shown below.If the model is zoomed to fill the Graphic window, the UCS icon will appear to be located off the origin point. It actually is not, it simply cannot display

where it should. To correct, zoom out to make the model display smaller and the icon will automatically display at the UCS origin.

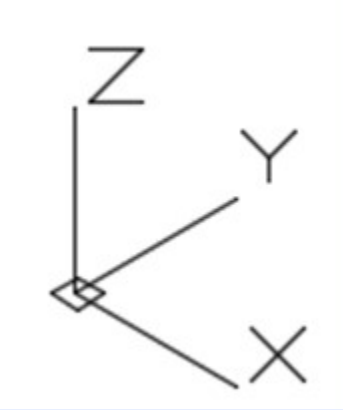


MUST KNOW: The coordinate location X0Y0Z0 is always referenced to the origin of the UCS. If the UCS is located at the WCS, its origin is the absoluteX0Y0Z0. If the UCS is at any other location, the coordinates location X0Y0Z0 is the relative X0Y0Z0. To work with absolute coordinates, the UCS **must** be located at the WCS. Use the UCS II toolbar or the UCS dialogue box and change the UCS location to World. When the UCS is at the WCS, the icon will display a small square.

Another way to work with absolute coordinates is to place an asterisk (*) preceding the coordinate value. i.e.

@*4,0,0 specifies a point four units in the X direction of the last point entered relative to the absolute X0Y0Z0 or the WCS.

@4,0,0 specifies a point four units in the X direction of the last point entered relative to the current UCS. *0,0,0 specifies absolute X0Y0Z0 of the WCS regardless of the current location of the UCS.



Key Principles

Key Principles in Module 4

1. The User Coordinate System (UCS) is the coordinate system used to construct 3D It can be located at the WCS or at any location or orientation in 3D space. Being able to locate and orientate the UCS anywhere is the secret of 3D modeling.
2. Another way to work with absolute coordinates is to place an asterisk (*) preceding the coordinate i.e.

@*4,0,0 specifies a point four units in the X direction of the last point entered relative to the absolute X0Y0Z0 or the WCS.

*0,0,0 specifies absolute X0Y0Z0 of the WCS regardless of the current location of the UCS.

Lab Exercise 4-1

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 04-1	3D Layout English	Inches

Step 1

Save and name the drawing: AutoCAD 3D Lab 04-1 as shown above.

Step 2

Draw all construction objects on layer: Construction and all model objects on layer: Model.

Step 3

Start your model with the current view in SE Isometric. If required, orbit it slightly with 3DFORBIT to help your line of sight.

Step 4

Draw a wireframe model of the object. (Figure Step 4A and 4B)

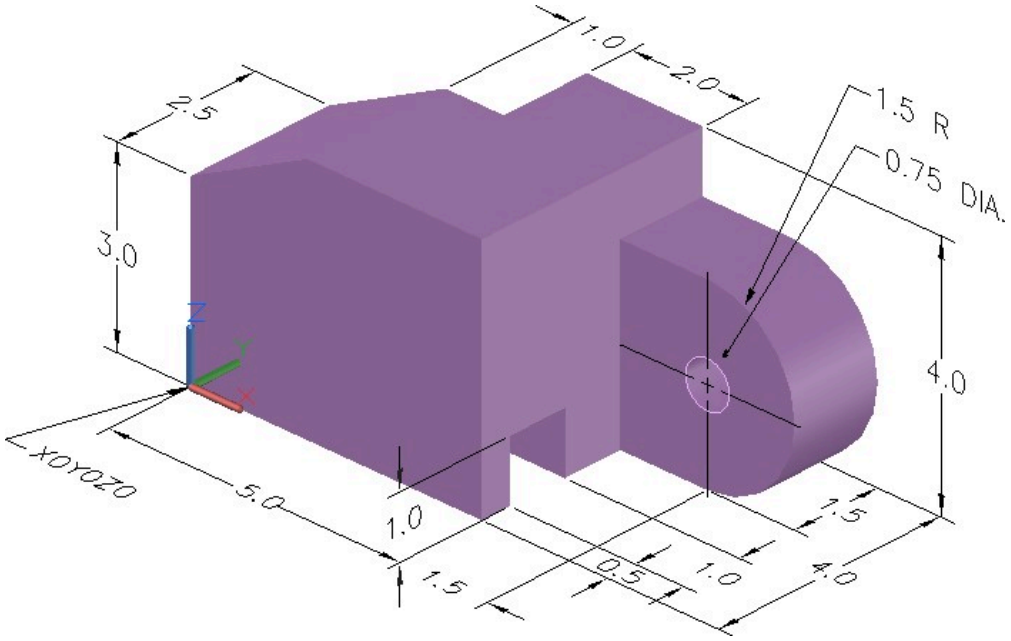


Figure Step 4A
Dimensioned Solid Model

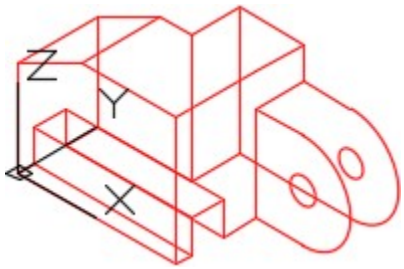


Figure 4B Completed Wireframe Model SE Isometric View

Step 5

When complete, freeze layer: Construction.

Step 6

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

Step 7

Change the current UCS to World and check the model with the key.

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.

Hint 1

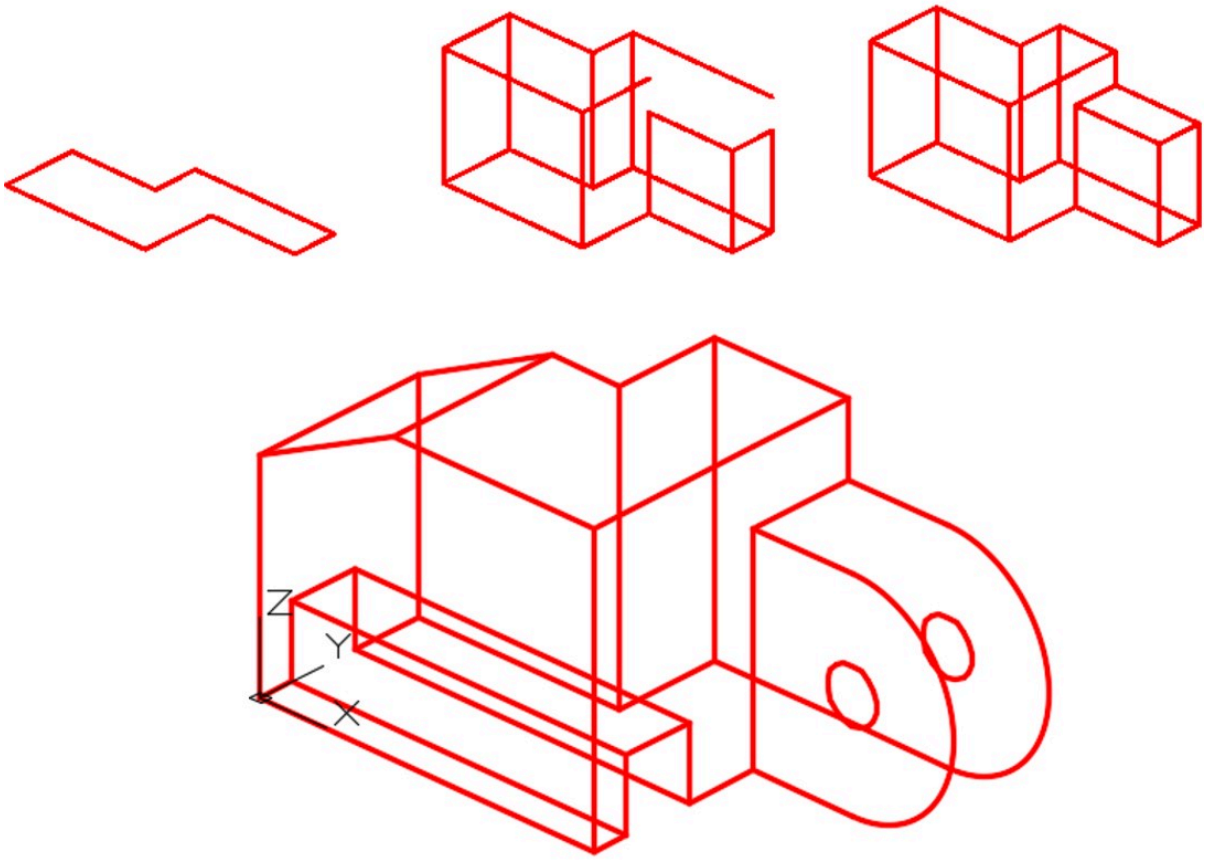


Figure Hint 1
Construction Steps

Lab Exercise 4-2

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 04-2	3D Layout Metric	Millimeters

Step 1

Save and name the drawing: AutoCAD 3D Lab 04-2, as shown above.

Step 2

Draw all construction objects on layer: Construction and all model objects on layer: Model.

Step 3

Start your model with the current view in SE Isometric. If required, orbit it slightly with 3DFORBIT to help your line of sight.

Step 4

Draw a wireframe model of the object. (Figure Step 4A, 4B, and 4C)

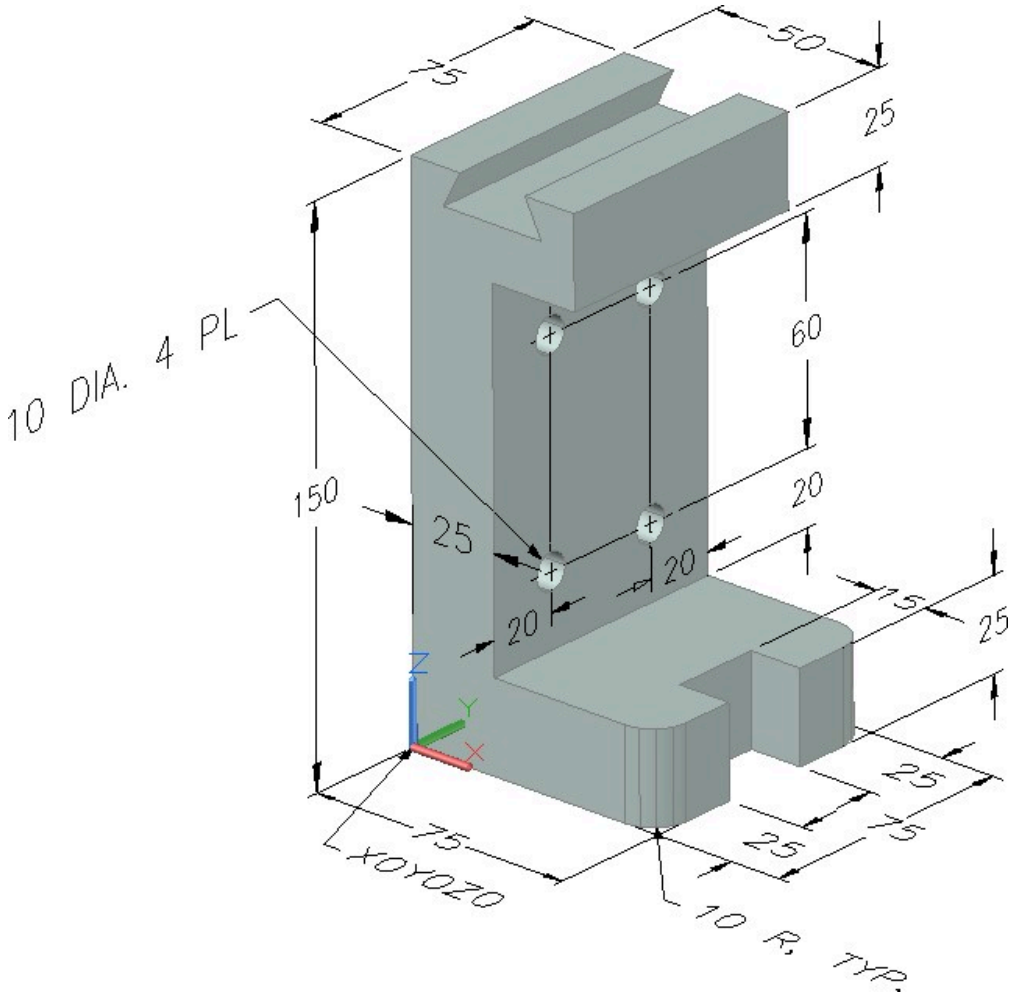


Figure Step 4A
Dimensioned Solid Model

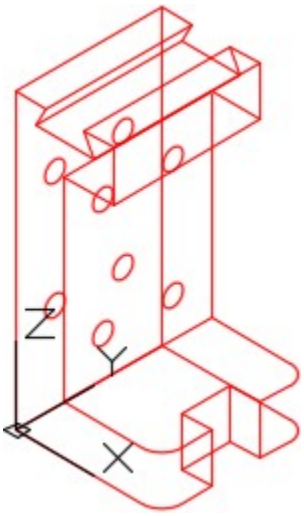


Figure Step 4B Completed
Wireframe Model SE
Isometric View

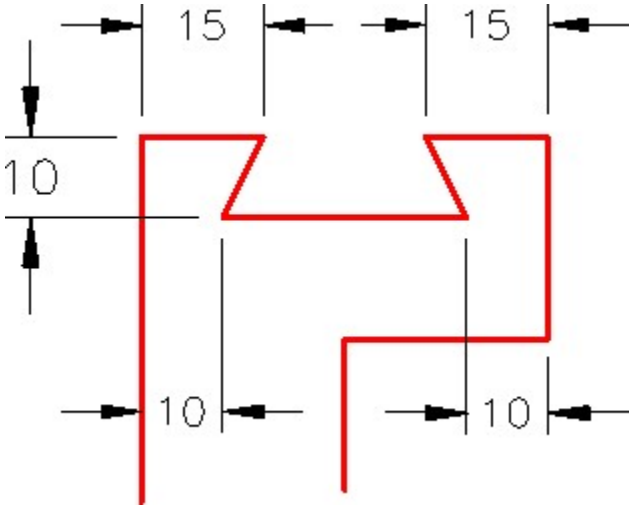


Figure Step 4C
Key Detail

AUTHOR'S COMMENTS: Start the first view on the Front UCS. Before you start drawing, change the location of the UCS to Front.

Step 5

When complete, freeze layer: Construction.

Step 6

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

Step 7

Change the current UCS to World and check the model with the key.

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.

Hint 1

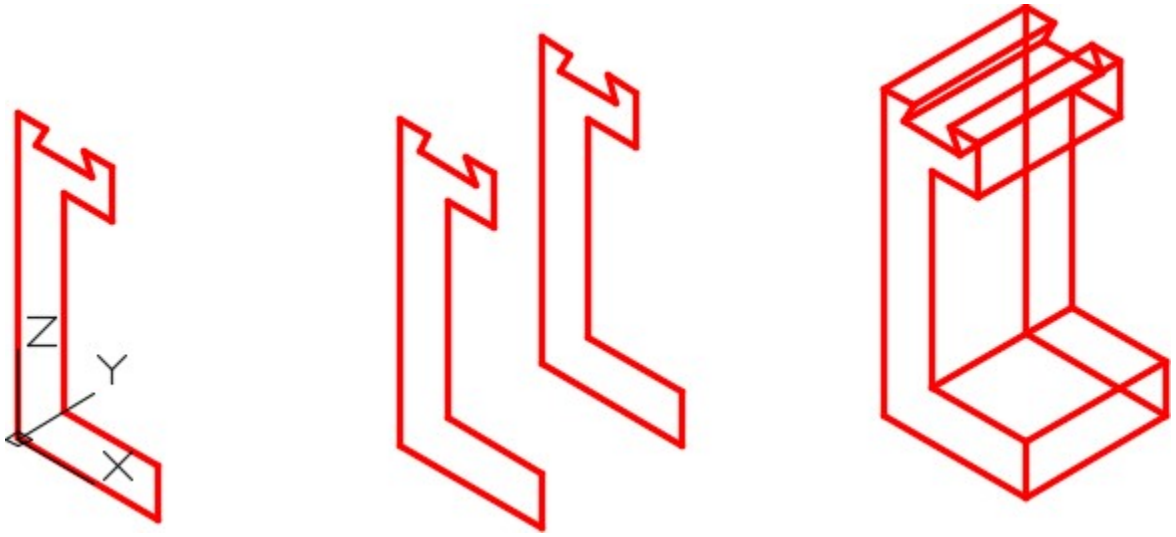


Figure Hint 1

Lab Exercise 4-3

Time allowed: 40 minutes.

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

Step 1

Save and name the drawing: AutoCAD 3D Lab 04-3, as shown above.

Step 2

Draw all construction objects on layer: Construction and all model objects on layer: Model.

Step 3

Start your model with the current view in SE Isometric. If required, orbit it slightly with 3DFORBIT to help your line of sight.

Step 4

Draw a wireframe model of the object. (Figure Step 4A, 4B, and 4C)

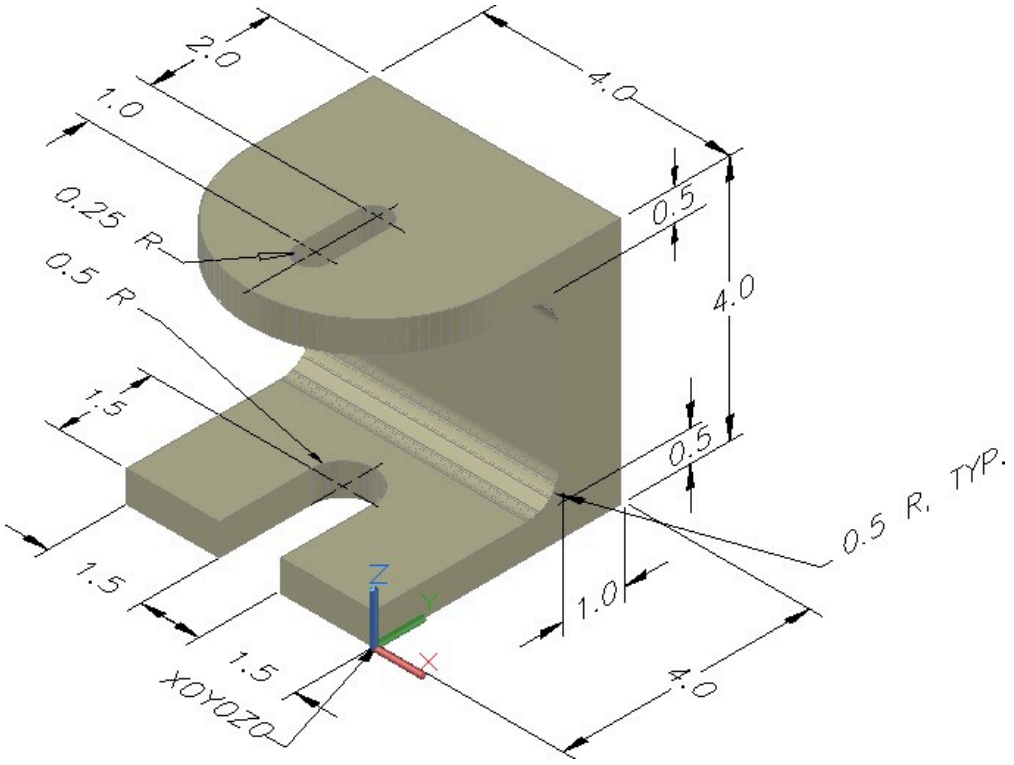


Figure Step 4A Dimensioned Solid Module SE Isometric View

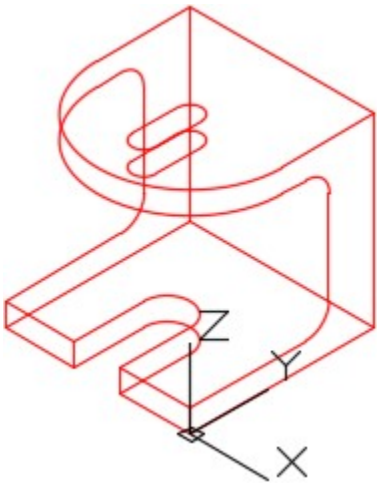


Figure Step 4B
Completed Wireframe Model SE Isometric View

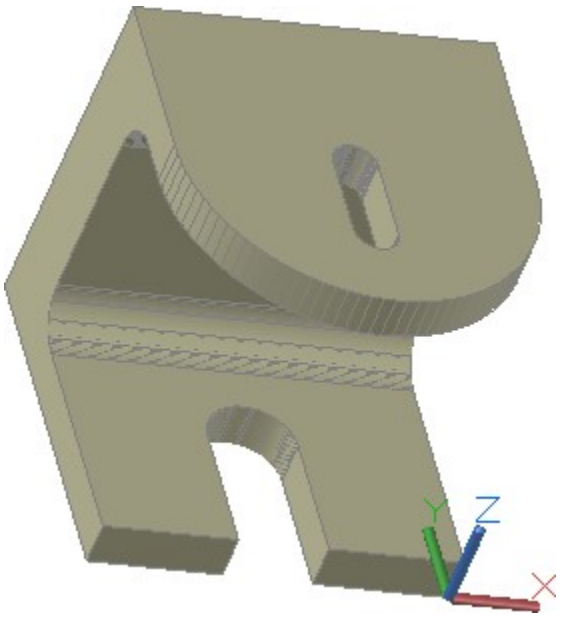


Figure Step 4C Solid Model Rotated View

AUTHOR'S COMMENTS: Start the first view on the Right UCS. Before you start drawing, change the location of the UCS to Right.

Step 5

When complete, freeze layer: Construction.

Step 6

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

Step 7

Change the current UCS to World and check the model with the key.

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.

Hint 1

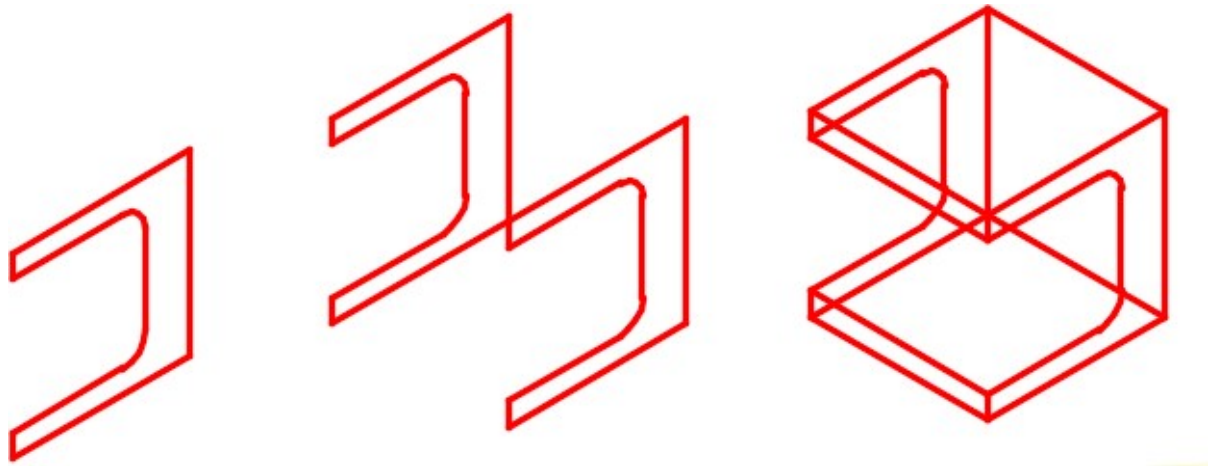


Figure Hint 1

Module 5: User Coordinate System – Part 2

Learning Outcomes

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

1. Describe and apply the UCS command to locate and orientate the UCS to any location necessary for model construction.
2. Draw 3D models using the UCS command to locate and orientate the UCS.

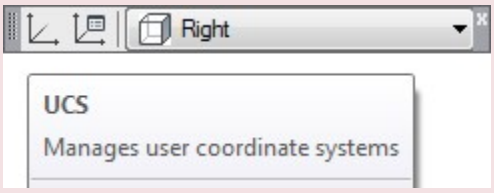
Locating the UCS in 3D Space

In Module 4, the UCSMAN command and the UCS II toolbar or ribbon were used to locate the UCS at preset orthographic locations on the model. In this module, the UCS command and the UCS toolbar or ribbon will be used to move and orientate the UCS to locations that are required to construct more complicated models. The most useful option is locating the UCS by selecting 3 points on the model. While it is still necessary to use the predefined orthographic UCS locations for model construction, not all models can be built using only them.

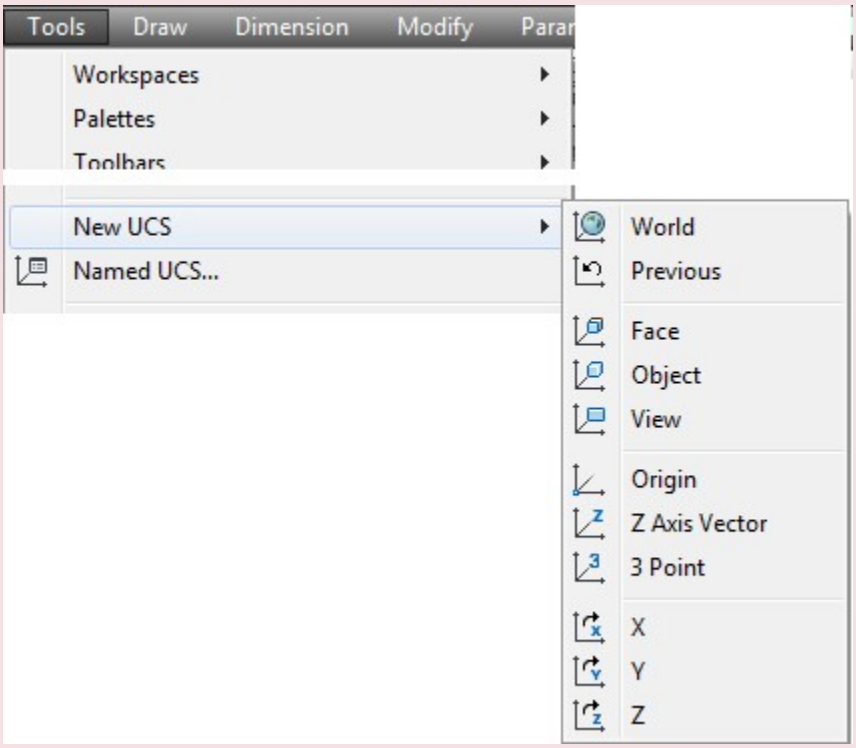
AutoCAD Command: UCS

The UCS command is used to locate and orientate the UCS on the model or in model space.

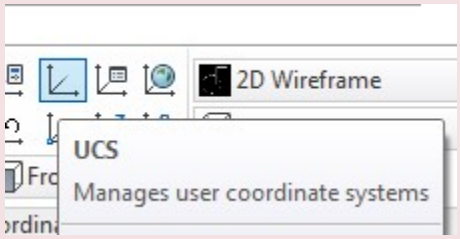
Shortcut: none



UCS II Toolbar



Tool Pull-down



Home Ribbon

WORK ALONG: Locating and Orientating the UCS Using the UCS Toolbar

Step 1

Using the NEW command, start a new drawing using the template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 05-1.

Step 3

Set the UCS to World. Enter the UNITS command and in the Units dialogue box, set the Insertion

Units to Inches. Using the INSERT command, insert the block: AutoCAD 3D Lab 01-1 at the coordinates 0,0,0.

Step 4

Explode the block you inserted in step 3.

Step 5

Set the current view to SE Isometric. Your model should appear as shown in the figure. (Figure Step 5).

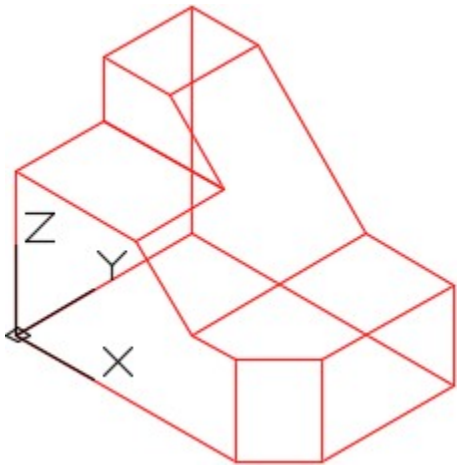


Figure Step 5

Step 6

Set the UCS location to Right. (Figure Step 6)

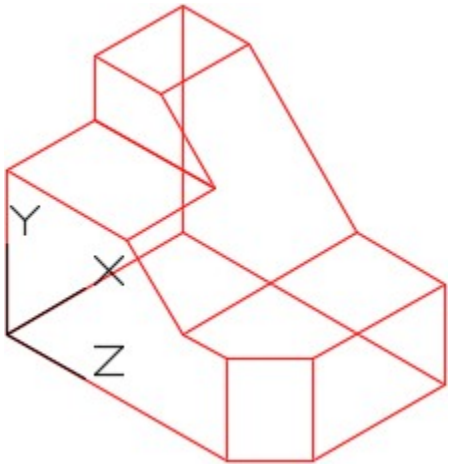


Figure Step 6

Step 7

Click the Origin icon and then snap to the end of the line shown in Figure Step 7A. Your model should now appear as shown in the figure. (Figure Step 7A and 7B)

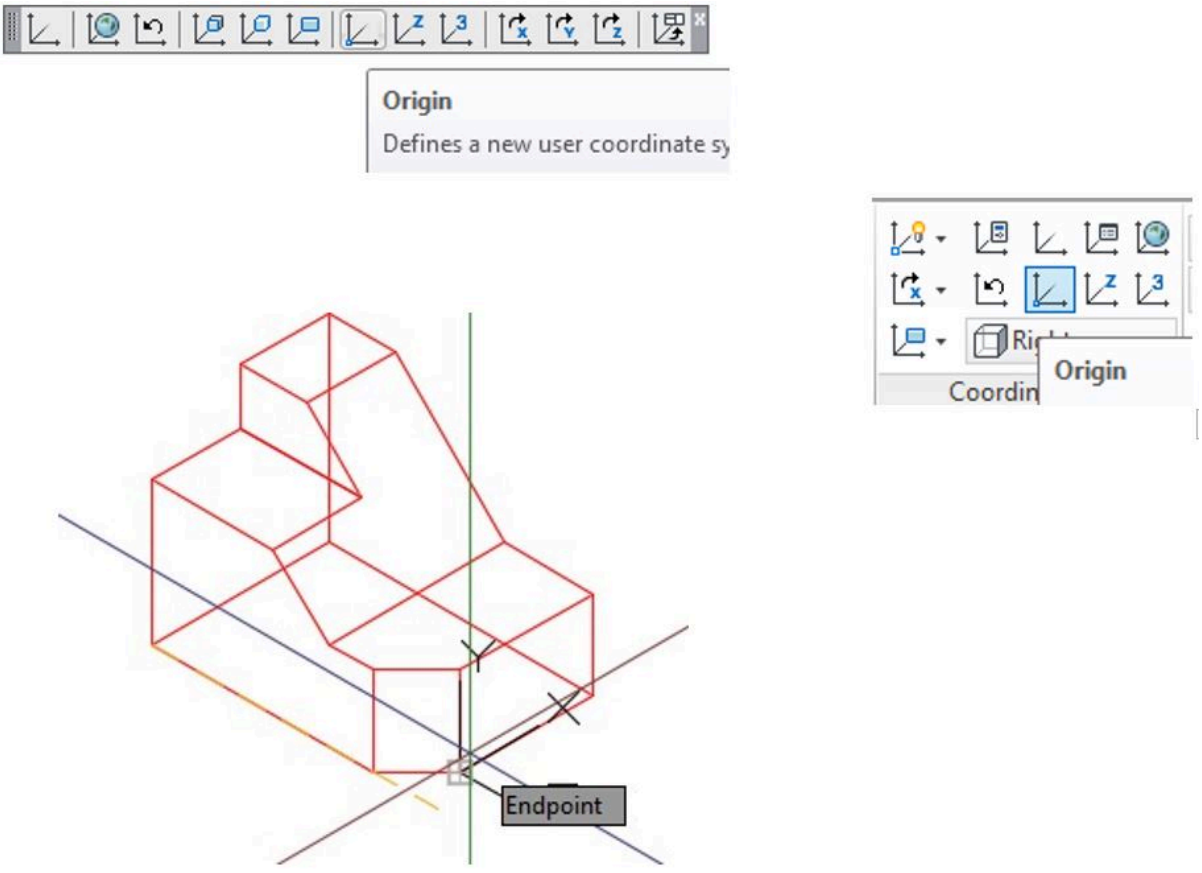


Figure Step 7A

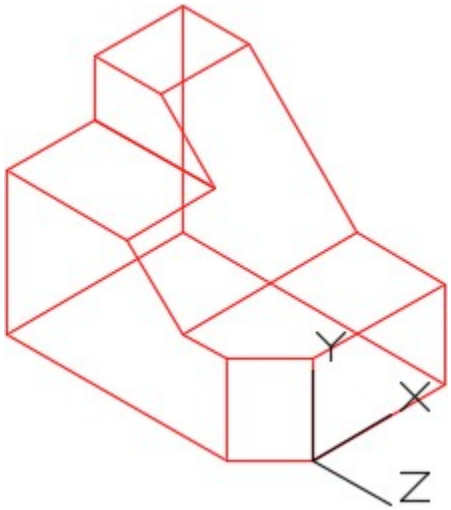


Figure Step 7B

Step 8

Enable object snap. Click the 3 Point icon. When prompted, select the end of the inclined line. This is the X0Y0Z0 of the UCS. (Figure Step 8)

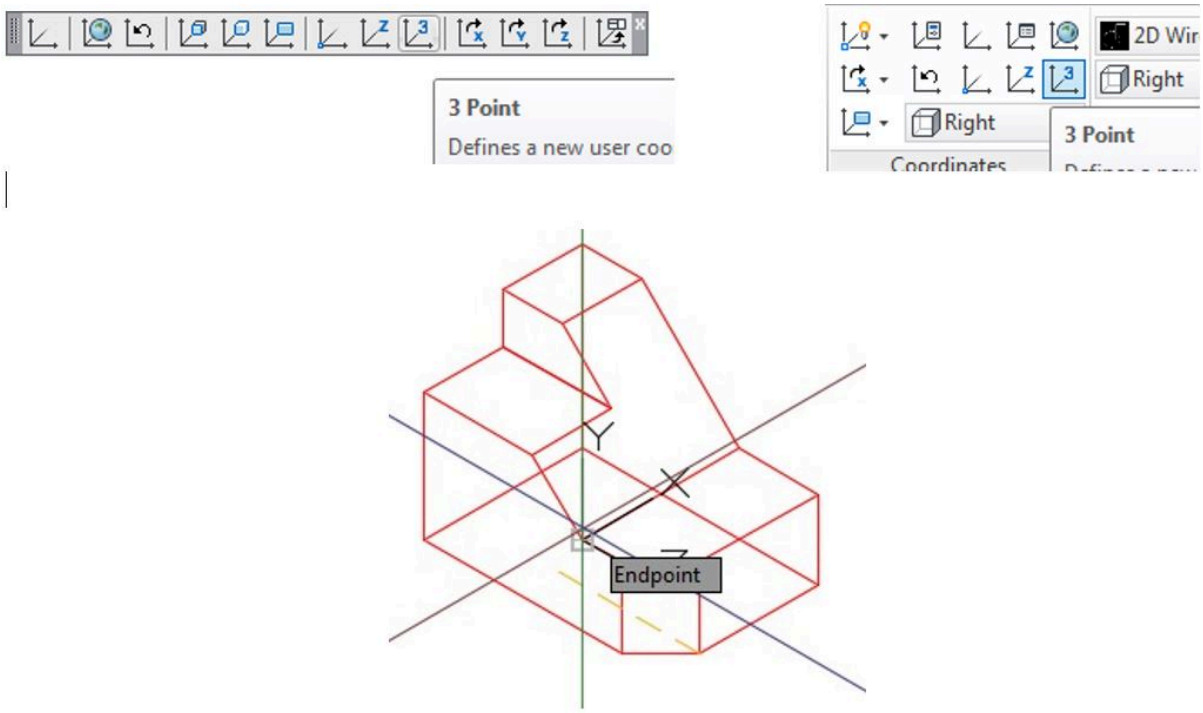


Figure Step 8

Step 9

For the second point, select end of the inclined line on the right side of the model. This is the positive X axis. (Figure Step 9)

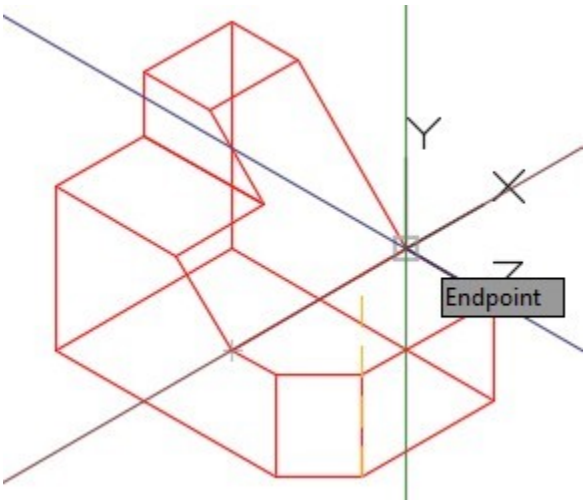


Figure Step 9

Step 10

For the third point, select the end of the inclined line on the top. This is the positive Y axis. Your model should now appear as shown in the figure. (Figure Step 10A and 10B)

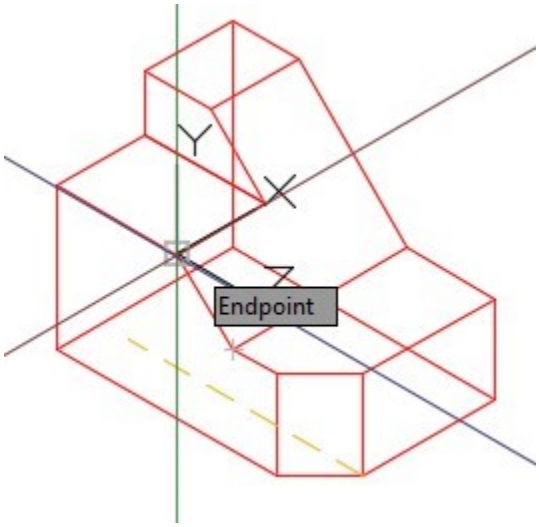


Figure Step 10A

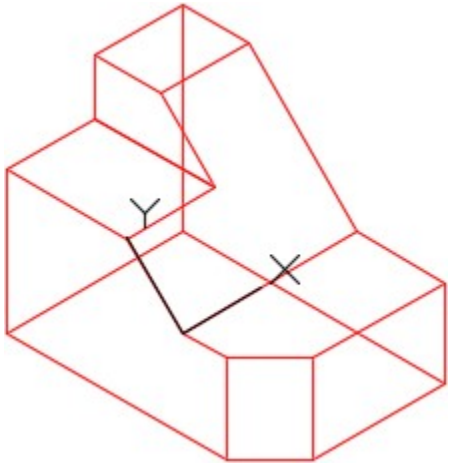


Figure Step 10B

AUTHOR'S COMMENTS: Use the 3 Point method of orientating the UCS as much as possible. It is a very easy to use and will speed your model drawing time. The order of selecting the points is important.

Pick 1 – The origin or X0Y0Z0.

Pick 2 – The positive X Axis.

Pick 3 – The positive Y axis.

Step 11

Using what you learned in Steps 8 to 10, locate the UCS as shown in the figure. Ensure that the UCS plane lies on the inclined plane. (Figure Step 11)

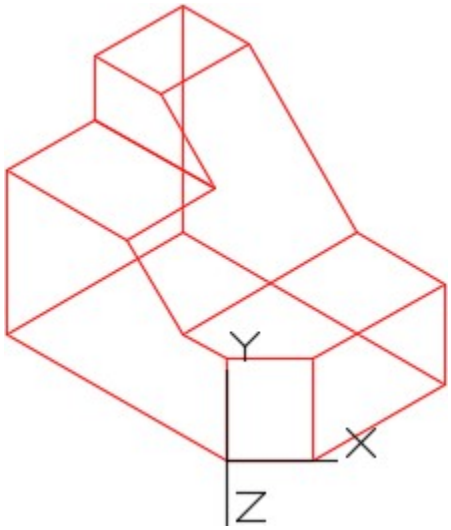


Figure Step 11

Step 12

Using what you learned in Step 8 to 10, locate the UCS as shown in the figure. Ensure that the UCS plane lies on the plane. (Figure Step 12.)

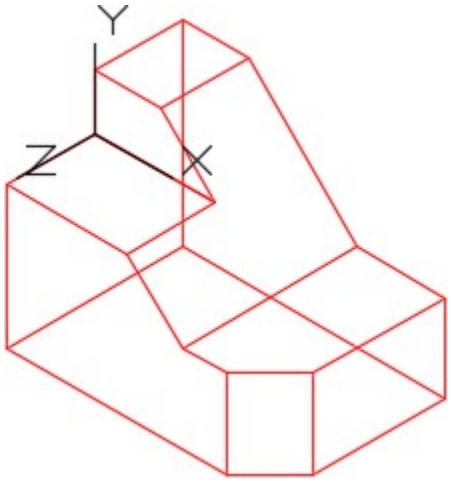


Figure Step 12

Step 13

Click the World UCS icon. The UCS icon will now be located as shown in the figure. (Figure Step 13)

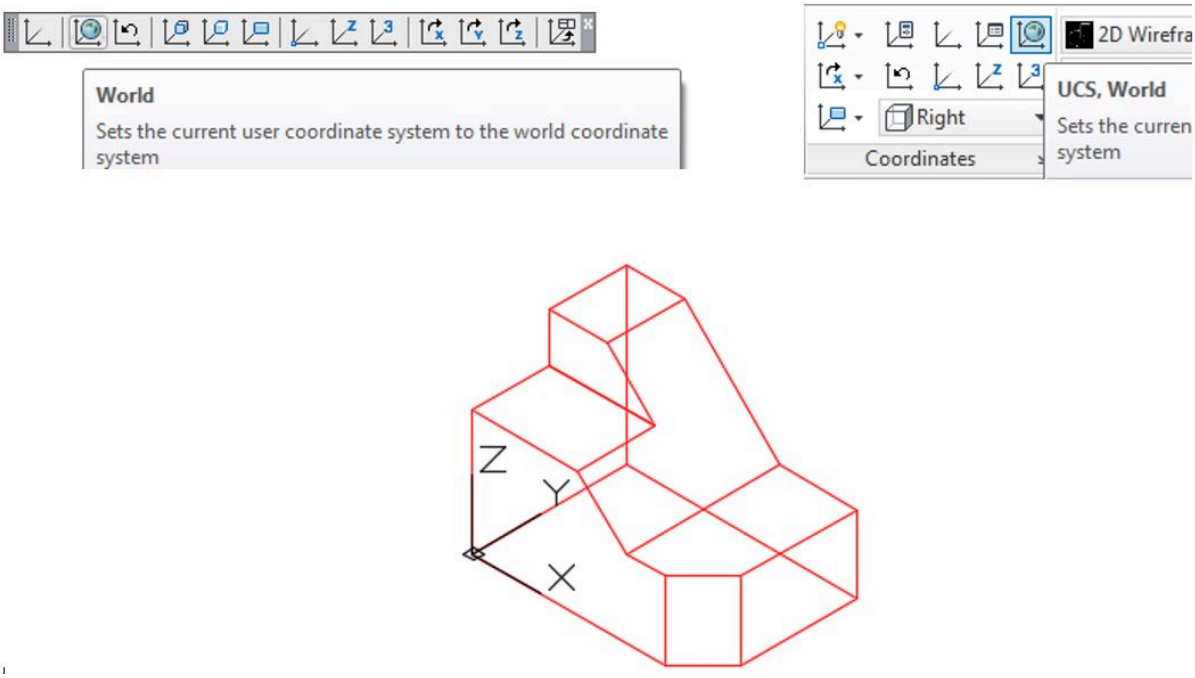
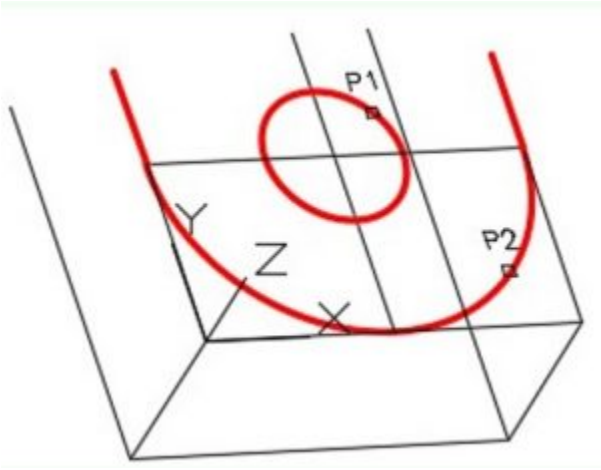


Figure Step 13

Step 14

Click the UCS Previous icon and the UCS should now return back to last location. (Figure Step 14)



Step 1

Step 15

Save and close the drawing.

USER TIP: Before starting to draw any 3D model, it is important for you to pick the best view to start with. The first view you draw is called the Base view. Usually it is the view with the most difficult contour. The more models you draw the easier it will be to select the best Base view.

WORK ALONG: Drawing a 3D Wireframe Model by Locating the UCS

Step 1

Using the NEW command, start a new drawing using the template: 3D Layout Metric.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 05-2. (Figure Step 2A, 2B, and 2C)

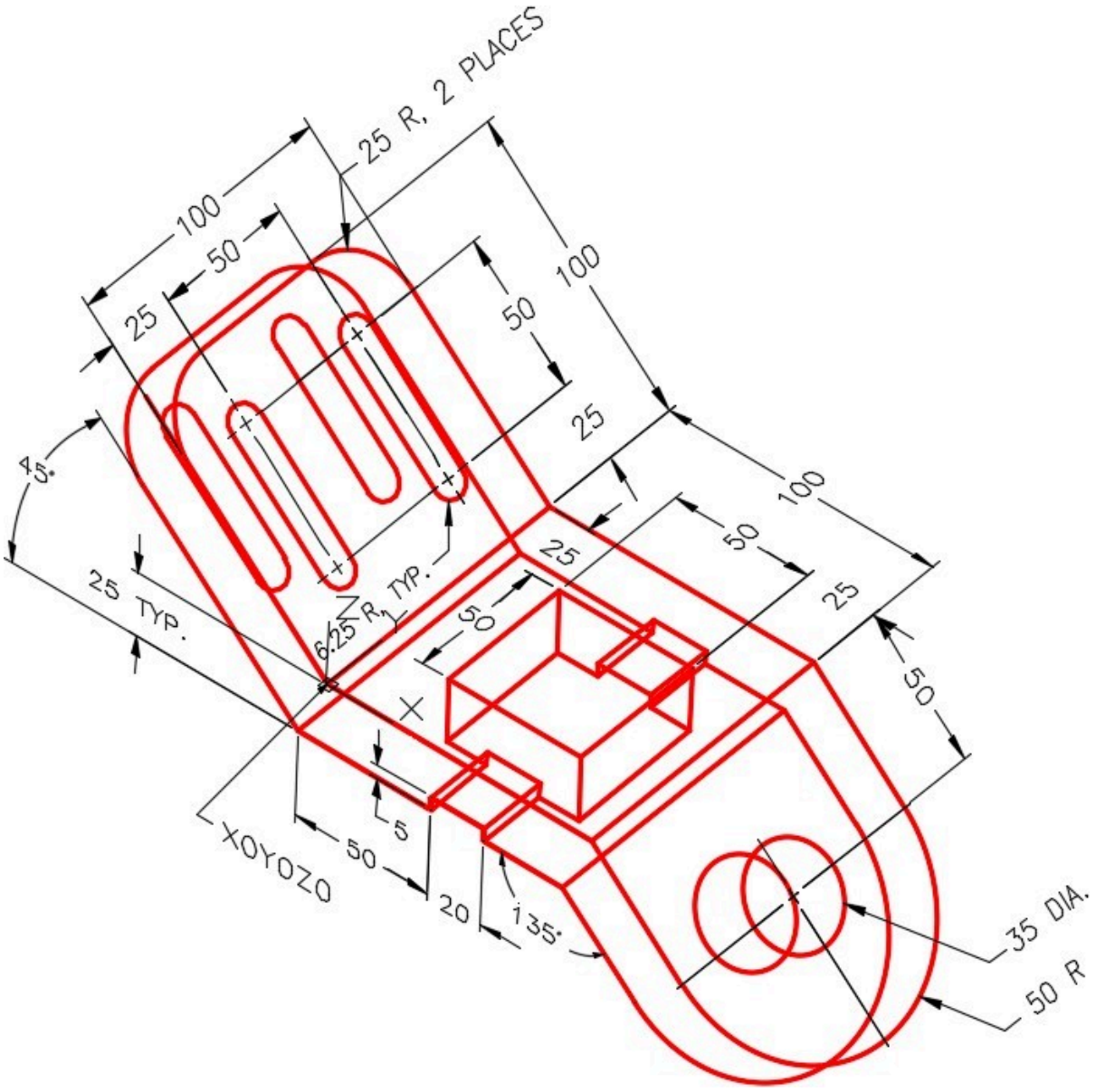
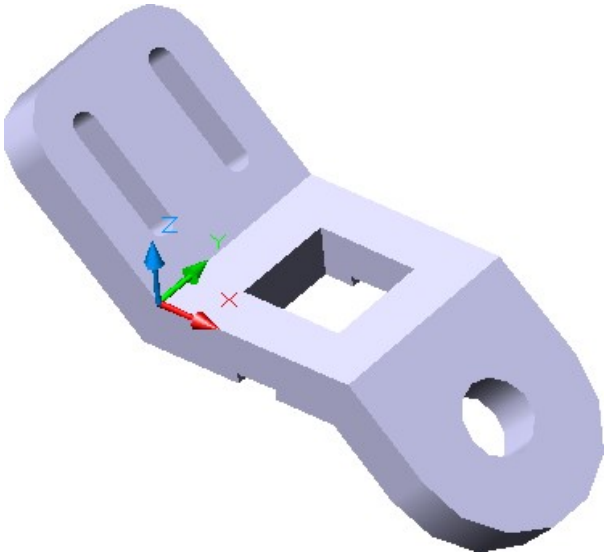


Figure Step 2A
Dimensioned Wireframe Model SE Isometric View



Solid Model – View from Top.

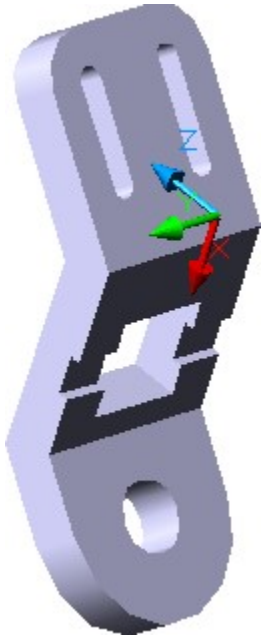


Figure Step 2C
Solid Model – View from Bottom.

Step 3

If you are using toolbar menus, you will be using the UCS, UCS II, and View toolbars. If you are using ribbon menus, enable the Home tab.

Step 4

Ensure that the UCS is located at World. (Figure Step 4)

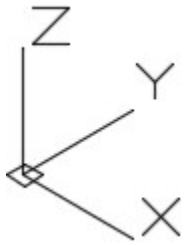


Figure Step 4

AUTHOR'S COMMENTS: Before you start drawing any model, you must first decide which view is the best to start your model. Usually it is the view with the most difficult contour. In this model, the Base view is the Front view.

Step 5

Change the UCS location to Front. (Figure Step 5)

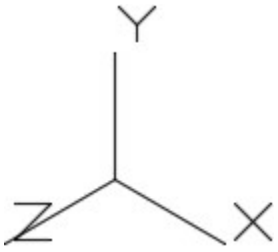


Figure Step 5

Step 6

Change the current view to Front. The UCS icon should now appear as shown in the figure. (Figure Step 6)

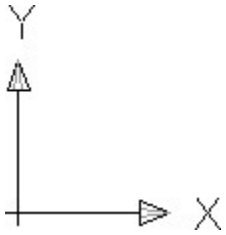


Figure Step 6

Step 7

Set layer: Model as the current layer. Draw the front contour of the model as shown in the figure. Drawing it in 2D is sometimes easier. It could have been drawn in the 3D view also. (Figure Step 7)

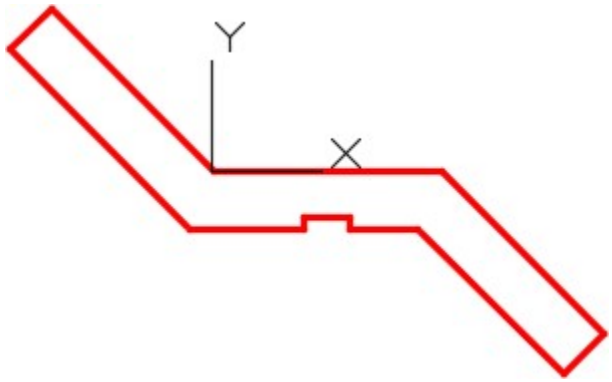


Figure Step 7

AUTHOR'S COMMENTS: It is ok to draw the first view of a model in a 2D view. As soon as you complete the Base view and are ready to copy the objects to make it three dimensional and from that point on, you must draw in a 3D view.

Step 8

Change the current view to SE Isometric. Your model should now appear as shown in the figure. (Figure Step 8)

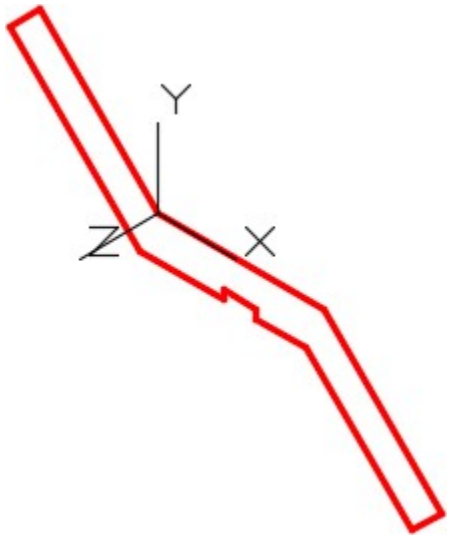


Figure Step 8

Step 9

Copy the lines 100 millimeters in the negative Z direction.

Step 10

Orbit the model slight as shown in the figure. Add the lines as shown in the figure. (Figure Step 10)

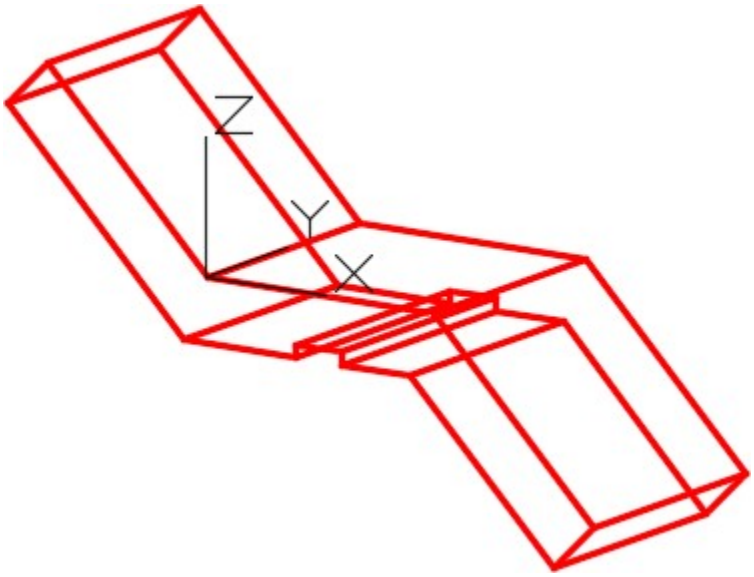


Figure Step 10

Step 11

Using the 3 Point method, locate the UCS to the top of the inclined plane. (Figure Step 11)

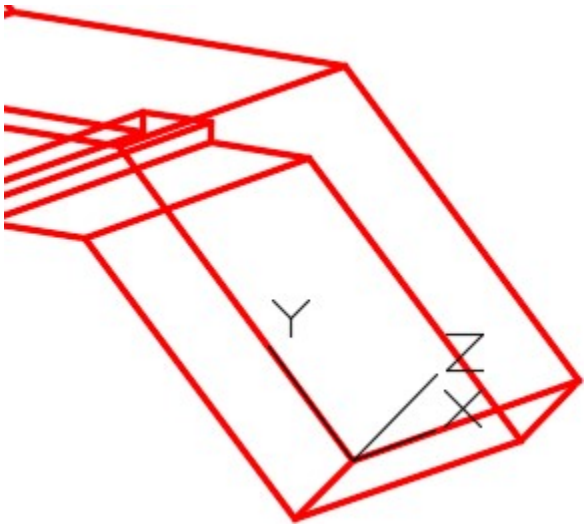


Figure Step 11

Step 12

Using OFFSET command, offset the existing lines to locate the centre of the circle and arc on the inclined plane. Change the lines to layer: Construction. Insert the circle and the arc and copy them 25

millimeters to the bottom plane. Before you do complete this step, read the User Tip at the end of this workalong. (Figure Step 12)

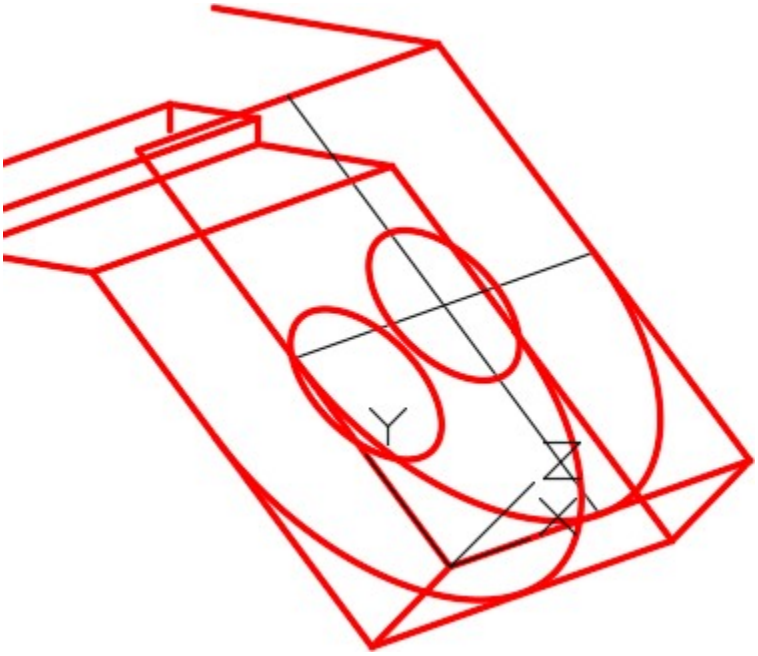


Figure Step 12

Step 13

Trim or delete the lines as required. Using the 3 Point method, locate the UCS to the upper incline plane. (Figure Step 13)

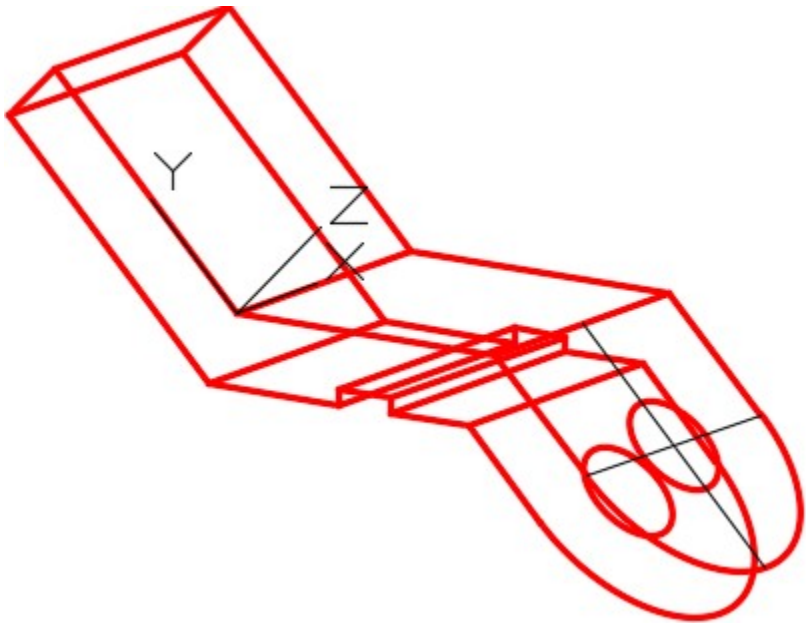


Figure Step 13

Step 14

Using the OFFSET command, offset the existing lines to locate the centre of the slots on the upper inclined plane. Change the offset lines to layer: Construction. Draw the slots and copy them 25 units to the bottom of the inclined plane. (Figure Step 14)

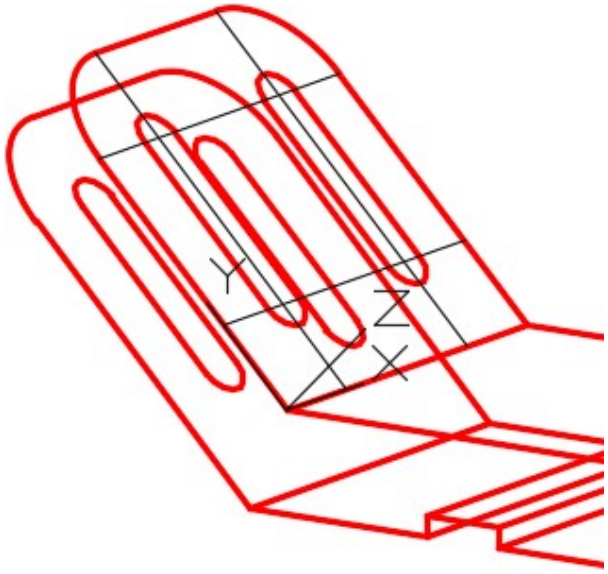


Figure Step 14

Step 15

Using the 3 Point method, locate the UCS to the top plane. (Figure Step 15)

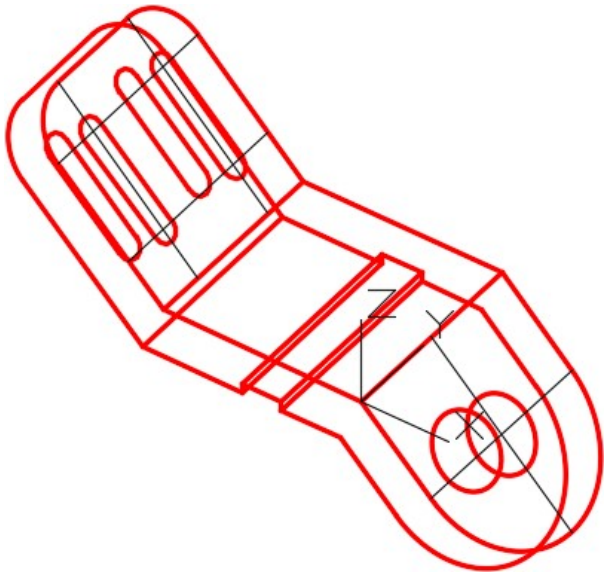


Figure Step 15

Step 16

Construct the square hole, copy it and trim the lines. (Step 16A and 16B)

Figure Step 16A

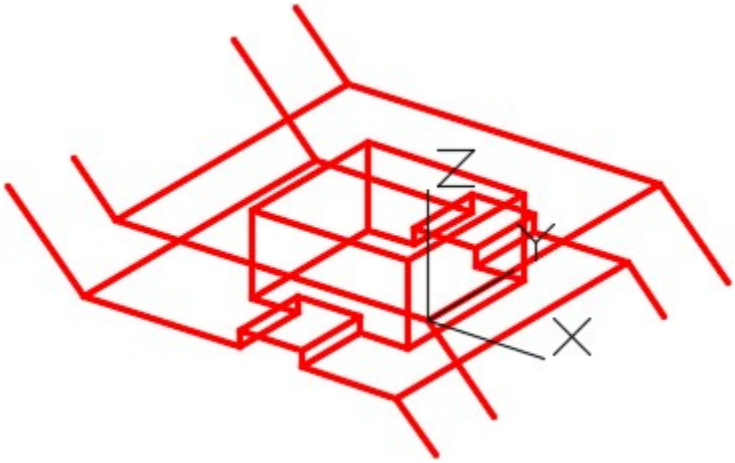


Figure Step 16B

Step 17

Freeze layer: Construction. Change the current UCS to World and the current view to SE Isometric. (Figure Step 17)

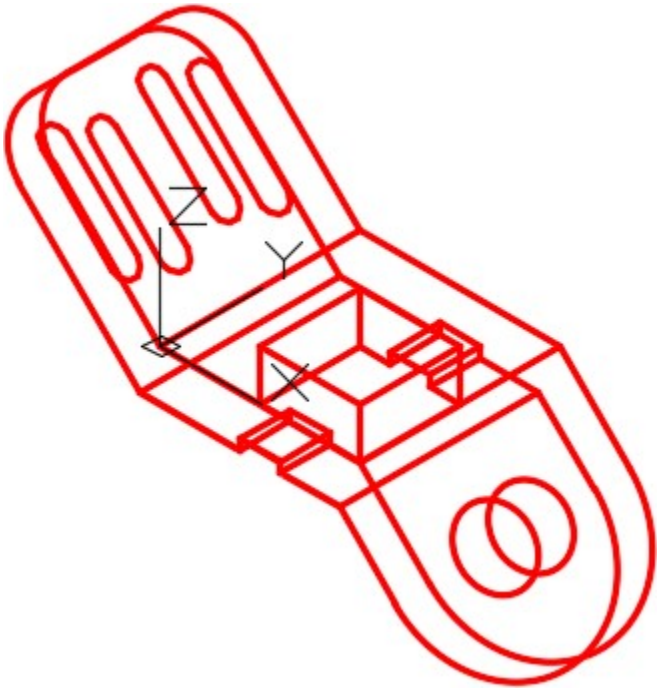


Figure Step 17

Step 18

Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Millimeters. Insert the key: AutoCAD 3D Workalong 05-2. It will overlay the model with a magenta model. If it overlays your model perfectly, your model is accurately drawn. (Figure Step 18)

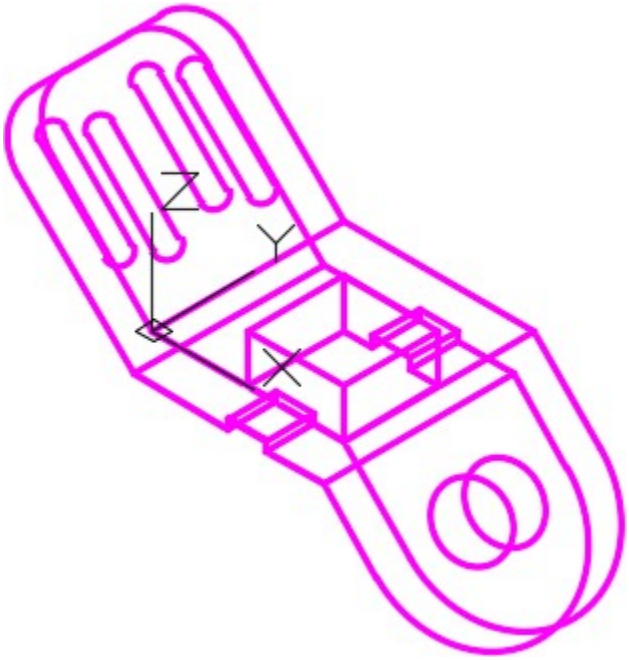


Figure Step 18

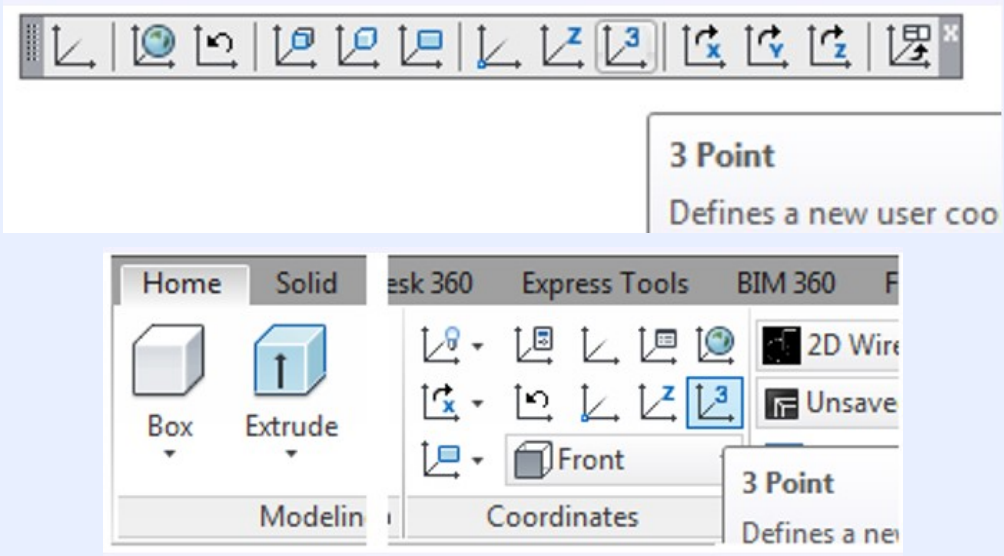
Step 19

Save and close the drawing.

MUST KNOW: The 3 Point icon is the easiest method of changing the UCS to lay on a plane. The order of selecting the 3 points is important. The order is as follows:

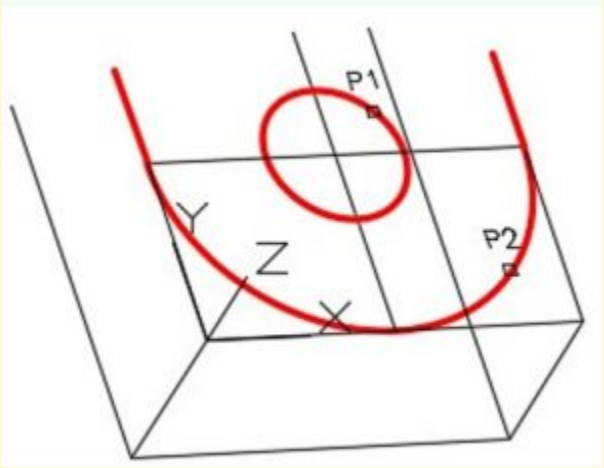
1. The X0Y0Z0 or origin of the UCS.
2. The direction of the positive X axis.
3. The direction of the positive Y axis.

Ensure that you enable object snap to snap to existing objects to locate the UCS correctly.

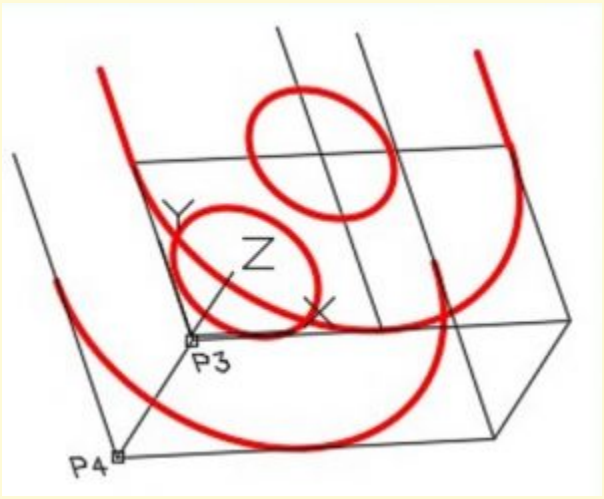


USER TIP: If you have to copy existing geometry in the Z direction, an easy way to do it is to use the COPY command and select the objects first (P1 and P2 – Step 1).

To indicate the copy distance and direction, snap to the ends of an existing line (P3 and P4 in Step 2).



Step 1



Step 2

MUST KNOW: The first view of a 3D wireframe model can be drawn in a 2D view. As soon as the objects are copied to make it 3 dimensional, the current view must be changed to a 3D view to complete constructing the model.

Key Principles

Key Principles in Module 5

1. Use the 3 Point method of orientating the UCS as much as possible. It is a very easy to use and will speed your model drawing time. The order of selecting the points is important. Pick 1 – The origin or X0Y0Z0, Pick 2 – The positive X Axis, Pick 3 – The positive Y axis.
2. Before you start drawing any model, you must first decide what is the best view to start drawing first. Usually it is view with the most complex contour. This is called the Base view.
3. The first view of a 3D wireframe model can be drawn in a 2D view. As soon as the objects are copied to make it three dimensional, the view must be changed to a 3D view before continuing the model construction.

Lab Exercise 5-1

Time allowed: 60 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 05-1	3D Layout English	Millimeters

Step 1

Save and name the drawing: AutoCAD 3D Lab 05-1.

Step 2

Draw a wireframe model of the object shown in the figure. (Figure Step 2A ,2B, 2C, 2D, 2E, 2F, and 2G)

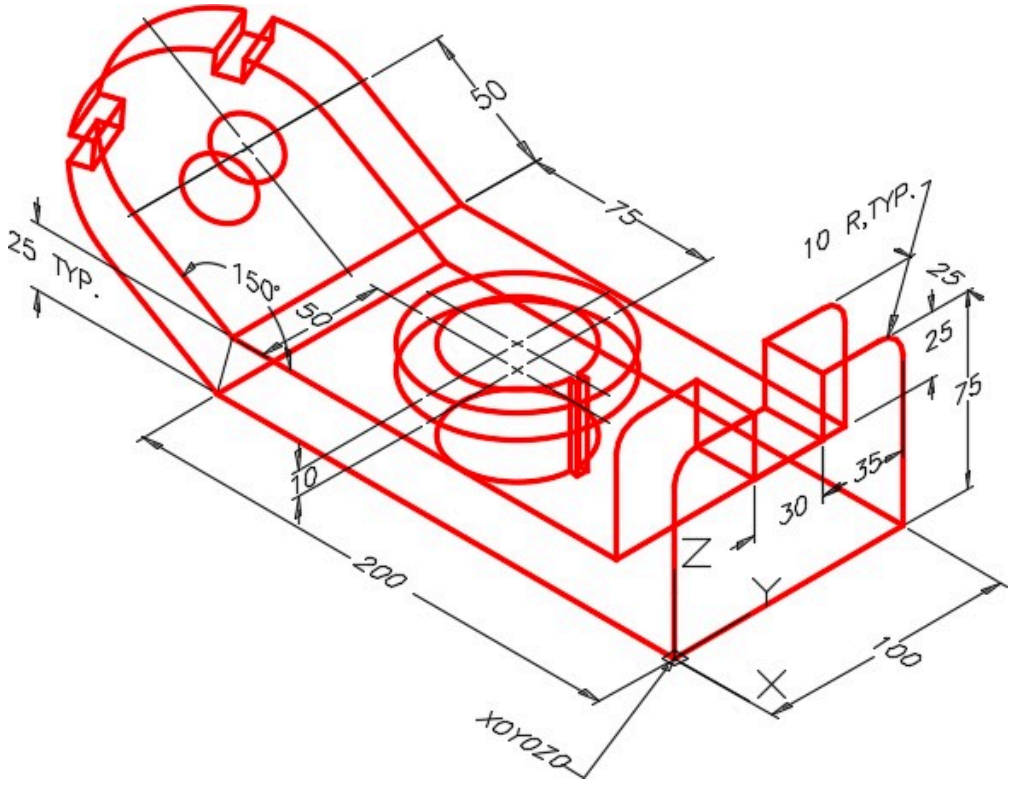


Figure Step 2A
Dimensioned Wireframe Model SE Isometric View

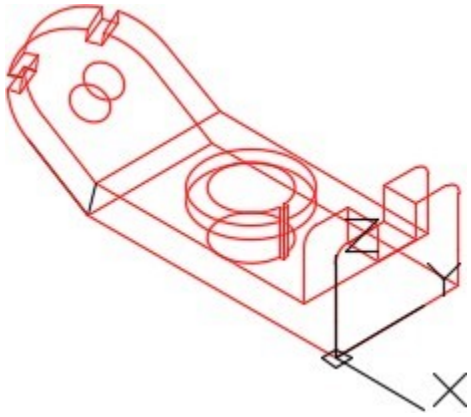


Figure Step 2B Completed Wireframe Model SE Isometric View

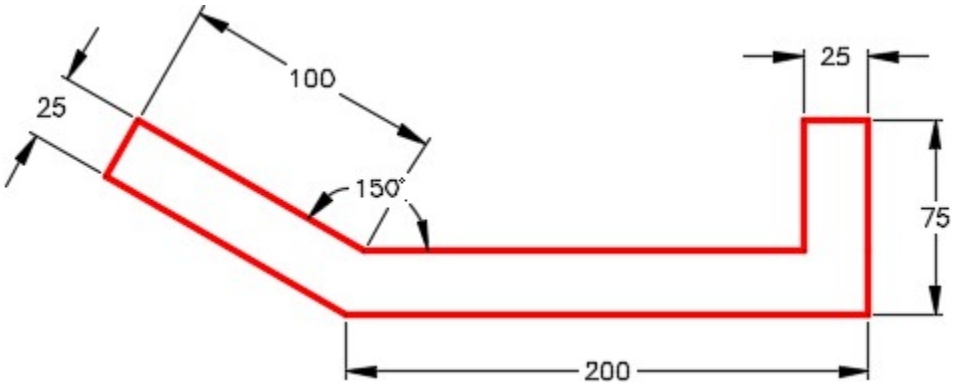


Figure Step 2C Front View – Base View

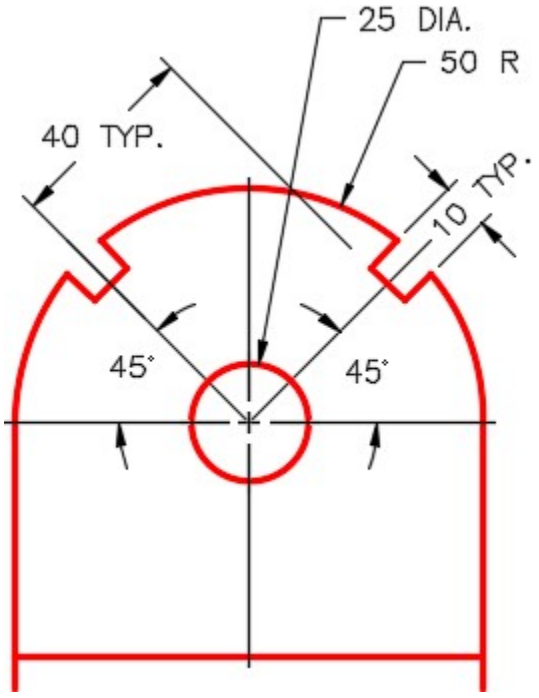


Figure Step 2D Detail of Inclined plane

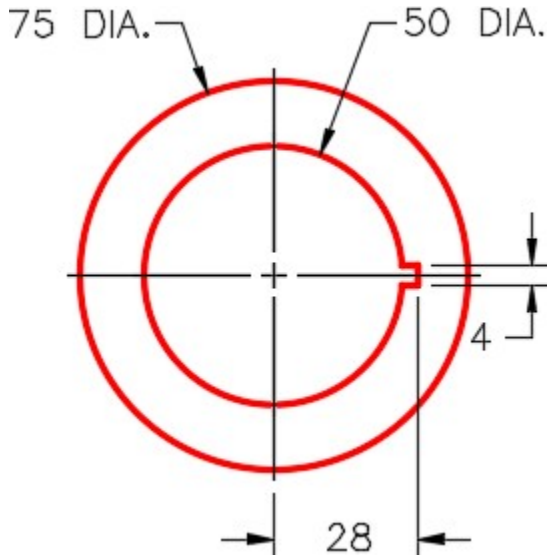


Figure Step 2E Detail of Keyway

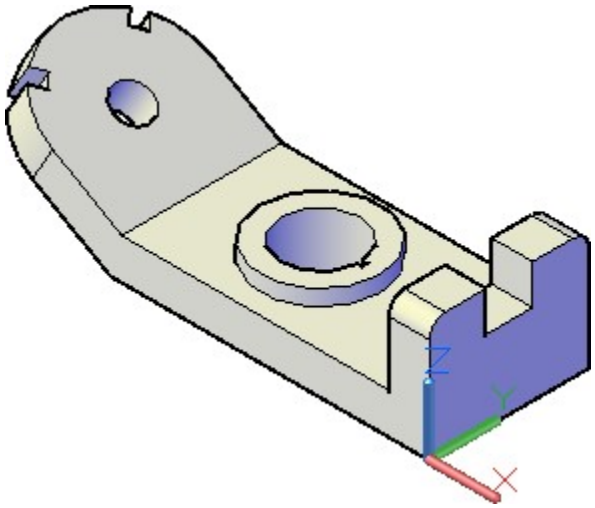


Figure Step 2F Solid Model – SE Isometric View

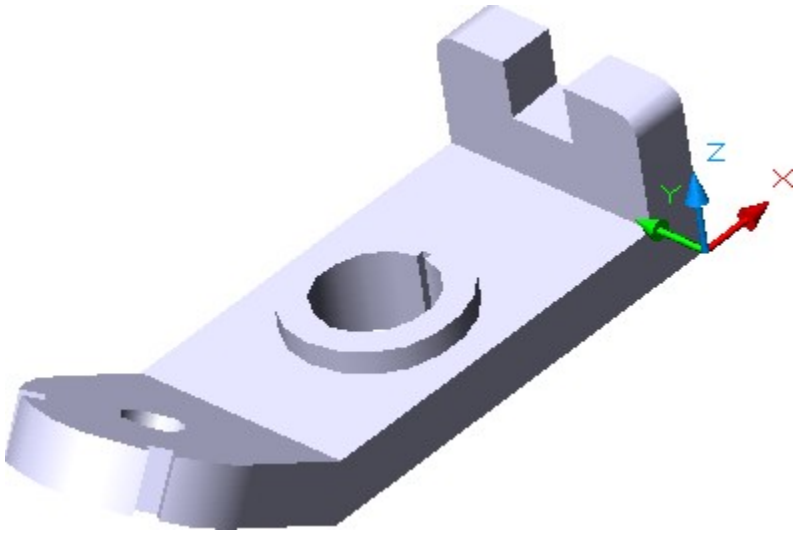


Figure Step 2G Solid Model – Rotated View

Step 3

Draw all construction objects on layer: Construction and all model objects on layer: Model.

Step 4

Start your model with the current view SE Isometric. If required, orbit it slightly with 3DFORBIT to help your line of sight.

> **AUTHOR'S COMMENTS:** Start the Base (first) view on the Front UCS. Before you start drawing the 2D view, change the current UCS to the Front.

Step 5

When complete, freeze layer: Construction.

Step 6

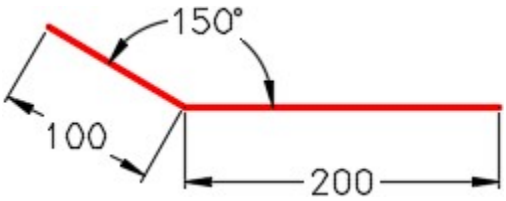
Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Millimeters.

Step 7

Change the current UCS to World and check the model with the key.

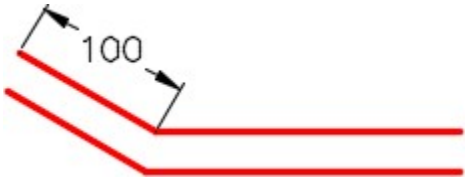
Hint 1

See the steps below to draw the Front view.



Step 1 – Draw the bottom lines first.

Step 2 – Offset the lines.



Step 3 – Trim the lines and change length of the top Line to 100 units.

Step 4 – Daw a line perpendicular at the end and extend the bottom inclined line.

Lab Exercise 5-2

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 05-2	3D Layout Metric	Millimeters

Step 1

Save and name the drawing: AutoCAD 3D Lab 05-2.

Step 2

Draw a wireframe model of the object shown in the figure. (Figure Step 2A, 2B, 2C, 2D, and 2E)

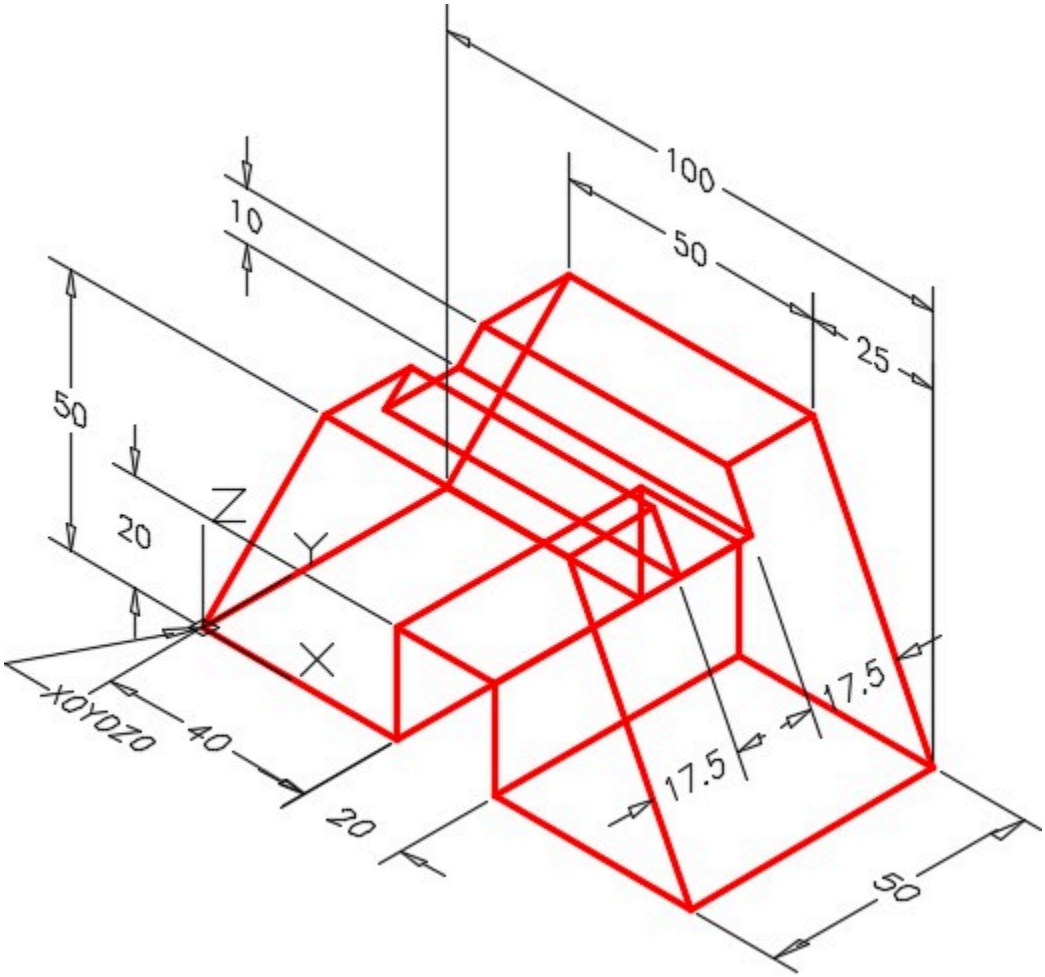


Figure Step 2A
Dimensioned Wireframe Model SE Isometric View

Step 3

Draw all construction objects on layer: Construction and all model objects on layer: Model.

Step 4

Start your model with the current view SE Isometric. If required, orbit it slightly with 3DFORBIT to help your line of sight.

AUTHOR'S COMMENTS: It is ok to draw the first view of a model in a 2D view. As soon as you complete the Base view and are ready to copy the objects to make it three dimensional and from that point on, you must draw in a 3D view.

Step 5

When complete, freeze layer: Construction.

Step 6

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

Step 7

Change the current UCS to World and check the model with the key.

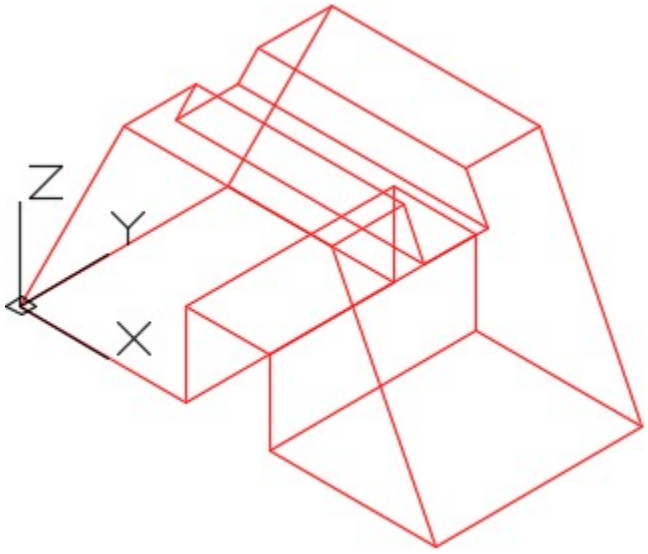


Figure Step 2B
Completed Wireframe Model SE Isometric View

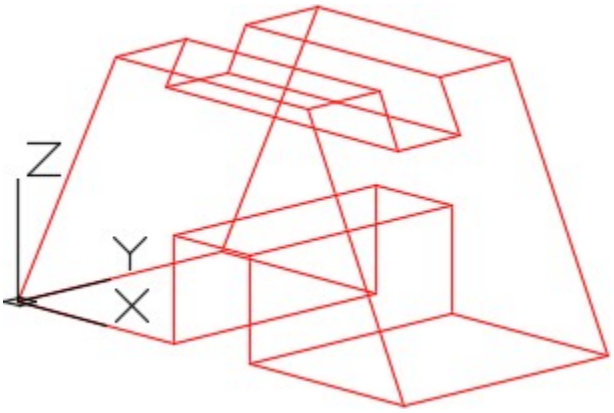


Figure Step 2C
Completed Wireframe Model Orbited View

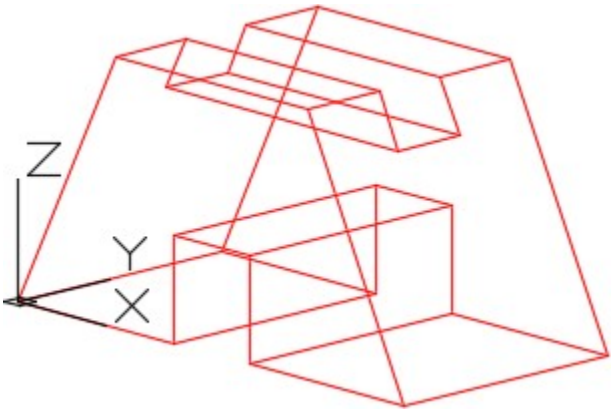


Figure Step 2C
Completed Wireframe Model Orbited View

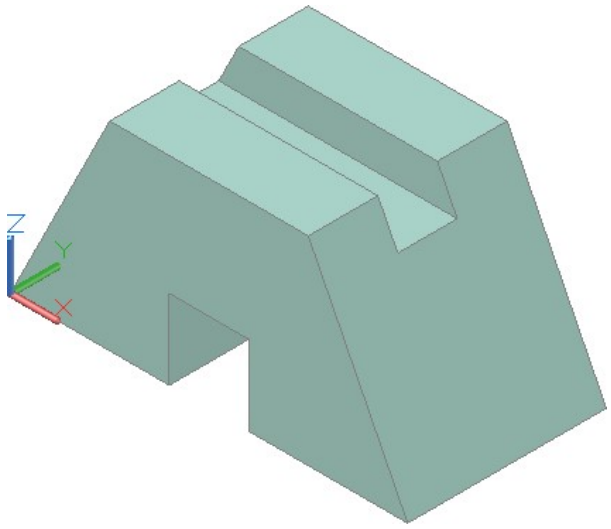


Figure Step 2E
Solid Model – SE Isometric View

Lab Exercise 5-3

Time allowed: 60 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 05-3	3D Layout English	Inches

Step 1

Save and name the drawing: AutoCAD 3D Lab 05-3.

Step 2

Draw a wireframe model of the object shown in the figure. (Figure Step 2A, 2B, 2C, 2D, 2E, 2F, and 2G)

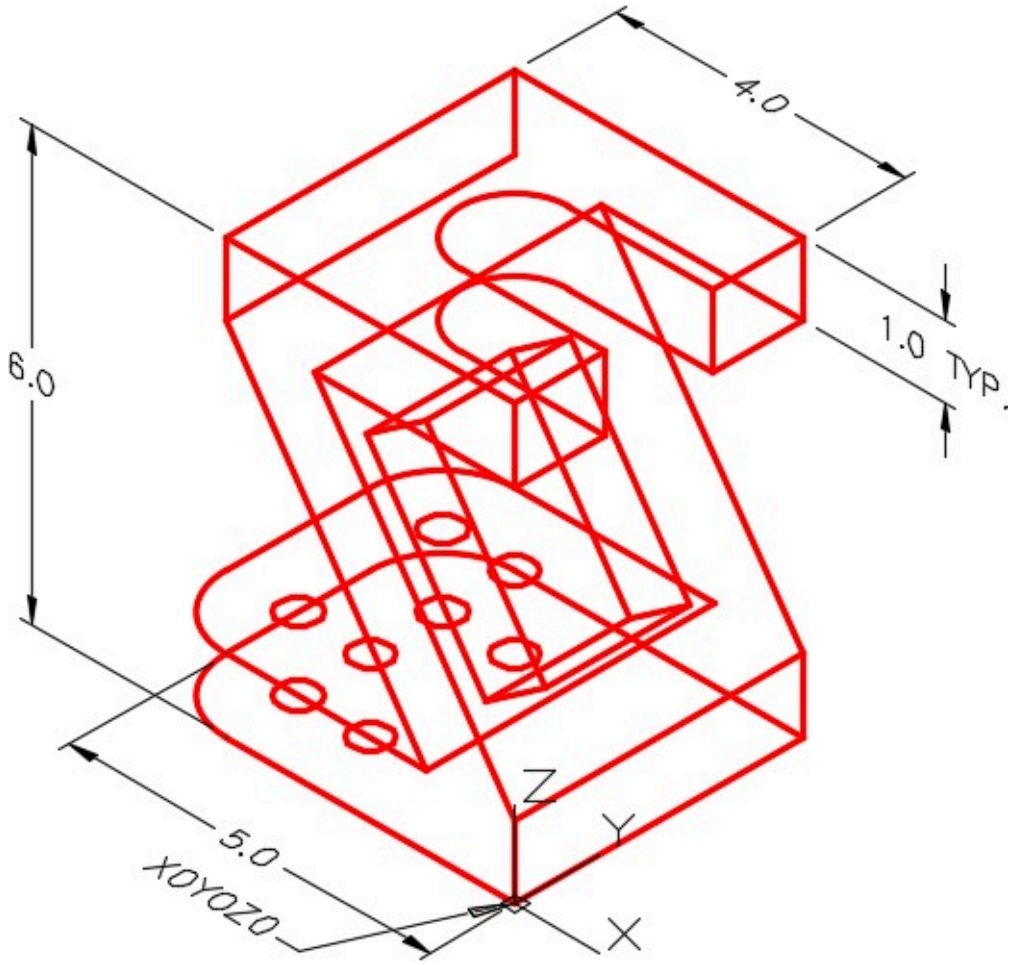


Figure Step 2A
Dimensioned Wireframe Model SE Isometric View

Step 3

Draw all construction objects on layer: Construction and all model objects on layer: Model.

Step 4

Start your model with the current view SE Isometric. If required, orbit it slightly with 3DFORBIT to help the line of sight.

AUTHOR'S COMMENTS: Start the Base (first) view on the Front. Before you start drawing, change the current UCS to Front.

Step 5

When complete, freeze layer: Construction.

Step 6

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

Step 7

Change the current UCS to World and check the model with the key.

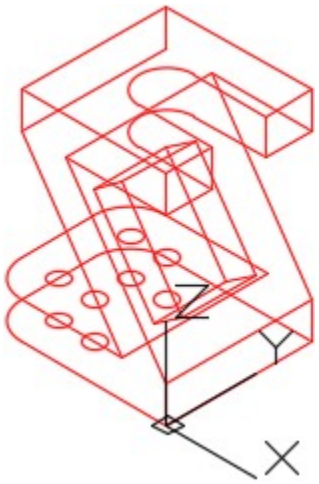


Figure Step 2B Completed Wireframe Model

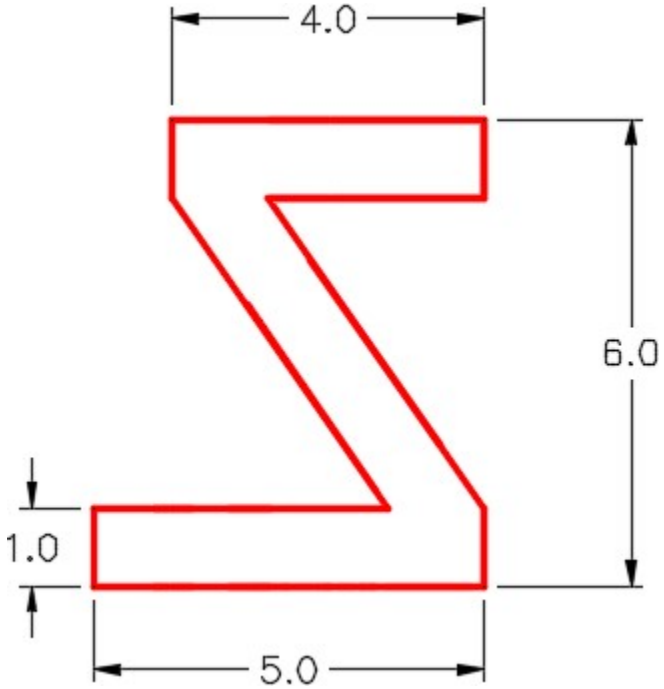


Figure Step 2C
Front View

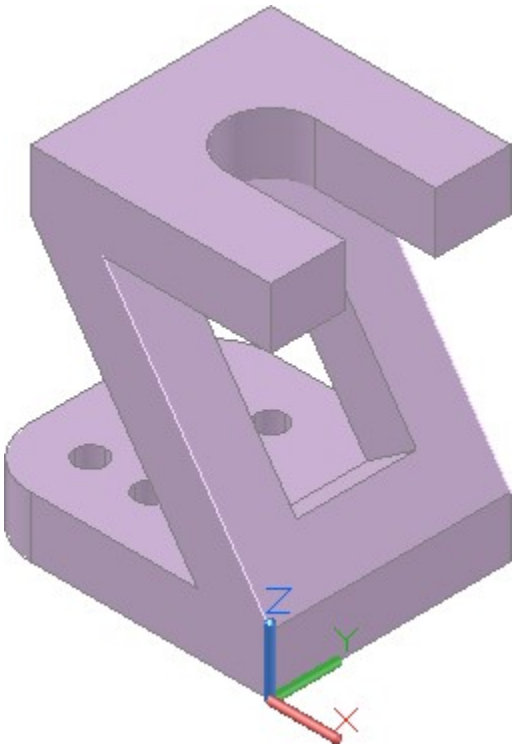


Figure Step 2D
Solid Model – SE Isometric View

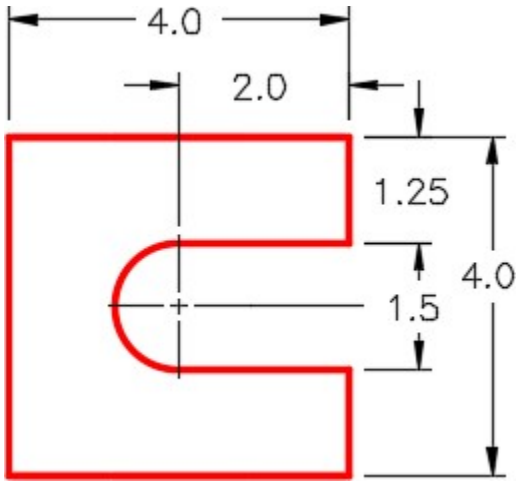


Figure Step 2E
Detail of Top

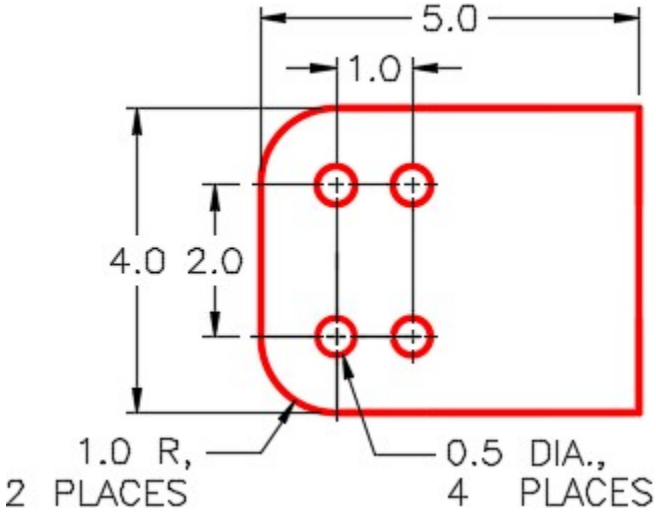


Figure Step 2F
Detail of Bottom

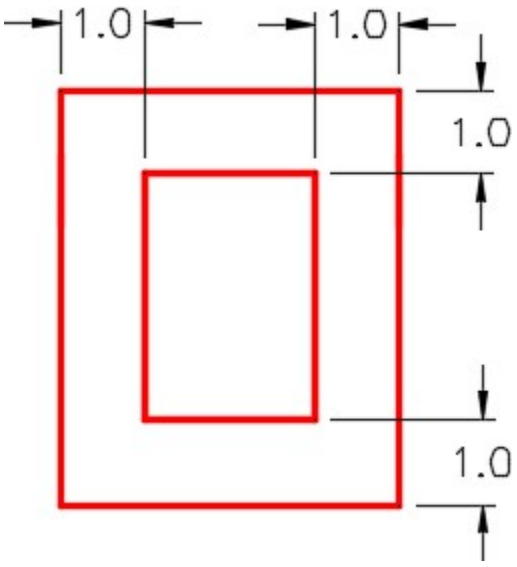


Figure Step 2G
Detail of Square Hole in Inclined

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

Hint 1

To draw the inclined line, see the steps below. (Figure Hint 1)

Step 1

Draw a construction line from corner to corner and a 1 diameter construction circle with the centre at the midpoint of the line.

Step 2

Draw two lines from the endpoints tangent to the circle.

Step 3

Extend the lines and trim the horizontal lines.

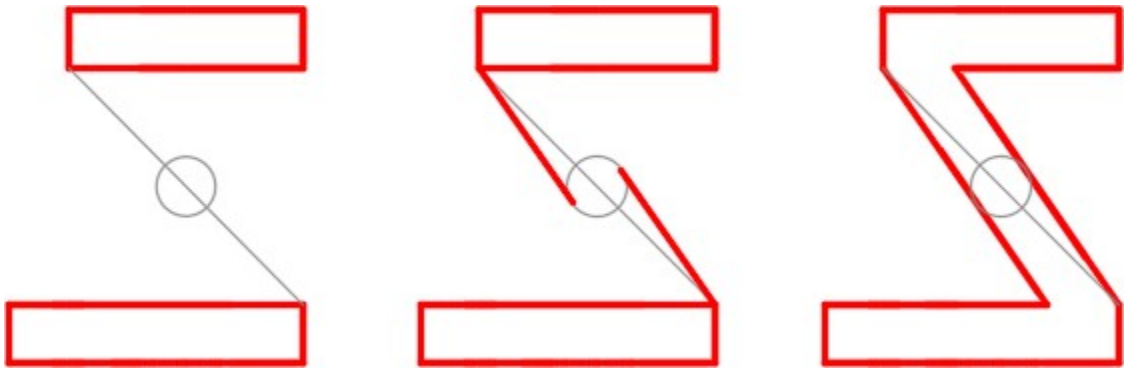


Figure Hint 1
Step 1 *Step 2* *Step 3*

Lab Exercise 5-4

Time allowed: 60 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 05-4	3D Layout English	Inches

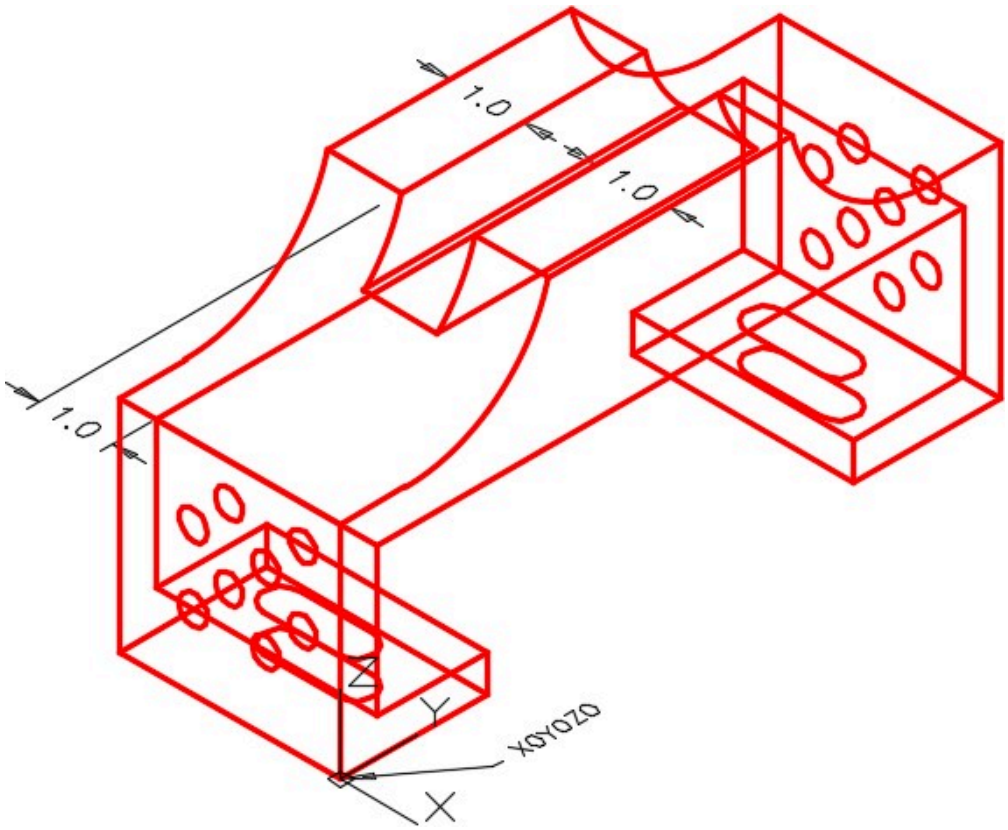


Figure Step 2A
Dimensioned Wireframe Model SE Isometric View

Step 1

Save and name the drawing: AutoCAD 3D Lab 05-4.

Step 2

Draw a wireframe model of the object. (Figure Step 2A, 2B, 2C, 2D, 2E, and 2F)

Step 3

Draw all construction objects on layer: Construction and all model objects on layer: Model.

Step 4

Start your model with the current view SE Isometric. If required, orbit it slightly with 3DFORBIT to help your line of sight.

AUTHOR'S COMMENTS: Start the Base view on the Right. Before you start drawing, change the current UCS to Right.

Step 5

When complete, freeze layer: Construction.

Step 6

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

Step 7

Change the current UCS to World and check the model with the key.

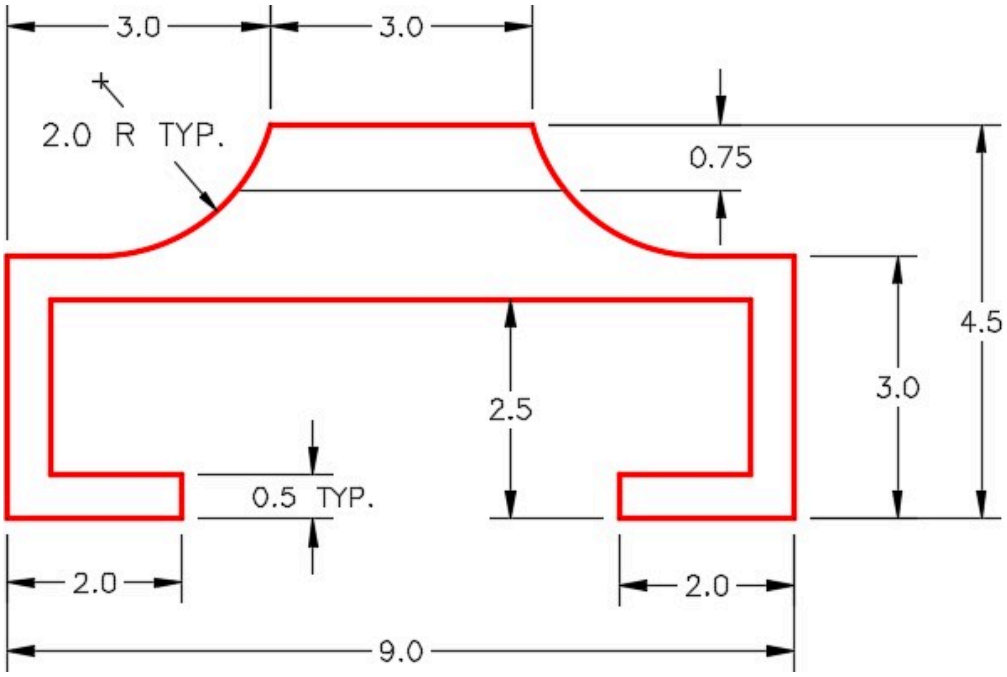


Figure Step 2B
Right Side View

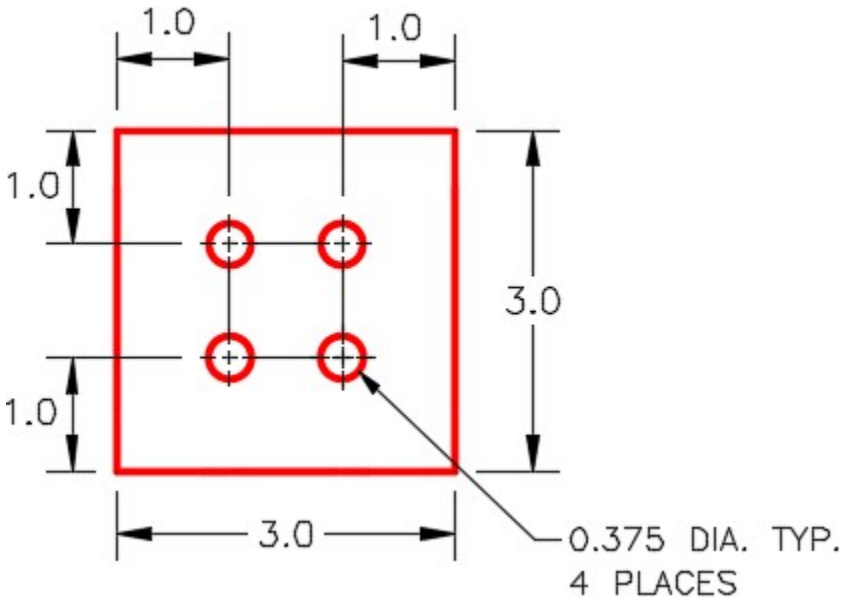


Figure Step 2C
Front and Back Detail TYP.

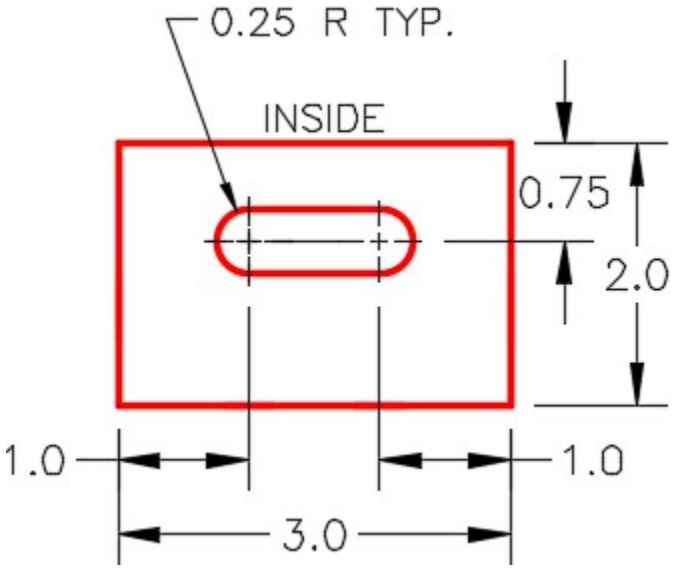


Figure Step 2D
Bottom Detail TYP.

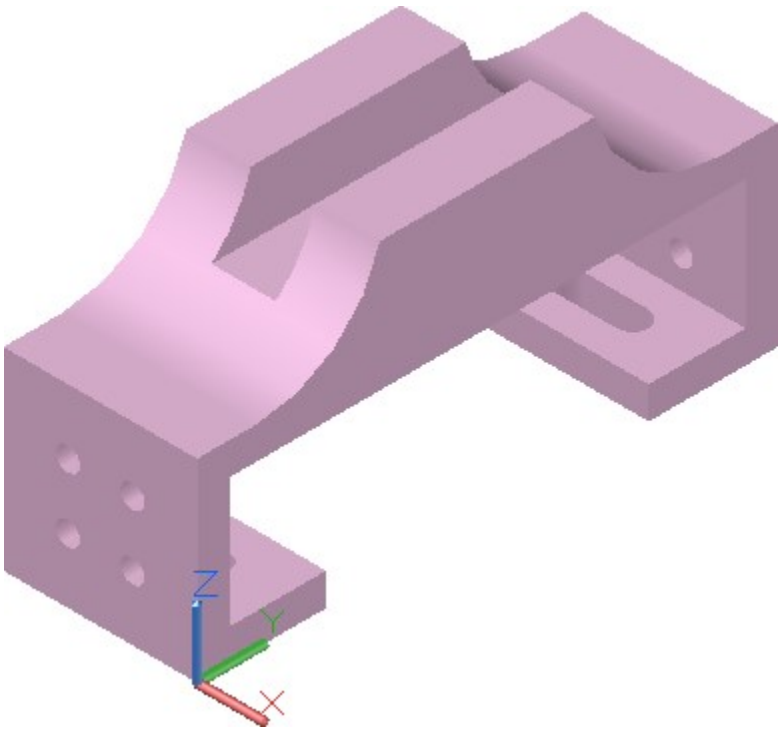


Figure Step 2E
Solid Model – SE Isometric View

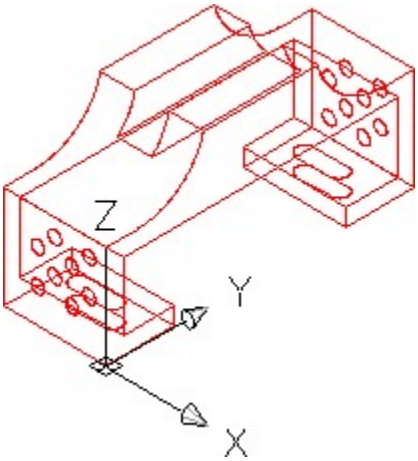


Figure Step 2F Completed
Wireframe Model
SE Isometric View

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

Hint 1

To draw the arc, you must draw the construction circle and lines to locate its centre. (Figure Hint 1)

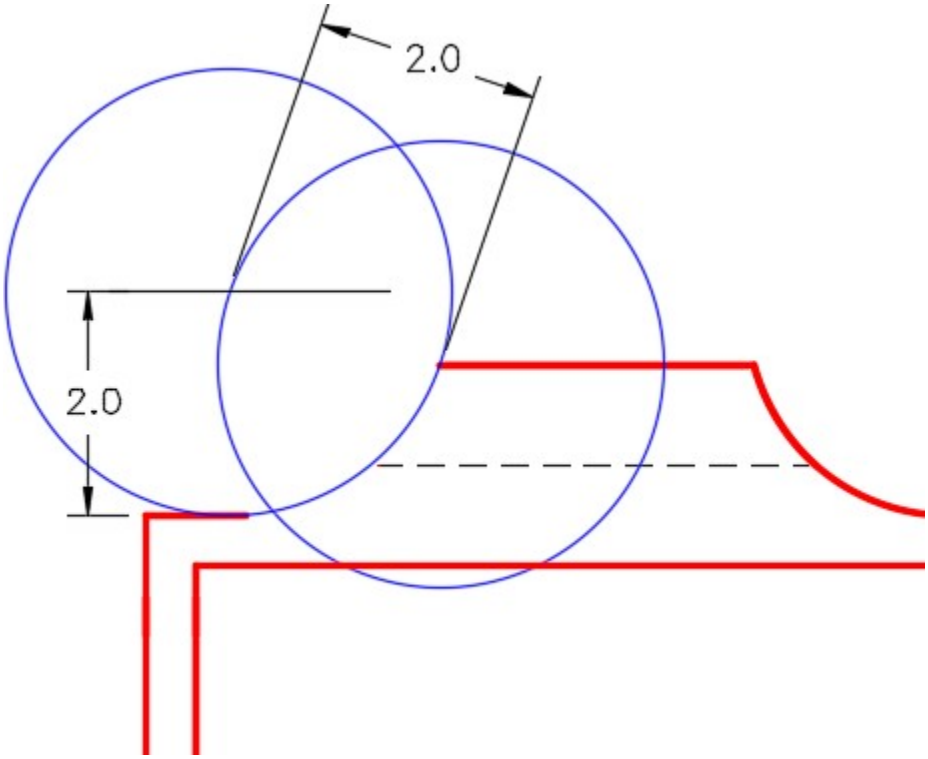


Figure Hint 1

Module 6: Competency Test No. 1 Open Book

Learning Outcomes

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

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

The AutoCAD 3D 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 coordinate locations would specify a point 2 units in the positive Y direction from the last point entered relative to the current UCS?
    A. @*2,0,0
    B. @&0,2,0
    C. @0,2,0
    D. @#0,2,0
    E. @0,0,2
2. What are the two distinct 3-dimensional coordinate systems used by AutoCAD?

A. The WCC and UCS.

B. The WCS and UCC.

C. The WSC and USC.

D. The WCS and UCS.

E. The UCS and USC.

3. What two commands can be used to restore a saved UCS location?

A. UCS and VIEW

B. UCSMAN and UCS

C. 3DORBIT and UCSMAN

D. VIEW and USC

E. UCSICON and UCSMAN

4. Which one of the following statements is a false statement when describing the User Coordinate System?

A. It's current location can be named, saved and restored in the future.

B. It can be located anywhere in 3D space.

C. It defines the User Construction Plane.

D. It is always located at the absolute coordinates X0Y0Z0.

E. It is used to construct 3D models

5. In Figure 6-1, what does the small square indicate? Choose the BEST answer.

A. The location of the last point entered.

B. The UCS is at the centre of the model.

C. The endpoint of the Z axis.

D. The location where X, Y and Z axis meet.

E. The UCS is located at the WCS.

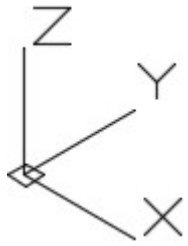


Figure 6-1

6. Which one of the following coordinate locations would specify a point 3 units in the Z direction from the last point entered relative to the absolute X0Y0Z0 or the WCS?

A. @*0,0,3

B. @-0,0,3

C. @3,0,0

D. @#0,0,3

E. @*3,0,0

7. Which one of the following is a false statement when describing a wireframe model?

A. It is a real-world 3D object represented by lines, circles, arcs and/or plines.

B. It is actually a 2 dimensional object.

C. It does not have any surfacing and is not solid.

D. You can see lines and curves right through it that would not appear in a solid object.

E. If it was a real object, you could put your finger through it.

8. Which one of the following statements is a false statement when describing the World Coordinate System? Choose the BEST answer.

   A. It is always located on the top view of the model.

   B. It can be located anywhere in 3D space.

   C. It is fixed and cannot be moved.

   D. It is always located at the absolute coordinates X0Y0Z0.

   E. It is not used to construct 3D models.

9. What command is used to define a new UCS location using the 3 Point UCS method?

   A. UCS

   B. UCSMAN

   C. 3DORBIT

   D. VIEW

   E. UCSICON

10. What direction does the thumb indicate in the right-hand rule?

    A. Positive Y

    B. Positive Z

    C. Negative Y

    D. Negative Z

    E. Positive X

Lab Exercise 6-1 OPEN BOOK

Drawing Name	**Template**	**Units**
AutoCAD 3D Lab 06-1	3D Layout English	Inches

Step 1

Save and name the drawing: AutoCAD 3D Lab 06-1.

Step 2

Draw a wireframe model of the object. (Figure Step 2A, 2B, 2C, 2D, 2E, and 2F)

Step 3

Draw all construction objects on layer: Construction and all model objects on layer: Model.

AUTHOR'S COMMENTS: Start the first view on the right side. Before you start drawing, change the UCS to the Right.

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

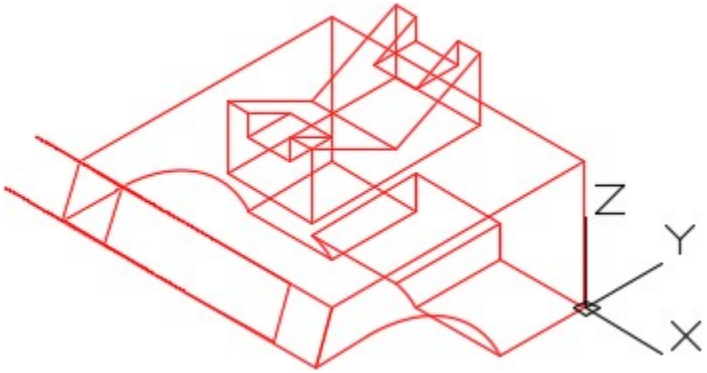


Figure Step 2A
Completed Wireframe Model – SE Isometric View

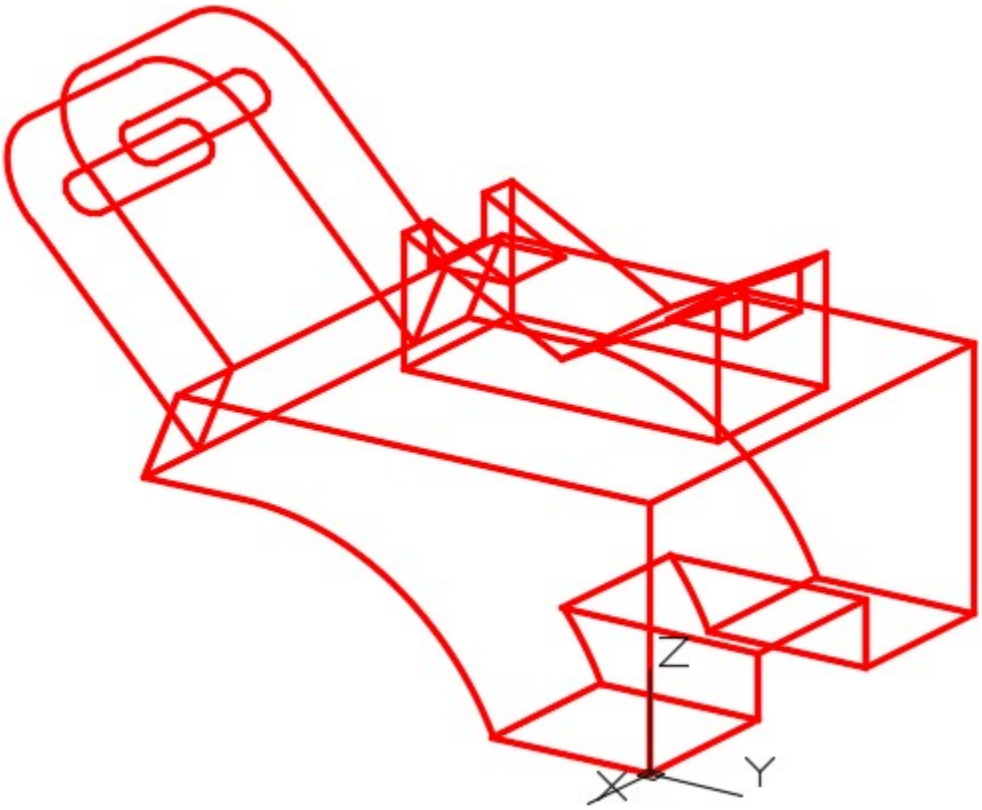


Figure Step 2B
The Model – NE Isometric View Slightly Orbited

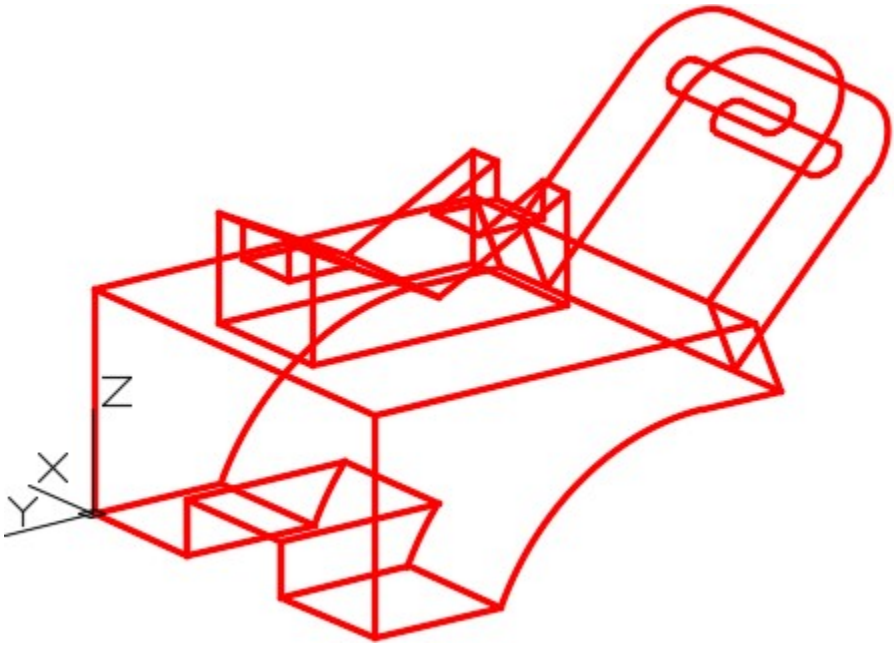


Figure Step 2C
The Model – NW Isometric Slightly Orbited

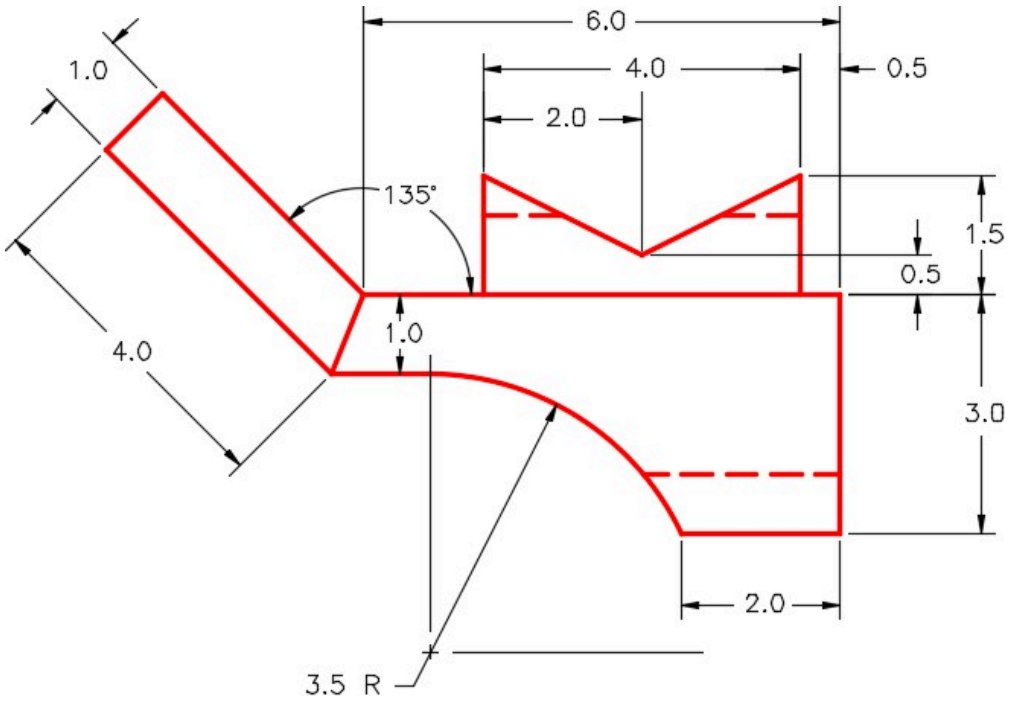


Figure Step 2D
Right Side View

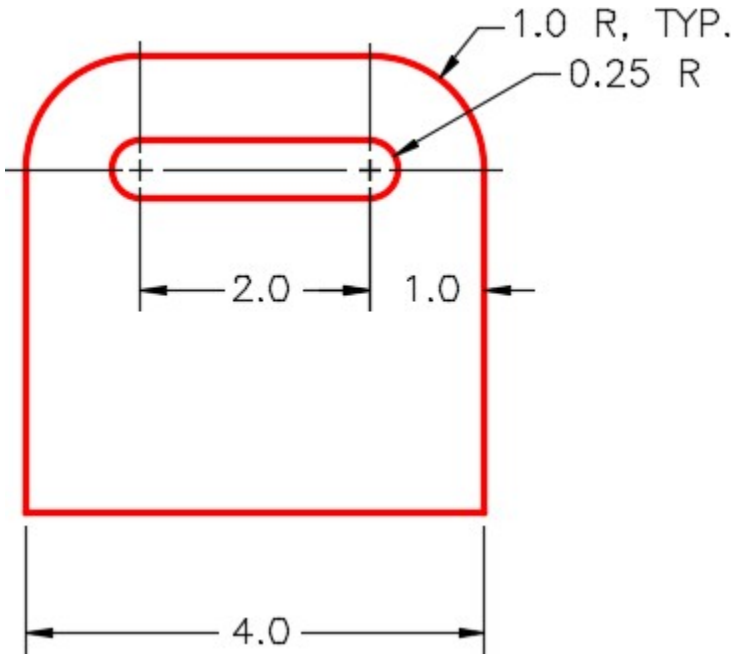


Figure Step 2E
Detail of Inclined View

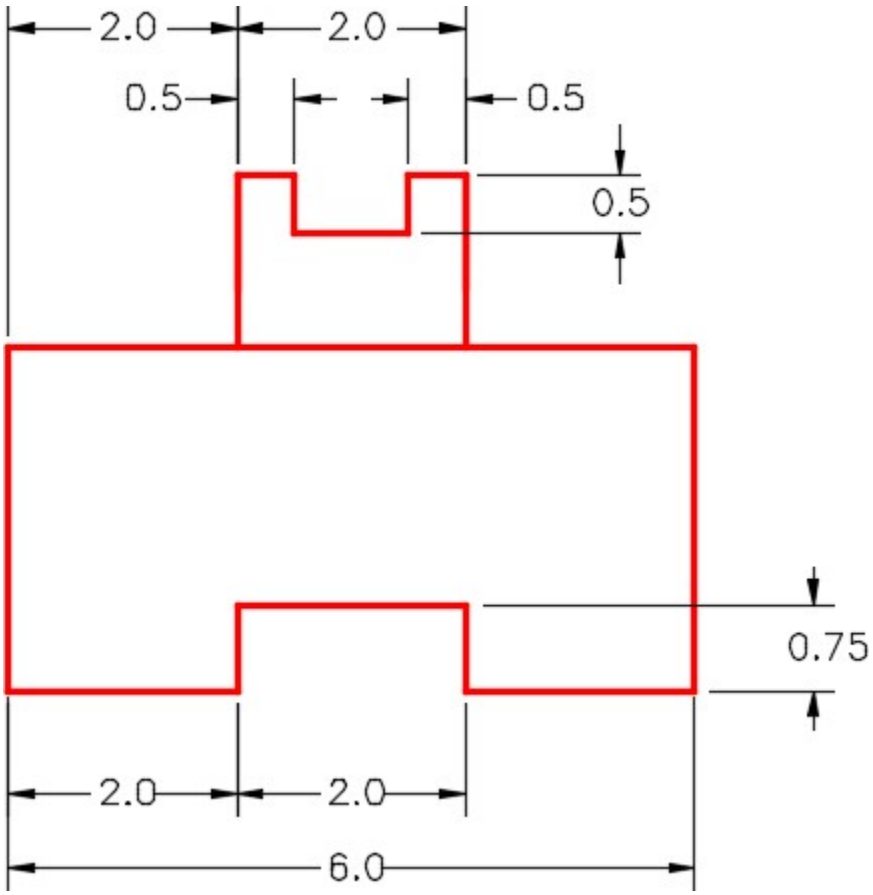


Figure Step 2F
Detail of Rear View

Step 4

When complete, freeze layer Construction.

Step 5

Change the view to SE Isometric. (Figure Step 5)

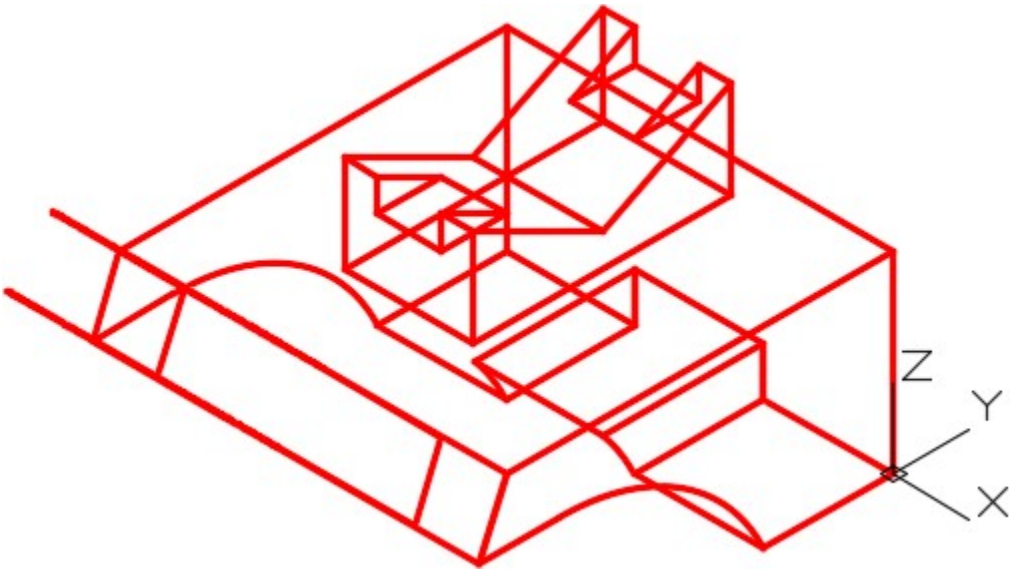


Figure Step 5
SE Isometric View

Part 2

Module 7: Visualizing Multiview Drawings

Learning Outcomes

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

1. Draw isometric drawings on an isometric grid using multiview drawings as a reference.
2. Construct 3D models in AutoCAD using multiview drawings.

Visualizing 3D Models

In the first six modules, all of the wireframe models that you constructed were referenced to a given 3D view of the model. Since most technical drawings used in the drafting and design world are 2D multiview drawings, 3D models must be able to be drawn using a multiview drawing as a reference to find the model's shape and dimensions. To construct a 3D model, you must be able to mentally visualize the 3D model using a multiview drawing as a reference.

A good way for you to learn to visualize a 3D model from a 2D multiview drawing is to first draw the model as an isometric drawing. By doing this, it is easier to form a mental image from the multiview drawing. After practicing this for while, you will be able to visualize and construct 3D models without drawing the isometric first.

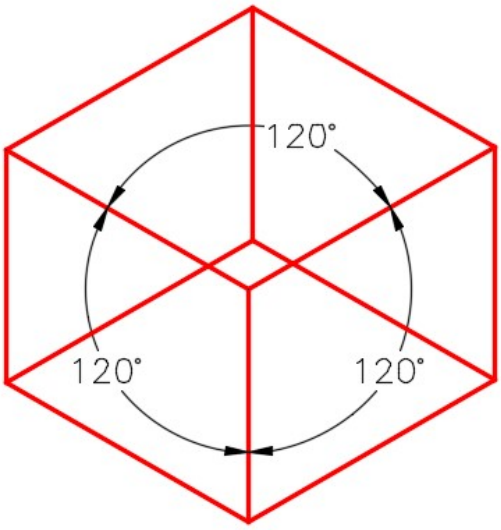


Figure 7-1
An Isometric Drawing

An *isometric drawing* is a 2-dimensional drawing that has the XYZ axis drawn at 120 degrees apart as shown in Figure 7-1. In this module, drawing the isometric on an isometric grid will be taught. An isometric grid has the grid lines drawn at 120 degrees as shown in Figure 7-2. Figure 7-3 shows a rectangular box drawn on the isometric grid.

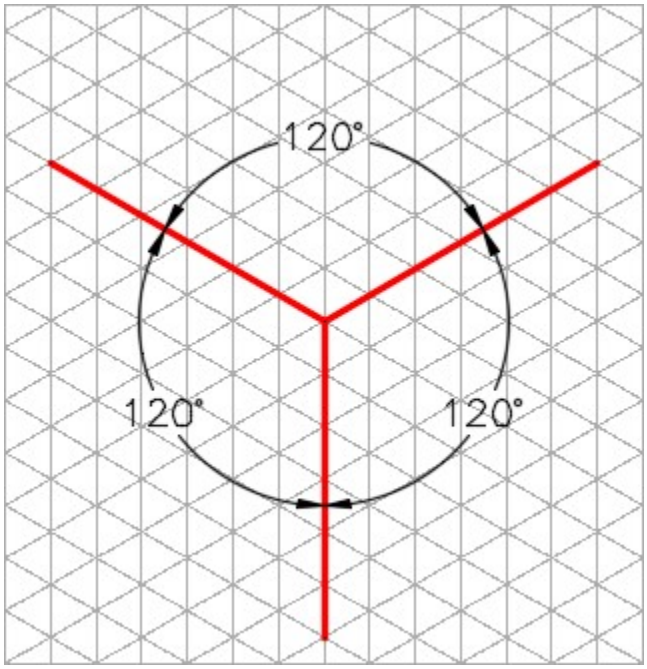


Figure 7-2 An Isometric Grid

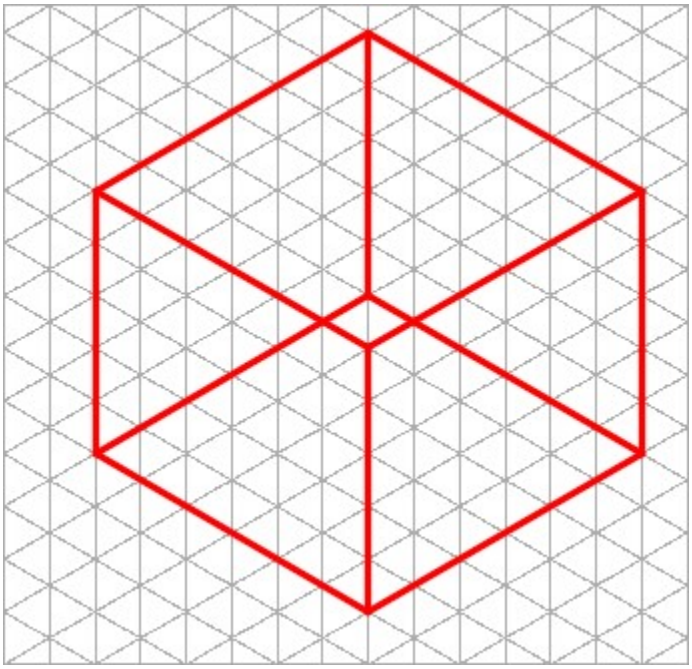


Figure 7-3
A Rectangular Box Drawn on an Isometric Grid

WORK ALONG: Visualizing 3D Models

Step 1

Using the NEW command, start a new drawing using the template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 07-1.

Step 3

Set the UCS to World and the view to Top. Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Inches. Using the INSERT command, insert the block: AutoCAD 3D Workalong 07-1 at the coordinates 0,0,0. Explode the block. The drawing should appear as shown in the figure. (Figure Step 3)

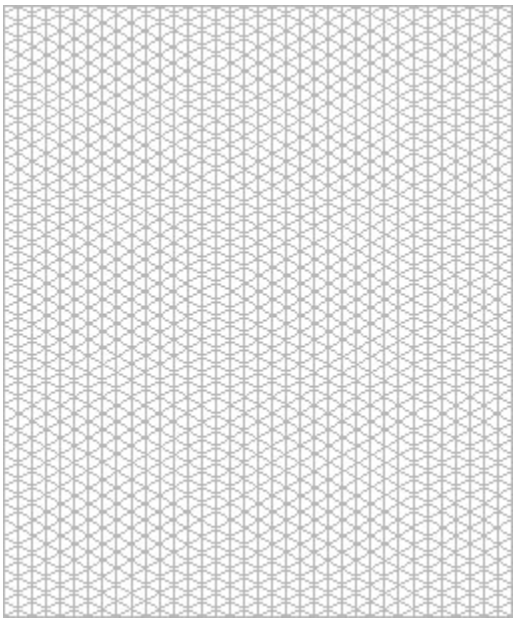


Figure Step 3

AUTHOR'S COMMENTS: In this workalong, you will be using AutoCAD to draw isometric drawings of objects on atwo-dimensional isometric grid. This is only a teaching tool to help you visualize 3D models using the multiview drawing of the object. Isometric drawings were used extensively in the drafting field before CAD was used.

With AutoCAD, you can draw in true 3D and isometric drawing are not really required. They are, however, a good teaching tool for this module.

Step 4

Change the layer of the isometric grid from layer: 0 to layer: Grid.

Step 5

Lock layer: Grid and set the lineweight of layer: Object to 0.35 mm. (Figure Step 5)

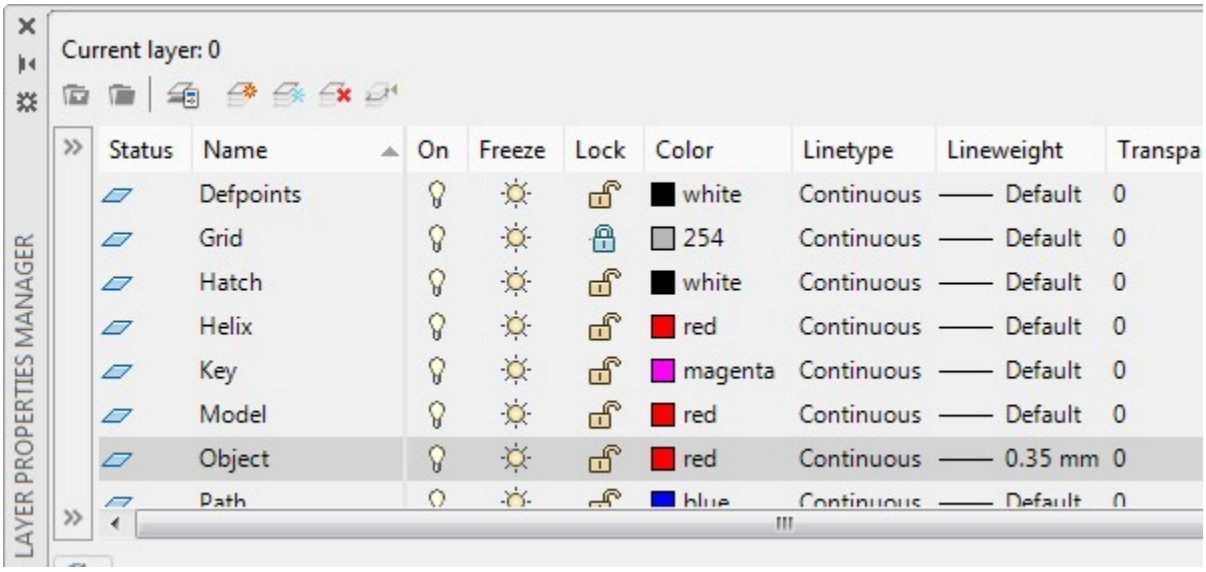


Figure Step 5

Step 6

Enable OSNAP and LWT. (Figure Step 6)

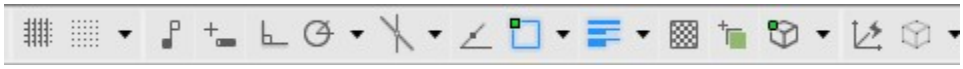


Figure Step 6

Step 7

Figure Step 7A shows the multiview drawing of Object 7-1. Set layer: Object as the current layer and draw a box 6 grids long, 5 grids wide and 4 grids high. Where it is drawn is not that important but it should be located somewhere in the top left corner. Ensure that all lines are snapped to the grid intersections. Since layer: Grid is locked, the lines can be freely moved on layer: Object, as required. (Figure Step 7A, 7B, and 7C)

AUTHOR'S COMMENTS: One grid on the multiview drawing equals one grid on the isometric.

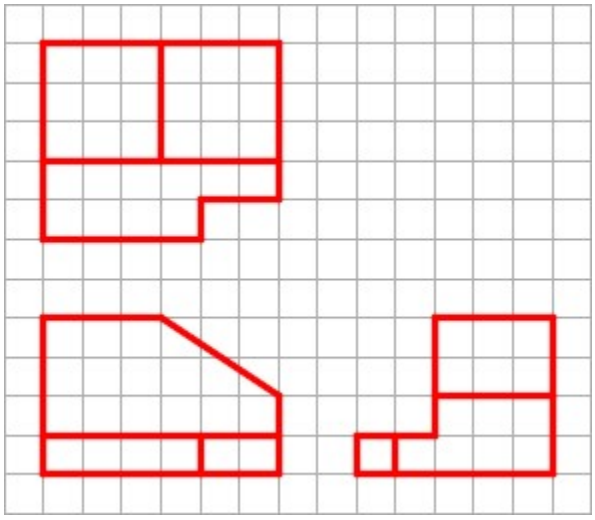


Figure Step 7A
Multiview Drawing – Object 7-1

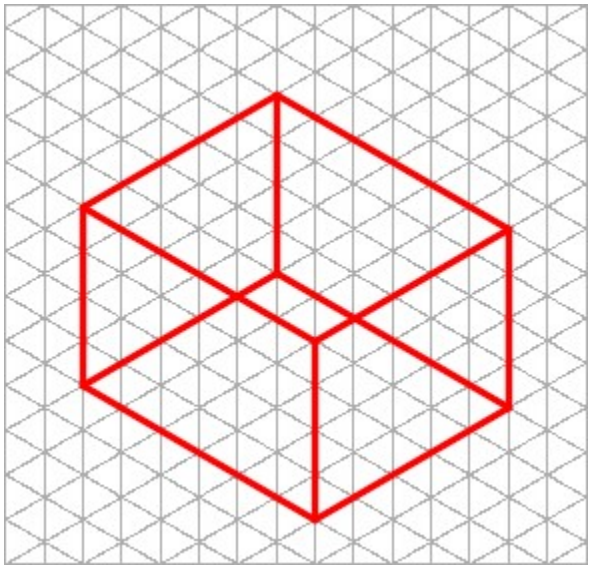


Figure Step 7B

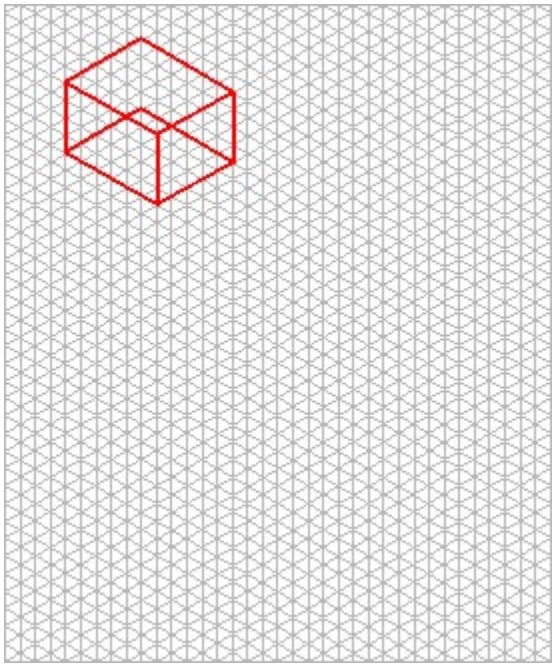


Figure Step 7C

AUTHOR'S COMMENTS: Solve one view at a time. Pick the view with the hardest contour to start with. In this case, start with the Front view.

Step 8

To draw the contour of the Front view, count the number of grids for the diagonal cut off. It is 3 grids on the X axis and 2 grids on the Z axis. Using those dimensions, draw the inclined lines on the front and rear view and add the lines to connect them. (Figure Step 8)

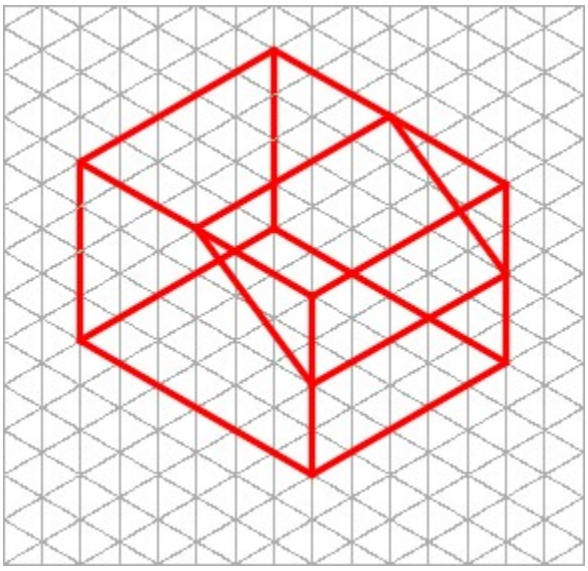


Figure Step 8

Step 9

Trim and delete the lines to complete the contour of the front view. (Figure Step 9)

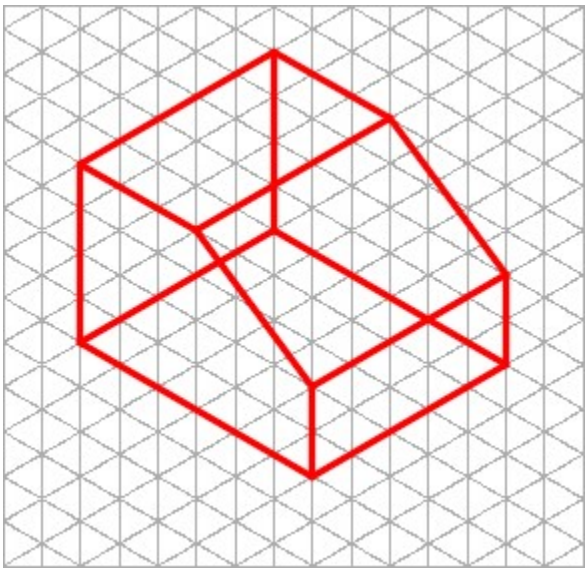


Figure Step 9

Step 10

Add the lines to cut out the contour of the Right Side view. (Figure Step 10)

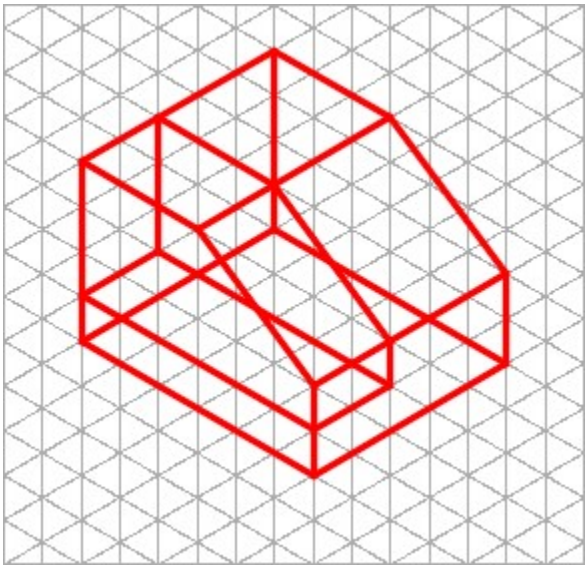


Figure Step 10

Step 11

Trim and delete the necessary lines to complete the Right Side view contour and then add the lines to form the Top view contour. (Figure Step 11)

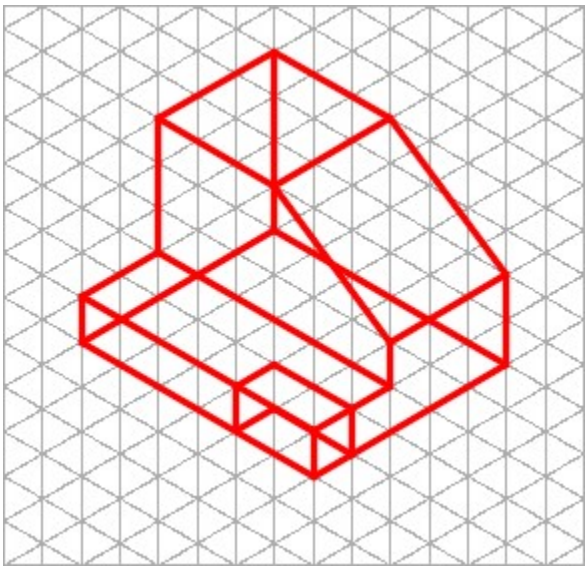


Figure Step 11

Step 12

Trim and delete any unwanted lines to complete the isometric object: Object 7-1. The drawing should appear similar to figure. (Figure Step 12A and 12B)

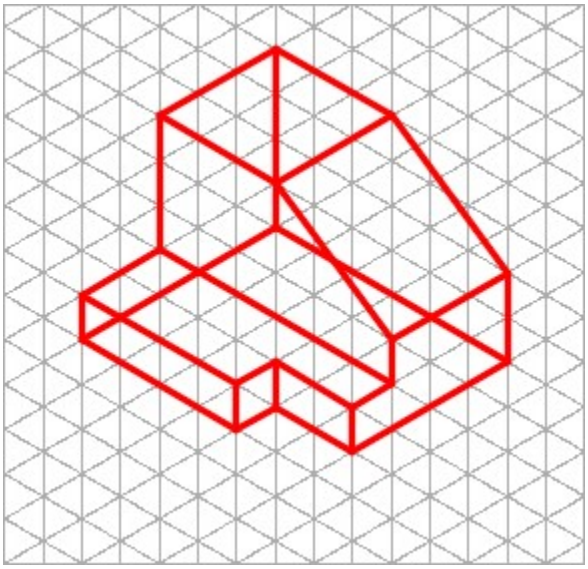


Figure Step 12A

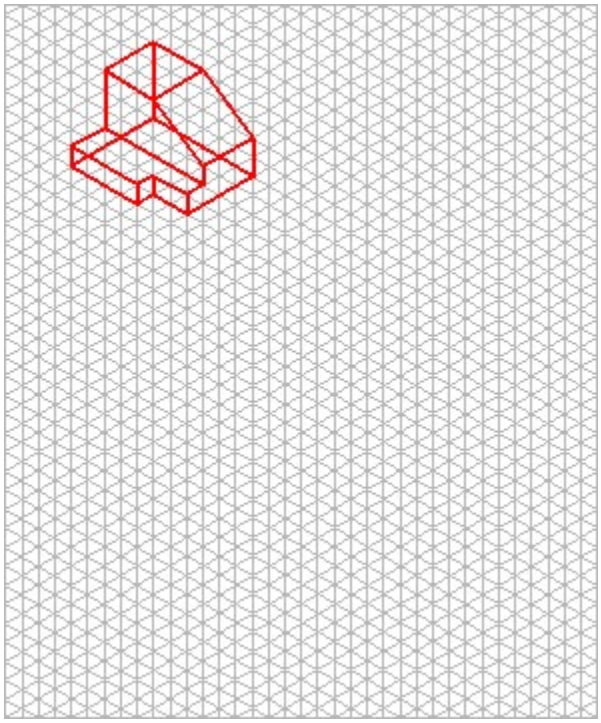


Figure Step 12B

Step 13

Using what was just taught, draw the isometric drawing of: Object 7-2. Draw it in the top right corner of the grid. (Figure Step 13)

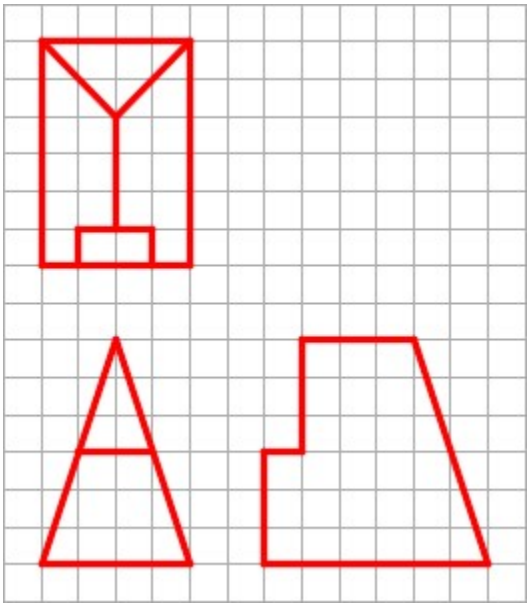


Figure Step 13

AUTHOR'S COMMENTS: Solve one view at time. Pick the view with the hardest contour to start with. In this case, start with the Front view.

Step 14

The figures show the necessary steps. Try to complete the isometric without looking at the figure. (Figure Step 14A, 14B, 14C, and 14D)

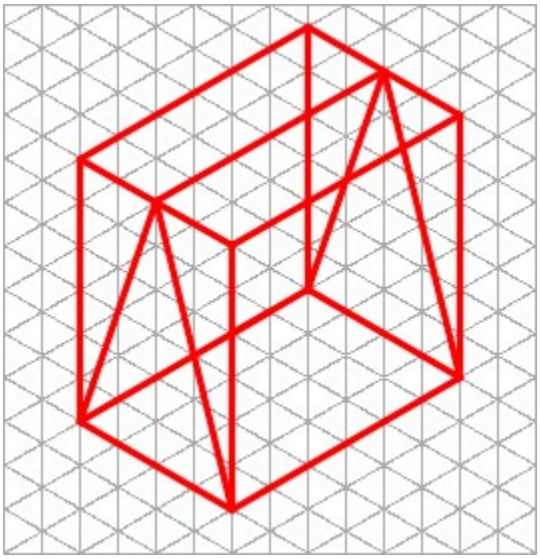


Figure Step 14A

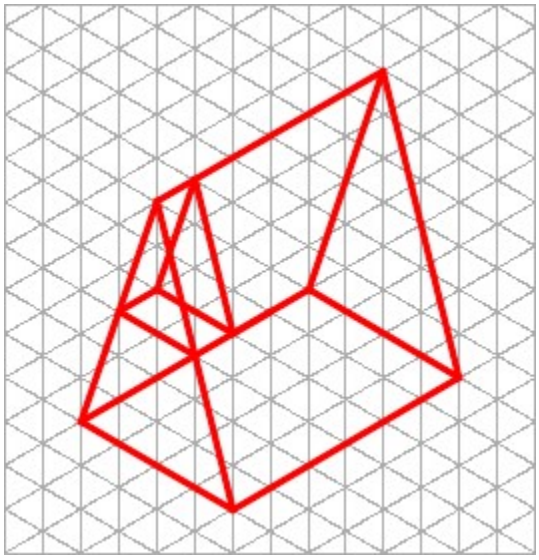


Figure Step 14B

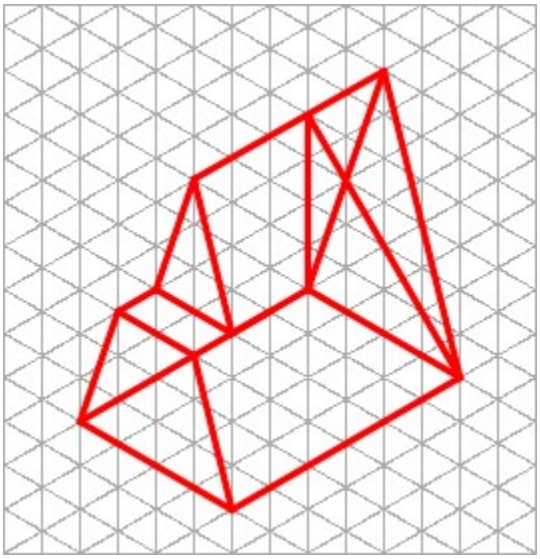


Figure Step 14C

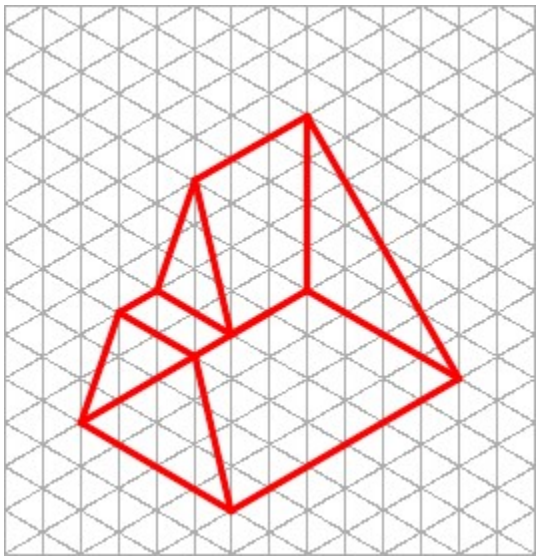


Figure Step 14D

Step 15

Your drawing should now appear similar to the figure. (Figure Step 15)

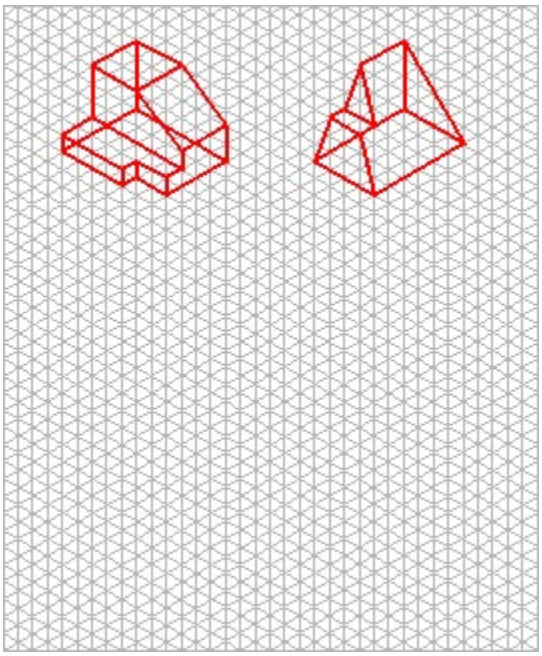
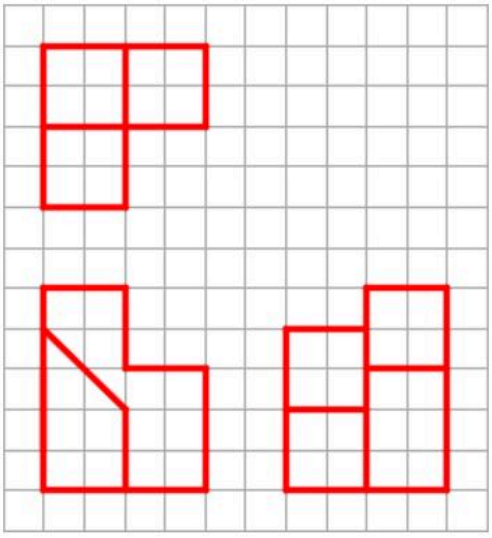


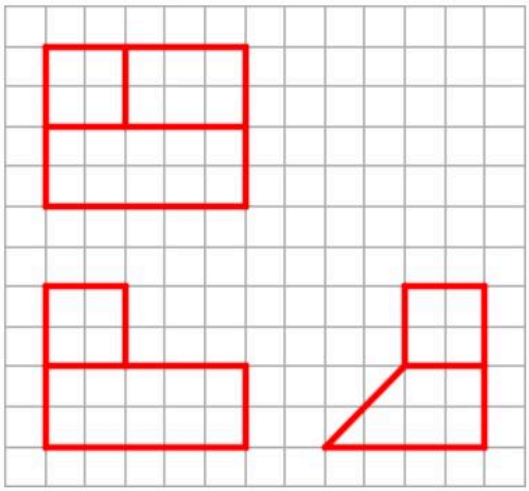
Figure Step 15

Step 16

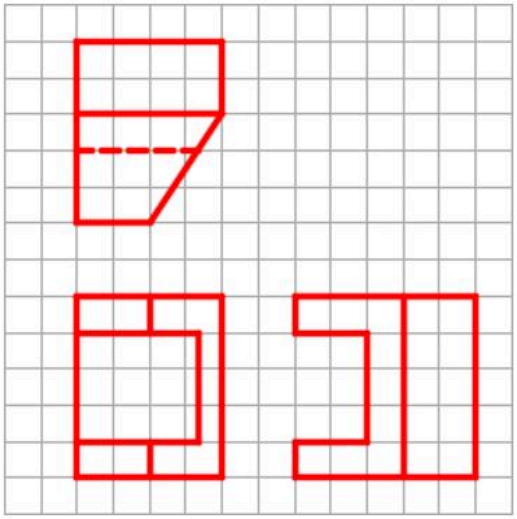
Using what was just taught, draw isometric drawings of the four objects: Object 7-3 to Object 7-6. For the answers, see Figure “Isometric drawing of Object 7-1 to Object 7-6” at the end of this chapter. Try to visualize the 3D model by looking at the multiview drawing and then draw the isometric. Do not look at the answers until you have done your best to complete the isometric drawing of each object. (Figure Step 16)



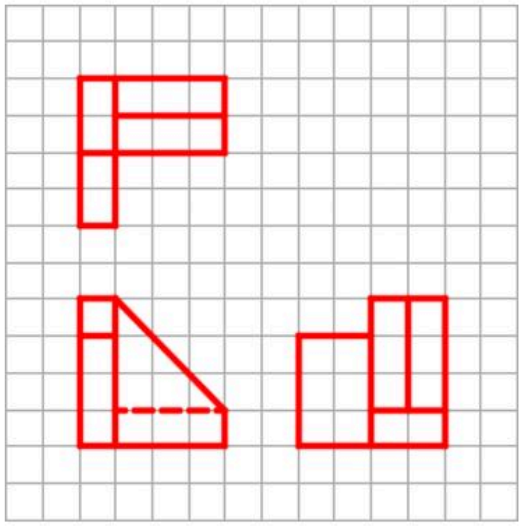
Object 7-3



Object 7-4



Object 7-5



Object 7-6

Figure Step 16

Step 17

Save and close the drawing.

Key Principles

Key Principles in Module 7

1. An isometric drawing is a 2-dimensional drawing that has the XYZ axis drawn 120 degrees
2. An isometric drawing is only used in the AutoCAD 3D book as a teaching They are mostly obsolete in the CAD design and drafting world.

Lab Exercise 7-1

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 07-1	3D Layout English	Inches

Step 1

Draw all construction objects on layer: Construction and all model objects on layer Model. When complete, freeze layer: Construction.

Step 2

Draw a wireframe model of the object. (Figure Step 2A, 2B, and 2C)

AUTHOR'S COMMENTS: Draw the Front view first.

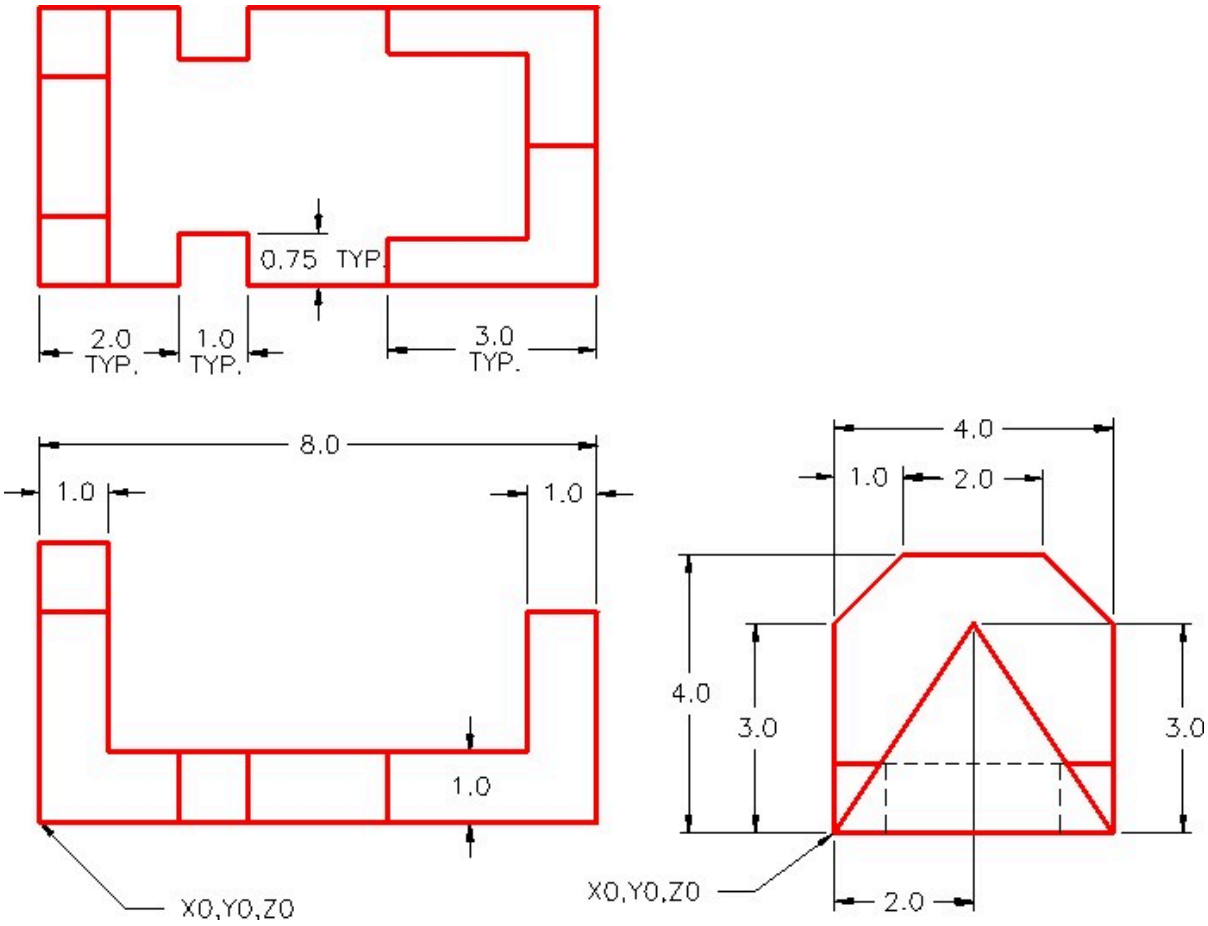


Figure Step 2A
Dimensioned Multiview Drawing

Step 3

Start with the current view SE Isometric. If required, orbit it slightly with 3DFORBIT to help your line of sight.

Step 4

Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Inches. Change the current UCS to World and check the model with the key.

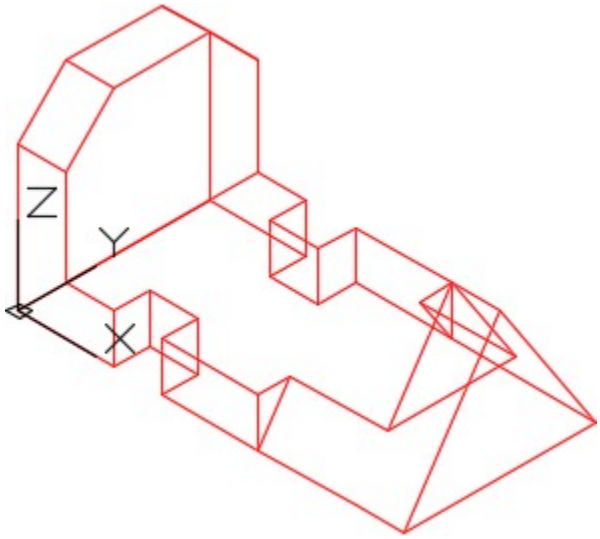


Figure Step 2B
Complete Wireframe Model

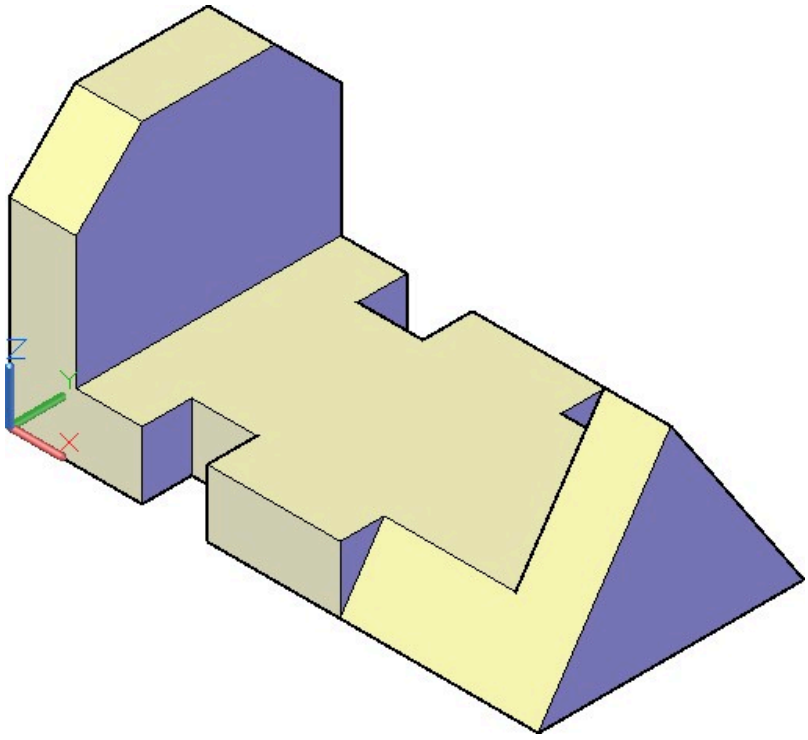
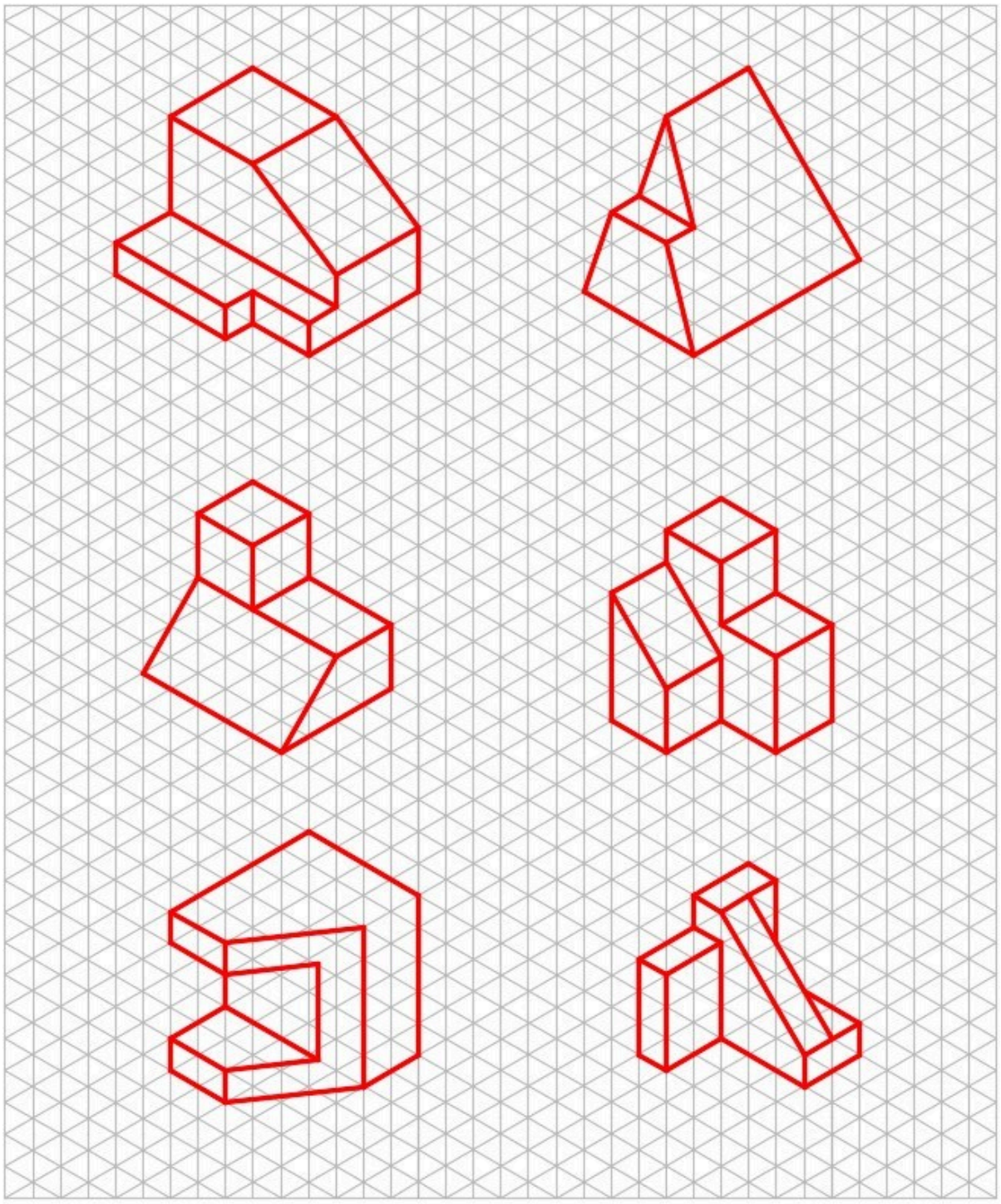


Figure Step 2C
Solid Model – SE Isometric View



Isometric drawing of Object 7-1 to Object 7-6

Module 8: Rotating 3D Models

Learning Outcomes

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

1. Apply the TRIM and EXTEND commands when trimming and extending in three dimensional wireframe modeling.
2. Apply the ROTATE3D command to rotate 3D models.

Rotating in 3D

AutoCAD has individual 2D and 3D rotate commands. The 2D rotate command is ROTATE and in 3D, it is ROTATE3D. In this module, using the ROTATE3D command will be taught. It is used to rotate the model in 3D space. The main difference between the two commands is that the ROTATE command uses a XY point to rotate around and the ROTATE3D command uses two XYZ points or an axis to rotated around. The ROTATE command can be used while 3D modeling as long as all of the objects being rotated are 2D objects and located on the current UCS. If 3D objects are going to be rotated in 3D space, the ROTATE3D command must be used. See Figure 8-1.

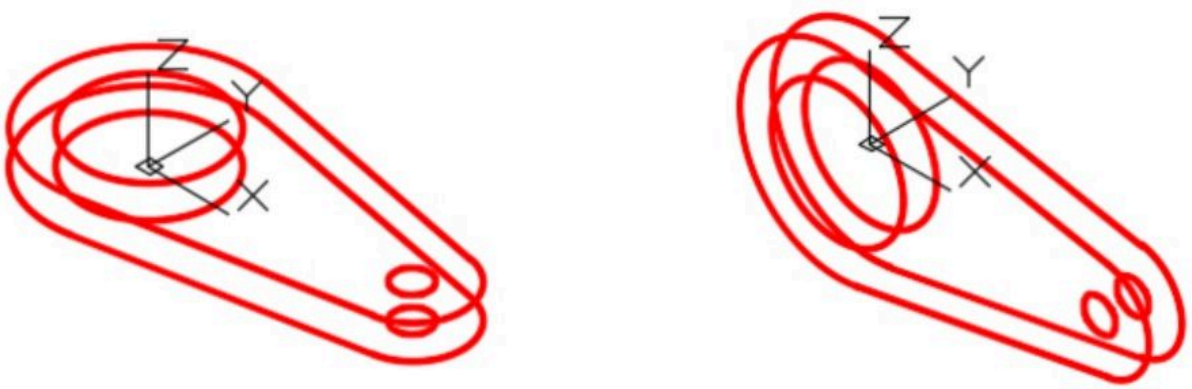


Figure 8-1
Rotating 3D Model in 3D Space

AutoCAD Command: ROTATE3D

The ROTATE3D command is used to rotate objects in 3D space around an axis or two XY points.

Shortcut: none

WORK ALONG: Rotating Wireframe Models in 3D Space

Step 1

Using the NEW command, start a new drawing using the template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 08-1.

Step 3

On layer: Model, draw a wireframe model of the object shown in the multiview drawing. (Figure Step 3A and 3B)

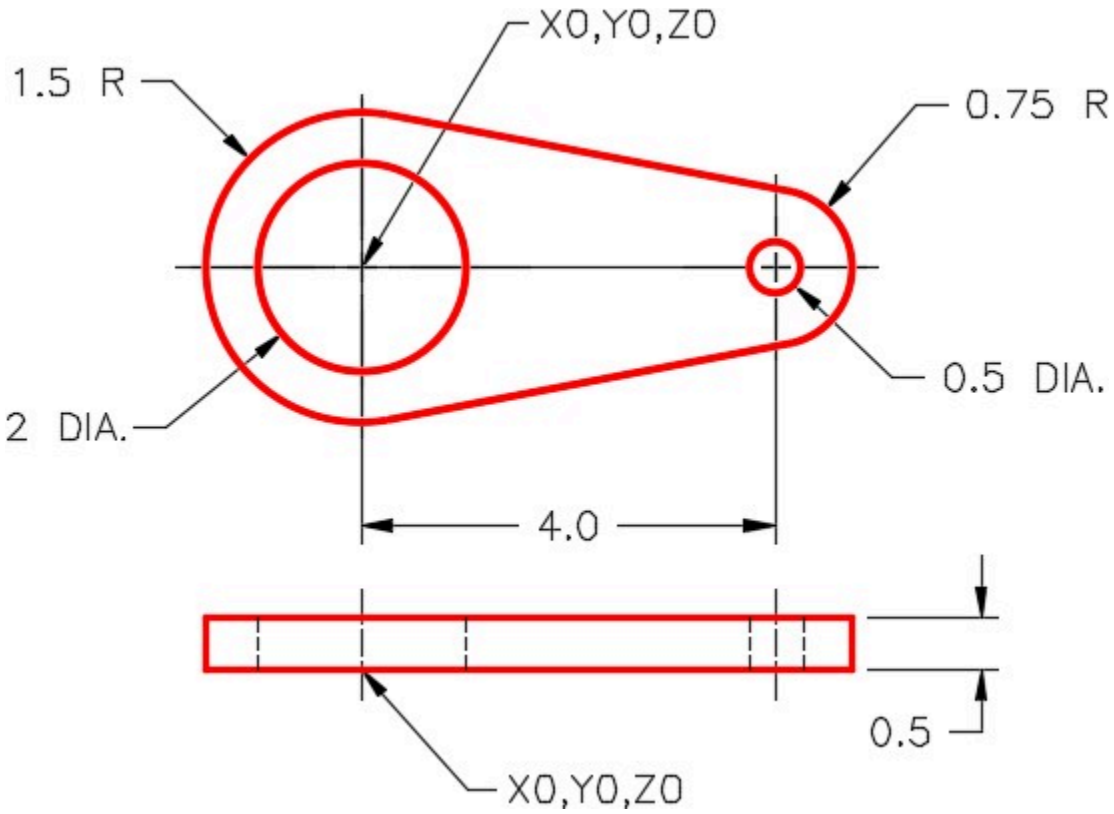


Figure Step 3A
Dimensioned Multiview Drawing

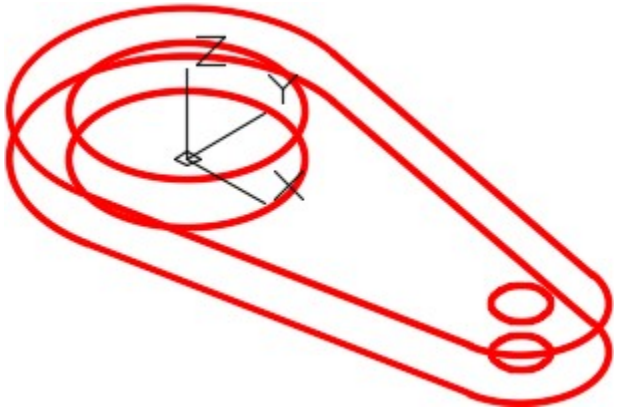


Figure Step 3B
SE Isometric View

Step 4

Enter the commands, as shown below, to set the TRIM and EXTEND commands while working in 3D.

Command: **TRIM**

Current settings: Projection=None, Edge=No Extend

(Note how in this case, the Projection is set to None and Edge to No Extend. Your computer might be different.)

Select cutting edges … *(Select an object, any object)*

Select objects or <select all>: *(Press Enter)*

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/eRase/Undo]: **P**

Enter a projection option [None/Ucs/View] <None>: **U**

(Enter U for Ucs.)

[Fence/Crossing/Project/Edge/eRase/Undo]: **E**

Enter an implied edge extension mode [Extend/No extend] <No Extend>: **E**

(Enter E for Edge and then another for Extend. That means that the intersection of the two objects that you are trimming or extending do not have to physically intersect. AutoCAD will extend them to find the apparent intersection for you.)

Command: **EXTEND**

Current settings: Projection=Ucs, Edge=Extend

Select cutting edges …

Select objects or <select all>:

(By entering the EXTEND command, you will now see the settings are set for this command as well.)

Command:

AUTHOR'S COMMENTS: The way the TRIM and EXTEND commands projects into 3D space can be configured in your AutoCAD software. These settings are saved in the registry which means they will affect all drawings modified on your computer. You only are required to set either the TRIM or the EXTEND command and the settings will be used by both. Complete Step 4 to set your TRIM and EXTEND commands for the AutoCAD 3D book.

Step 5

Enter the TRIM command as shown below. The Current settings: should be set as shown. If not, go back and redo Step 4. If they match, abort the command.

Command: **TRIM**

Current settings: Projection=UCS, Edge=Extend

Select boundary edges …

Step 6

Enter the ROTATE3D, as shown below, to rotate the model 90 degrees counterclockwise around the X axis. (Figure Step 6)

Command: **ROTATE3D**

Current positive angle: ANGDIR=counterclockwise ANGBASE=0

Select objects:

Specify opposite corner: 22 found

(Using a window, select all the objects)

Select objects:

Specify first point on axis or define axis by [Object/Last/View/Xaxis/Yaxis/Zaxis/2points]: **X**

(Rotate around the X axis.)

Specify a point on the X axis <0,0,0>:

(Press Enter to select 0,0,0 as the base point of the rotation.)

Specify rotation angle or [Reference]: **90**

(The angle is positive since looking along the X axis towards 0,0,0, counterclockwise is the direction you want to rotate.)

Command:

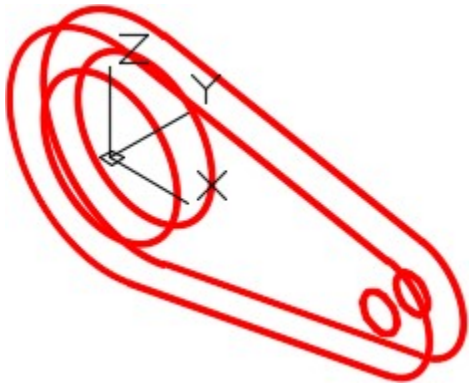


Figure Step 6

AUTHOR'S COMMENTS: Unlike the ROTATE command, where the objects are rotated around a point, the ROTATE3D command rotates the objects around two XYZ points or an axis.

Step 7

Using what you learned in Step 6, rotate the model 90 degrees around the Y axis. This rotation will have be negative 90 degrees since the rotation is clockwise. (Figure Step 7)

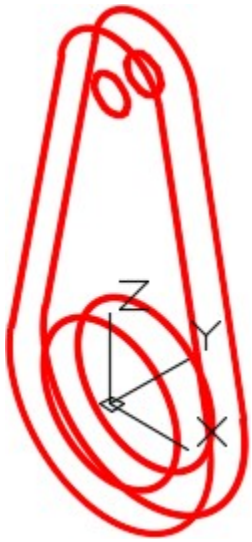


Figure Step 7

AUTHOR'S COMMENTS: To find whether the rotation direction is either counterclockwise or clockwise, look at 0,0,0 from the positive end of the axis that you are rotating around.

Step 8

Change the UCS to the front and locate it at the centre of the circle. (Figure Step 8)

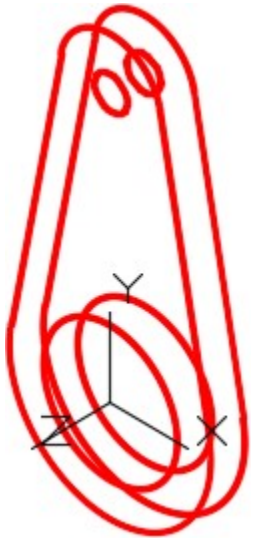


Figure Step 8

Step 9

On layer: Construction, draw a line from 0,0,0 at any angle and any length. The line you draw does not have to match the figure exactly. Draw it by eye. (Figure step 9)

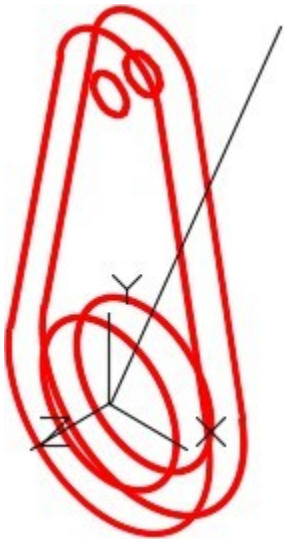


Figure Step 9

Step 10

Change to the Front view to ensure that the line was drawn correctly. (Figure Step 10)

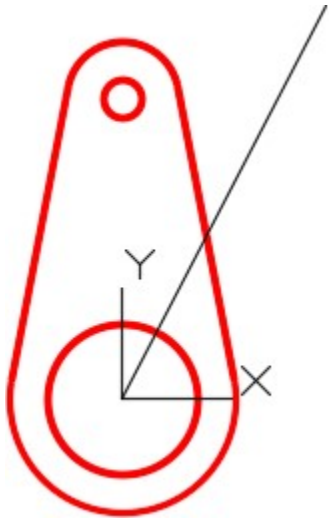


Figure Step 10

MUST KNOW: The TRIM and EXTEND commands should have the following default settings when working in 3D:

Current settings: Projection=UCS, Edge=Extend

Step 11

Change the current view to SE Isometric and enter the ROTATE3D command, as shown below, to rotate the model using the Reference option. (Figure Step 11)

Command: **ROTATE3D**

Current positive angle: ANGDIR=counterclockwise ANGBASE=0

Select objects:

(Select all of the model objects in a window. Do not include the reference line that you just drew in the selection.)

Specify opposite corner: 22 found

Select objects:

Specify first point on axis or define axis by

[Object/Last/View/Xaxis/Yaxis/Zaxis/2points]: **Z**

(Rotating around the Z axis.)

Specify a point on the Z axis <0,0,0>:

Specify rotation angle or [Reference]: **R**

Specify the reference angle <0>: *(cen)* **P1**

Specify second point: *(cen)* **P2**

Specify the new angle: *(cen)* **P1**

(Note that you have to select the P1 location twice.)

Specify second point: *(end)* **P3**

Command:

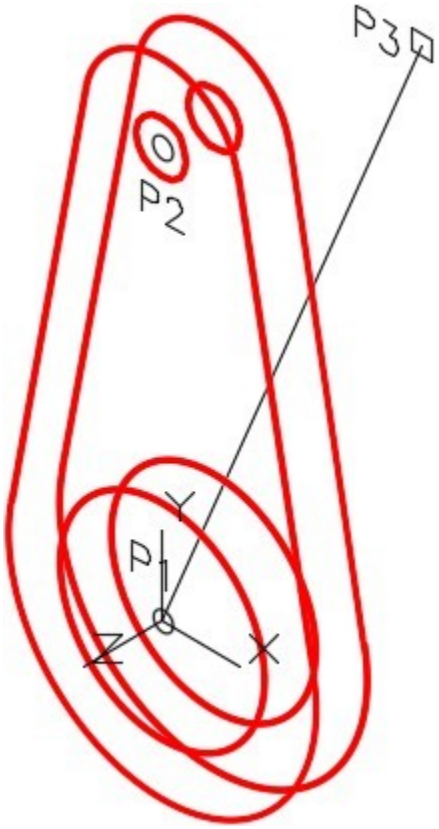


Figure Step 11

MUST KNOW: Unlike the ROTATE command where the object is rotated around a point, the ROTATE3D uses two XYZ points to rotated around. You can use the X, Y or Z axis or two endpoints of a line.

Step 12

The wireframe model should now appear as shown in Figure Step 12A. Change the current view to Front to ensure that the model was rotated correctly. (Figure Step 12A and 12B)

Figure Step 12A

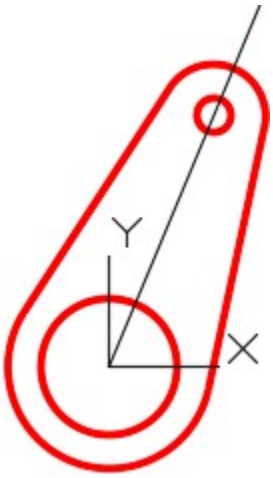


Figure Step 12B

AUTHOR'S COMMENTS: There is no key for this workalong and any future workalongs in theAutoCAD 3D book. Keys are supplied are for the lab exercises. The reason for this is I want to slowly remove your reliance on using keys to check the accuracy of your models. Since keys are not available to you after you complete this book, you must learn how to check the accuracy of your models on your own.

Step 13

Save and close the drawing.

USER TIP: When using the ROTATE3D command, ensure that the ANGDIR and ANGBASE system variables are set as shown below. These setting will display when you enter the command, as you can see below. After you become more familiar using 3D, you can change these settings. While working on the AutoCAD 3D book, leave them set as shown.

Command: **ROTATE3D**

Current positive angle: ANGDIR=counterclockwise ANGBASE=0

Key Principles

Key Principles in Module 8

1. While working in 3D, the TRIM and EXTEND commands should have the following default settings: Projection=UCS, Edge=Extend
2. The ROTATE command, used for a 2D rotate, uses a XY point to rotate For a 3D rotate using the ROTATE3D command, one of the axises, a line or two XYZ points must be specified to rotate around.
3. To find whether the 3D rotation direction is counterclockwise or clockwise, look at 0,0,0 from the positive end of the axis you are rotating around.

Lab Exercise 8-1

Time allowed: 60 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 08-1	3D Layout English	Inches

Step 1

Draw all construction objects on layer: Construction and all model objects on layer: Model.

Step 2

Draw a wireframe model of the object shown in the figure. (Figure Step 2A, 2B, 2C,and 2D)

Step 3

When complete, freeze layer: Construction.

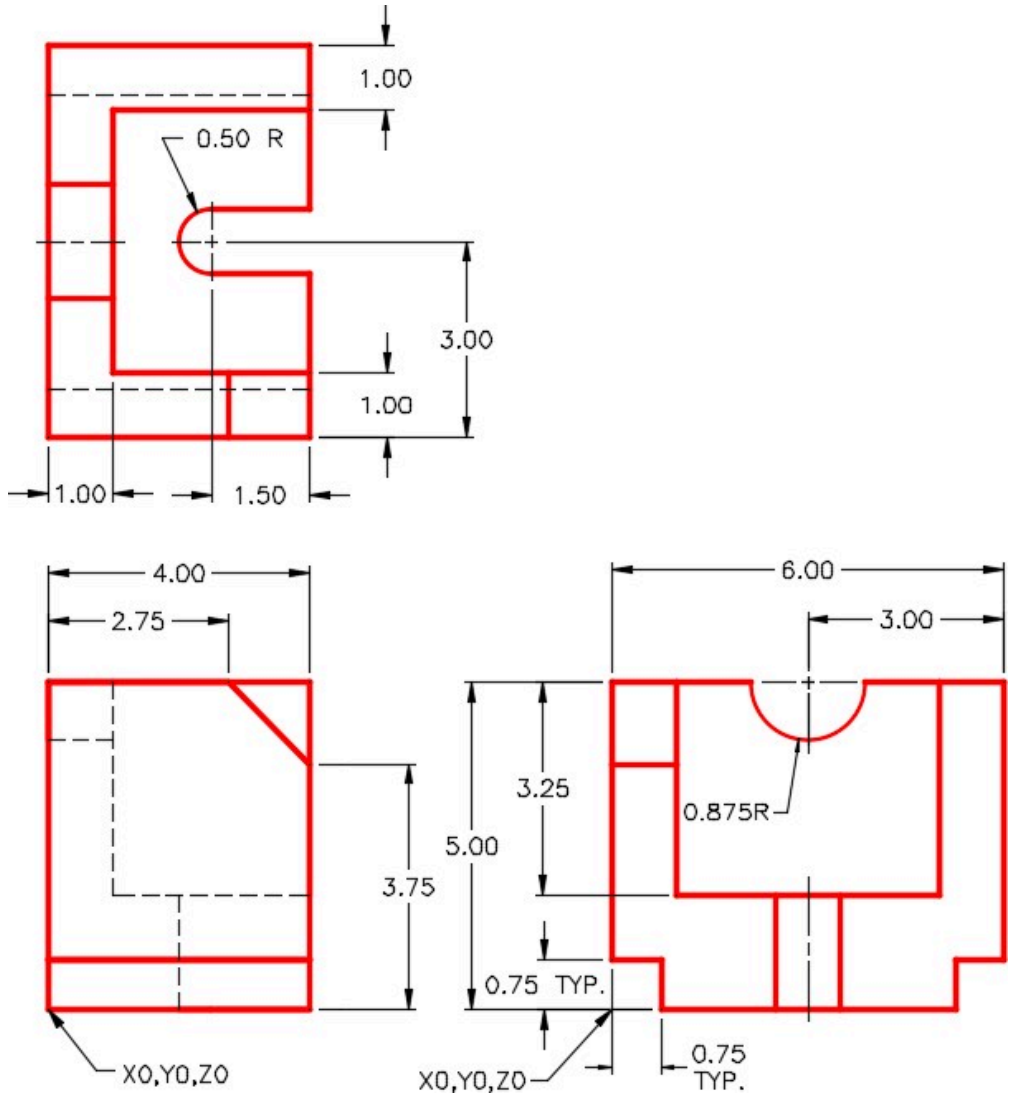


Figure Step 2A
Dimensioned Multiview Drawing

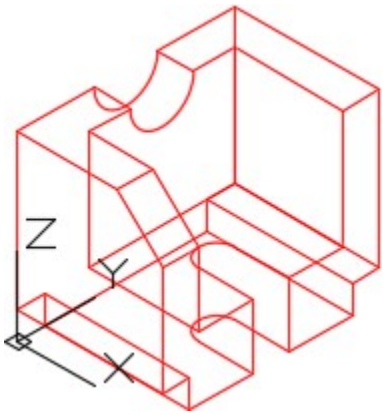


Figure Step 2B
Completed Wireframe Model SE
Isometric View

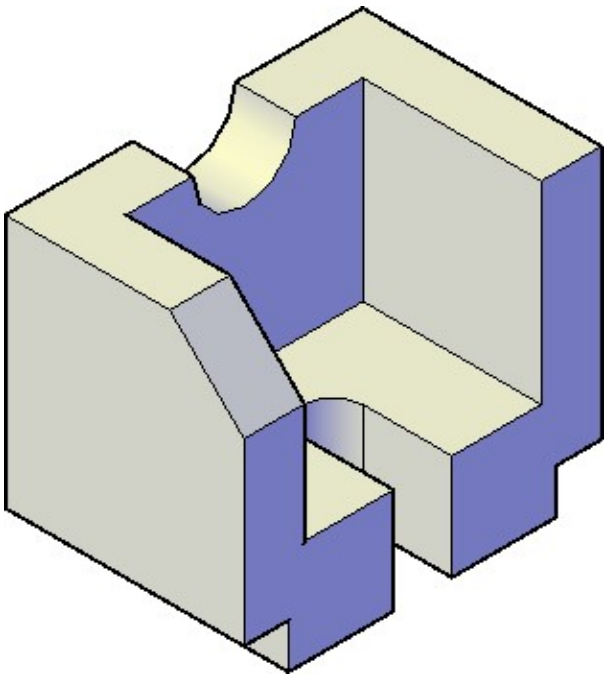


Figure Step 2C
Solid Model – SE Isometric View

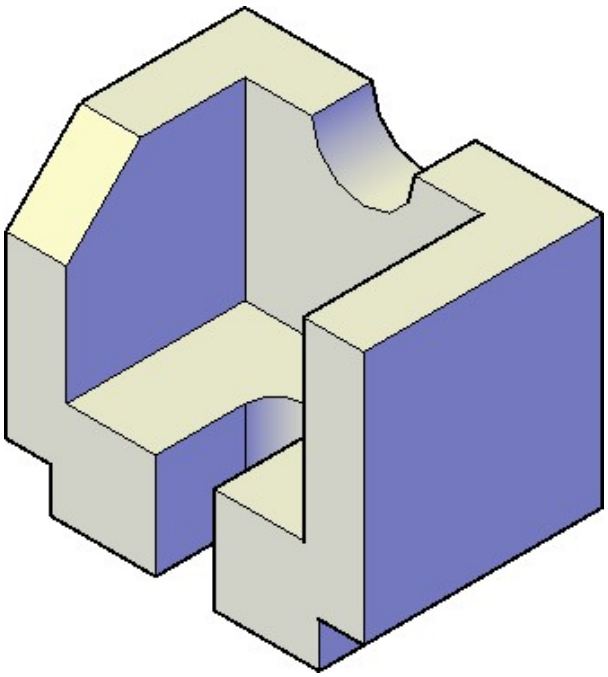


Figure Step 2D
Solid Model – NE Isometric View

Step 4

Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Inches. Change the location of the UCS to World and check the model with the key.

Step 5

Ensure that the USC is set to World and the current view is SE Isometric. (Figure Step 5)

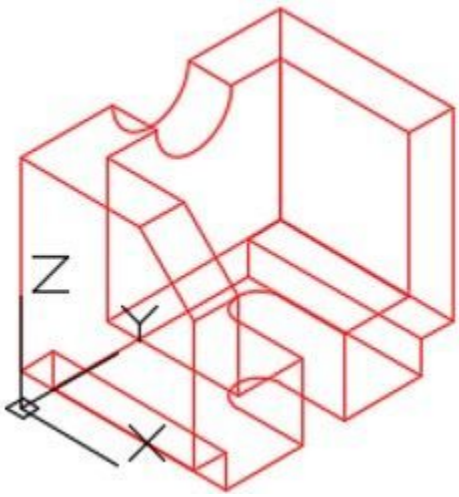


Figure Step 5

Step 6

Rotate the model 90 degrees around the X axis. (Figure Step 6)

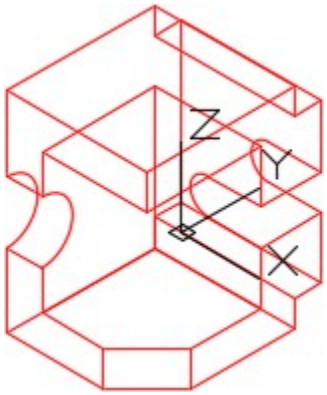


Figure Step 6

Step 7

Rotate the model 90 degrees around the Y axis. (Figure Step 7)

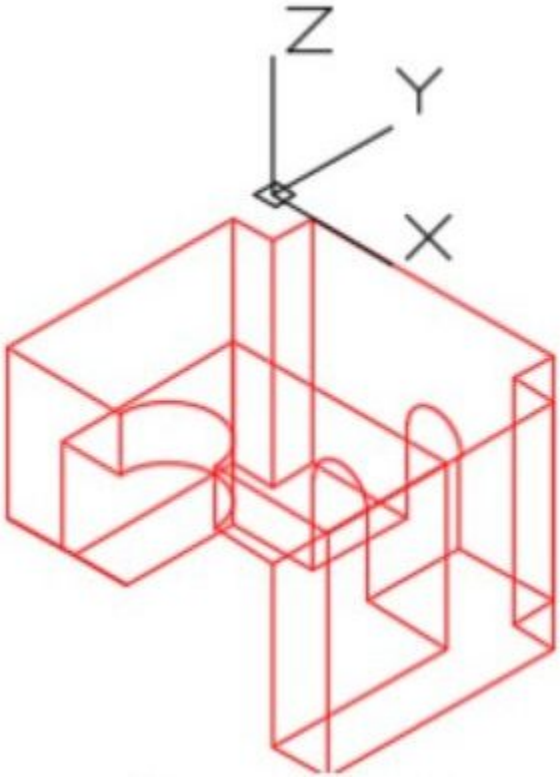


Figure Step 7

Step 8

Rotate the model 90 degrees around the X axis. (Figure Step 8)

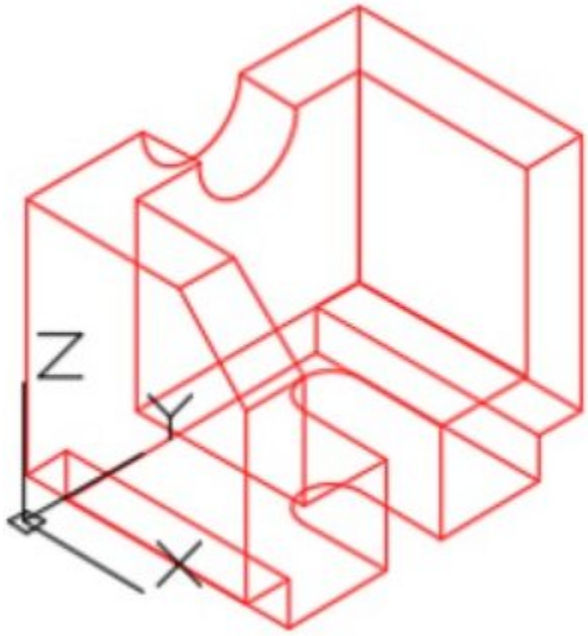


Figure Step 8

Step 9

Rotate the model 90 degrees around the Z axis. (Figure Step 9)

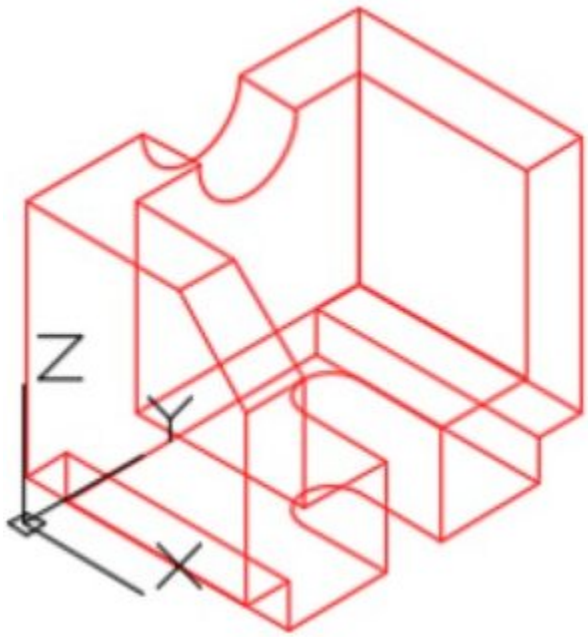


Figure Step 9

AUTHOR'S COMMENTS: If you take note of where the WCS is located in each figure and where is was located in the previous figure, it will help you select which axis to rotate around.

Step 10

Save and close the drawing.

Lab Exercise 8-2

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 08-2	3D Layout English	Inches

Step 1

Draw all construction objects on layer: Construction and all model objects on layer: Model.

Step 2

Draw a wireframe model of the object shown in the figure. (Figure Step 2A, 2B, 2C, and 2D)

Step 3

When complete, freeze layer: Construction.

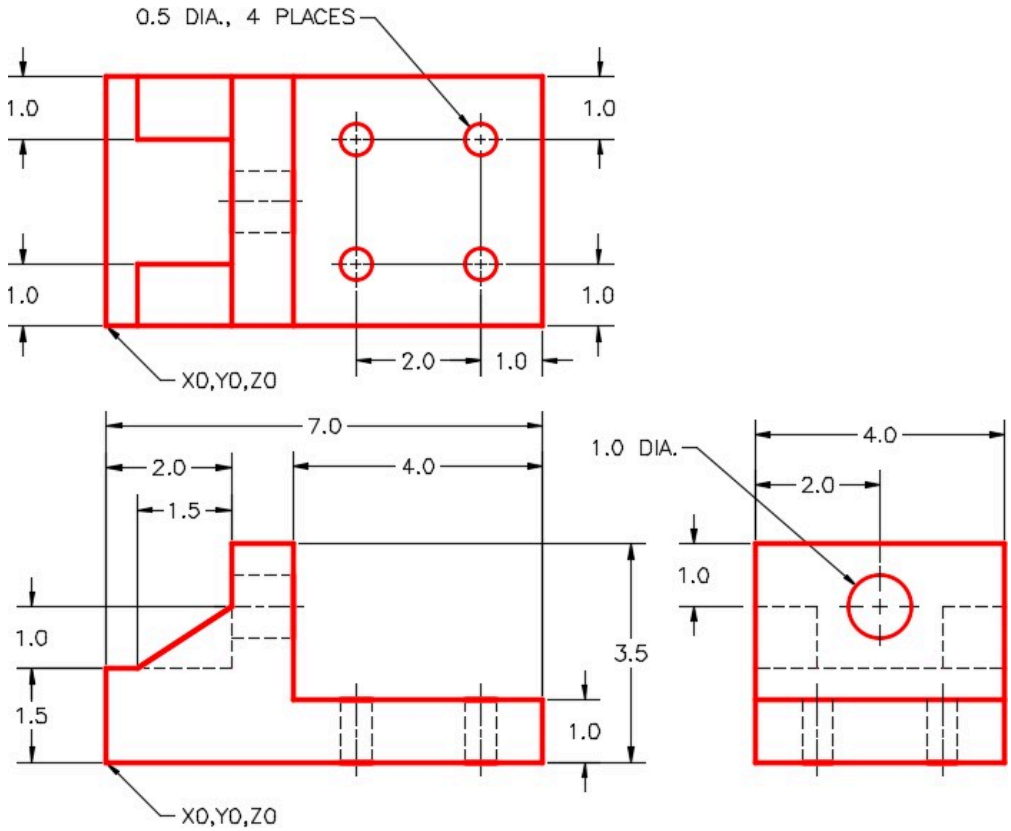


Figure Step 2A
Dimensioned Multiview Drawing

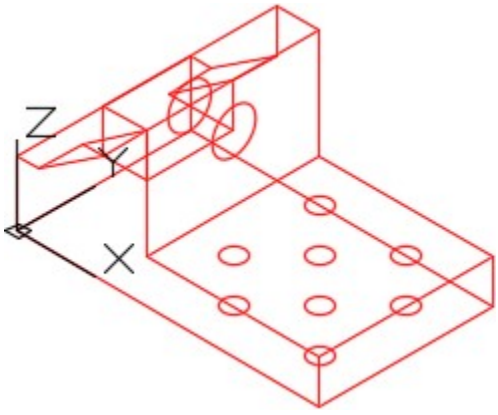


Figure Step 2B
Completed Wireframe Model SE Isometric View

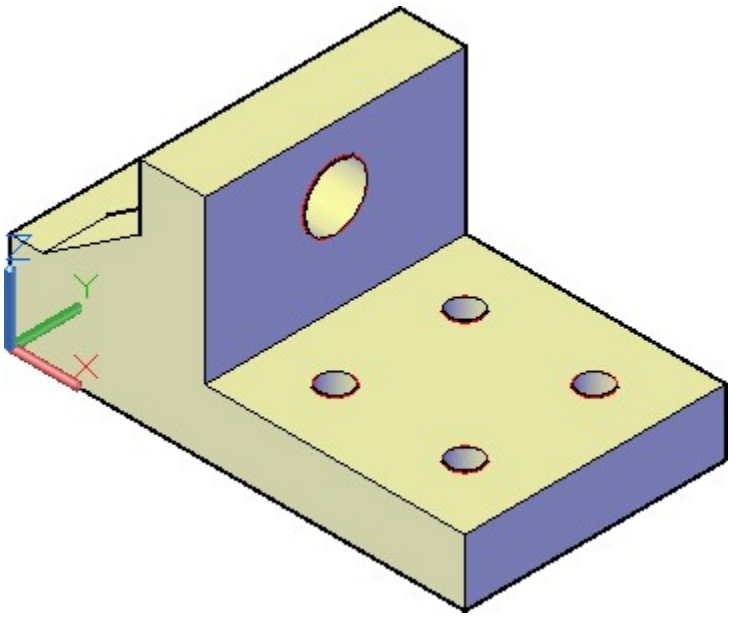


Figure Step 2C
Solid Model SE Isometric View

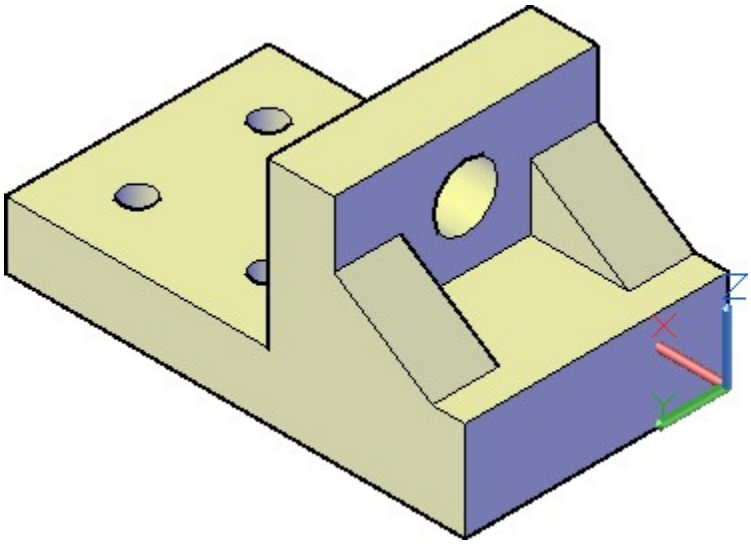


Figure Step 2D
Solid Model NW Isometric View

Step 4

Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Inches. Change the location of the UCS to World and check the model with the key.

Step 5

Save and close the drawing.

Module 9: Arraying and Mirroring 3D Models

Learning Outcomes

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

1. Apply the 3DARRAY and MIRROR3D commands to construct and modify 3D models.

Modifying 3D Objects

When constructing 3D models, the only time the 2D commands ARRAY and MIRROR can be used is if all the objects that are selected in the command lie on the 2D plane of the current UCS. To array or mirror objects in 3D space, the corresponding 3D commands must be used.

3D Array

A 3D array, using the 3DARRAY command, is very similar to a 2D array, using the ARRAY command, that was taught in the AutoCAD 2D book. A rectangular and a polar 3D array can be preformed.

Rectangular Array

A 2D rectangular array uses rows and columns. In 3D, the third dimension of the array is called a *level*. Rows are along the Y axis, columns are along the X axis and levels are along the Z axis. Distance between the rows, columns and levels can be positive or negative. See Figure 9-1.

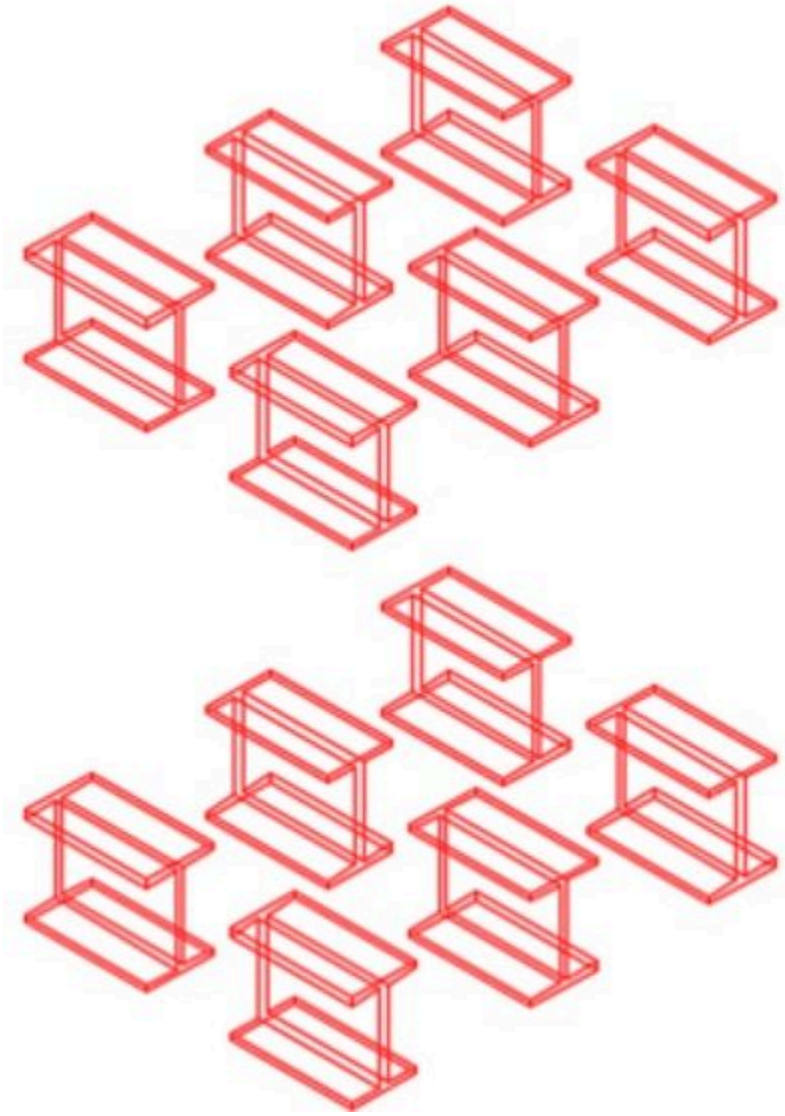


Figure 9-1
3D Rectangular Array

Polar Array

A 2D polar array only requires a XY point to array around while a 3D polar array, two XYZ points, a line or an axis must be specified to array around. See Figure 9-2.

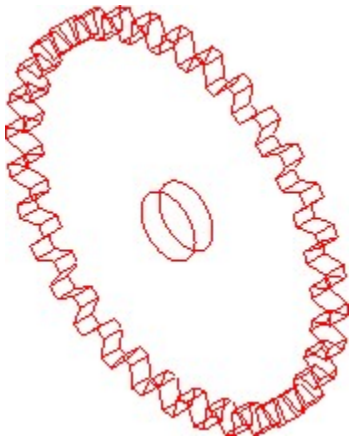


Figure 9-2
3D Polar array

3D Mirroring

A 3D mirror, using the MIRROR3D command, is very similar to a 2D mirror, using the MIRROR command, that was taught in the AutoCAD 2D book. A 2D mirror requires two XY points or a line to mirror around while a 3D mirror requires three XYZ points or a plane to mirror around. See Figure 9-3.

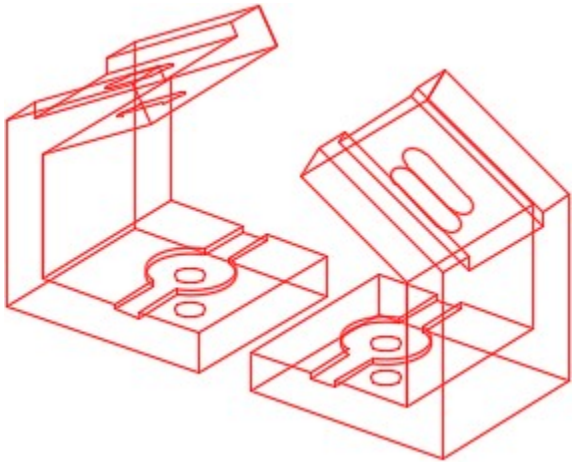


Figure 9-3
3D Mirror

AutoCAD Command: 3DARRAY

The 3DARRAY command is used to array an object(s) in 3D space.
Shortcut: none

AutoCAD Command: MIRROR3D

The MIRROR3D command is used to mirror an object(s) in 3D space.
Shortcut: none

WORK ALONG: Creating a 3D Rectangular Array

Step 1

Using the NEW command, start a new drawing using the template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 09-1.

Step 3

Set layer: Model as the current layer.

Step 4

Using the figures as a reference, draw the wireframe model. Note the location of the WCS. (Figure Step 4A and 4B)

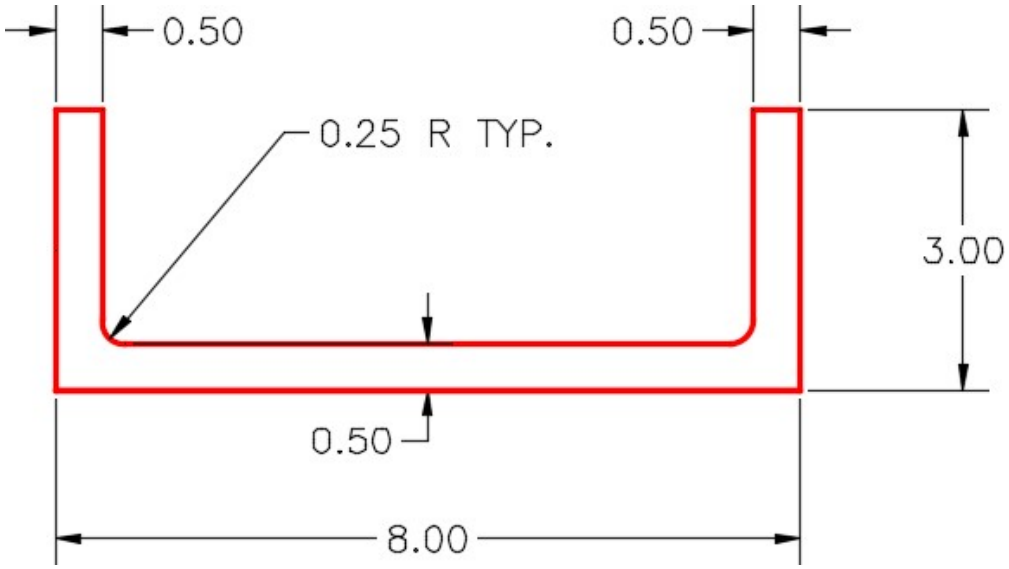


Figure Step 4A
Right Side View

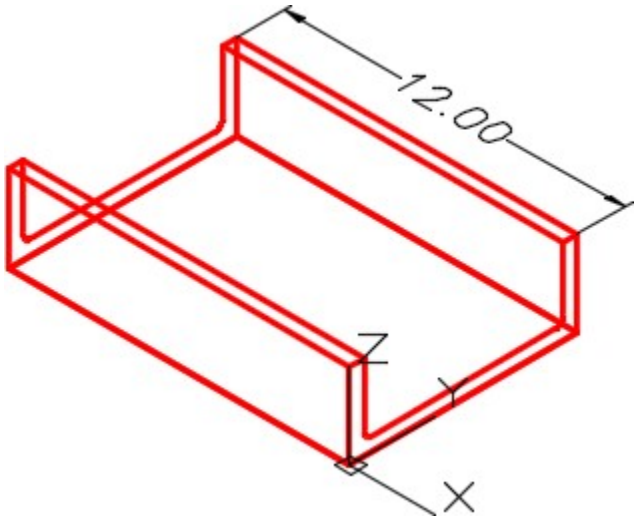


Figure Step 4B
Wireframe Model

Step 5

Using the figures as a reference, draw the circle and then copy it to create a hole through the Front view. (Figure Step 5A, 5B, and 5C)

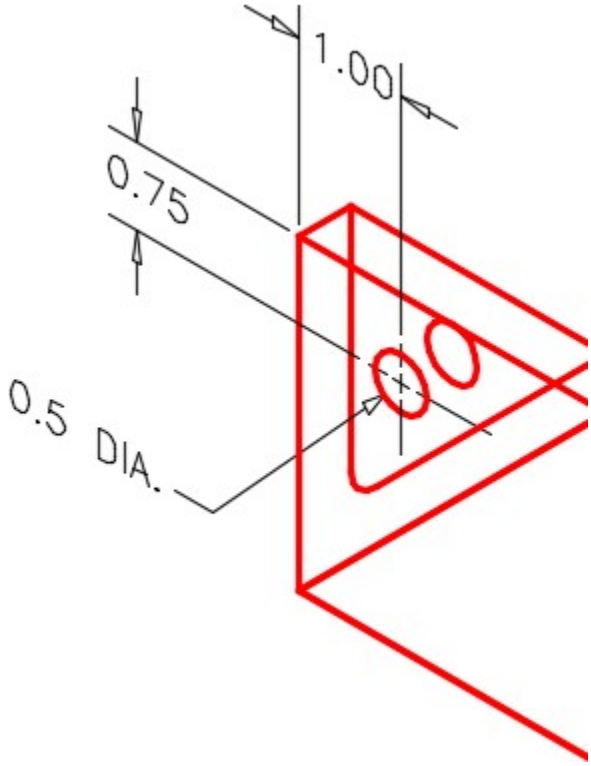


Figure Step 5A
Hole Detail

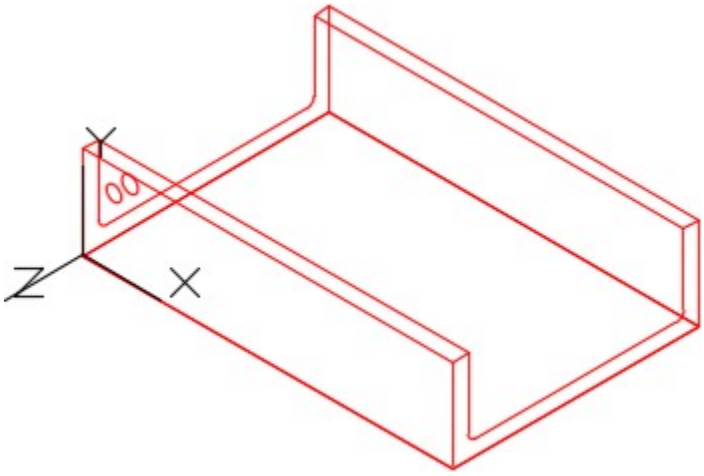


Figure Step 5B
Wireframe Model

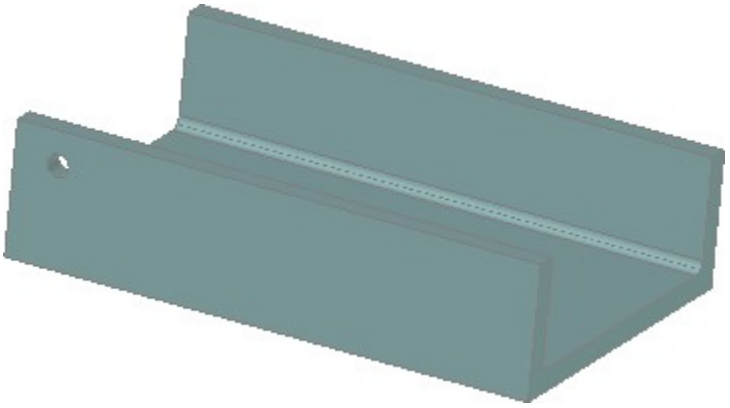


Figure Step 5C
Solid Model

MUST KNOW: When you use the 3DARRAY command, ensure that object snap is disabled as it may affect the array by snapping to an unwanted existing object.

Step 6

Study the figure. It shows the hole pattern that will be used in the 3D ARRAY command. (Figure step 6)

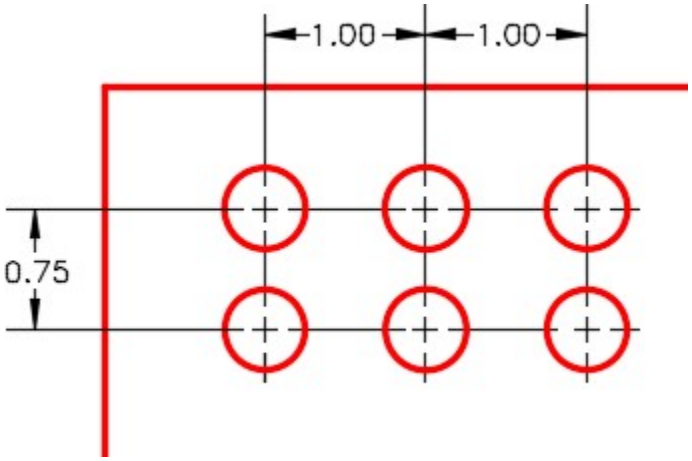


Figure Step 6

AUTHOR'S COMMENTS: Before you execute the 3DARRAY command, you must set the UCS to the correct orientation and location. In this case, you will be doing a rectangular array, therefore, you will want the columns to be along the X axis, the rows along the Y axis and the levels along the Z axis.

Step 7

Ensure that Osnap is disabled.

Step 8

Ensure that the current UCS is set to the Front and enter the 3DARRAY command, as shown below, to array the holes. Your finished drawing should appear as shown. (Figure Step 8A and 8B)

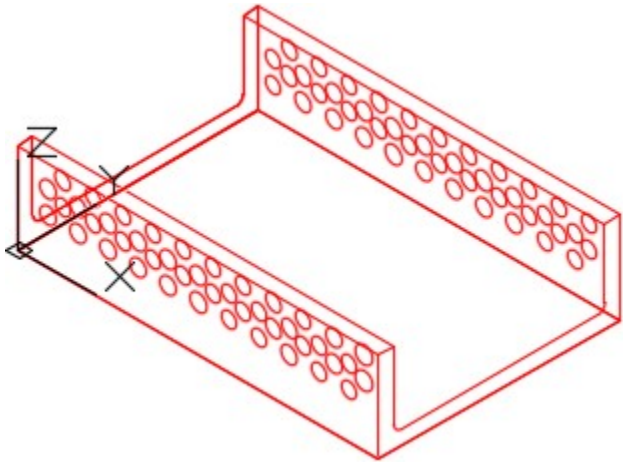


Figure Step 8A

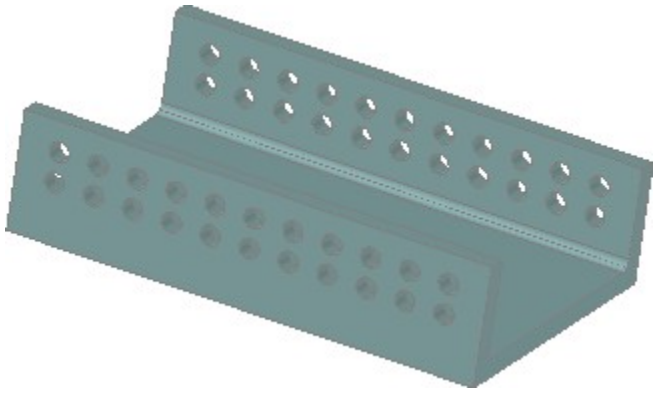


Figure Step 8B
Solid Model – Rotated View

Command: **3DARRAY**

Select objects:

Specify opposite corner: 2 found

(Select the two circles.)

Select objects:

Enter the type of array [Rectangular/Polar] <R>:

(Rectangular array)

Enter the number of rows (—) <1>: **2**

Enter the number of columns (|||) <1>: **11**

Enter the number of levels (…) <1>: **2**

Specify the distance between rows (—): **-0.75**

(In the negative Y direction.)

Specify the distance between columns (|||): **1**

(In the positive X direction.)

Specify the distance between levels (…): **-7.5**

(In the negative Z direction.)

Command:

WORK ALONG: Creating a 3D Polar Array

Step 1

Using the NEW command, start a new drawing using the template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 09-2.

Step 3

Draw all model objects on layer: Model and all construction objects on layer: Construction. (Figure Step 3A, 3B, and 3C)

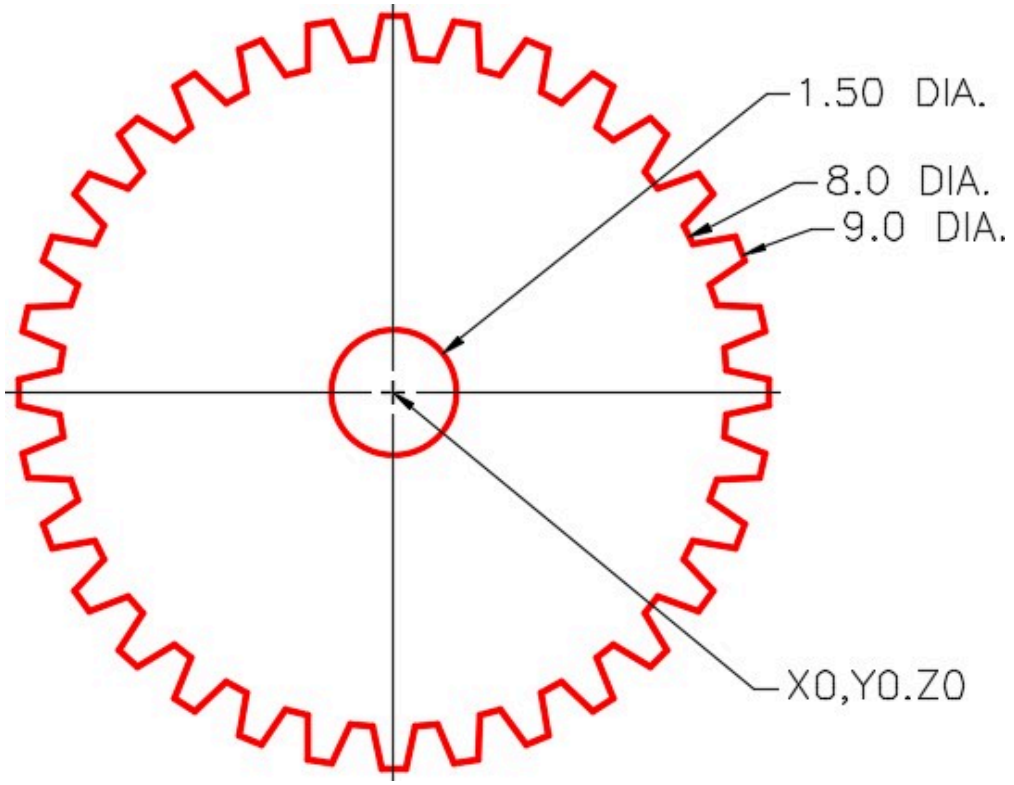


Figure Step 3A
Multiview Drawing – Gear is 0.5 Inches Thick

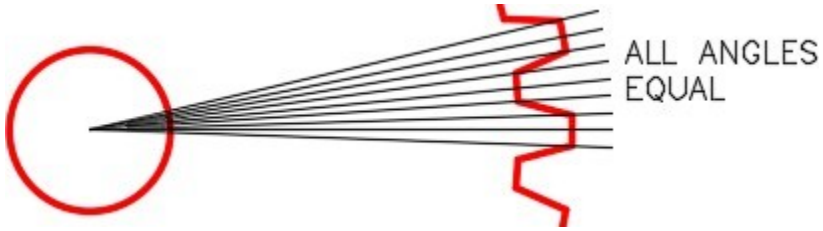


Figure Step 3B
Detail of Gear Teeth

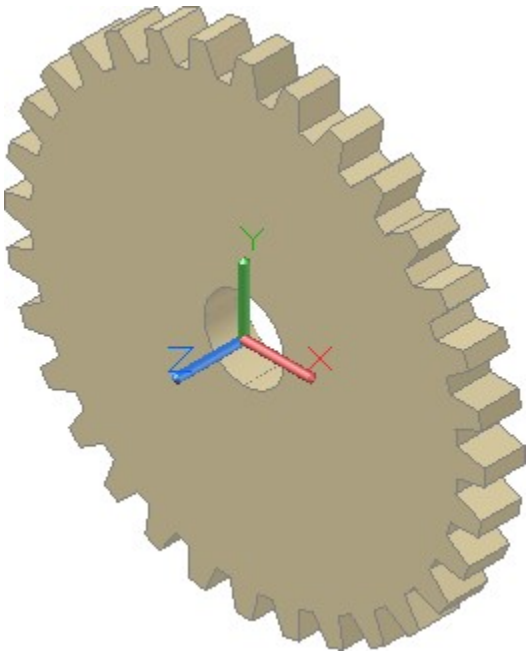


Figure Step 3C
Solid Model – SE Isometric View

Step 4

Set the current UCS to Front and change the current view to Front. Draw three circles and a construction line from the centre to the quad as shown in the figure. Take all of the sizes from the multiview drawing. (Figure Step 4)

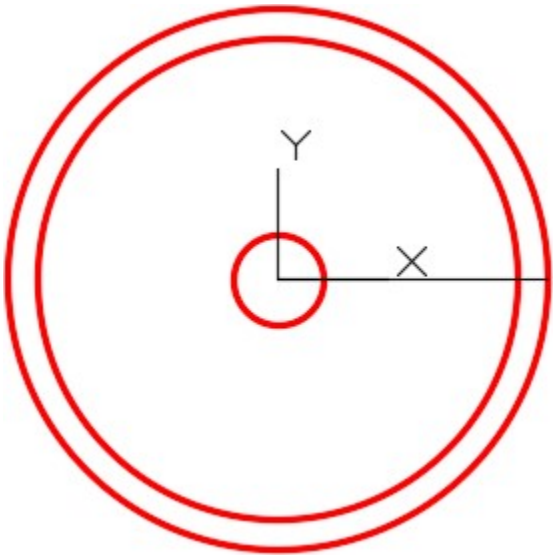


Figure Step 4

Step 5

Using the ARRAY command, array the construction line 192 times around. Use the centre of the circle as the basepoint for the array. (Figure Step 5)

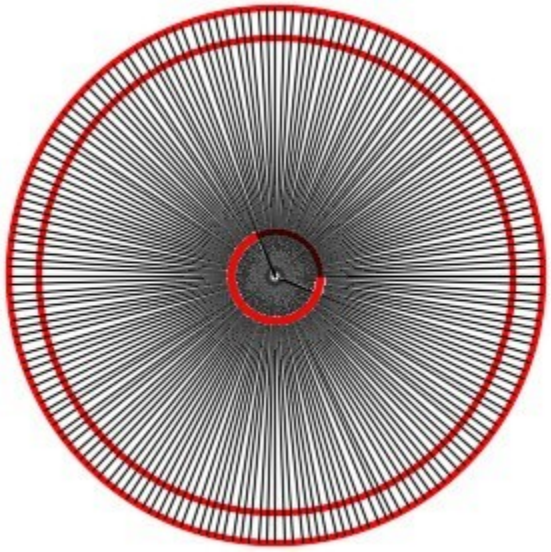


Figure Step 5

AUTHOR'S COMMENTS: Why was the line arrayed 192 times? If you count the gear teeth, you will find there are 32. Count the number of parts (angled lines) one gear tooth requires from the same point on one tooth to the same point on the next tooth. That equals six. Therefore, 6 times 32 equals 192.

Step 6

On layer: Model, draw two gear teeth by drawing lines from the intersection of the lines and the circles. (Figure Step 6)

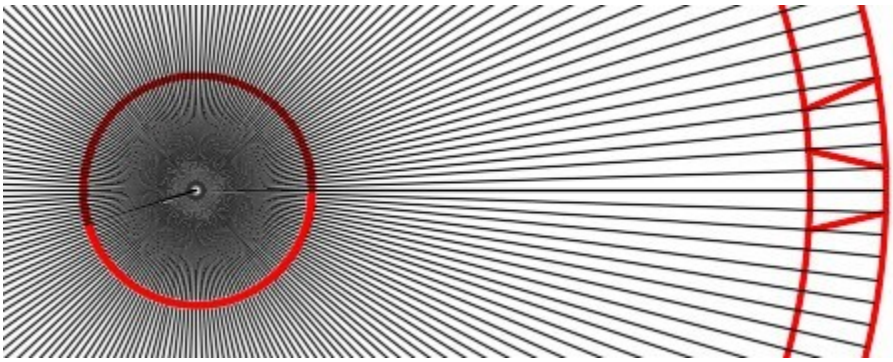


Figure Step 6

Step 7

Turn layer: Construction off and your drawing should appear as shown in the figure. (Figure Step 7)

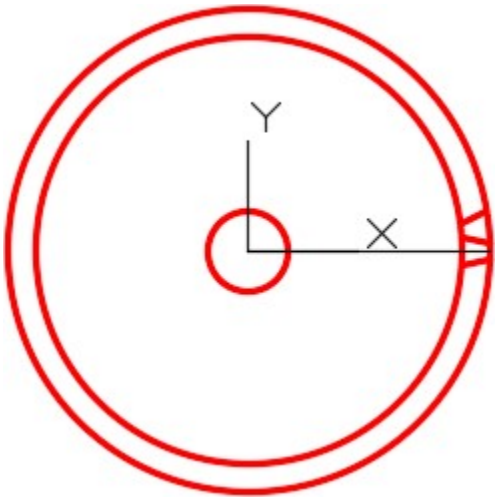


Figure Step 7

Step 8

Trim the two outer circles to form the repeat pattern of one gear tooth. (Figure Step 8)

Figure Step 8

AUTHOR'S COMMENTS: An array repeat pattern is the location from a specified location on one tooth to the same location on the next tooth.

Step 9

Change the current view to SE Isometric. (Figure Step 9)

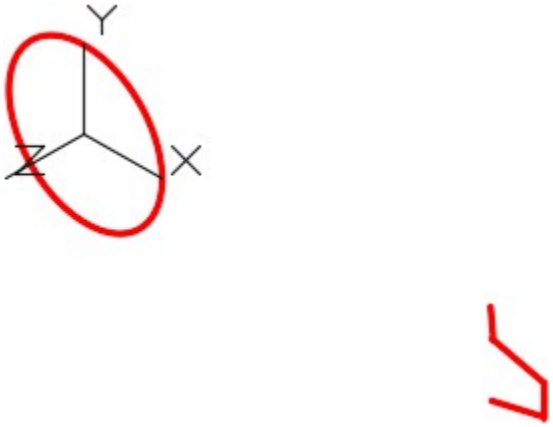


Figure Step 9

Step 10

Copy all objects 0.5 inches in the -Z direction. 0.5 is the thickness of the gear. (Figure Step 10)

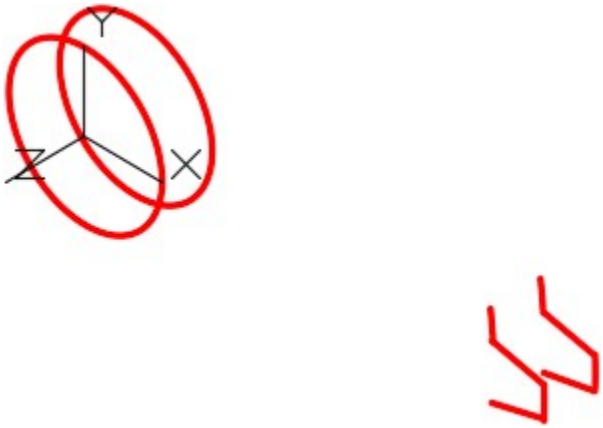


Figure Step 10

Step 11

Add lines to complete the gear tooth. (Figure Step 11)

Figure Step 11

Step 12

Enter the 3DARRAY command, as shown below, to complete the wireframe model of the gear. (Figure Step 12A and 12B)

Command: **3DARRAY**

Select objects:

Specify opposite corner: 12 found

(Select the lines that make one gear tooth in a window.)

Select objects:

Enter the type of array [Rectangular/Polar] <R>:**P**

(Polar array.)

Enter the number of items in the array: **32**

Specify the angle to fill (+=ccw, -=cw) <360>:

(Accept the default.)

Rotate arrayed objects? [Yes/No] <Y>:

(Accept the default.)

Specify centre point of array: (cen) **P1**

Specify second point on axis of rotation: (cen) **P2**

(From centre to centre of the circles defines the endpoints of a line to be used as the axis of the array.)

Command:

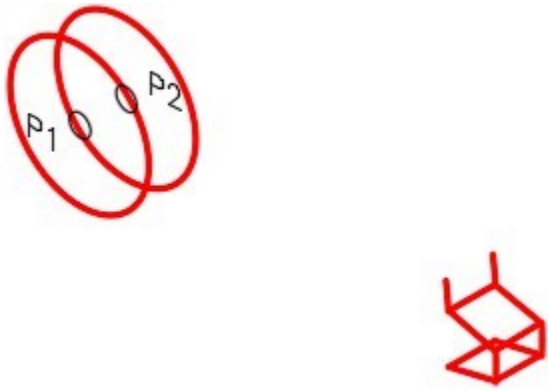


Figure Step 12A

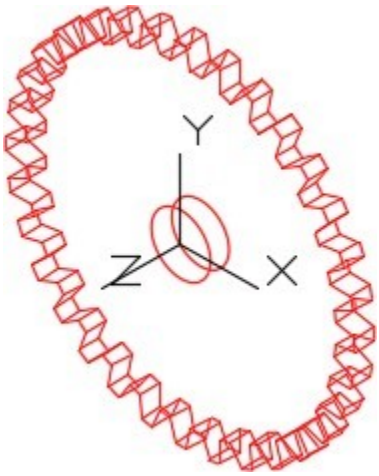


Figure Step 12B

Step 13

Save and close the drawing.

WORK ALONG: Creating a 3D Mirror

Step 1

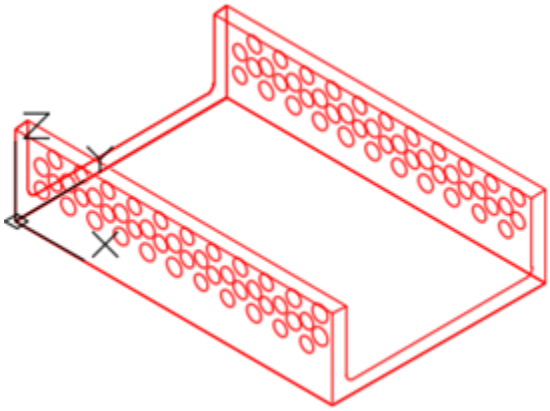


Figure Step 1

Open the drawing: AutoCAD 3D Workalong 09-1. The drawing should appear as shown in the figure. (Figure Step 1)

Step 2

Using the SAVEAS command, save the drawing with the name: AutoCAD 3D Workalong 09-3.

Step 3

Enter the MIRROR3D command, as shown below, to mirror the model. (Figure Step 3)

Command: **MIRROR3D**

Select objects:

Specify opposite corner: 124 found

(Select all object in the model in a window.)

Select objects:

Specify first point of mirror plane or [Object/Last/Zaxis/View/XY/YZ/ZX/3points] <3points>: *(end)* **P1**

Specify second point on mirror plane: *(end)* **P2**

Specify third point on mirror plane: *(end)* **P3**

Delete source objects? [Yes/No] <N>:

(Accept the default.)

Command:

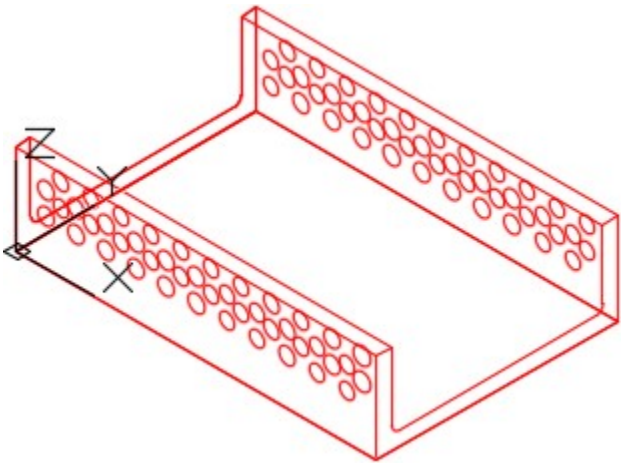


Figure Step 3

AUTHOR'S COMMENTS: If you join the three points with an imaginary line, they form a plane which is used by the MIRROR3D command to mirror the model around. Figure 9-4 shows the plane.

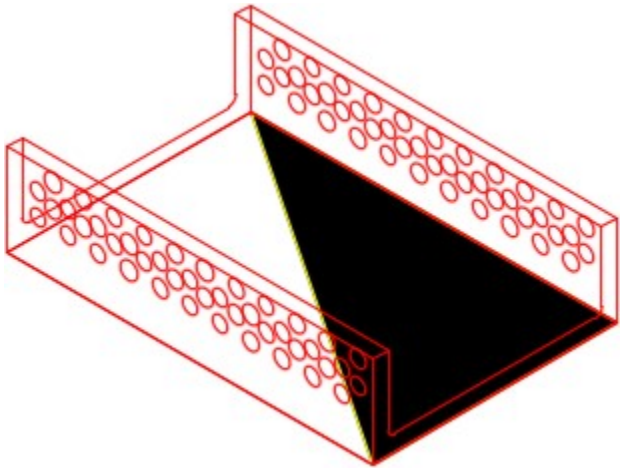


Figure 9-4
Mirror Plane

Step 4

The completed mirrored model is shown in the figure. (Figure Step 4A and 4B)

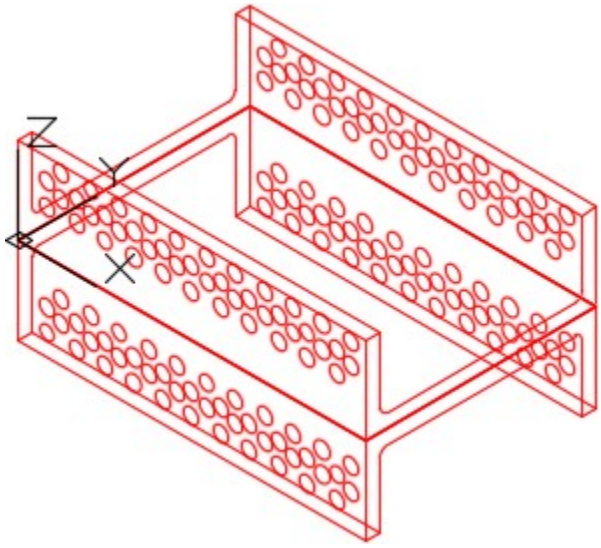


Figure Step 4A

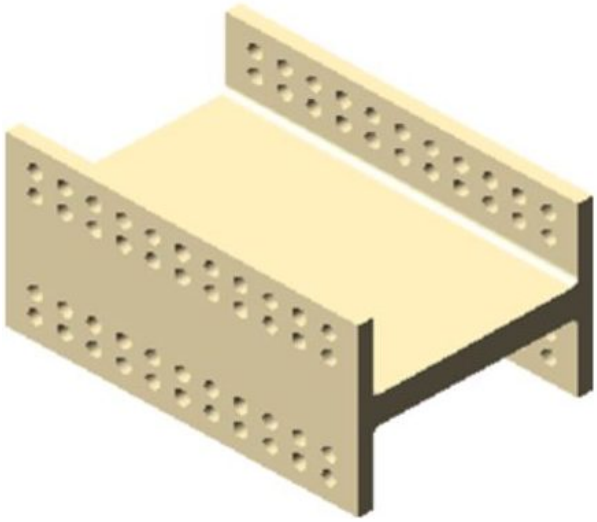


Figure Step 4B
Solid Model – SE Isometric View

Step 5

Save and close the drawing.

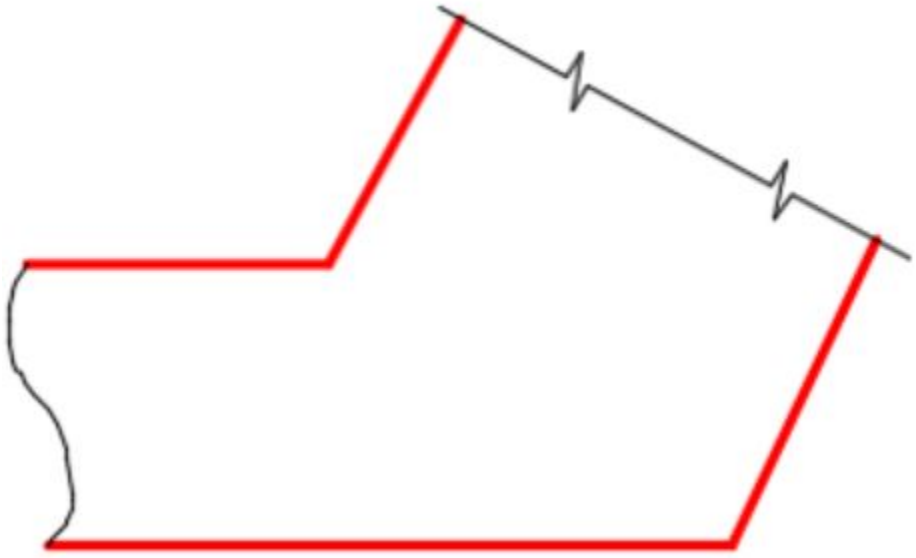
Drafting Lesson: Break Lines

To simplify or speed up drawing orthographic views of a model, sometimes views are only partially

drawn. In these cases, the cut off portion of the view is not required for the reader to visualize or construct the model. See the figures for examples of short and long break lines.

Long Break
(Drawn by Eye)

Short Break
(Drawn by Eye)



Example Of Break Lines

Key Principles

Key Principles in Module 9

1. The third dimension of a 3D array is called a level.
2. When rotating in 3D, you must rotate around two XYZ points or a line.
3. When mirroring in 3D, you must mirror around a three XYZ points or a plane.
4. Always disable object snap when executing a 3D array.

Lab Exercise 9-1

Time allowed: 50 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 09-1	3D Layout Metric	Millimeters

Step 1

Draw all construction objects on layer: Construction.

Step 2

Draw all model objects on layer: Model.

Step 3

Draw a wireframe model of the object shown in the figure. (Figure Step 3A and 3B)

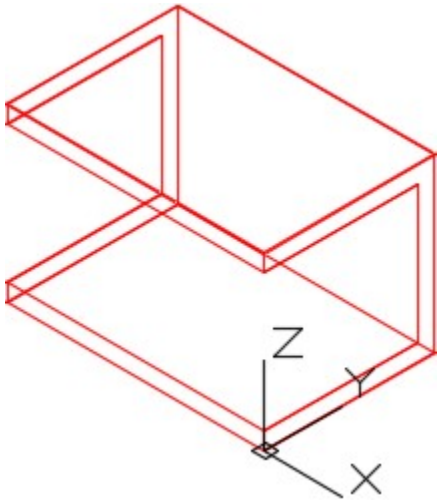


Figure Step 3A
SE Isometric View

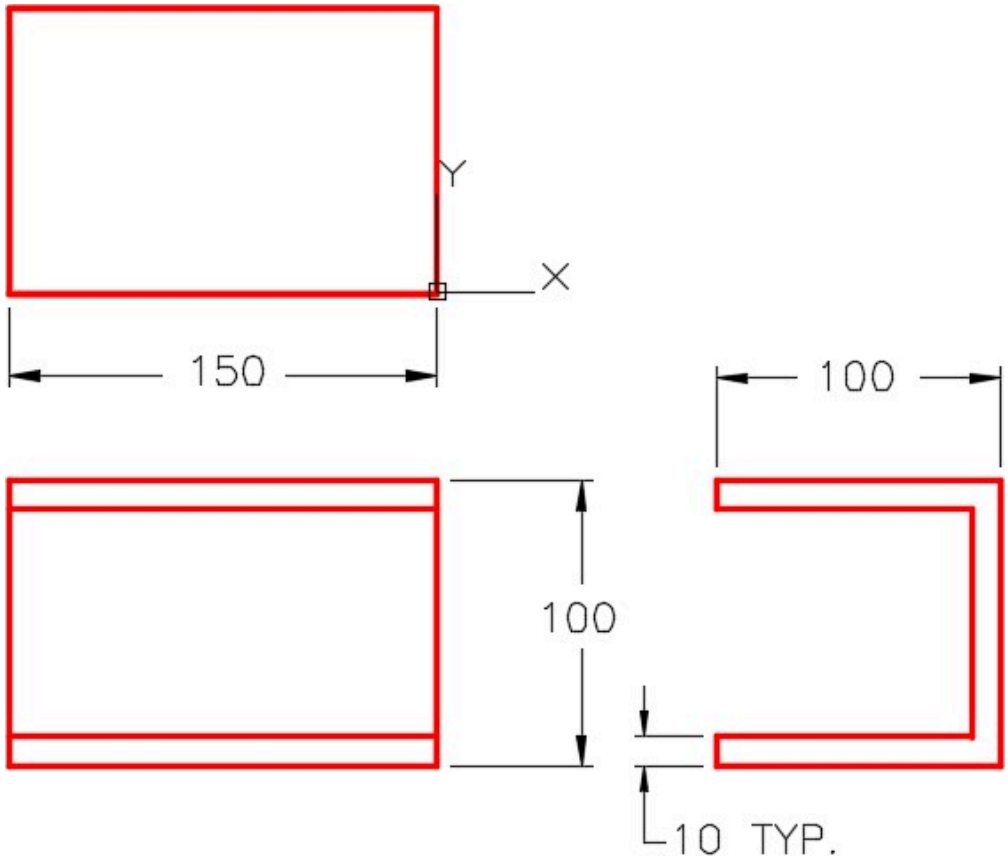


Figure Step 3B
Dimensioned Multiview Drawing

Step 4

Using the figure as a reference, anywhere in model space, draw a wireframe model of one nut and one bolt. (Figure Step 4A and 4B)

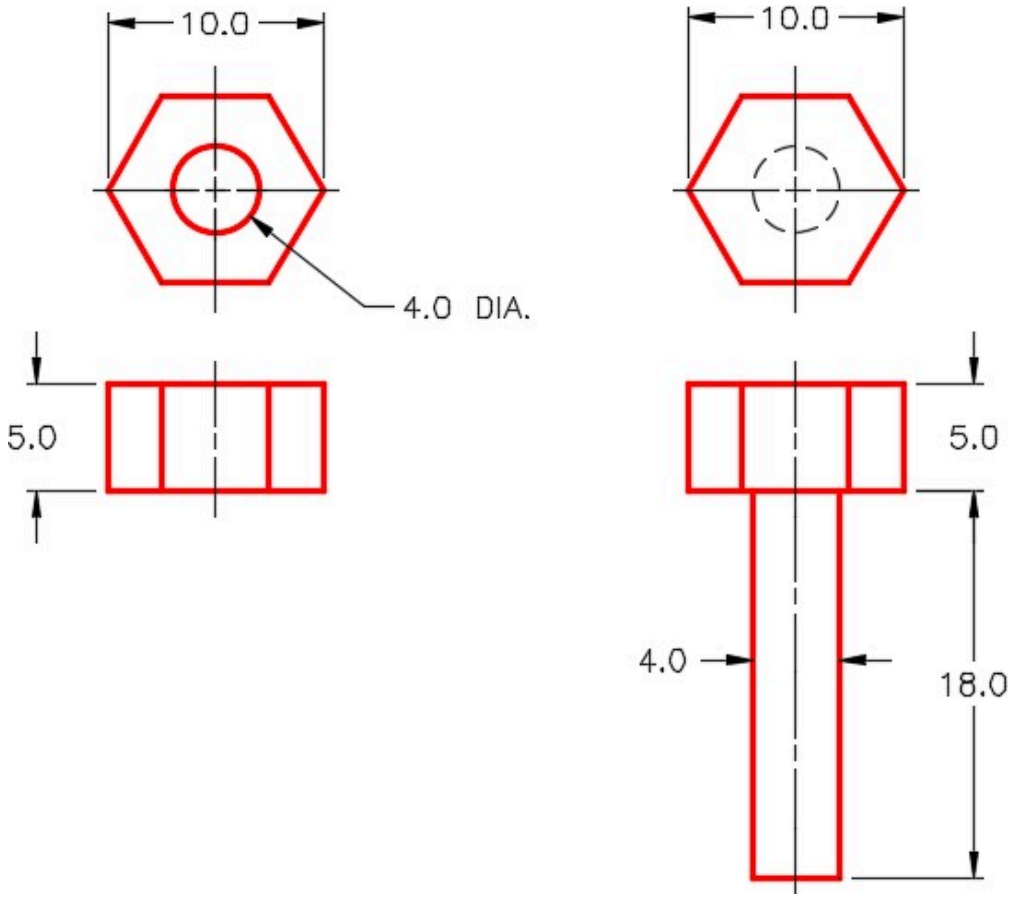


Figure Step 4A
Dimensioned Multiview Drawings of Nut and Bolt

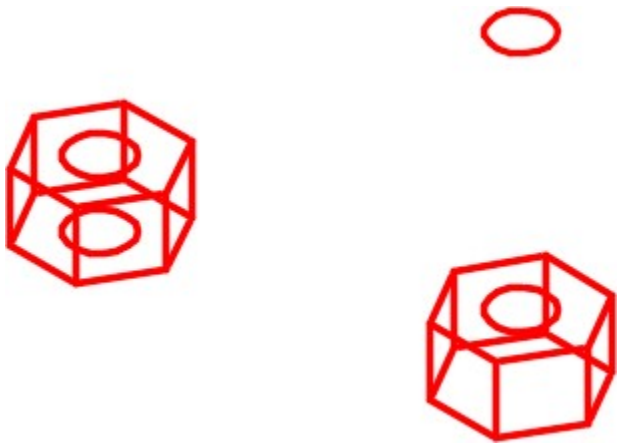


Figure Step 4B
Wireframe Models of Nut and Bolt

AUTHOR'S COMMENTS: Note how the bolt is drawn with the head down and thread up.

Step 5

Move one bolt and one nut to the bottom outside bolt location as shown in the figure. The bolt comes up from the bottom and nut is on top of the plane. (Figure Step 5A, 5B, 5C, and 5D)

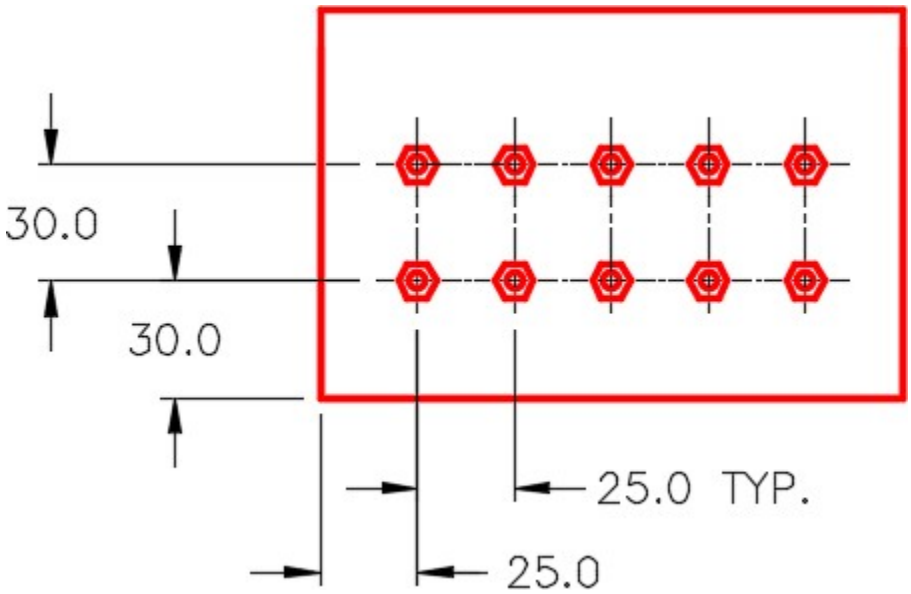


Figure Step 5A
Bolt Pattern – Bottom View

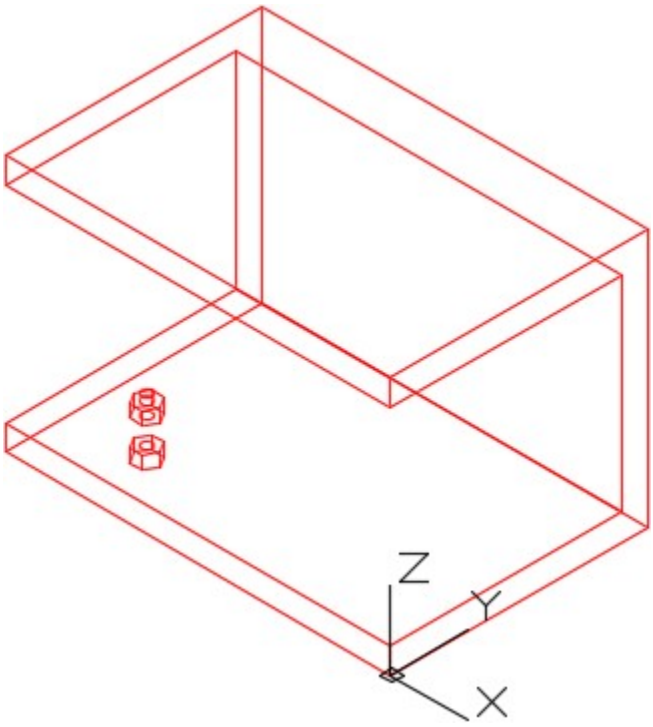


Figure Step 5B

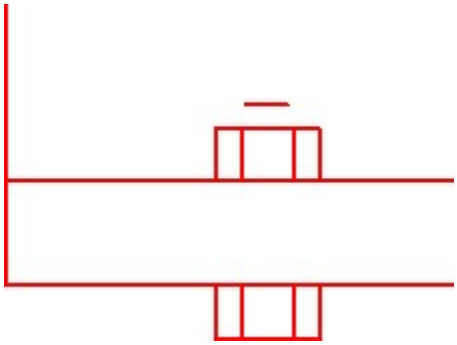


Figure Step 5C

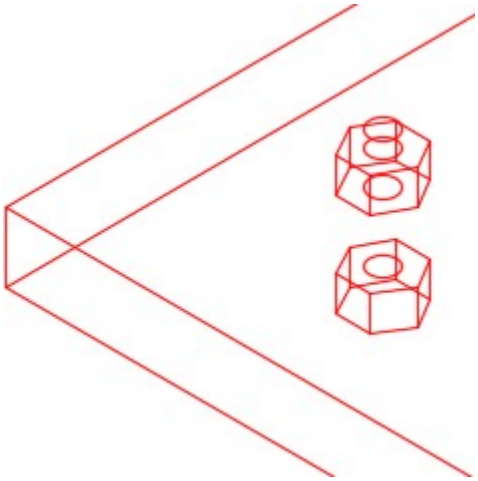


Figure Step 5D

Step 6

Array the nut and bolt to match the figure. (Figure Step 6)

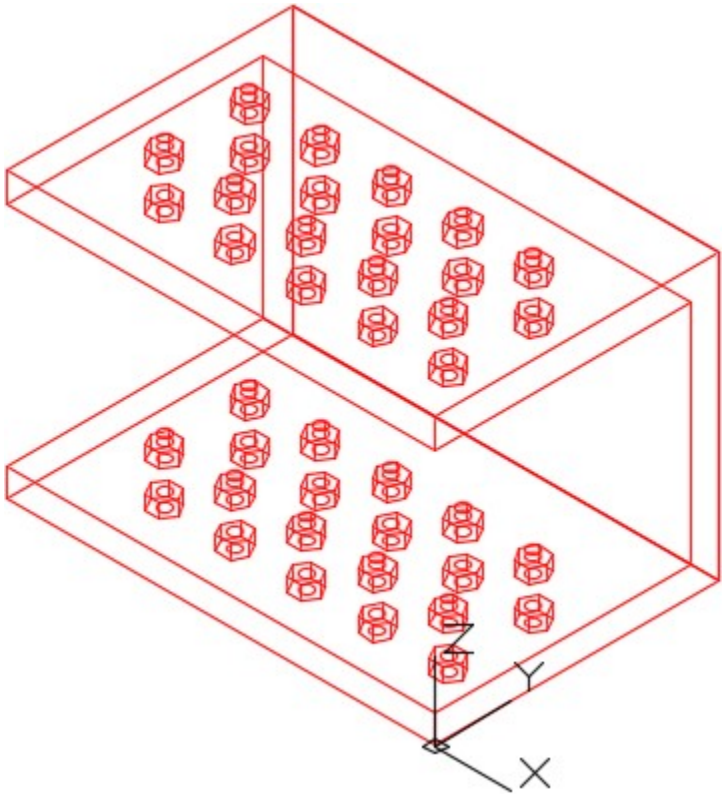


Figure Step 6

Step 7

Mirror the completed model. (Figure Step 7)

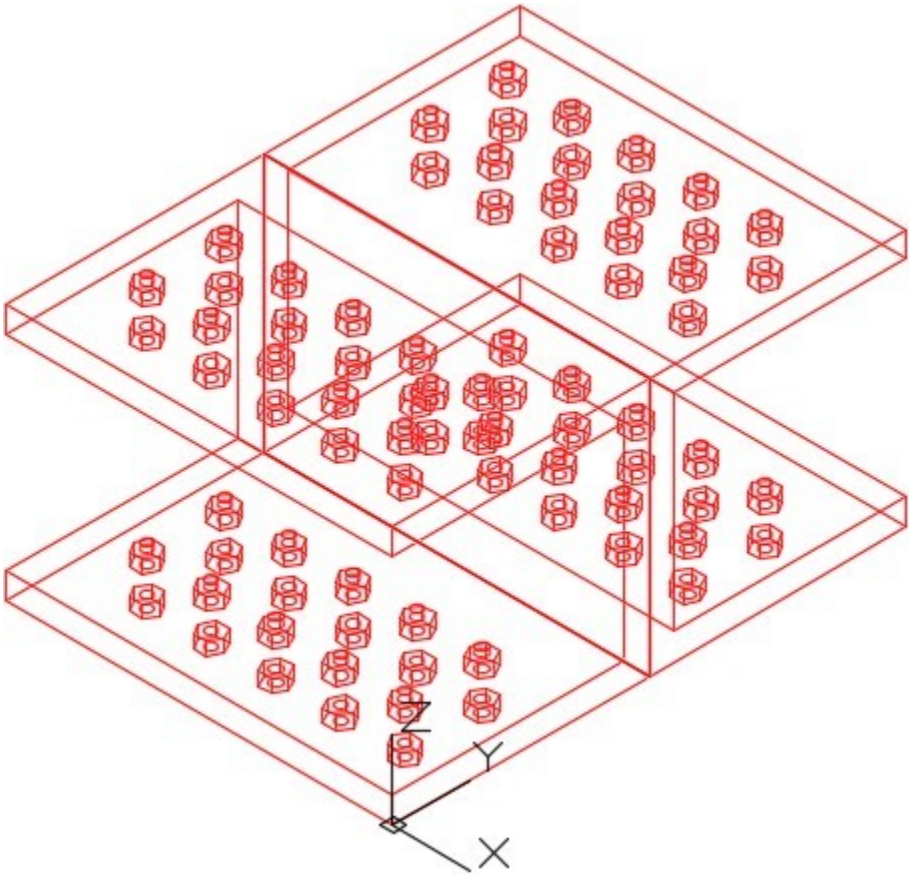


Figure Step 7

Step 8

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

Step 9

Change the current UCS to World and check the model with the key.

Step 10

Save and close the drawing.

Lab Exercise 9-2

Time allowed: 50 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 09-2	3D Layout English	Inches

Step 1

Draw a wireframe model of the centre octagon and one arm as shown in the figures. Details of the arm are in Figure Step 1D Detail B and Figure Step 1E Detail A. (Figure Step 1A, 1B, 1C, 1D, and 1E)

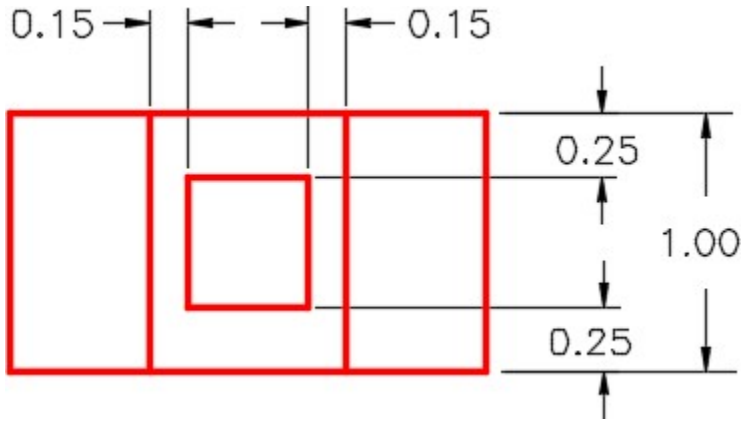


Figure Step 1A

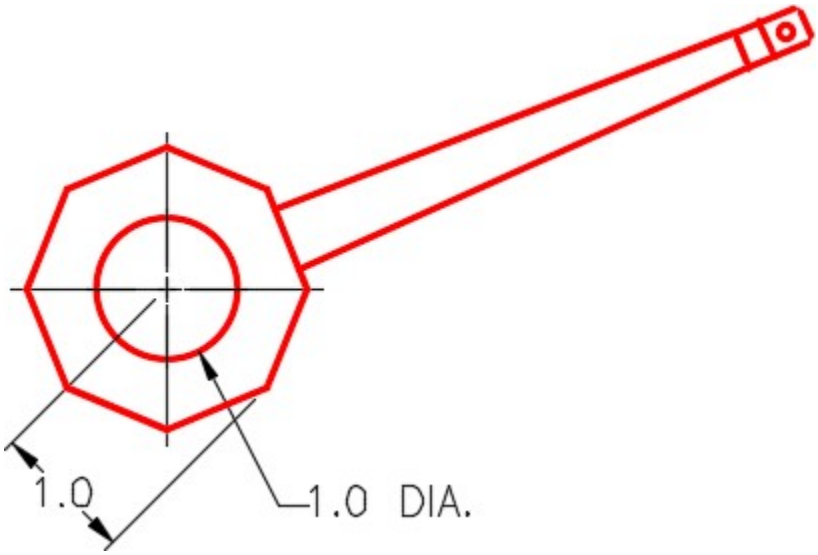


Figure Step 1B

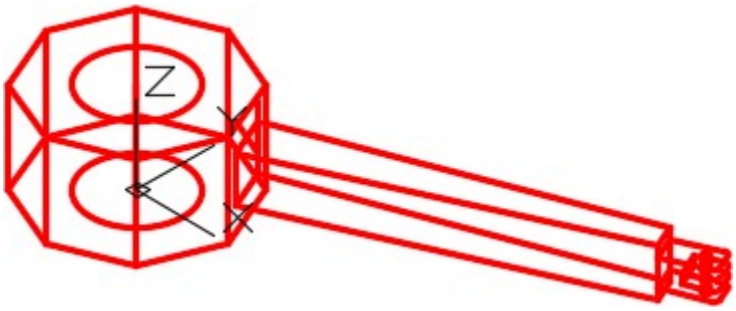


Figure Step 1C

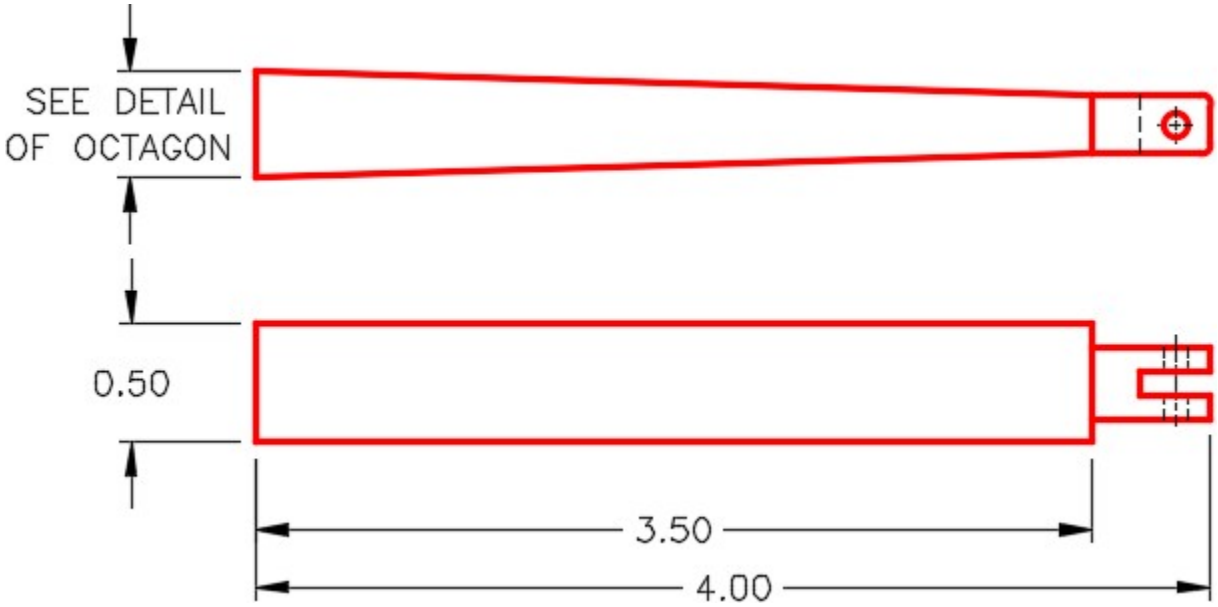


Figure Step 1D Detail B – Dimensioned Multiview Drawing of Arm

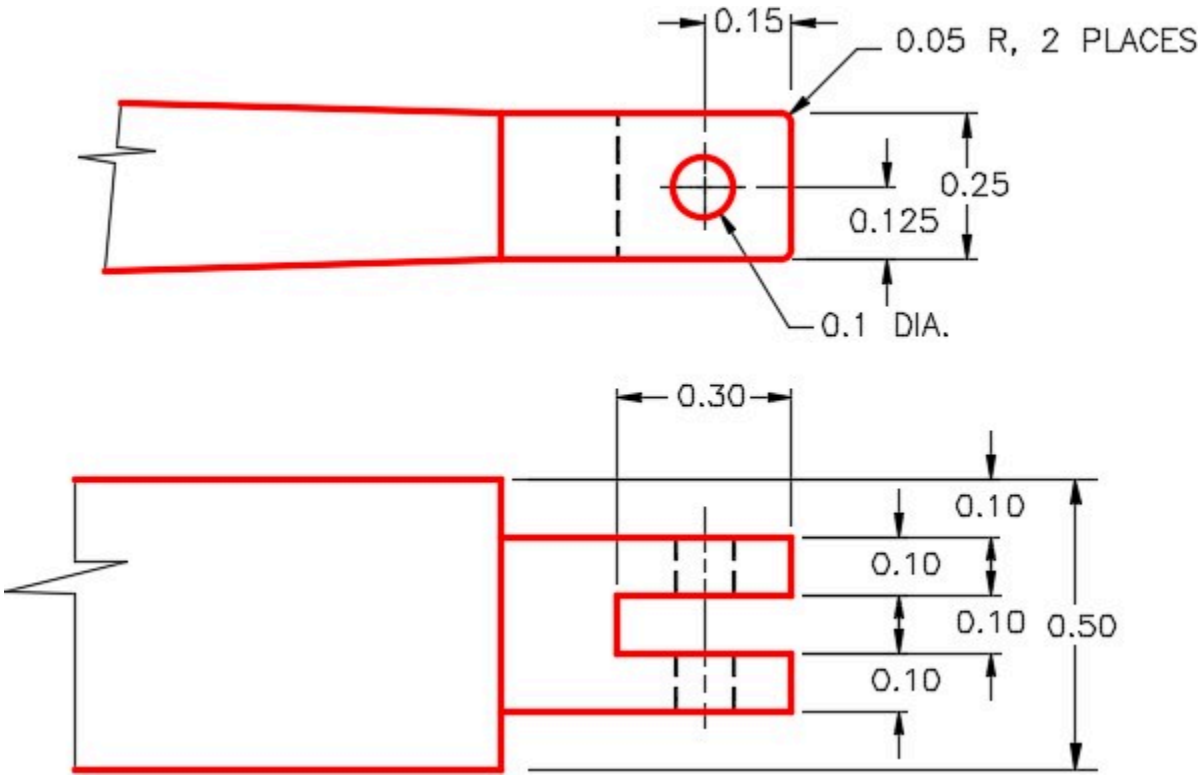


Figure Step 1E Detail A – Dimensioned Multiview Drawing of Arm

Step 2

Array the arm as shown in the figure. (Figure Step 2)

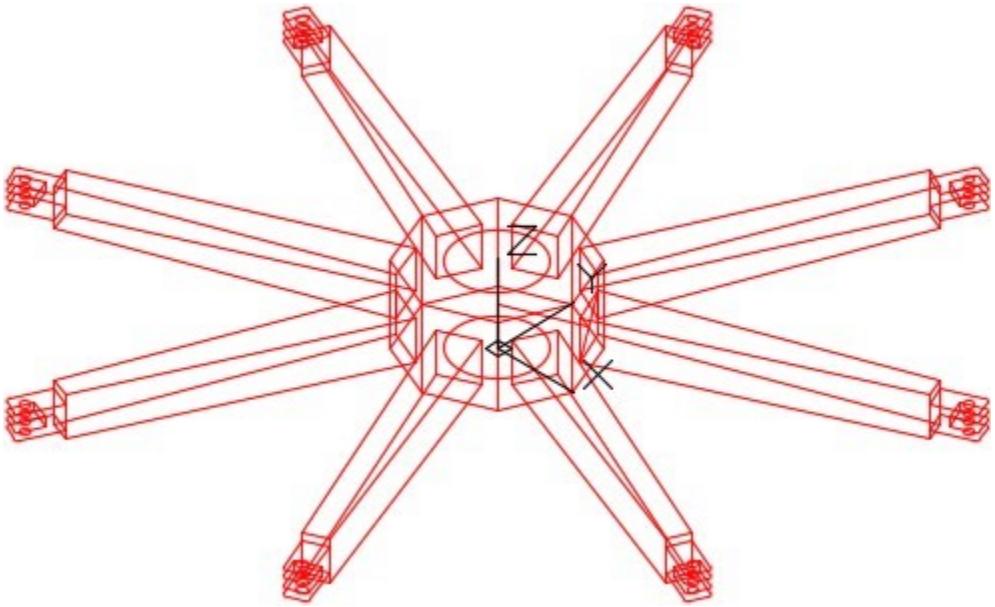


Figure Step 2

Step 3

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

Step 4

Change the current UCS to World and check the model with the key.

Step 5

Save and close the drawing.

Lab Exercise 9-3

Time allowed: 50 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 09-3	3D Layout English	Inches

Step 1

On layer: Model, draw a wireframe model as shown in the figure. See the multiview drawing in Figure 1 Step B. (Figure Step 1A and 1B)

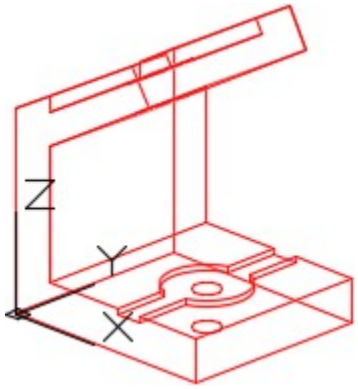


Figure Step 1A

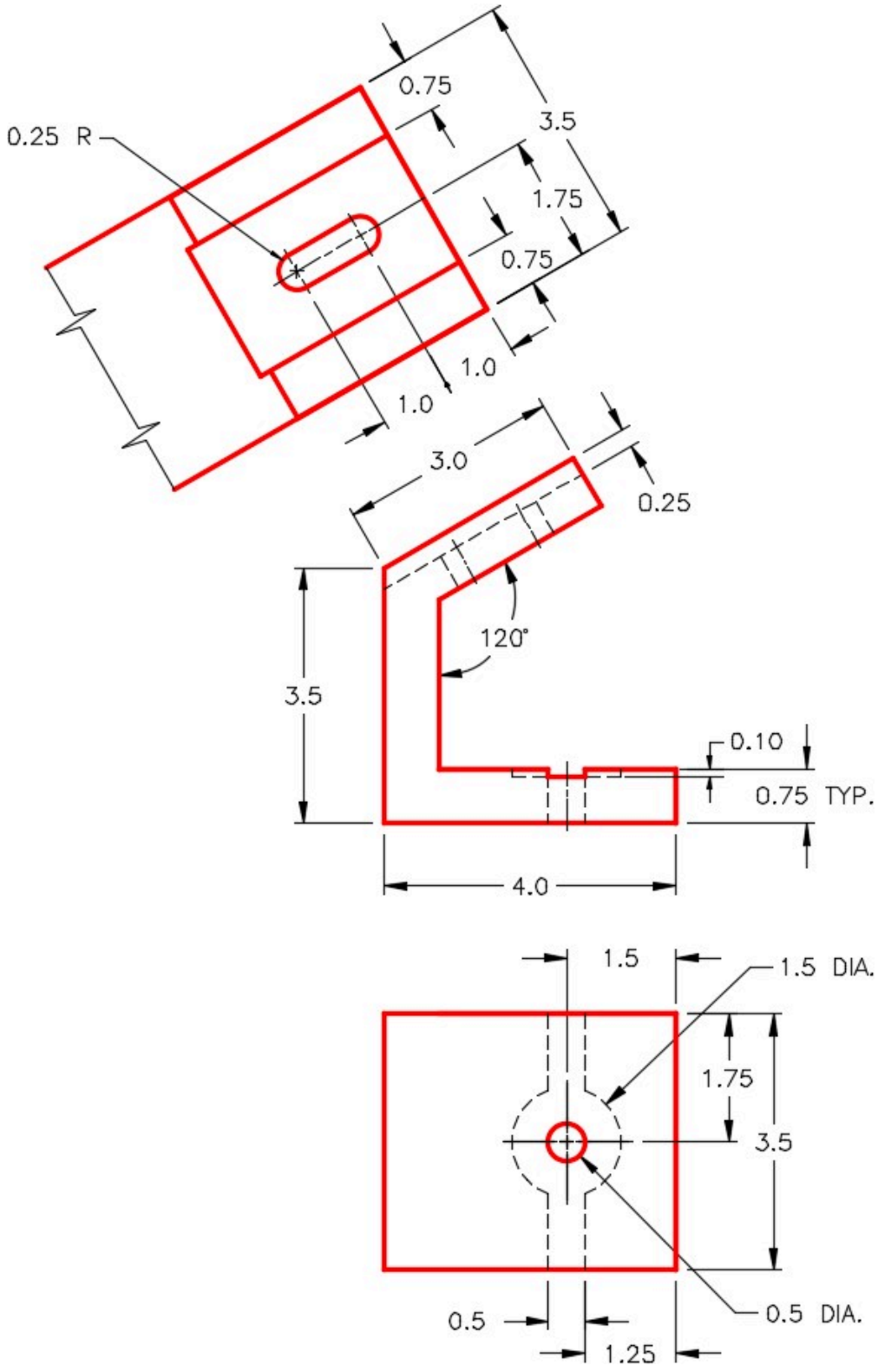


Figure Step 1B
Dimensioned Multiview Drawing

Step 2

Mirror the wireframe drawn in Step 1. Erase the extra lines and clean up the model to create one model. (Figure Step 2)

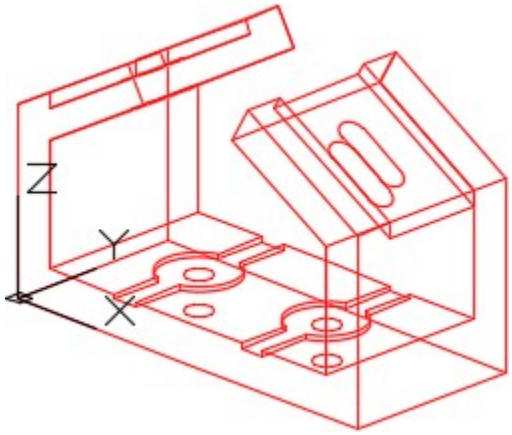


Figure Step 1A

Step 3

Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Inches. Change the current UCS to World and check the model with the key.

Step 4

Turn layer: Key off.

Step 5

Note the location of the UCS and rotate the model. (Figure Step 5)

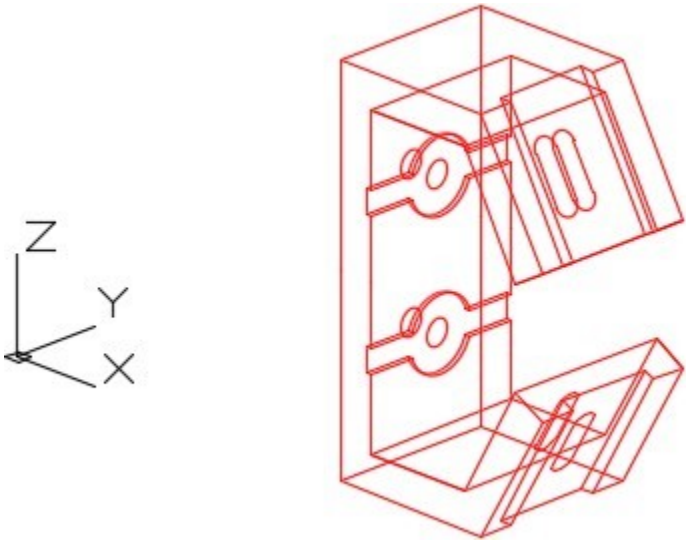


Figure Step 5

Step 6

Note the location of the UCS and rotate the model. (Figure Step 6)

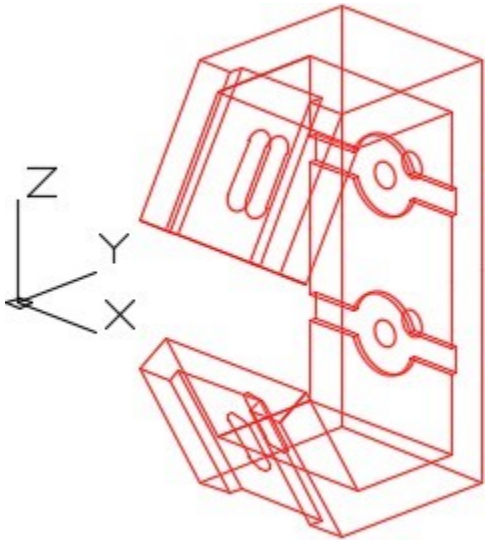


Figure Step 6

Step 7

Save and close the drawing.

Module 10: Viewing 3D Models – Part 2

Learning Outcomes

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

1. Apply the VSCURRENT command to display 3D surface or solid models in the five predefined visual styles.
2. Apply the more advanced options of the 3DORBIT command.
3. Use the system variable VPCONTROL to enable the display the Viewport Controls menu and describe how this menu is used to set the current view and orientation of the model.
4. Enable the display of the ViewCube and describe how it used to change the display and orientation of the model.

Visual Styles

In the first nine modules, drawing wireframe models was taught. A wireframe model is a real-world 3D object represented by lines, circles, arcs, and/or plines located along the edges of the model. For that reason, you can see right through the model and see objects that would not be visible if the model was surfaced or solid. Think of it as the skeleton of a model.

In Modules 12 to 30, constructing surface and solid models is taught. AutoCAD software comes complete with five predefined visual styles. In this module, viewing surface and solid models using these five styles is covered.

The VSCURRENT command is used to set the current visual style that the model will be displayed. See Figure 10-1. You can change the current visual style as required.

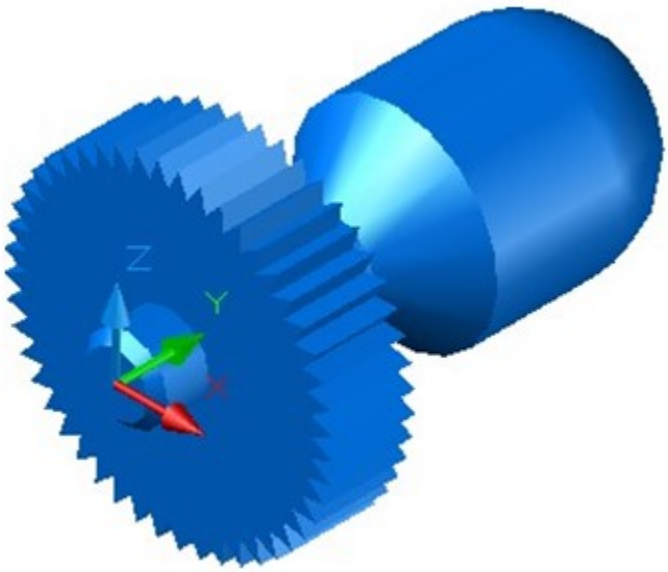


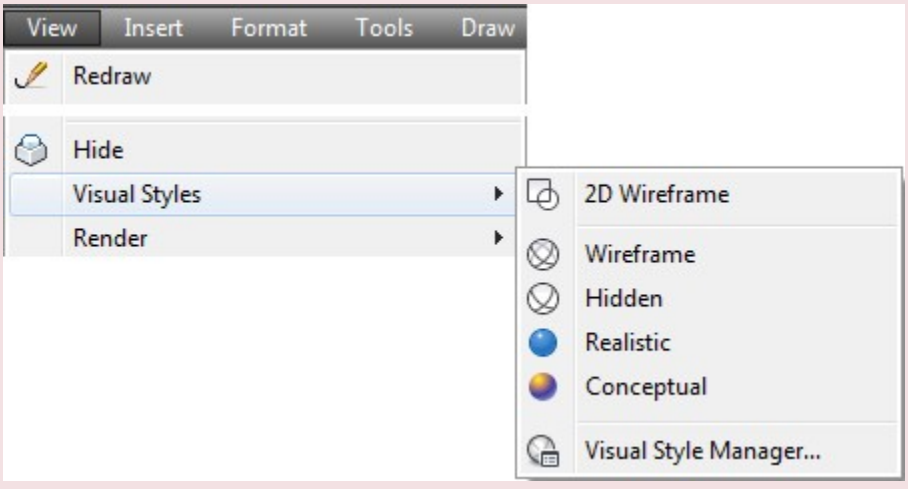
Figure 10-1
Realistic Visual Style of a Solid Model

AutoCAD Command: VSCURRENT

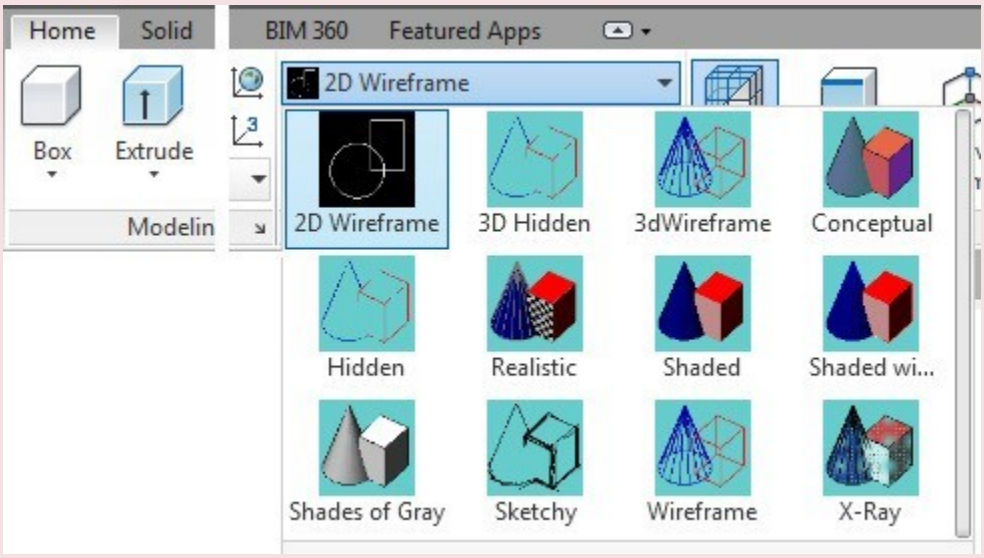
The VSCURRENT command is used to set the current visual style (shade mode) to be applied to a surface or solid model.

Shortcut: **VS**

Visual Style Toolbar



View Pull-down



Home Ribbon

WORK ALONG: Setting the Current Visual Style of a Model

Step 1

Using the NEW command, start a new drawing using the template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 10-1.

Step 3

Enter the UNITS command. In the Units dialogue box, set the Insertion Units to Inches. With the current UCS set in World, use the INSERT command to insert the block:

AutoCAD 3D Workalong 10-1 at the coordinates 0,0,0. Explode the block. Be careful to only explode it once.

Step 4

Set the current view to SE Isometric. Your model should now appear as shown in the figure. (Figure Step 4)

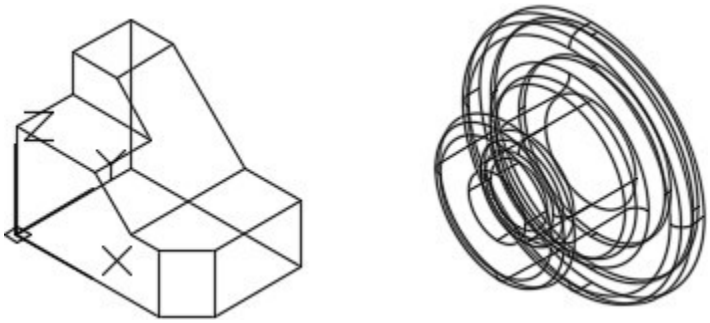


Figure Step 4

Step 5

Change the layer of both solid models to layer: Model.

Step 6

You should now have two solid models. To check this, open the Properties window and select one of the solid models. When selected, the object type in the top box of the Properties windows should indicate that the object is a 3D Solid. If the object type is anything else, exit and delete the drawing. Start this workalong again from Step 1. Be careful when using the EXPLODE command to explode the block only once. (Figure Step 6)

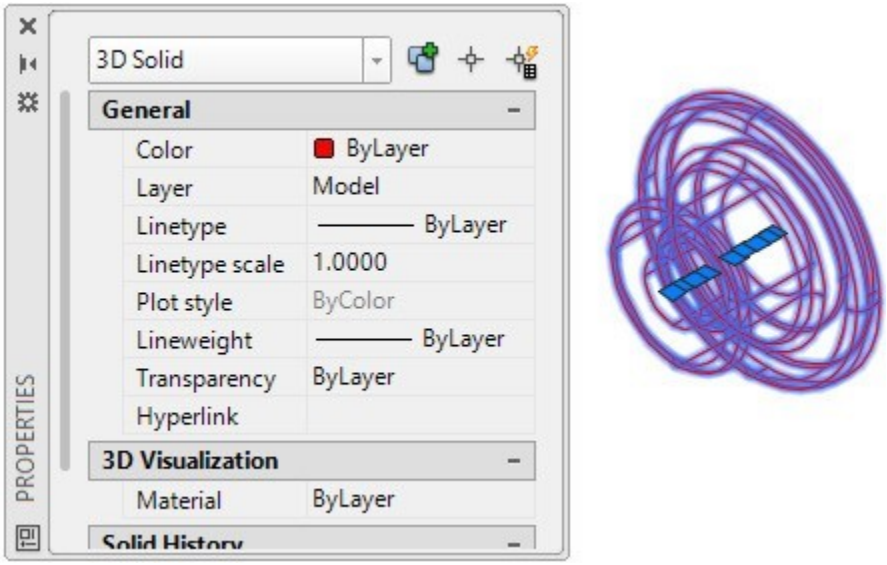


Figure Step 6

Step 7

If you are using Toolbars menus, enable the Visual Styles toolbar. If you are using the Ribbon Menus, click the Home tab. Enable the 2D Wireframe visual style icon. You can tell when the current visual style is 2D Wireframe by the appearance of the UCS icon. (Figure Step 7)

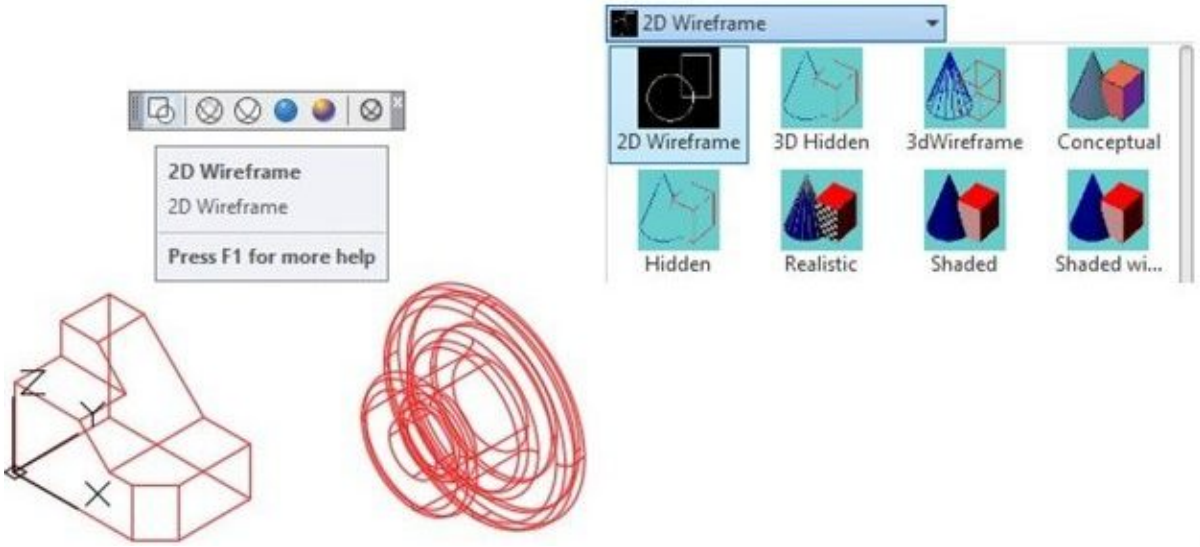


Figure Step 7

Step 8

Click the 3D Wireframe visual style icon. You can tell when the current visual style is 3D Wireframe by the appearance of the UCS icon. (Figure Step 8)

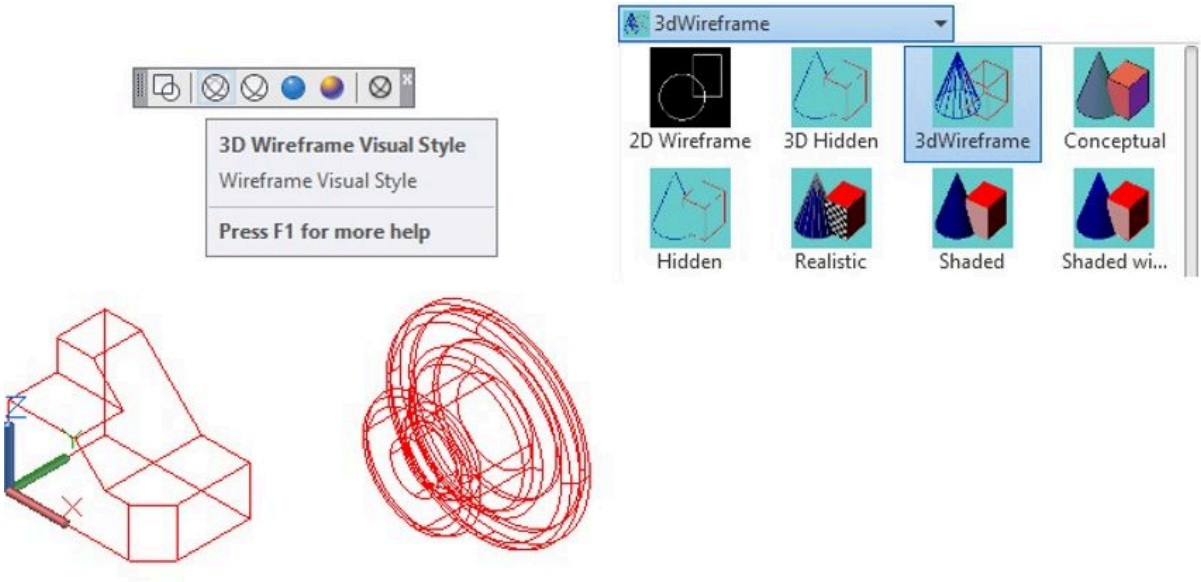


Figure Step 8

AUTHOR'S COMMENTS: Except for the appearance of the UCS icon, the 2D Wireframe and the 3D Wireframe visual style appear similar. There are important differences but they have nothing to do with their appearance. These differences are taught in AutoCAD 3D Advanced book.

Step 9

Click the 3D Hidden visual style icon. Note how when the current visual style is 3D Hidden, the model appears solid and only displays the visible objects. (Figure Step 9)

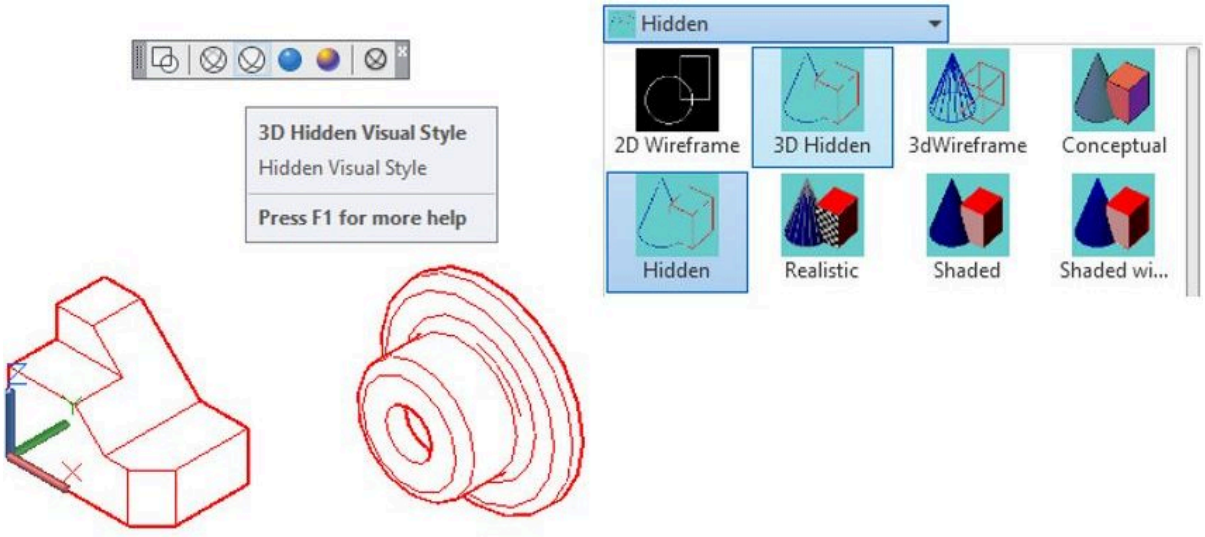


Figure Step 9

Step 10

Change the layer of both solid models to layer: Solid 1.

Step 11

Click the Realistic visual style icon and note how when the current visual style is Realistic, the appearance of the model is the most basic realistic view of a shaded solid or surfaced model. (Figure Step 11)

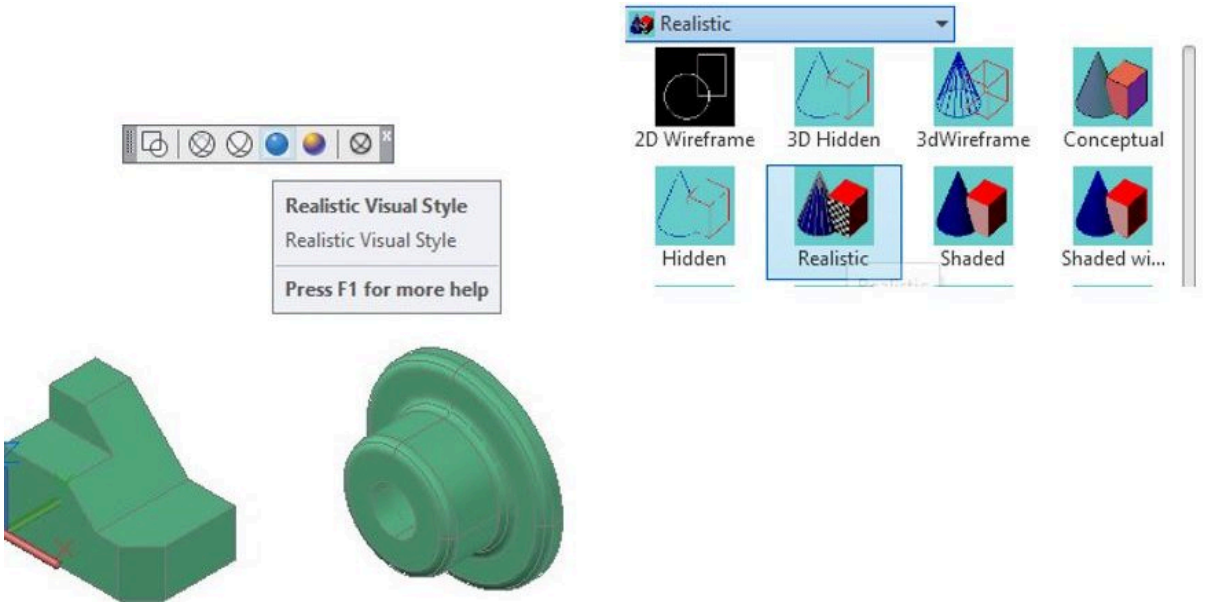


Figure Step 11

Step 12

Click the Conceptual visual style icon and note how when the current visual style is Conceptual, the appearance of the solid model is more of an artistic look. (Figure Step 12)

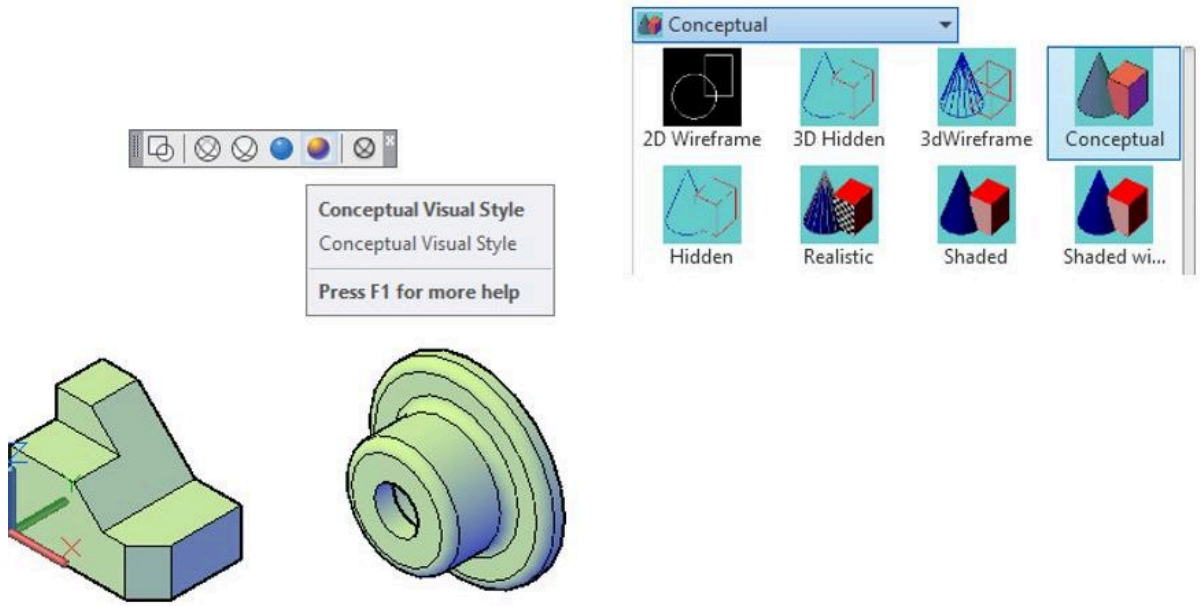


Figure Step 12

AUTHOR'S COMMENTS: Some models display better in the Conceptual visual style.

Step 13

Delete the cylindrical solid model leaving only one solid model. Change the current visual style to Realistic and your model should now appear as shown in the figure. (Figure Step 13)

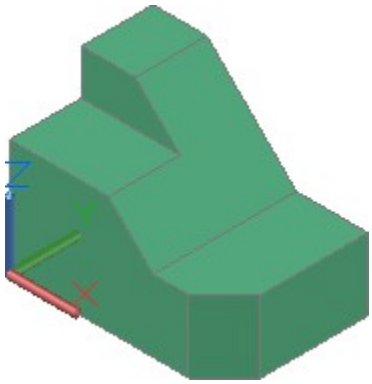
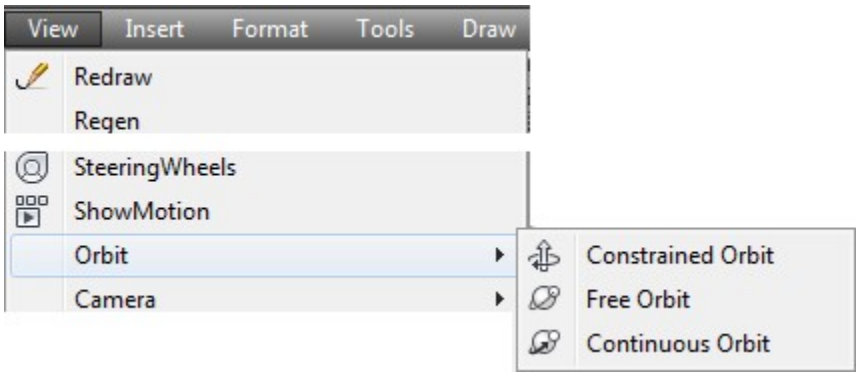


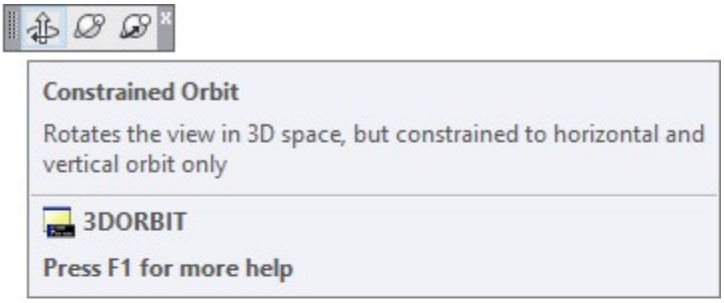
Figure Step 13

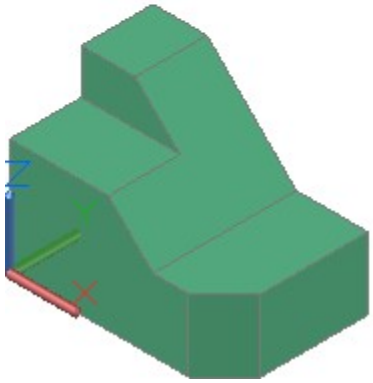
Step 14

If you are using Toolbar menus, enable the display of the Orbit toolbar. If you are using Ribbon menus, use the View-Orbit pull-down menu. Click the Constrained Orbit icon. (Figure Step 14A and 14B).



View Pull-down
Figure Step 14A





Orbit Toolbar
Figure Step 14B

Step 15

When constrained orbit command is active, it displays the icon as shown in the figure. (Figure Step 15)

Figure Step 15

Step 16

Orbit the model and note how this command affects the position of the model. Press Esc to exit the command.

Step 17

Click the Free Orbit icon and orbit the model. Note how this command affects the position of the model. Press Esc to exit.

Step 18

Click the Continuous Orbit icon and use it to orbit the model. Read the Author's Comments. (Figure Step 18)

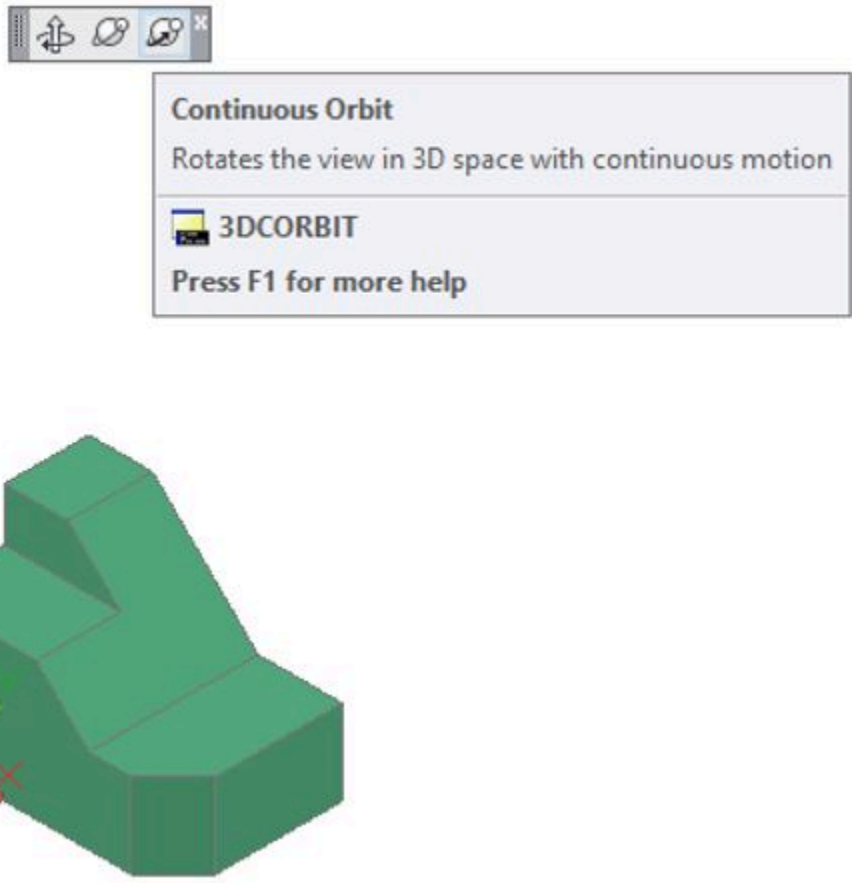


Figure Step 18

AUTHOR'S COMMENTS: The continuous 3D orbit feature is an interesting and fun feature in AutoCAD. After you enter this command, the Continuous 3D Orbit icon, Figure 10-2, will appear on the Graphic window. Hold down the mouse's pick button, spin the model and release the button. The model will start to rotate continuously until you stop it by clicking in the Graphic window or by pressing the Esc key. The faster you spin it, the faster the model rotates.

Figure 10-2
Continuous 3D Icon

Step 19

Set the current view to SE Isometric.

Step 20

Save and close the drawing.

AUTHOR'S COMMENTS: There are several additional predefined visual styles available in AutoCAD and you can also create, name, and save your own visual styles. This is taught in the AutoCAD 3D Advanced book.

MUST KNOW: Surface and solid models can be viewed in many visual styles. Models can be viewed as a wireframe, a hidden, or a solid. In a hidden style, the model will appear as it would in real life. It obstructs the lines and curves that are behind the visible surfaces. In the solid style, the model is shaded. A shaded model will be shaded with the same color as the model.

The VSCURRENT command controls which visual style is used to display the model. It will remain displayed in the that visual style until the current visual style is changed. Even if the drawing is closed and opened again, the visual style of the model remains as it was last set.

A model must be constructed as a surface or a solid to be viewed hidden or shaded. A wireframe model will only display in the 2D Wireframe or 3D Wireframe visual style. Constructing surface and solid models is taught in Modules 11 to 30.

AutoCAD System Variable: VPCONTROL

Enables or disables the display of the Viewport Controls menu.

Shortcut: **none**

WORK ALONG: Using the Viewport Controls Menu

Step 1

Open the drawing: AutoCAD 3D Workalong 10-1. (Figure Step 1)

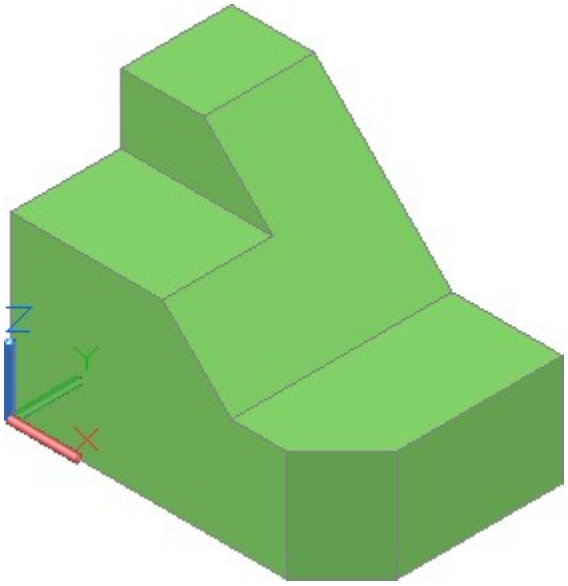


Figure Step 1

Step 2

Enter the system variable VPCONTROL and set it to ON, as shown below.

Command: **VPCONTROL**

Enter new value for VPCONTROL <OFF>: **ON**

Command:

Step 3

Step 2 will enable the display of the Viewport Controls menu located in the top left corner of the Graphic window. (Figure Step 3A, 3B, and 3C)

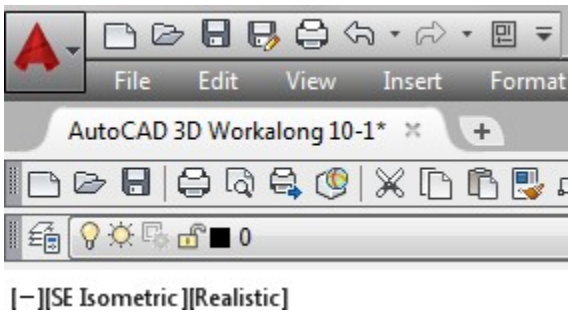


Figure Step 3A
Toolbar Menu

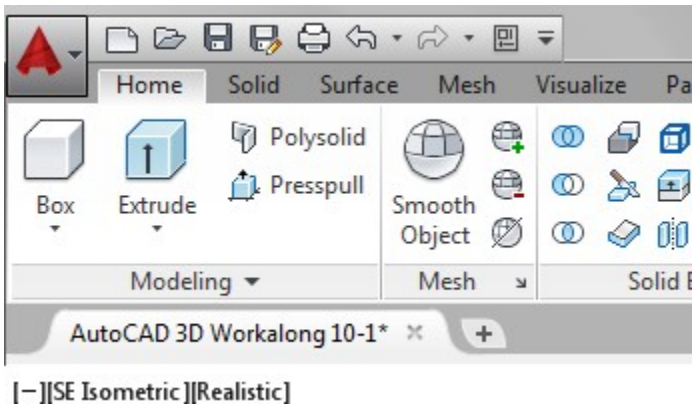


Figure Step 3B
Ribbon Menu

Figure Step 3C

Step 4

Click the small dash on the left side of the menu. This will pull down a list of menu items. Ensure that all items are disabled, as shown in the figure. (Figure Step 4)

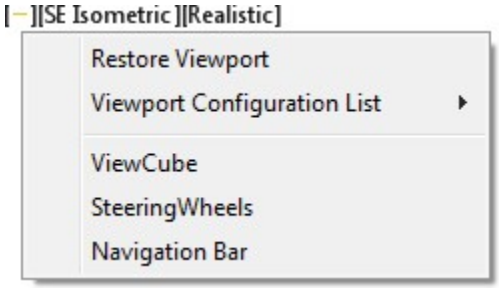


Figure Step 4

Step 5

Click the View menu. It should read SE Isometric. This will pull down the list of the predefined views as shown in the figure. (Figure Step 5)

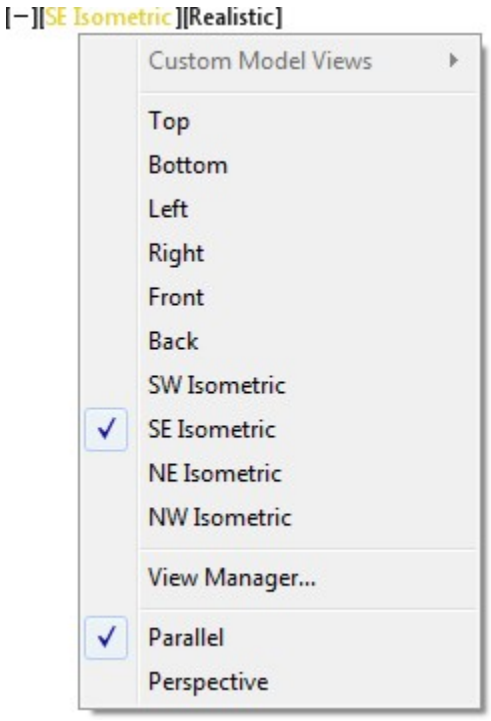


Figure Step 5

AUTHOR'S COMMENTS: The Parallel view type should be always enabled while working in the AutoCAD 3D book. The AutoCAD 3D Advanced book teaches Perspective views.

Step 6

Enable Top to change the current view of the model to the Top. (Figure Step 6A and 6B)

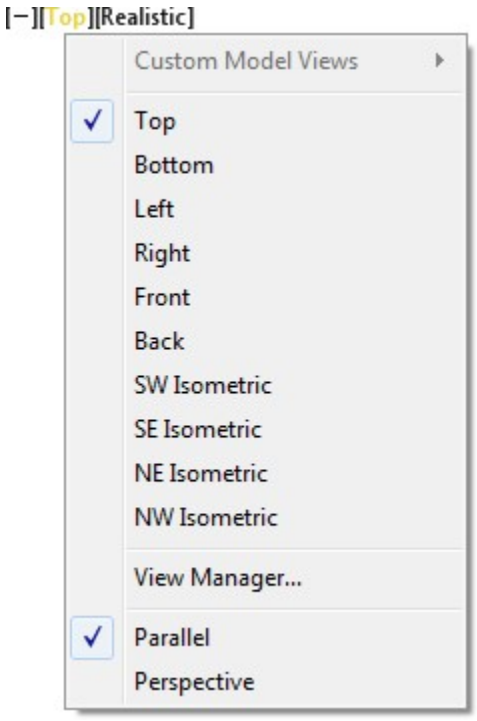


Figure Step 6A

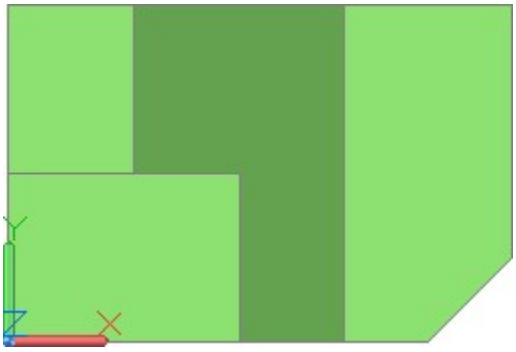


Figure Step 6B

Step 7

Pull down the View menu and set the current view to SE Isometric. (Figure Step 7A and 7B)

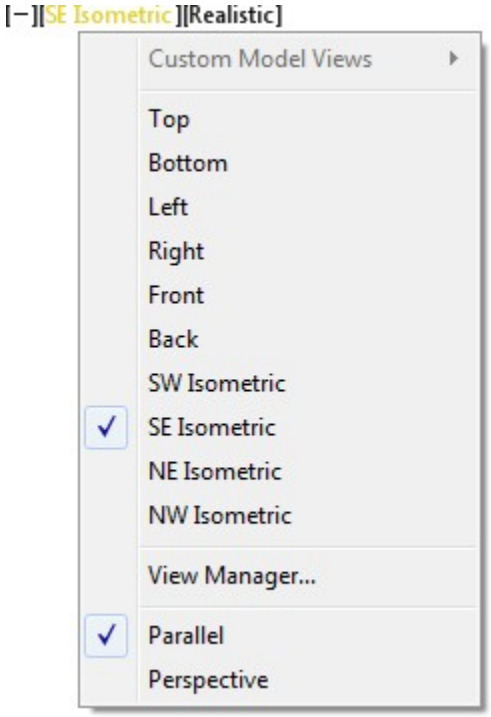


Figure Step 7A

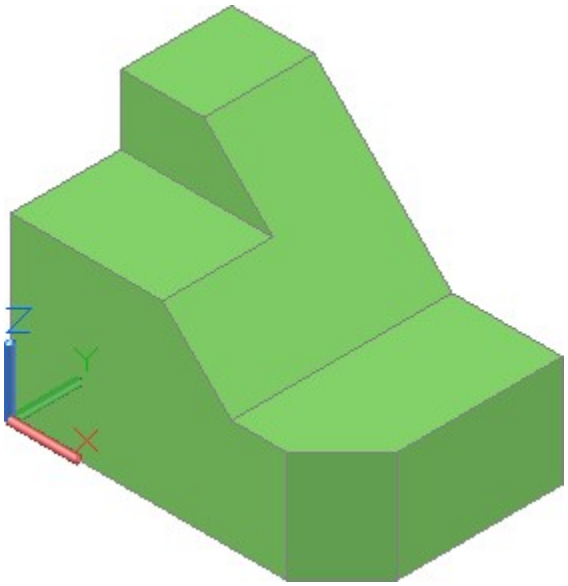


Figure Step 7B

Step 8

Click the Visual Style menu and in the pull down list, set the current visual style to 2D Wireframe. (Figure Step 8A and 8B)

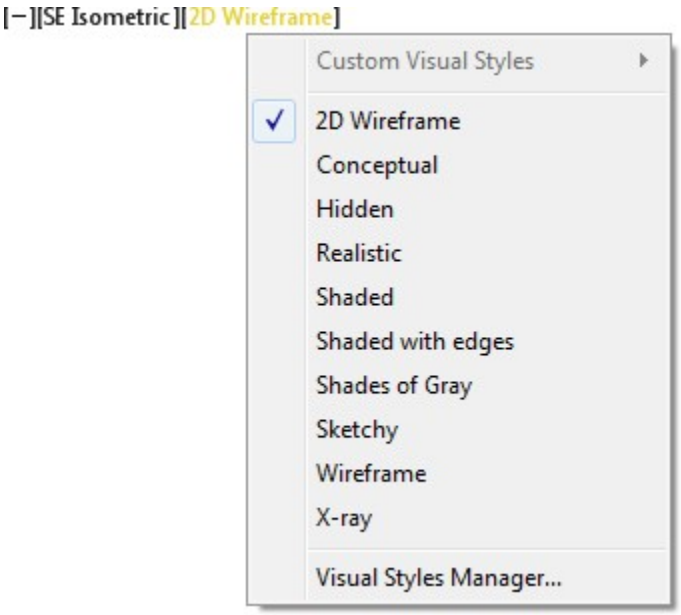


Figure Step 8A

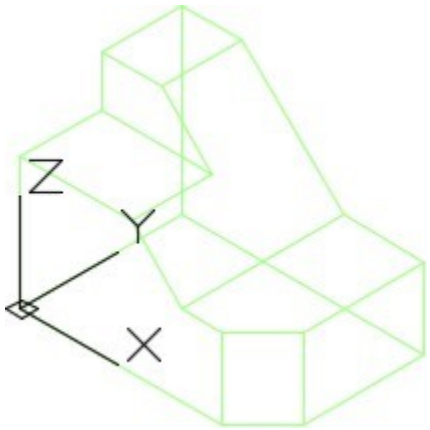


Figure Step 8B

Step 9

Click the Visual Style menu and in the pull down list, set the current visual style to Conceptual. (Figure Step 9A and 9B)

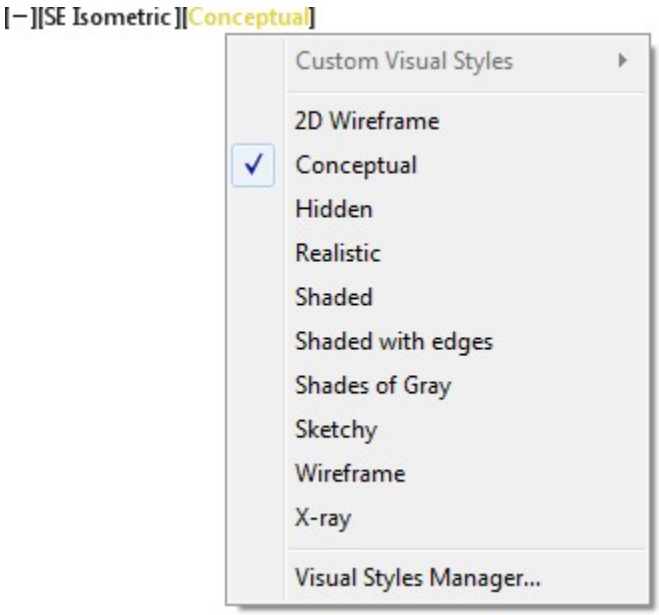


Figure Step 9A

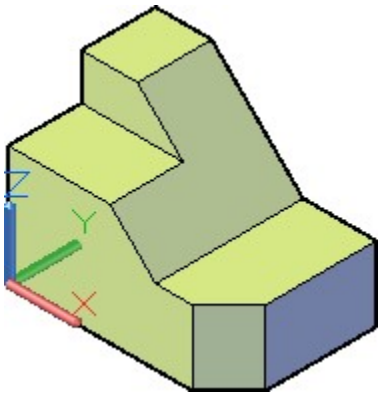


Figure Step 9B

Step 10

Click the small dash on the left side of the menu to pull down the menu. Enable Navigation Bar. The Navigation Bar will display on the right side of the Graphic window as shown in the figure. (Figure Step 10A and 10B)

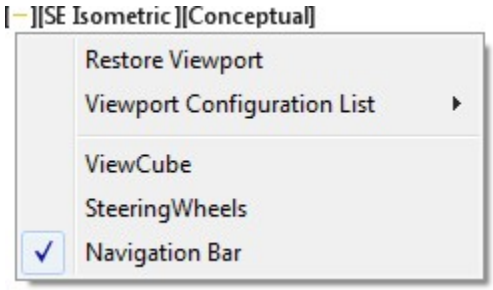


Figure Step 10A

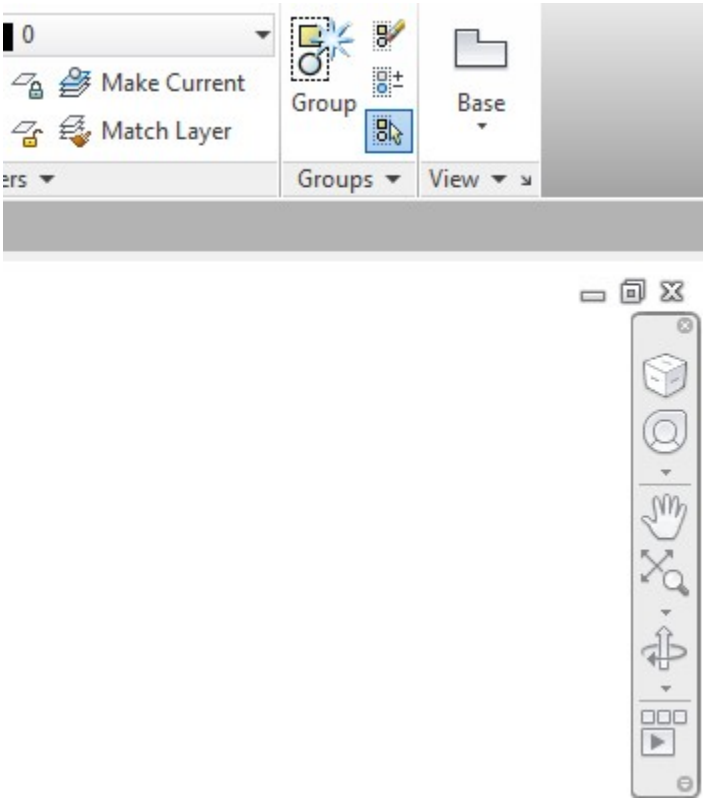


Figure Step 10B

Step 11

Click the small arrow located at bottom right corner of the Navigation Bar to pull down its menu. Ensure that all six item are enabled as shown in the figure. (Figure Step 11)

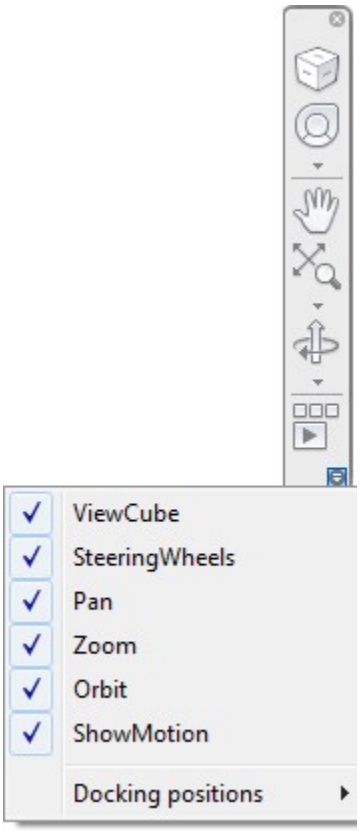


Figure Step 11

Step 12

Click the small arrow again located at the bottom of the Navigation Bar to pull down its menu. Click the Docking positions item to display the flyout menu. Ensure the enabled items match the figure. (Figure Step 12)

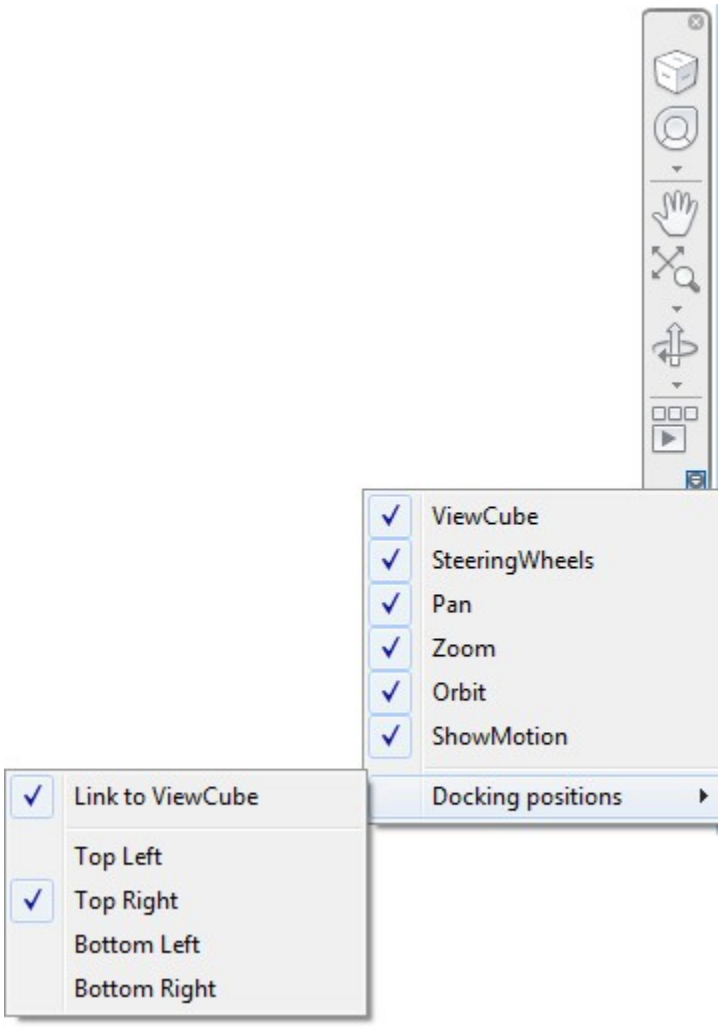


Figure Step 12

Step 13

Open the Options dialogue box using the OPTIONS command. Enable the All other visual styles item under the Display the ViewCube, as shown in the figures. Click OK to close the dialogue box. (Figure Step 13A and 13B)

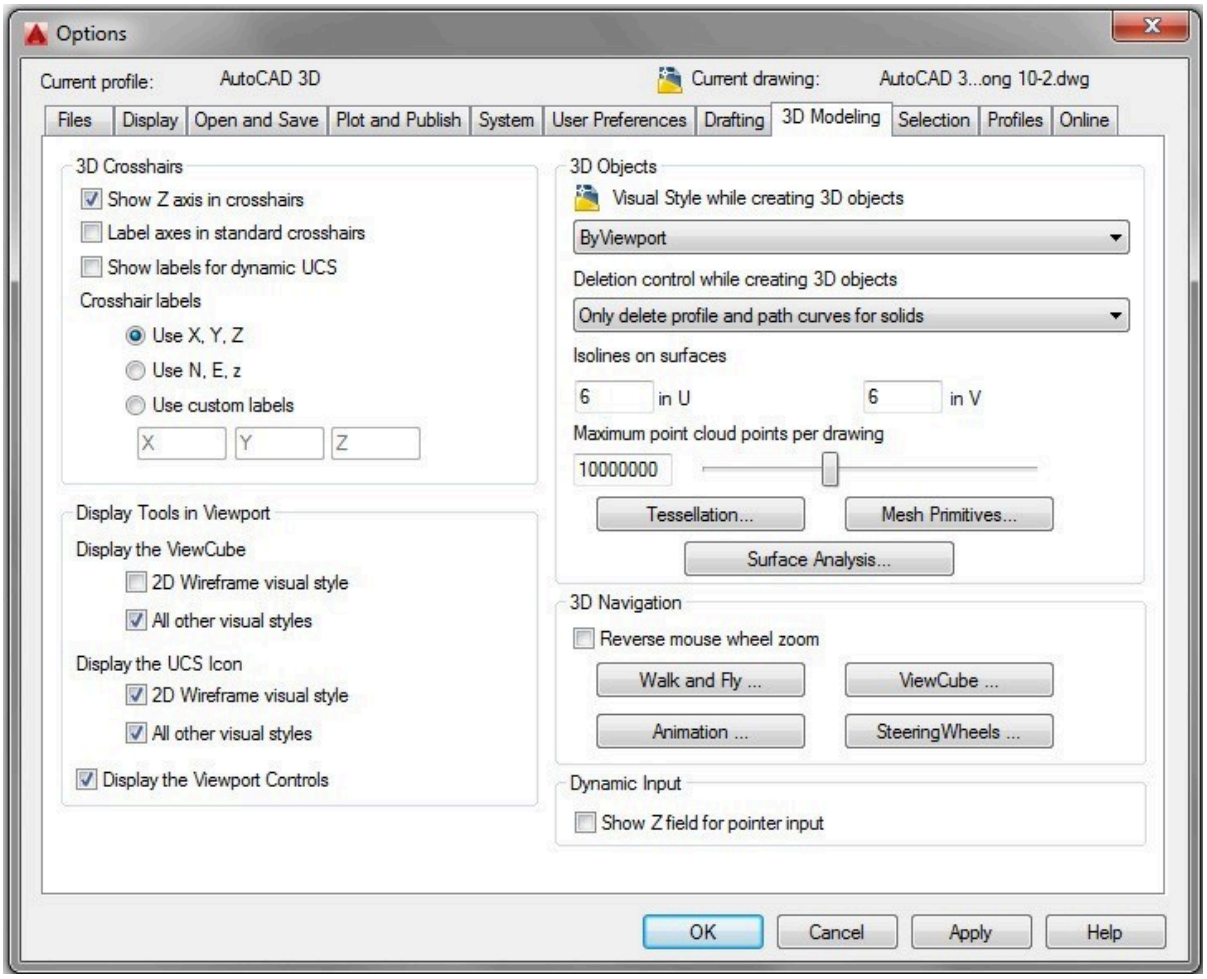


Figure Step 13A

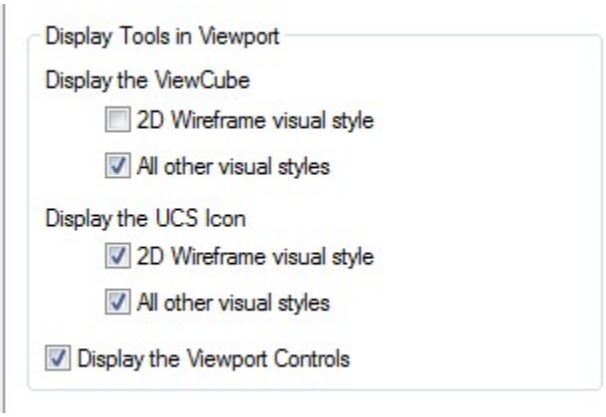


Figure Step 13B

Step 14

Click the small dash on the left side of the menu to pull down the menu. Enable ViewCube. The ViewCube will display on the right side of the Graphic window, as shown in the figure. (Figure Step 14A and 14B)

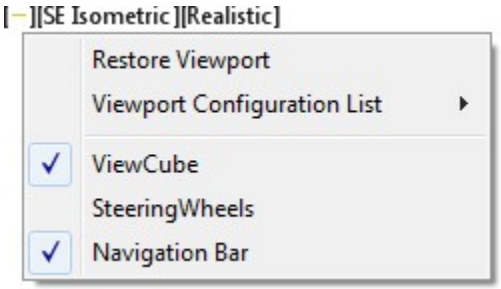


Figure Step 14A

Figure Step 14B

Step 15

Save and close the drawing.

ViewCube is a valuable AutoCAD 3D tool that provides visual feedback of the current orientation of the model and a very quick and effective method of changing your viewpoint of the model. See Figure 10-3.

ViewCube can be used to restore and define the Home view of a model, switch between views of the model, projection modes and change its interactive behavior and appearance.

Figure 10-3
ViewCube

AutoCAD Command: NAVVCUBE

The NAVVCUBE command controls the visibility and display properties of the ViewCube tool.

Shortcut: **none**

WORK ALONG: Using ViewCube

Step 1

Open the drawing: AutoCAD 3D Workalong 10-1. (Figure Step 1)

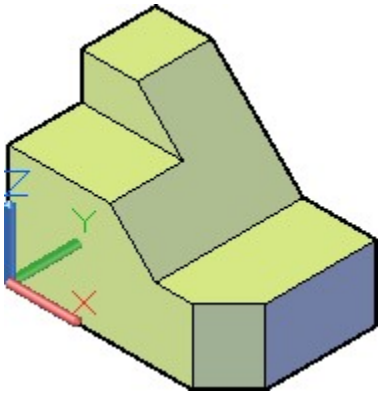


Figure Step 1

Step 2

Click the small dash on the left side of the Viewport Controls menu. Ensure that ViewCube and Navigation Bar are enabled as shown in the figure. (Figure Step 2)

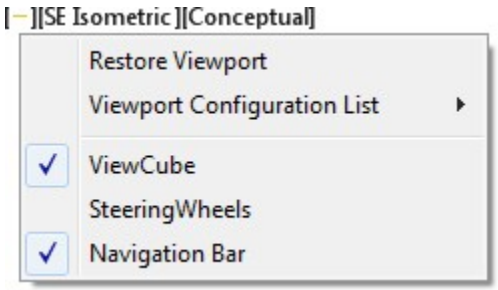


Figure Step 2

Step 3

The Graphic window should appear similar to the figure. (Figure Step 3)

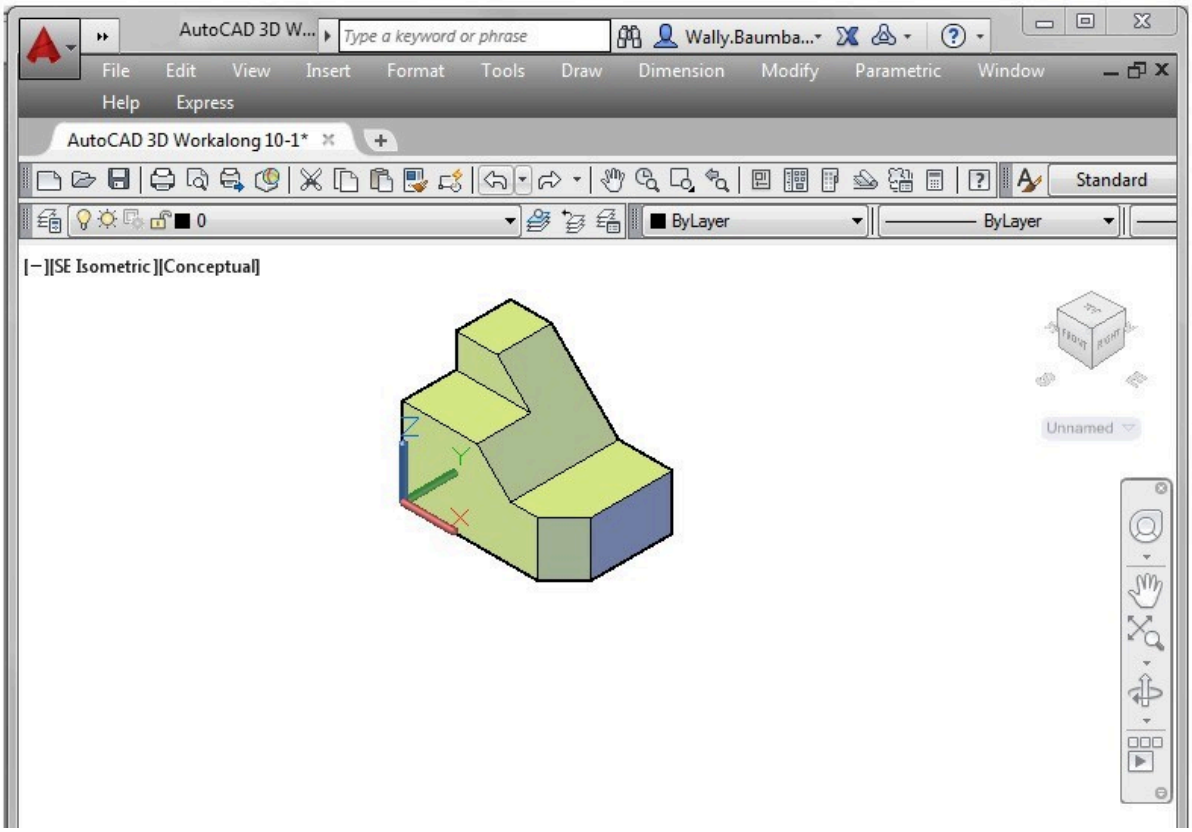


Figure Step 3

AUTHOR'S COMMENTS: ViewCube will not display when the current visual style is set to 2D Wireframe.

Step 4

Ensure that the current view is set to SE Isometric.

Step 5

Click the arrow icon located on the lower right side of ViewCube. (Figure Step 5)

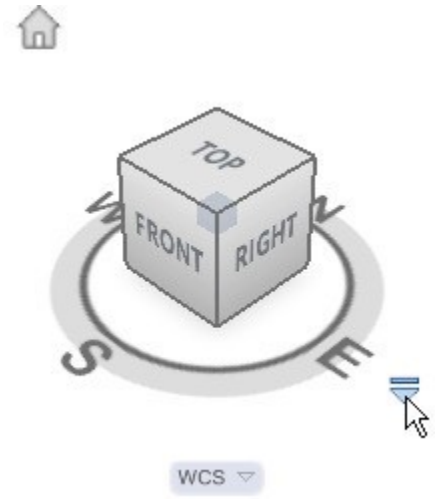


Figure Step 5

AUTHOR'S COMMENTS: If the arrow is not currently displayed, move the cursor onto the ViewCube.

Step 6

In the pull-down menu, click Set Current View as Home. (Figure Step 6)

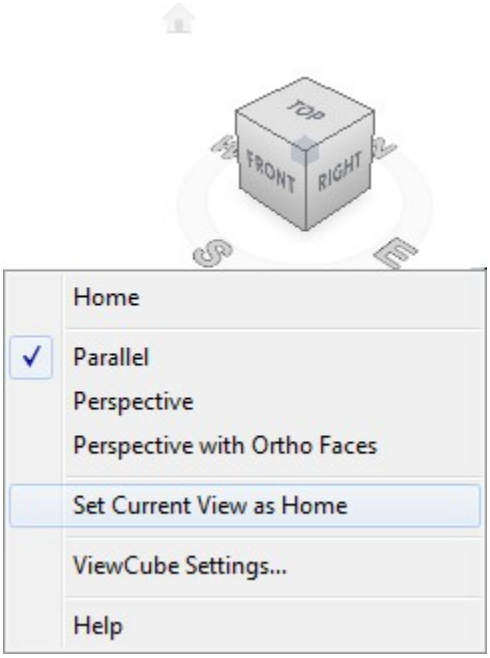


Figure Step 6

AUTHOR'S COMMENTS: Setting the Home view allows you to easily return to a known view or orientation of the model.

Step 7

Click the arrow again and click ViewCube Settings (Figure Step 7)

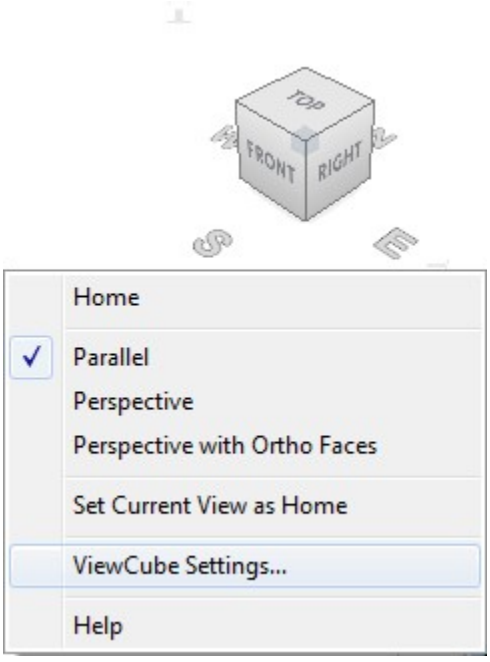


Figure Step 7

Step 8

Set the ViewCube Settings dialogue box as shown in the figure. (Figure Step 8)

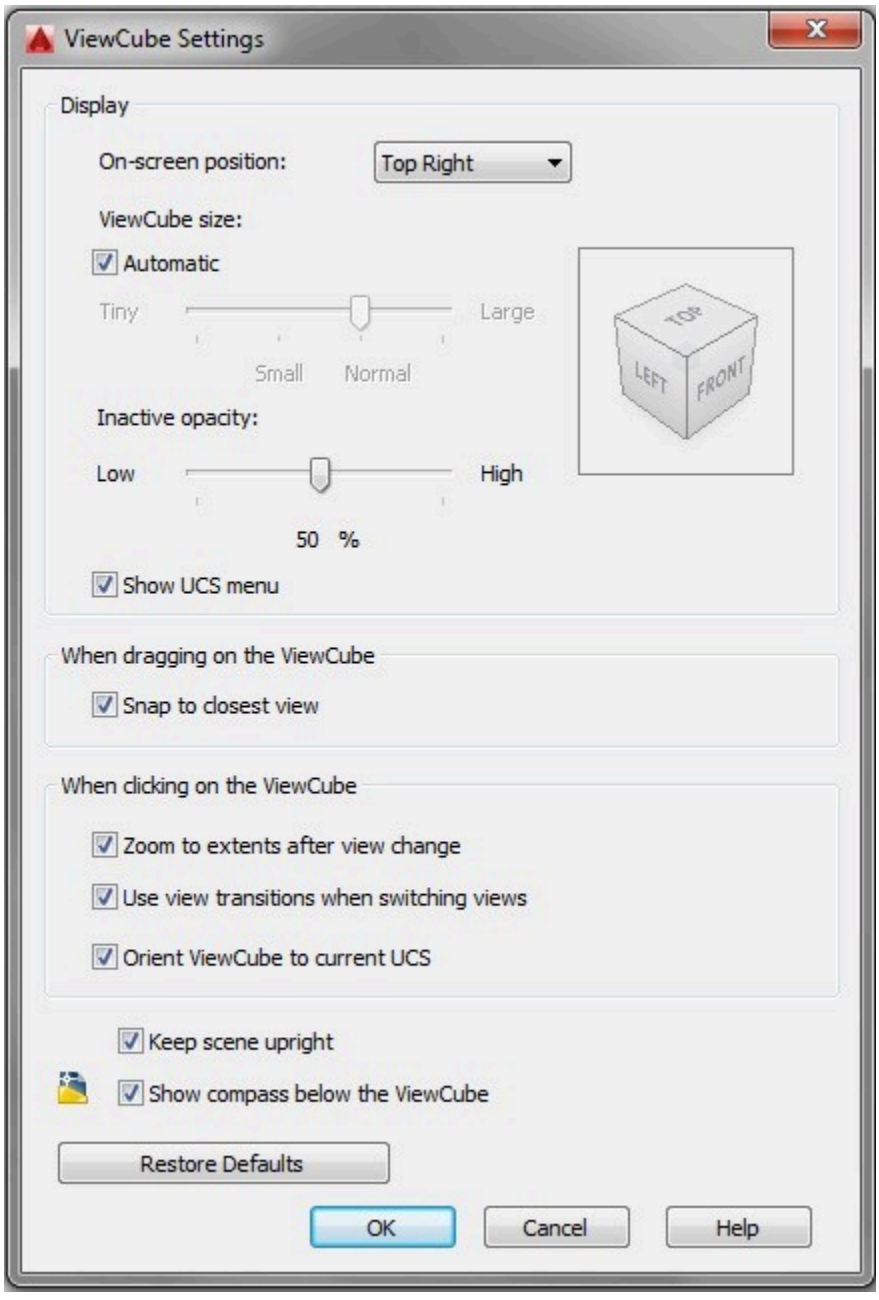


Figure Step 8

Step 9

Move the cursor onto the top of ViewCube. When it turns blue, click it. The solid model's current view will change to the Top view. (Figure Step 9A and 9B)

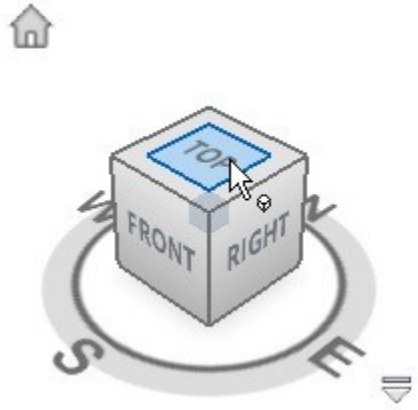


Figure Step 9A

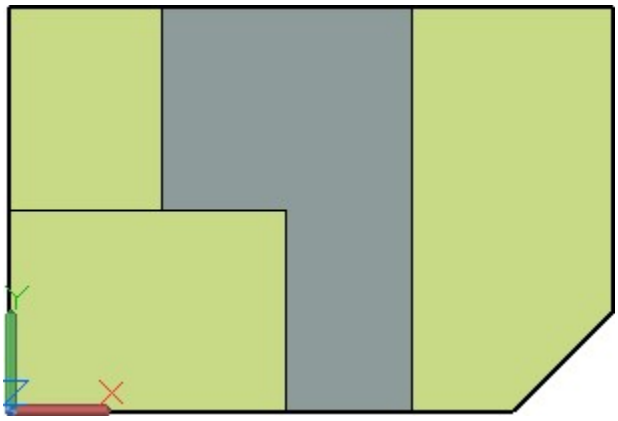


Figure Step 9B

Step 10

Click the Home icon on ViewCube. It is the small house in the upper left corner. The model will now display the home view. For this model, it is SE Isometric. (Figure Step 10A and 10B)

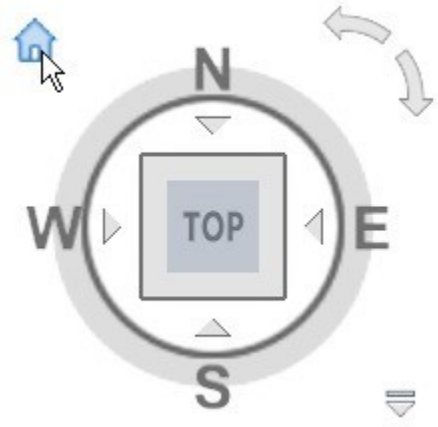


Figure Step 10A

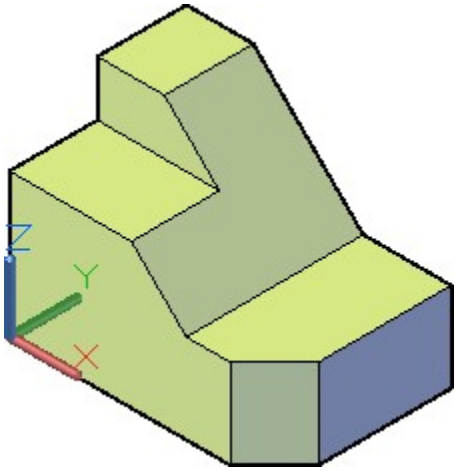


Figure Step 10B

Step 11

Using what you just learned, change the current view to Front. (Figure Step 11)

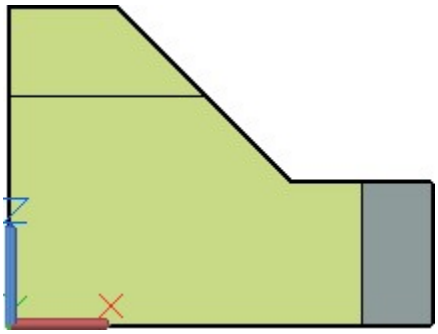


Figure Step 11

Step 12

Change the current view to Right. (Figure Step 12)

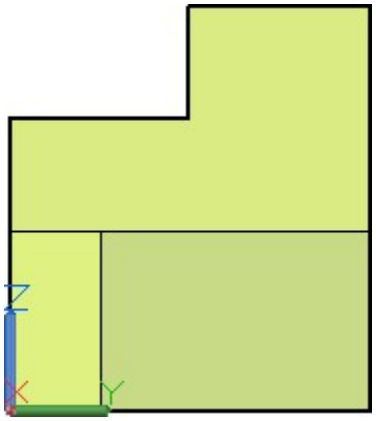


Figure Step 12

Step 13

Change the current view to the Home view. (Figure Step 13)

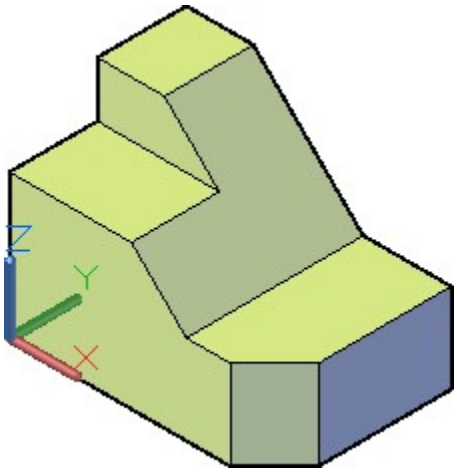
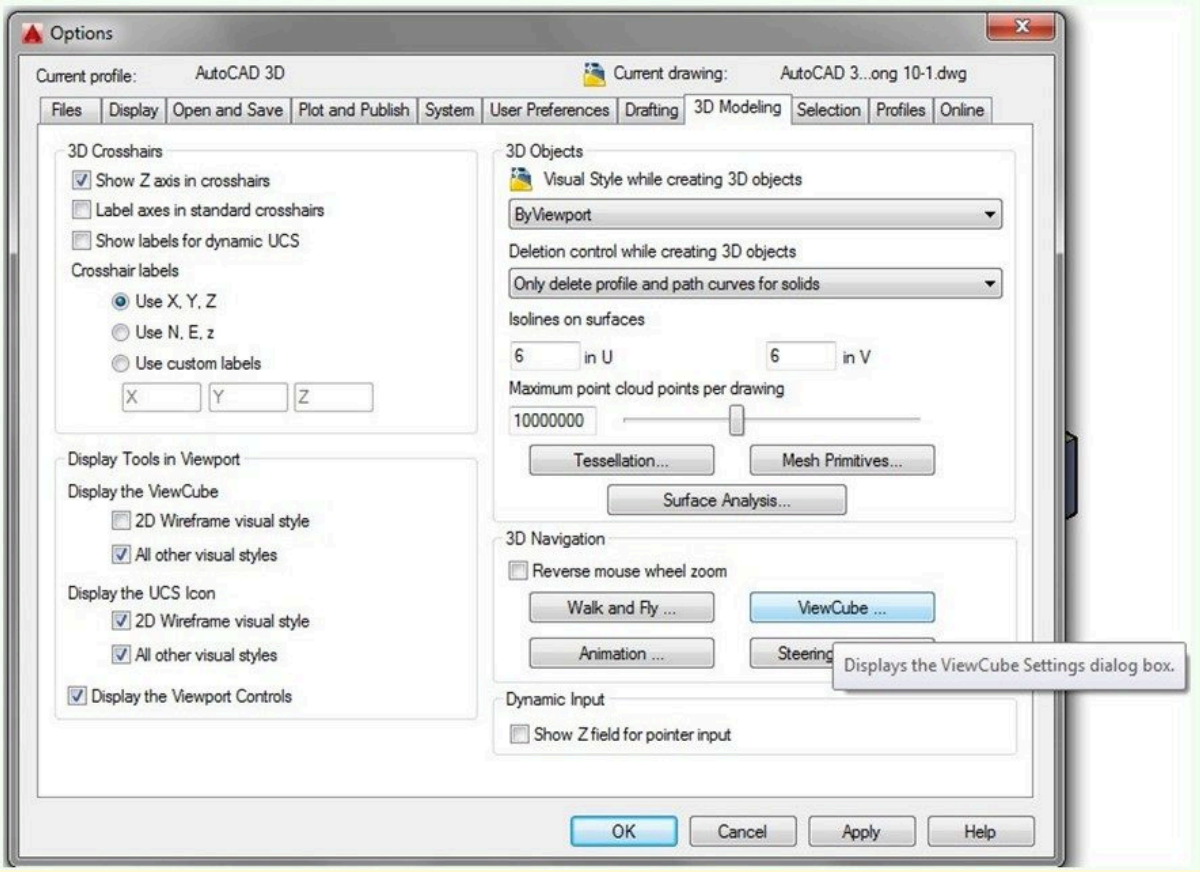


Figure Step 13

Step 14

Save and close the drawing.

USER TIP: ViewCube's settings can also be adjusted in the Options dialogue box. See figure below.



Key Principles

Key Principles in Module 10

1. Surface and solid models can be viewed in many visual Models can be viewed as a wireframe, a hidden, or a solid. A shaded model will be shaded with the same color as the model.
2. The Viewport Controls menu is used to easily change the models orientation and visual style.
3. ViewCube is a valuable AutoCAD 3D tool that provides visual feedback of the current orientation of the model and a very quick and effective method of changing your viewpoint of the model.
4. ViewCube can be used to restore and define the Home view of a model, switch between views of the model, projection modes, and change its interactive behavior and appearance.

Module 11: Competency Test No. 2 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 3D 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. Where would the UCS plane be located if you used the Object UCS icon to select the current UCS?
    A. It would lie perpendicular to the plane the object selected was on.
    B. It would be on the WCS.
    C. It would lie parallel with the current view.
    D. It would lie on the plane that the selected object was on.
    E. It would lie perpendicular to the WCS.
2. What type of axis must you select to mirror a 3 dimensional object?

A. level
B. plane
C. line
D. circle
E. point.

3. What command is used to polar array an object in 3D space?
   A. 3DARRAY
   B. ARRAY
   C. REVOLVE
   D. ROTATE
   E. 3DORBIT
4. What is the Z-axis array called in a 3D array?
   A. Level
   B. Axis
   C. Row
   D. Elevation
   E. Column
5. On what axis will the row be on in a 3D ARRAY?
   A. X
   B. Both the X and Y
   C. Z
   D. In the negative Z direction
   E. Y
6. How many degrees apart are the XYZ axis in a 2 dimensional isometric drawing?
   A. 60
   B. 90
   C. 120
   D. 180
   E. 360
7. How must a model be constructed if you want to view it in hidden mode using the VSCURRENT command? Choose the BEST answer.
   A. As a solid model.
   B. As a 3D wireframe.

C. As a 2D wireframe.

D. As a surface model.

E. As either a surface or solid model.

8. What is the current visual style shown in Figure 11-1?

A. 2D wireframe

B. 3D wireframe

C. Hidden

D. Flat

E. Gouraud

9. What is the current visual style shown in Figure 11-2?

A. 2D wireframe

B. 3D wireframe

C. Hidden

D. Flat

E. Gouraud

10. What type of axis must you select to rotate a 3 dimensional object?

A. level

B. plane

C. line

D. circle

E. point

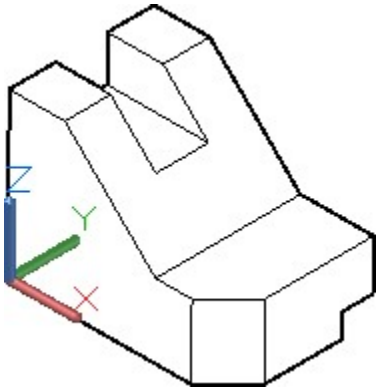


Figure 11-1

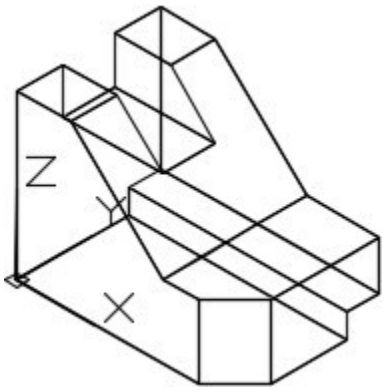


Figure 11-2

Lab Exercise 11-1 OPEN BOOK

Drawing Name	Template	Units
AutoCAD 3D Lab 11-1	3D Layout English	Inches

Step 1

On layer: Model, draw a wireframe model of the object shown in the multiview drawing. Set the current view to SE Isometric. (Figure Step 1A, 1B, 1C, and 1D)

AUTHOR'S COMMENTS: Draw the right side view first.

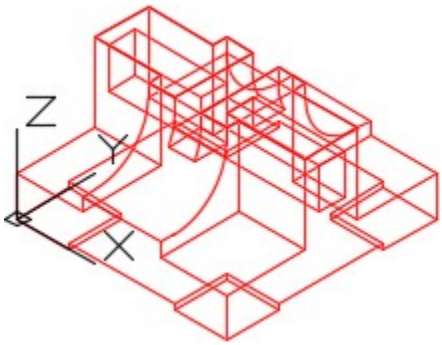


Figure Step 1A
Wireframe Model

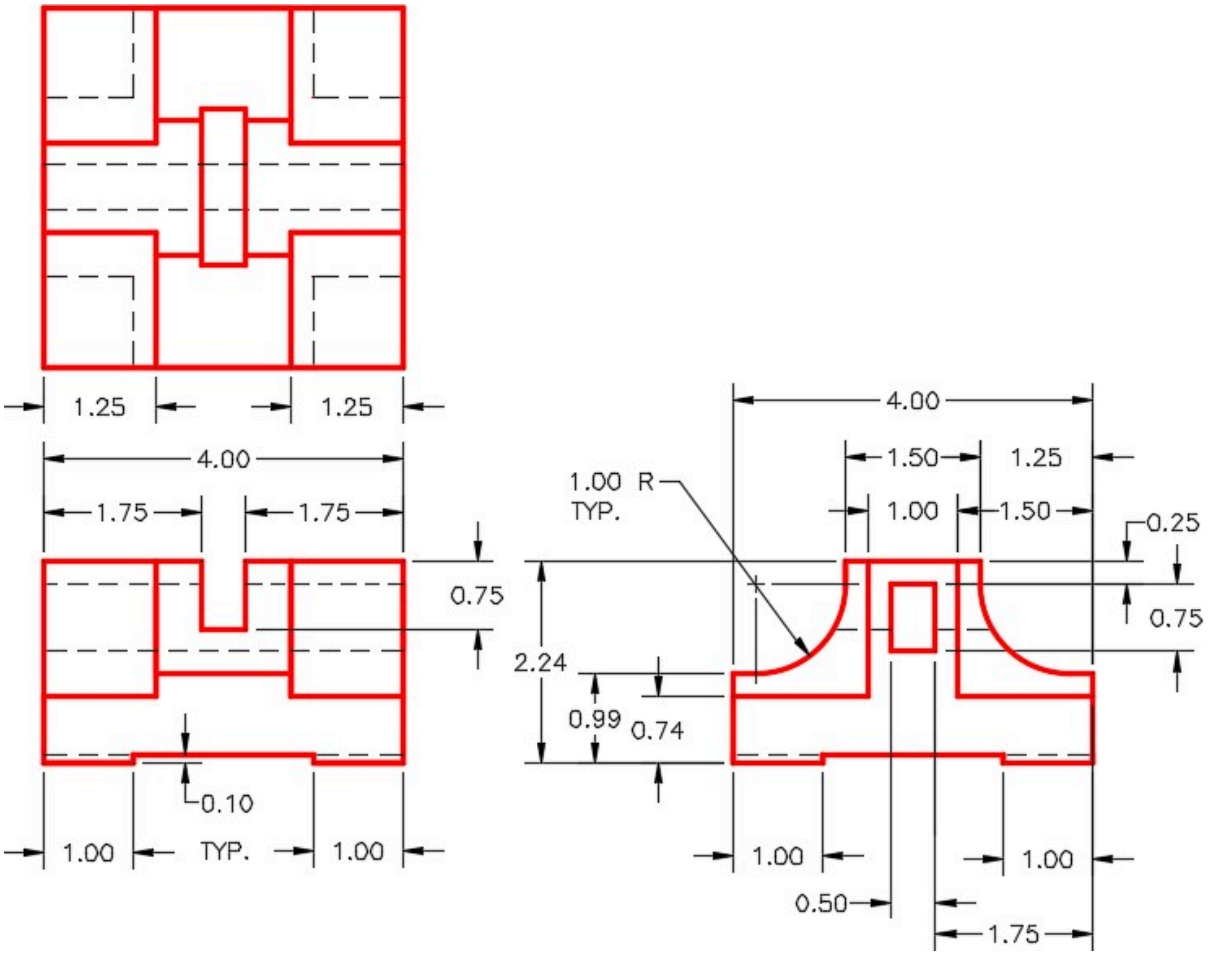


Figure Step 1B
Dimensioned Multiview Drawing

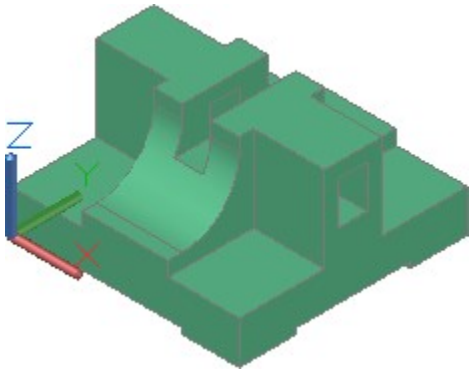


Figure Step 1C
Solid Model

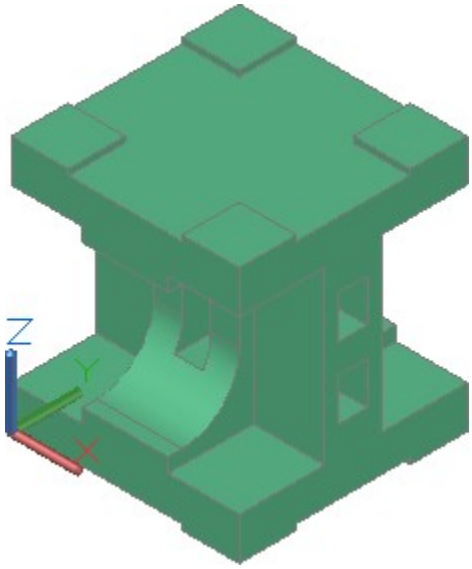


Figure Step 1D
Solid Model

Step 2

Mirror the original model so that the mirrored copy lies on top of the existing model as shown in the figure. (Figure Step 2)

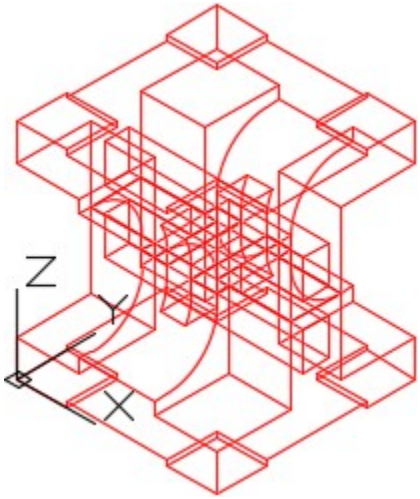


Figure Step 2

Step 3

Rotate the model 90 degrees as shown in the figure. Note the location of the WCS in relation to the model, after the rotation. (Figure Step 3)

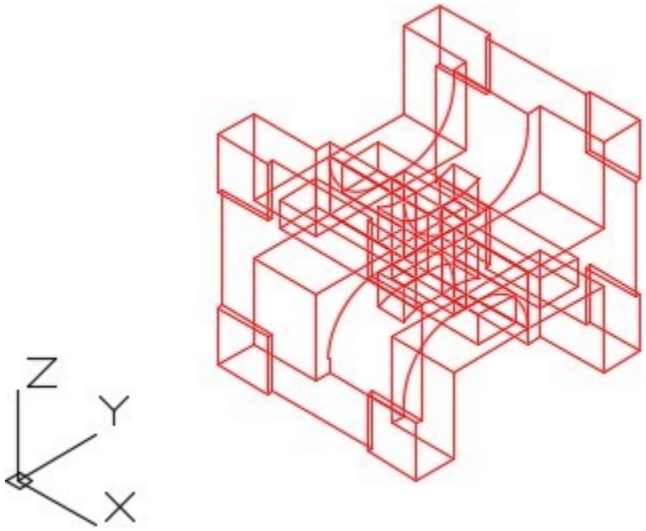


Figure Step 3

Step 4

Move the model to locate the front corner on the WCS. (Figure Step 4)

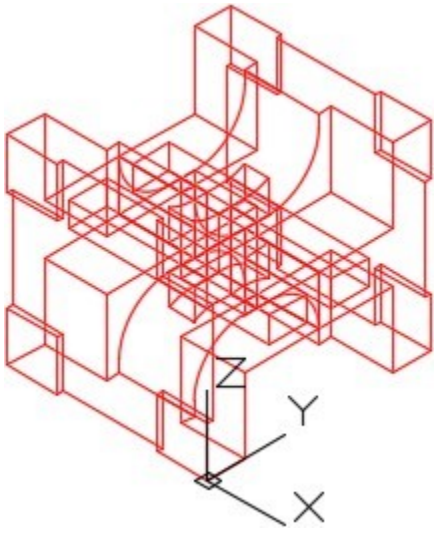


Figure Step 4

Step 5

Array the model to match the figure. Keep the location of the WCS in mind when you are executing the array. (Figure Step 5)

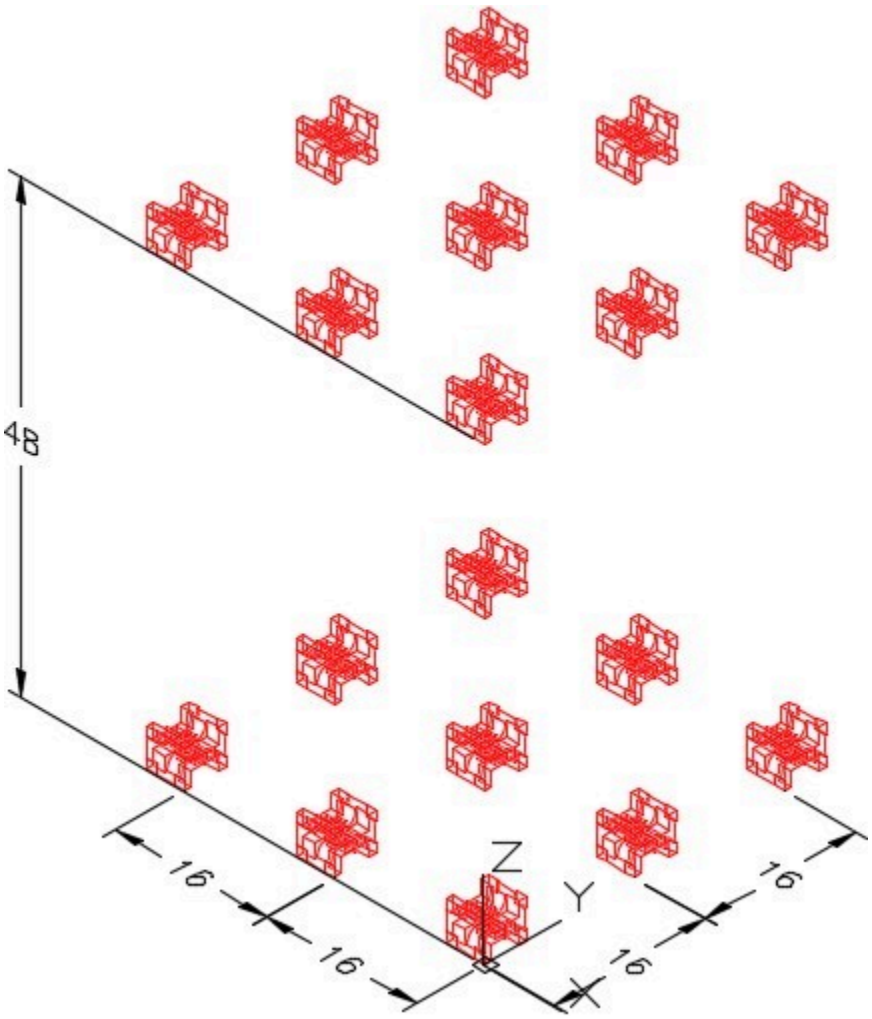


Figure Step 5

Step 6

On layer: Construction, draw a line from absolute coordinates X20,Y100,Z0 to the absolute coordinates X-5,Y100,Z0. (Note that X is negative 5). Before you draw it, ensure that the current UCS is World and keep in mind that the coordinates are absolute so do not use an @.

Step 7

Copy, rotate and locate one of the models at the end of the line you drew in Step 6. Locate it at the absolute coordinates X-5,Y100,Z0 (Note that X is negative 5) as shown in the figure. (Figure Step 7A and 7B)

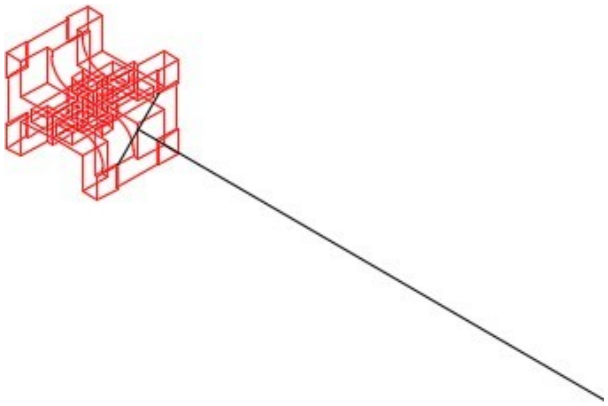


Figure Step 7A

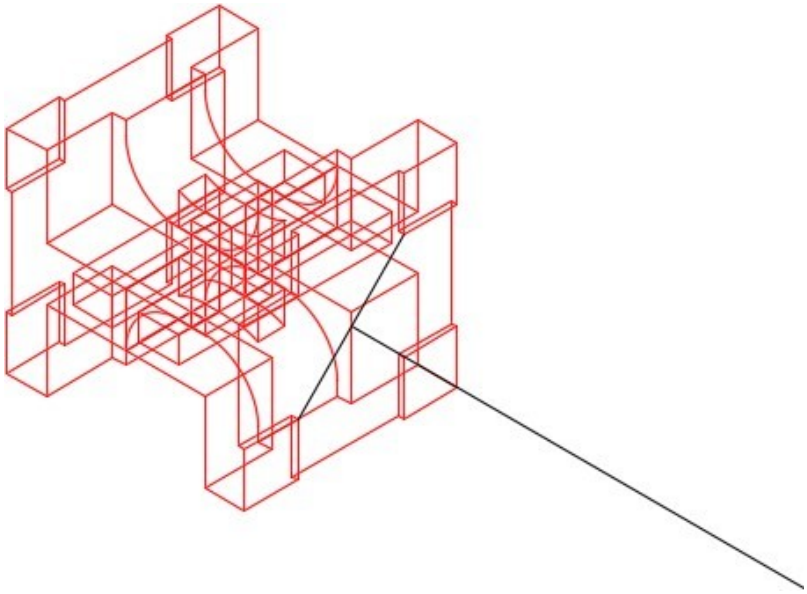


Figure Step 7B

AUTHOR'S COMMENTS: Draw a diagonal construction line from one corner of the pad on bottom of the model to the corner of the other pad. The line drawn in Step 6, ends at the midpoint of this diagonal line.

Step 8

Array the model as shown in the figure. (Figure Step 8)

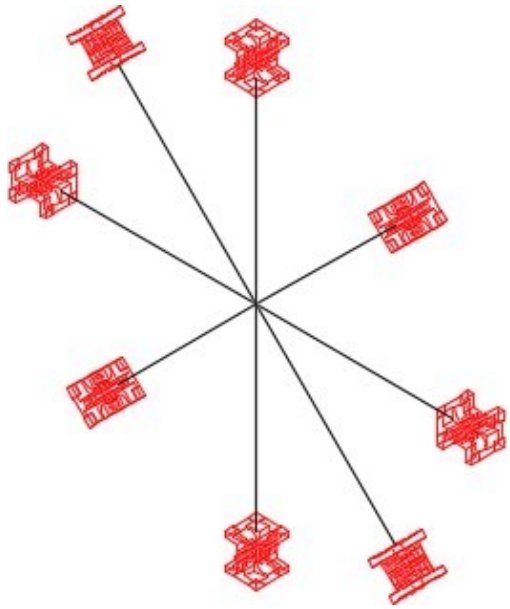


Figure Step 8

AUTHOR'S COMMENTS: Ensure that you locate the UCS in the correct location and disable object snap before you preform the array.

Step 9

Your complete model should match the figure. (figure Step 9)

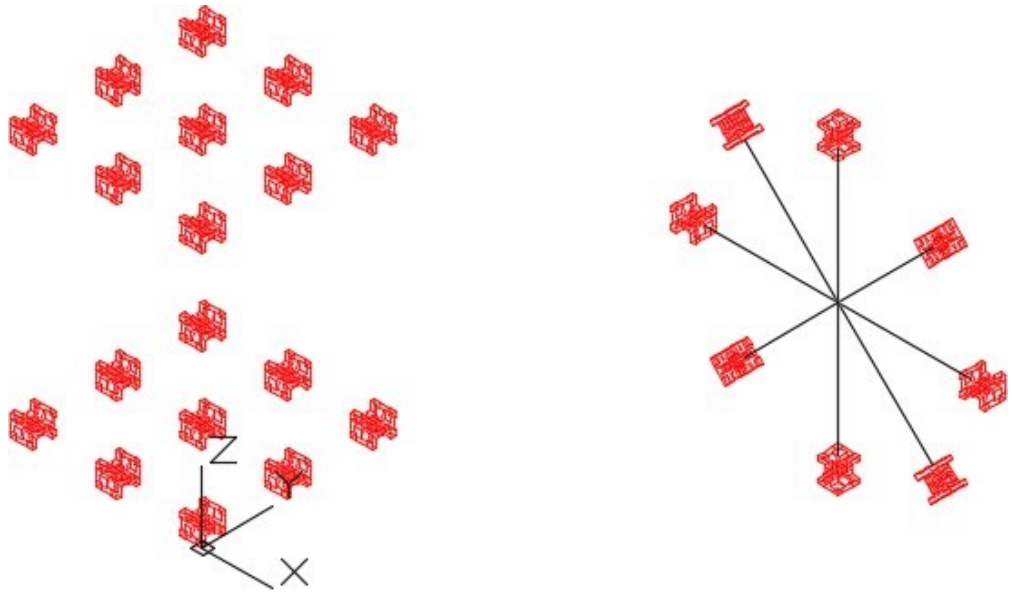


Figure Step 9

Step 10

Save and close the drawing.

Part 3

Module 12: Surface Modeling – Part 1

Learning Outcomes

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

1. Describe surface faces and meshes and explain when and why they are used.
2. Apply the 3DFACE command to create simple face surfaced models.

Surface Models

A *surface model* is a 3D model with all of its outer surfaces covered with a thin coating. Think of it as an empty box with a balloon skin. All the sides that define its shape are covered but it is hollow on the inside. Unlike a solid model, the volume, weight, and centre of gravity of a surface model cannot be calculated by AutoCAD. Solid modeling is taught in Modules 17 to 31.

There are many reasons why surface models are constructed rather than solid models. A surface model can be used to create objects that have complex curved shapes which are difficult to construct as a solid models. For industries such as the automobile, aircraft, and ship design, surfaces are used as a crucial part of their CAD design. For 3D mapping, surface meshes are used to define the 3D physical shape of the earth.

Surface Types

A surface can be either created with a face or a mesh or a combination of the two. A *surface face* is a good method to create simple surfaces while a *surface mesh* works for all surfaces including complex and curved surfaces. Figure 12-1 shows a model covered with a surface mesh. Faces and meshes can be combined, on the same model, when surfacing a model by creating some of the surfaces with faces and some with meshes. Once a model is surfaced, it can be shaded as shown in Figure 12-2.

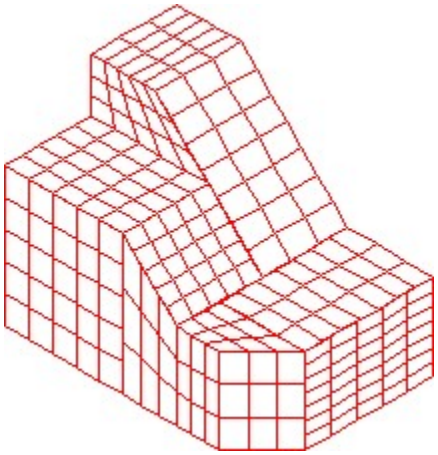


Figure 12-1
Model with Surface Mesh

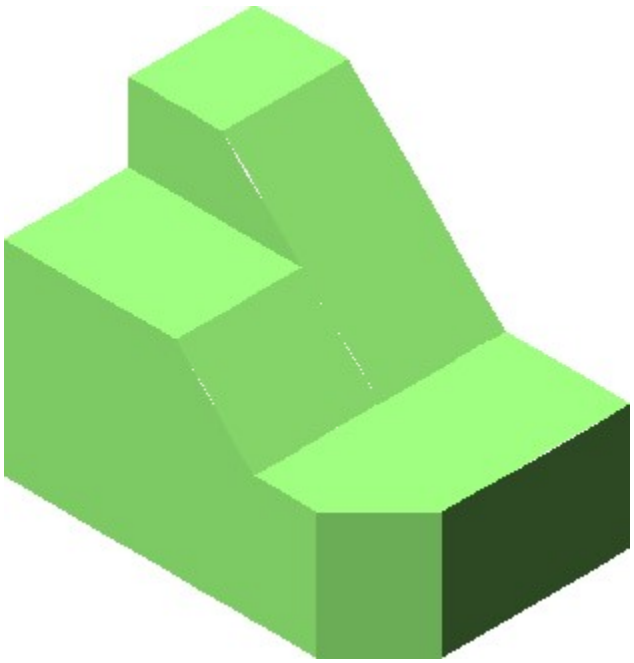
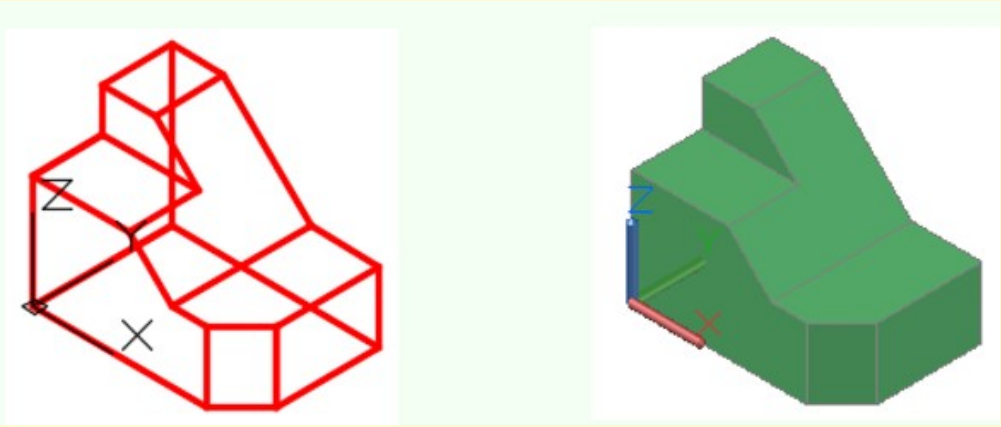


Figure 12-2
Shaded Surfaced Model

USER TIP: Although a model can be created with surfaces alone, it is much easier to construct it as a wireframe model first. Draw the wireframe on layer Model and when it is complete, create the surfaces and then freeze layer Model.

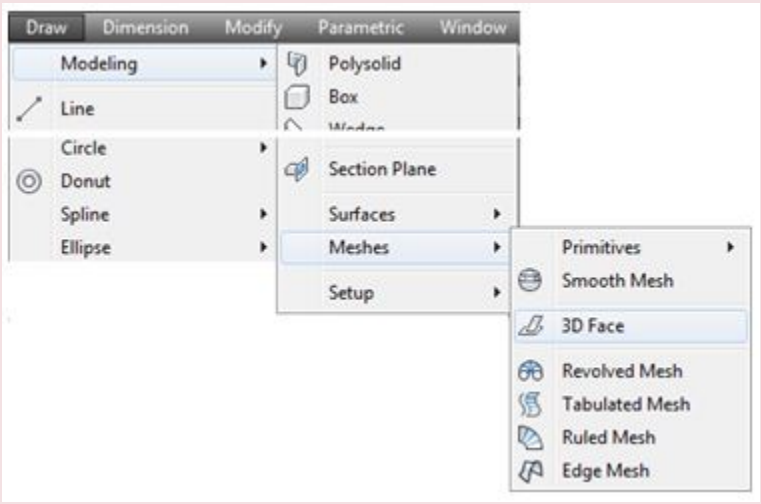


Wireframe Model *Surfaced Model*

AutoCAD Command: 3DFACE

The 3DFACE command is used to create a surface face bounded by a minimum of three and a maximum of four edges.

Shortcut: **3F**



USER TIP: There is no undo in the 3DFACE command. Be very careful when selecting the endpoints and plan the best order to create the surfaces before entering the command. If you make a mistake inside the command, all surface faces created in that command will be lost and you must reenter the command and start over.

WORK ALONG: Inserting Surface Faces

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 12-1.

Step 3

Set the current UCS to World. Enter the UNITS command and in the Units dialogue box, set the Insertion Units to Inches. Using the INSERT command, insert the block:

AutoCAD 3D Lab 01-2 at the coordinates 0,0,0. Explode the block and set the current view to SE Isometric. (Figure Step 3)

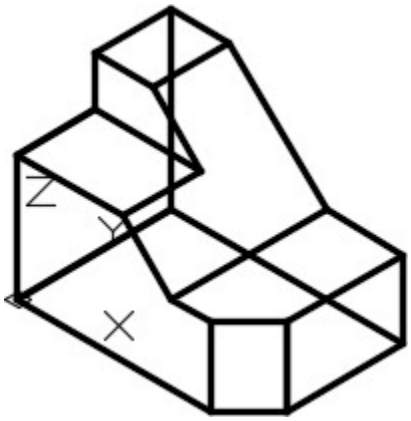


Figure Step 3

Step 4

Change the layer of all wireframe lines to layer: Model. (Figure Step 4)

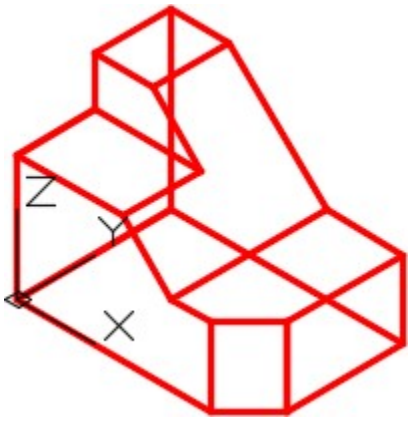


Figure Step 4

AUTHOR'S COMMENTS: The 3DFACE command is used to create a surface face bounded by a minimum of three and a maximum of four edges. The faces can be joined together with visible or invisible edges. By using invisible edges, several surfaces can be created to appear as one large surface.

Step 5

On layer: Construction, draw the construction lines to help you insert the surfaces. Keep in mind that the maximum number of sides that a face can be bound by is 4. Draw the lines on all six sides. (Figure Step 5)

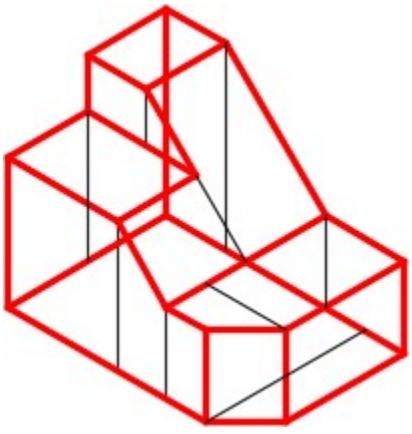


Figure Step 5

AUTHOR'S COMMENTS: Instead of drawing all the construction lines from scratch, try copying existing wireframe lines, using object snap, to locate them exactly and then change their layer to Construction.

Step 6

Set layer: Surface Face as the current layer. Enter the 3DFACE command, as shown below, to place a surface face on the wireframe model. Ensure you snap to the endpoints of the wireframe lines. (Figure Step 6)

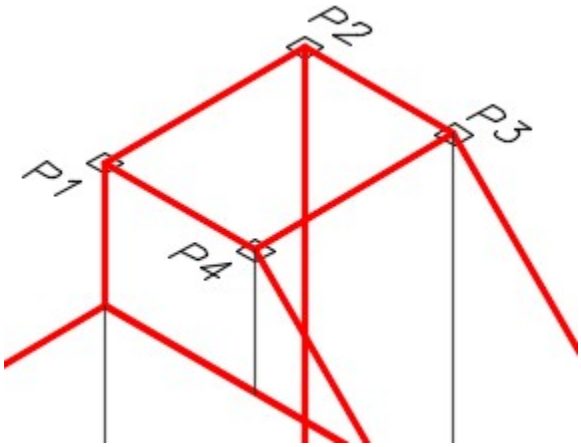


Figure Step 6

Command: **3DFACE**

Specify first point or [Invisible]: *(end)* **P1**

Specify second point or [Invisible]: *(end)* **P2**

Specify third point or [Invisible] <exit>: *(end)* **P3**

Specify fourth point or [Invisible]: *(end)* **P4**

(The surface face is defined by snapping to endpoints of existing wireframe lines. Pick points moving around the surface. Do not crisscross.)

Command:

AUTHOR'S COMMENTS: Surface face construction can be completed faster by inserting several faces in the same command. Since a face can only have a maximum of four sides, there are times when two or more faces must joined together to surface a plane on the model. A shaded model will appear much cleaner if the edges between the faces are inserted with invisible edges.

Step 7

Enter the 3DFACE command as shown below. In this command, insert the common edges on the same plane as invisible. The figure shows the two edges (marked with an I) that should be inserted as invisible. (Figure Step 7)

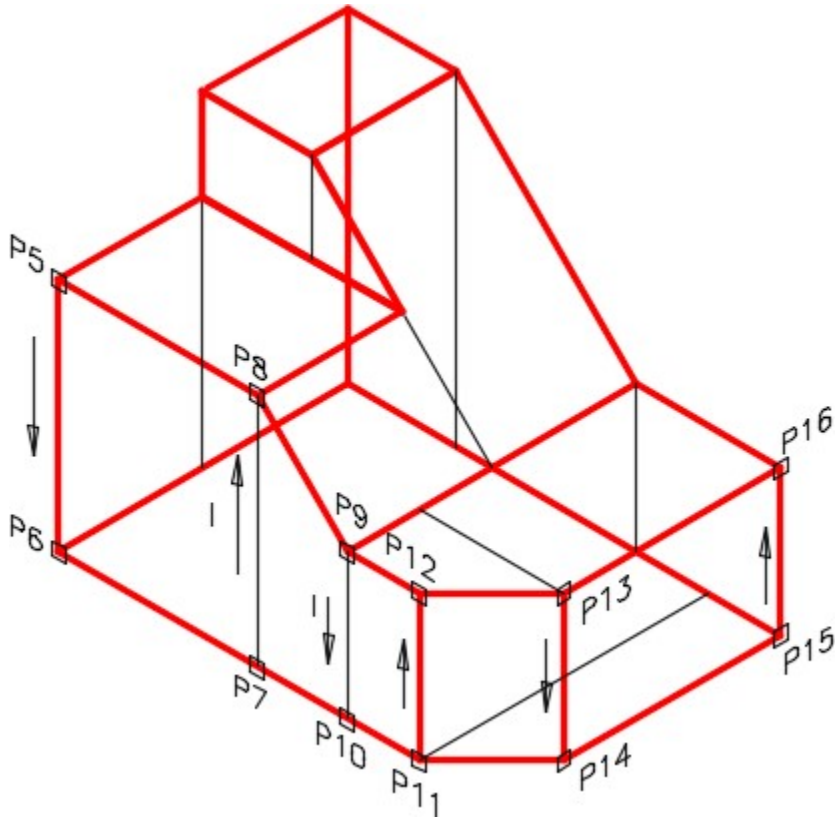


Figure Step 7

Command: **3DFACE**

Specify first point or [Invisible]: *(end)* **P5**

Specify second point or [Invisible]: *(end)* **P6**

Specify third point or [Invisible] <create three-sided face>: **I**

(You want the last edge to be invisible.)

Specify third point or [Invisible] <exit>: *(end)* **P7**

Specify fourth point or [Invisible] <create three-sided face>: *(end)* **P8**

Specify third point or [Invisible] <create three-sided face>: **I**

(You want the last edge to be invisible.)

Specify third point or [Invisible] <exit>: (*end*) **P9**

Specify fourth point or [Invisible] <create three-sided face>: (*end*) **P10**

Specify fourth point or [Invisible] <create three-sided face>: (*end*) **P11**

Specify fourth point or [Invisible] <create three-sided face>: (*end*) **P12**

Specify fourth point or [Invisible] <create three-sided face>: (*end*) **P13**

Specify fourth point or [Invisible] <create three-sided face>: (*end*) **P14**

Specify fourth point or [Invisible] <create three-sided face>: (*end*) **P15**

Specify fourth point or [Invisible] <create three-sided face>: (*end*) **P16**

Specify third point or [Invisible] <exit>:

Command:

Step 8

To check the surfaces you just inserted, set the current visual style to Realistic as shown in the figure. (Figure Step 8)

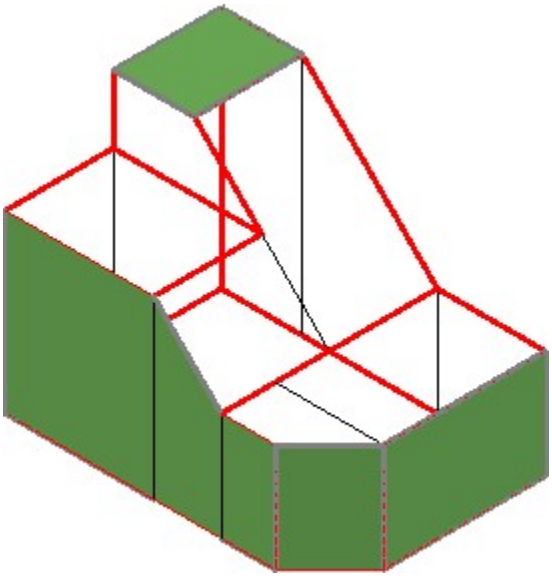


Figure Step 8

Step 9

Change the current visual style to 3D Wireframe.

Step 10

Using the 3DFACE command, add faces to completely surface the model. Ensure that you apply faces on the back and bottom. Turn layers: Model and Construction off.

Step 11

Change the current visual style to Realistic. Using the 3DFORBIT command, ensure that the complete model is surfaced. (Figure Step 11A and 11B)

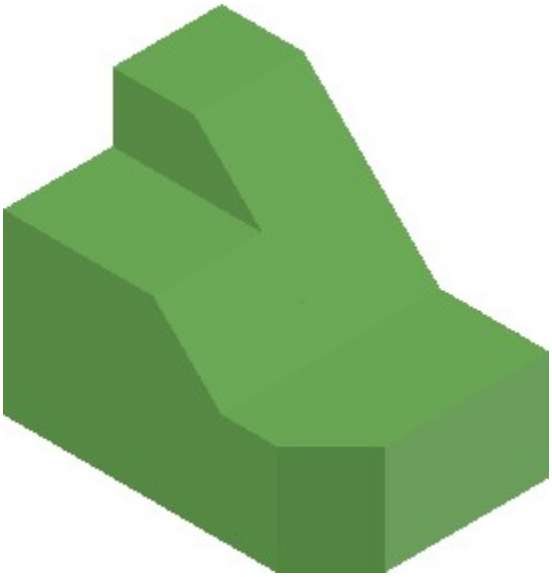


Figure Step 11A

Figure Step 11B

Step 12

Save and close the drawing.

AUTHOR'S COMMENTS: Do not be concerned if your shaded model does not match the figures in the

module exactly. There are many visual style settings that can be set to change the look of the shaded model. For now, all you should be concerned about is being able to surface the model. The advanced features of customizing visual styles are taught in the AutoCAD 3D Advanced book.

Key Principles

Key Principles in Module 12

1. A surface model is a 3D model with all of its outer surfaces covered with a A surface can be either created with a face or a mesh. A surface face is a good method to create simple surfaces while a surface mesh works for all surfaces including complex and curved surfaces.
2. Faces and meshes can be combined when surfacing a model by creating some of the surfaces with faces and some with
3. The 3DFACE command is used to create a surface face bounded by a minimum of three and a maximum of four The faces can be joined together with visible or invisible edges to make the faces appear as large as necessary.

Lab Exercise 12-1

Time allowed: 60 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 12-1	3D Layout English	Inches

Step 1

On layer: Model, draw a wireframe model of the object shown in the figures. (Figure Step 1A, 1B, and 1C)

Step 2

On layer: Surface Face, create surface faces on all surfaces including the back and bottom. Make all surface edges invisible.

Step 3

Freeze layers: Construction and Model.

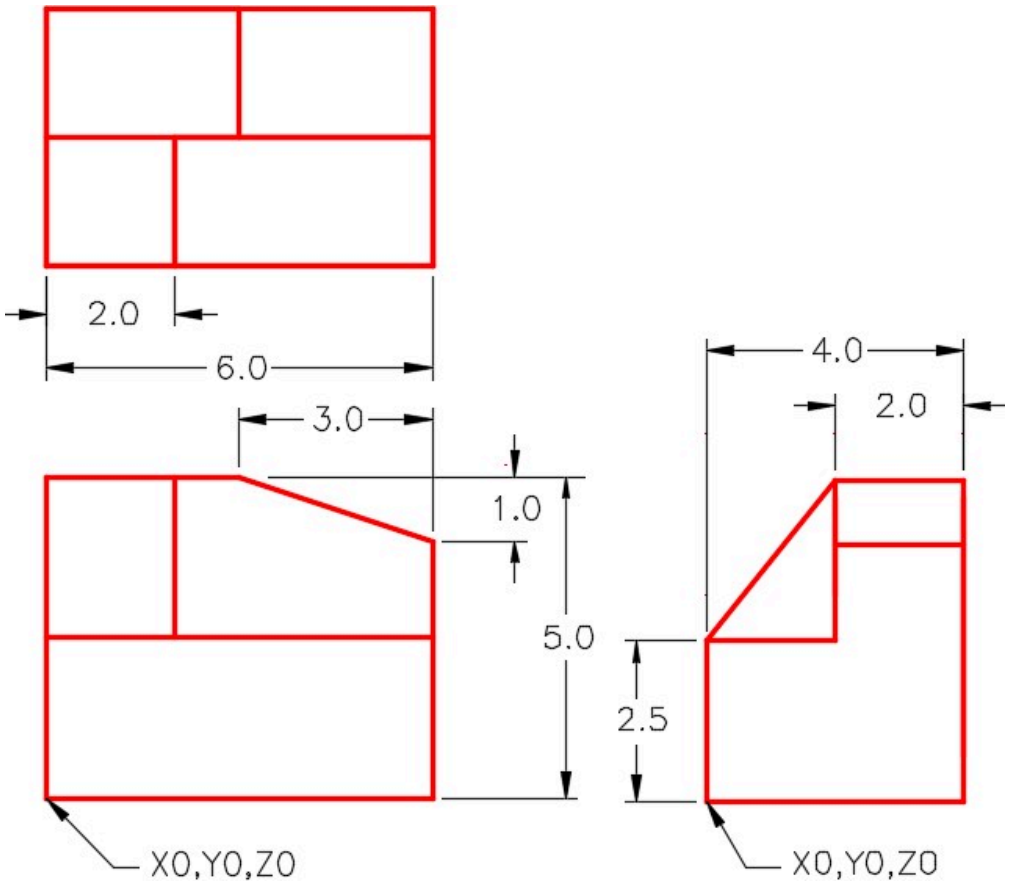


Figure Step 1A
Dimensioned Multiview Drawing

Step 4

See the current visual style to Realistic. Using 3DFORBIT, check it for completeness.

Step 5

Set the Insertion Units, change the current UCS to World and check the model with the key.

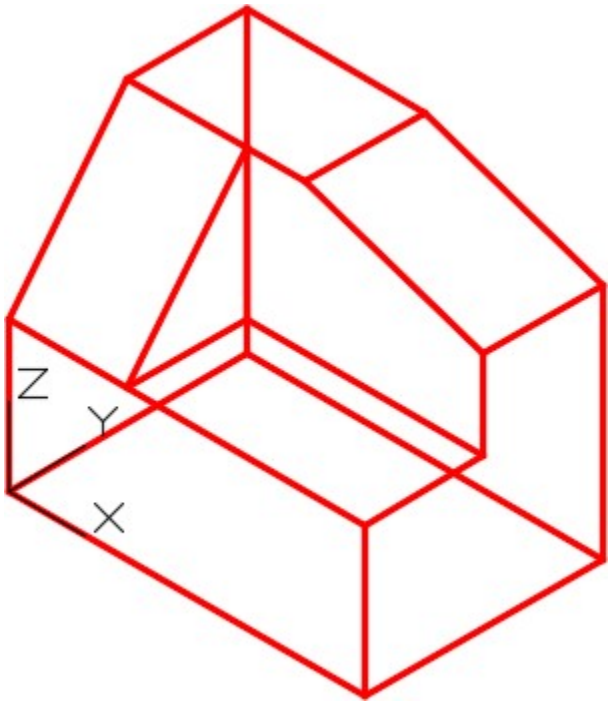


Figure Step 1B Wireframe of Model SE Isometric View

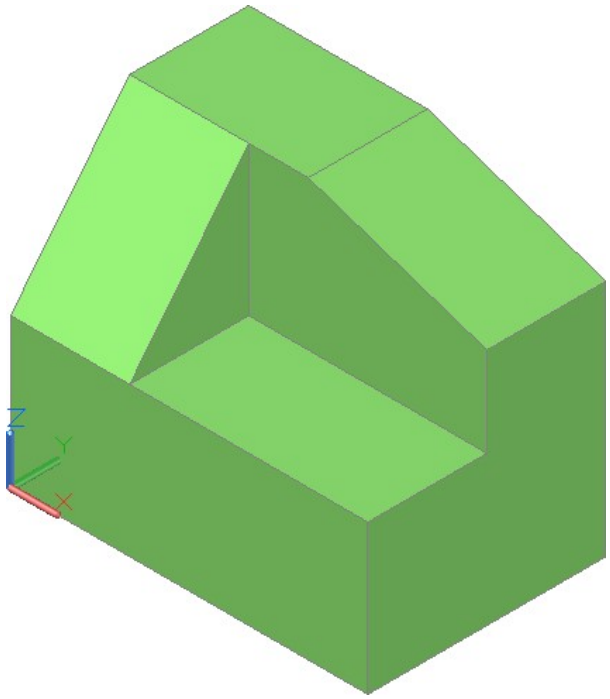


Figure Step 1C Shaded Model

Step 6

Save and close the drawing.

AUTHOR'S COMMENTS: Do not be concerned if your shaded model does not match the figures in the module exactly. There are many visual style settings that can be set to change the look of the shaded model. For now, all you should be concerned about is being able to surface the model.The advanced features of customizing visual styles are taught in the AutoCAD 3D Advanced book.

Lab Exercise 12-2

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 12-2	N/A	Millimeters

Step 1

Open the drawing: AutoCAD 3D Lab 05-2.

Step 2

Using the SAVEAS command, save the drawing with the name: AutoCAD 3D Lab 12-2.

Step 3

On layer: Surface 3, create surface faces on each side including the back and bottom. (Figure Step 3A and 3B)

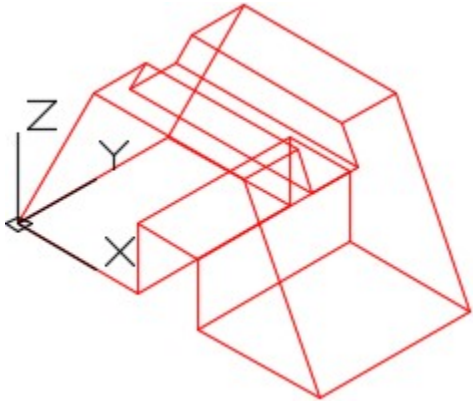


Figure Step 3A
Wireframe Model

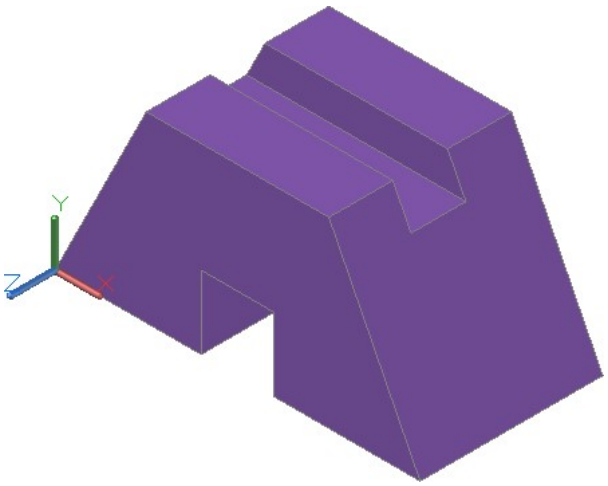


Figure Step 3B
Surfaced Model – Shaded

Step 4

Freeze layers: Construction and Model.

Step 5

See the current visual style to Realistic. Using 3DFORBIT, check it for completeness.

Step 6

Save and close the drawing.

AUTHOR'S COMMENTS: Do not be concerned if your shaded model does not match the figures in the module exactly. There are many visual style settings that can be set to change the look of the shaded model. For now, all you should be concerned about is being able to surface the model. The advanced features of customizing visual styles are taught in the AutoCAD 3D Advanced book.

Lab Exercise 12-3

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 12-3	N/A	Inches

Step 1

Open the drawing: AutoCAD 3D Lab 07-1.

Step 2

Using the SAVEAS command, save the drawing with the name: AutoCAD 3D Lab 12-3.

Step 3

On layer: Surface 2, create surface faces on each side including the back and bottom. Surface all of it with mesh surfaces. (Figure Step 3A and 3B)

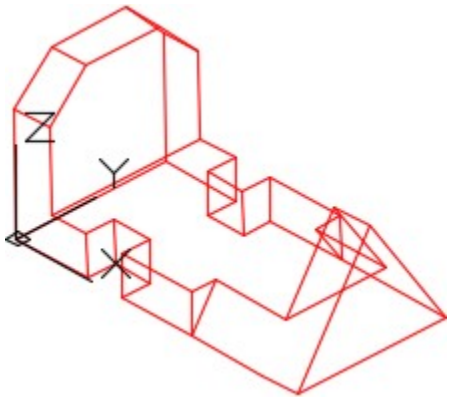


Figure Step 3A
Wireframe Model

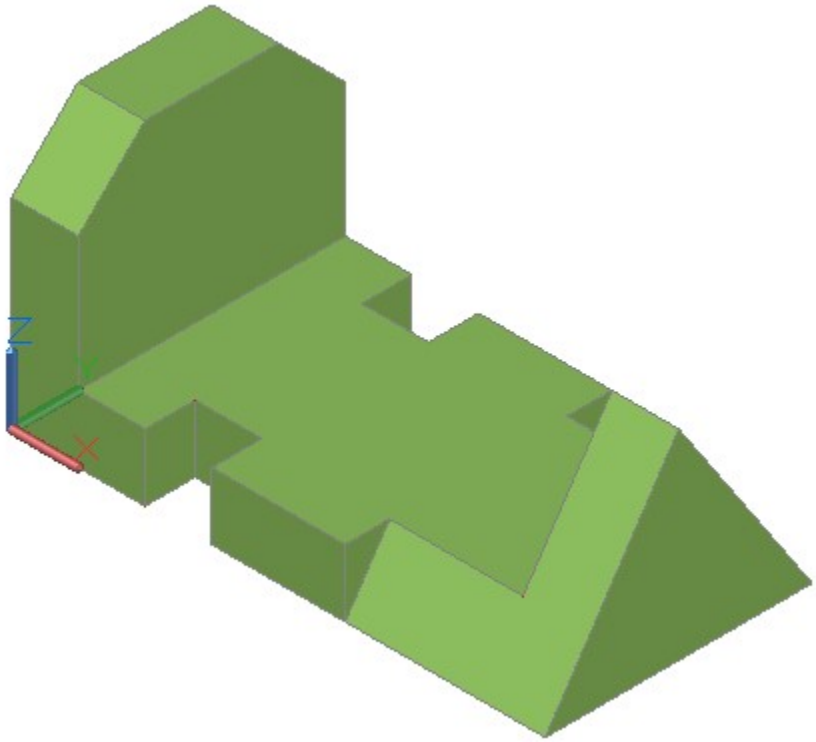


Figure Step 3B
Surfaced Model – Shaded

Step 4

When complete, freeze layers: Construction and Model.

Step 5

See the current visual style to Realistic. Using 3DFORBIT, check it for completeness.

Step 6

Save and close the drawing.

AUTHOR'S COMMENTS: Do not be concerned if your shaded model does not match the figures in the module exactly. There are many visual style settings that can be set to change the look of the shaded model. For now, all you should be concerned about is being able to surface the model. The advanced features of customizing visual styles are taught in the AutoCAD 3D Advanced book.

Module 13: Surface Modeling – Part 2

Learning Outcomes

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

1. Describe geometry defined meshes including rule surfaces, revolved surfaces, and tabulated surfaces.
2. Apply the RULESURF, REVSURF, and TABSURF commands to apply mesh surfaces to 3D models.

Geometry Defined Surface Meshes

Geometrically defined surface meshes use existing geometry that must be created before the surfaces. In almost all cases, the wireframe of the model is used as the existing geometry. There are four geometrically defined surface meshes that can be created in AutoCAD. They are the ruled surface, revolved surface, tabulated surface, and edge surface. The commands to create these meshes are RULESURF, REVSURF, TABSURF, and EDGESURF. The EDGESURF command is taught in Module 15.

To create a geometry defined surface mesh, start with the wireframe geometry and ensure that it is on its own layer. Surface the wireframe model on all sides, making sure that all exposed sides have a surface covering them. Think of the model as an object that must be made water tight.

Even the inside of a hole going through the model must have a surface applied on it. Place the surface meshes on their own layer. When the surfaced model is complete, freeze the layer containing the wireframe or the geometry leaving only the layer with the surface mesh displayed.

Ruled Surface

A *ruled surface*, inserted with the RULESURF command, is the most commonly used method to surface a model. To place a ruled surface, simply select two lines, a line and an arc, two arcs, or two circles to place the surface between. See Figure 13-1.

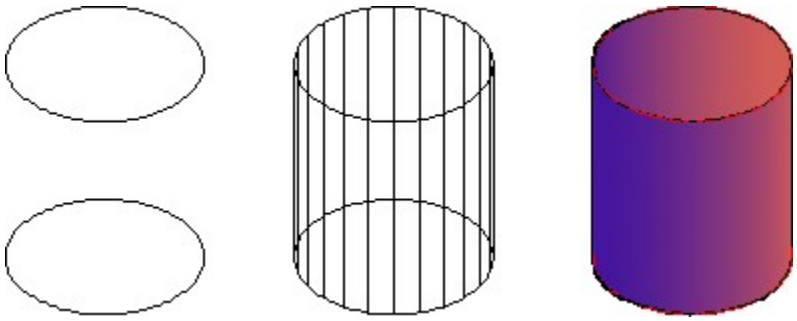


Figure 13-1
Ruled Surface

Revolved Surface

A *revolved surface*, inserted with the REVSURF command, is a surface created by revolving a profile around an axis. The profile can be a line, arc, circle, or an open or closed 2D polyline or 3D polyline. See Figure 13-2. The axis must be a line or an open 2D or 3D polyline. If a polyline is used as the axis, the REVSURF command will simply use a straight line between the start point and end point of the polyline.

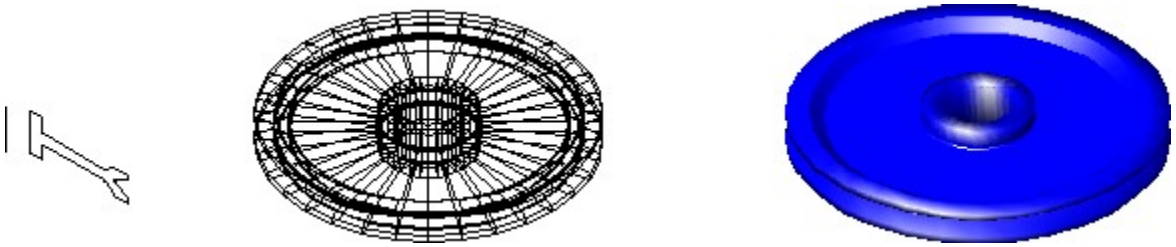


Figure 13-2
Revolved Surface

Tabulated Surface

A *tabulated surface*, inserted with the TABSURF command, is a surface created by projecting a profile along a path. The profile defines the surface of the mesh as it follows the path. See Figure 13-3. The profile can be a line, arc, circle, ellipse, or an open or closed 2D or 3D polyline. The path can be a line or a polyline. If a polyline is used as the axis, the TABSURF command will simply use a straight line between the start point and end point of the polyline. The surface is drawn starting at the point on the profile closest to the point selected in the command.

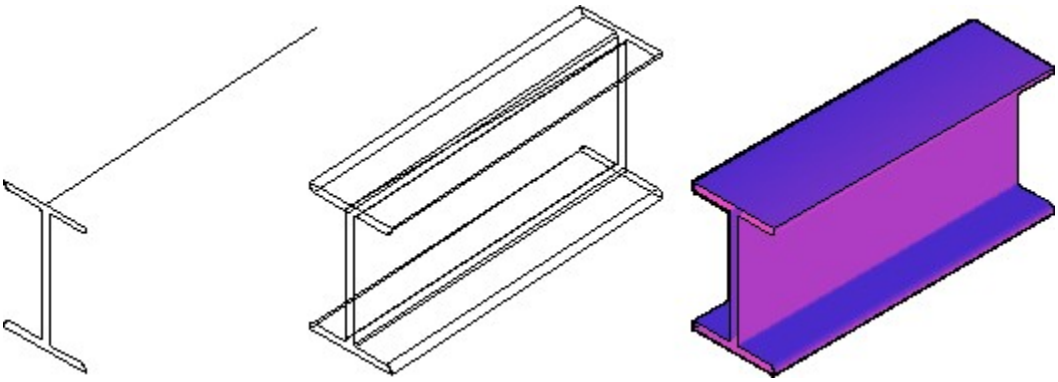


Figure 13-3
Tabulated Surface

Setting the Mesh Density

The *mesh density* represented by the lines, circles, or arcs that make the rows and columns of the surface mesh are controlled with the system variables SURFTAB1 and SURFTAB2. On flat surfaces, the density of the mesh is not that important but with curved or irregular shaped surfaces, the density is very important since the higher the setting, the more segments are used when creating circles and arcs. If the mesh is not dense enough, small gaps will be left where a curved surface meets a flat surface. See Figure 13-4.

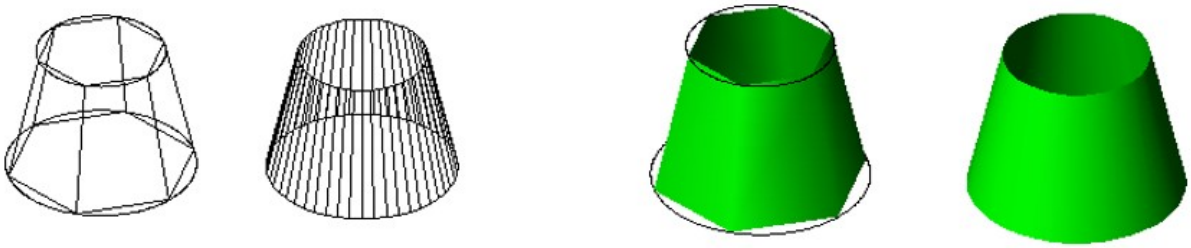


Figure 13-4
SURFTAB Settings

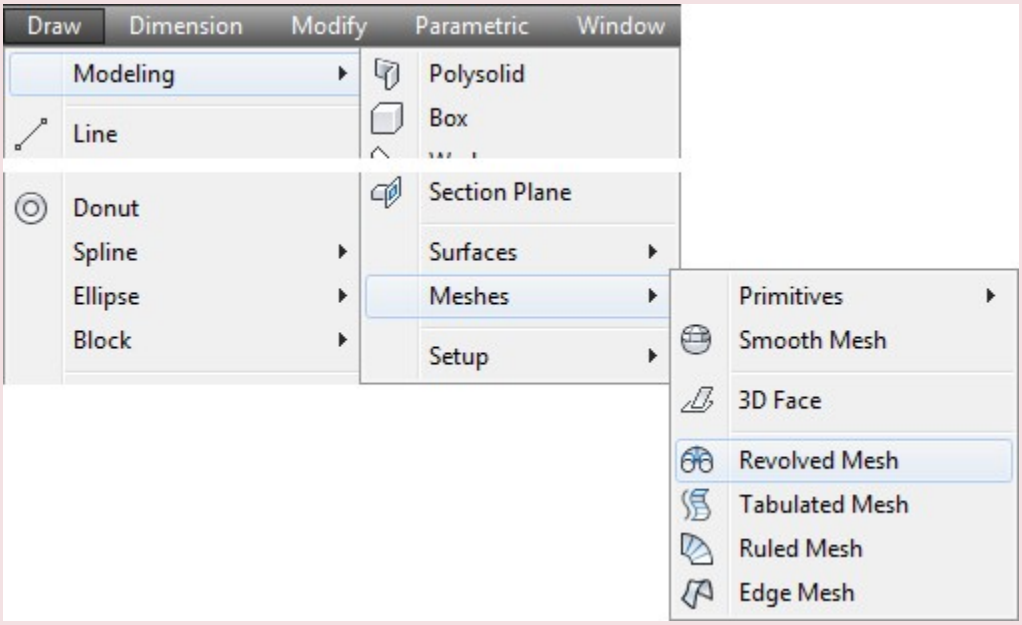
AutoCAD Command: RULESURF

The RULESURF command is used to create a rule surface between two existing edges.

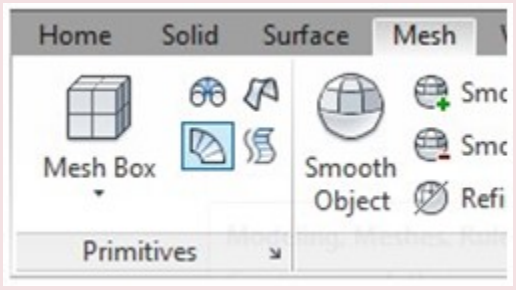
Shortcut: none

SURFTAB1 = YES

SURFTAB2 = No Effect



Draw Pull-down



Mesh Ribbon

WORK ALONG: Inserting Ruled Surfaces

Step 1

Using the NEW command, start a new drawing using template: 3D Layout Metric.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 13-1.

Step 3

Set the current layer to: Model and the current visual style to 2D Wireframe.

Step 4

Draw a wireframe model of the multiview drawing. (Figure Step 4A and 4B)

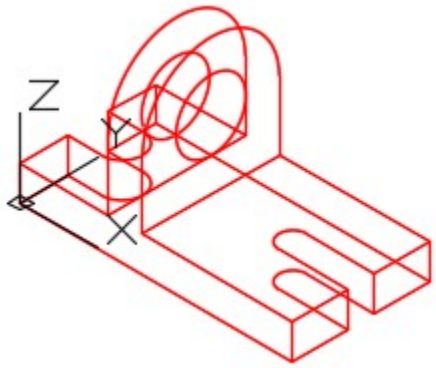


Figure Step 4A
Wireframe Model

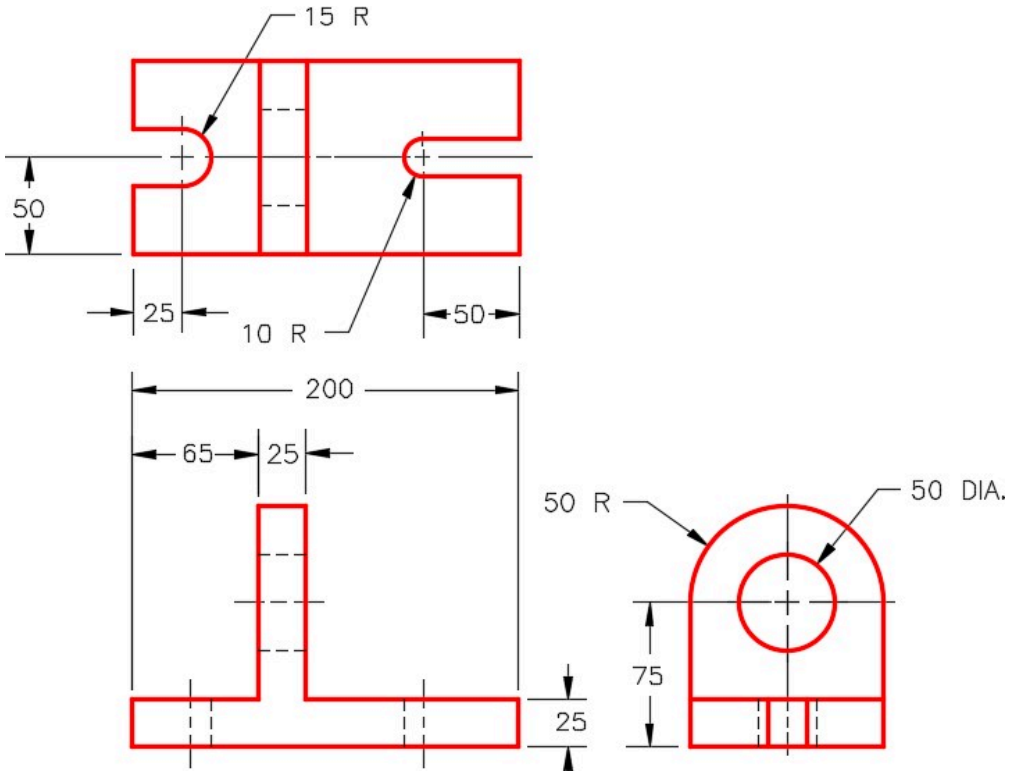


Figure Step 4B
Dimensioned Multiview Drawing

Step 5

Set the system variable SURFTAB1 to 24 as shown below: Command: **SURFTAB1**

Enter new value for SURFTAB1 <8>: **24**

Command:

AUTHOR'S COMMENTS: Setting SURFTAB1 to 24 sets the mesh density to 24 for the ruled surfaced inserted.

Step 6

On layer: Construction, draw a line along the top edge of each side of the model. (Figure Step 6)

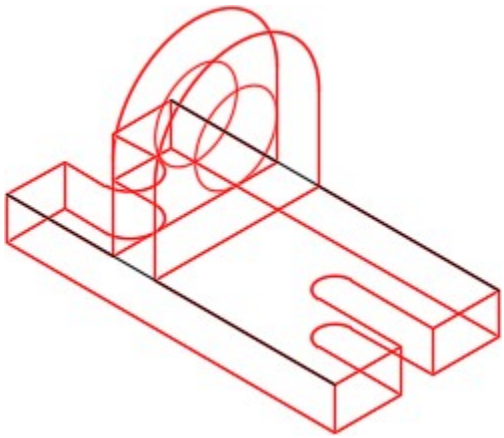


Figure Step 6

AUTHOR'S COMMENTS: On some models, construction lines must be drawn on the wireframe to make it easier to insert surfaces.

Step 7

Set layer: Surface 1 as the current layer. Enter the RULESURF command, as shown below, to insert a ruled surface. (Figure Step 7A and 7B)

Command: **RULESURF**

Current wire frame density: SURFTAB1=24

Select first defining curve: **P1**

Select second defining curve: **P2**

Command:

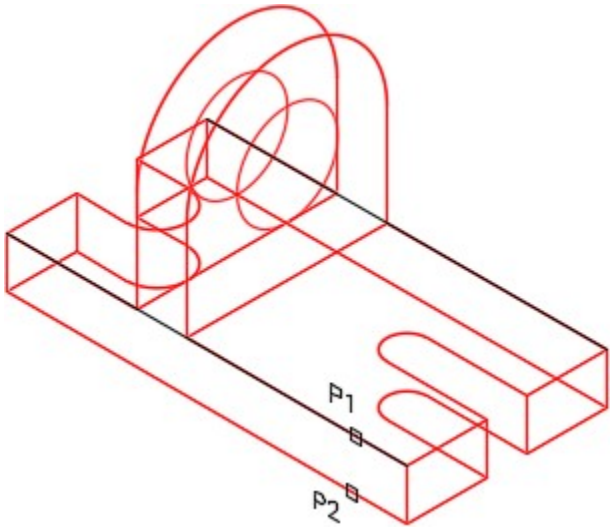


Figure Step 7A

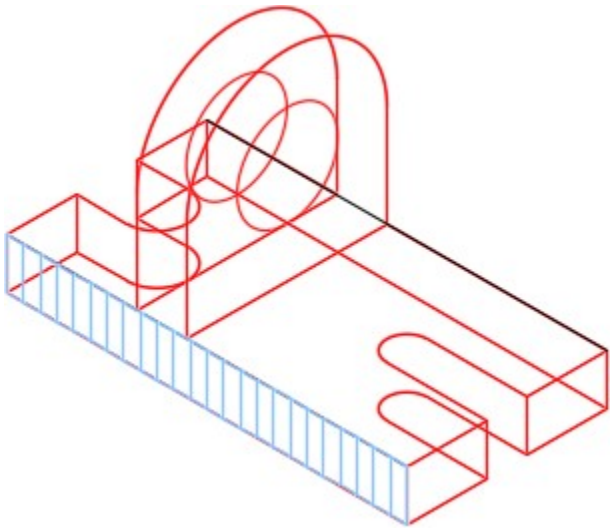


Figure Step 7B

Step 8

Repeat the RULESURF command and insert surfaces all around the edge of the model. (Figure Step 8)

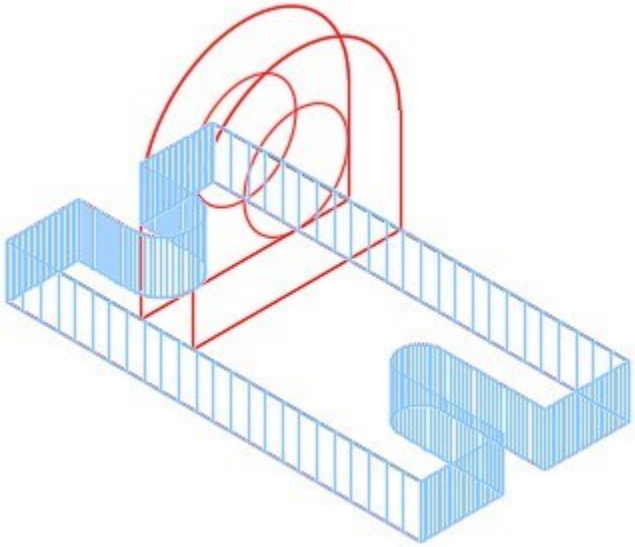


Figure Step 8

Step 9

Turn layer: Surface Off off and lock layers: Model and Construction. Select all of the surfaces inserted in Steps 7 and 8 and change their layer to: Surface Off. (Figure Step 9)

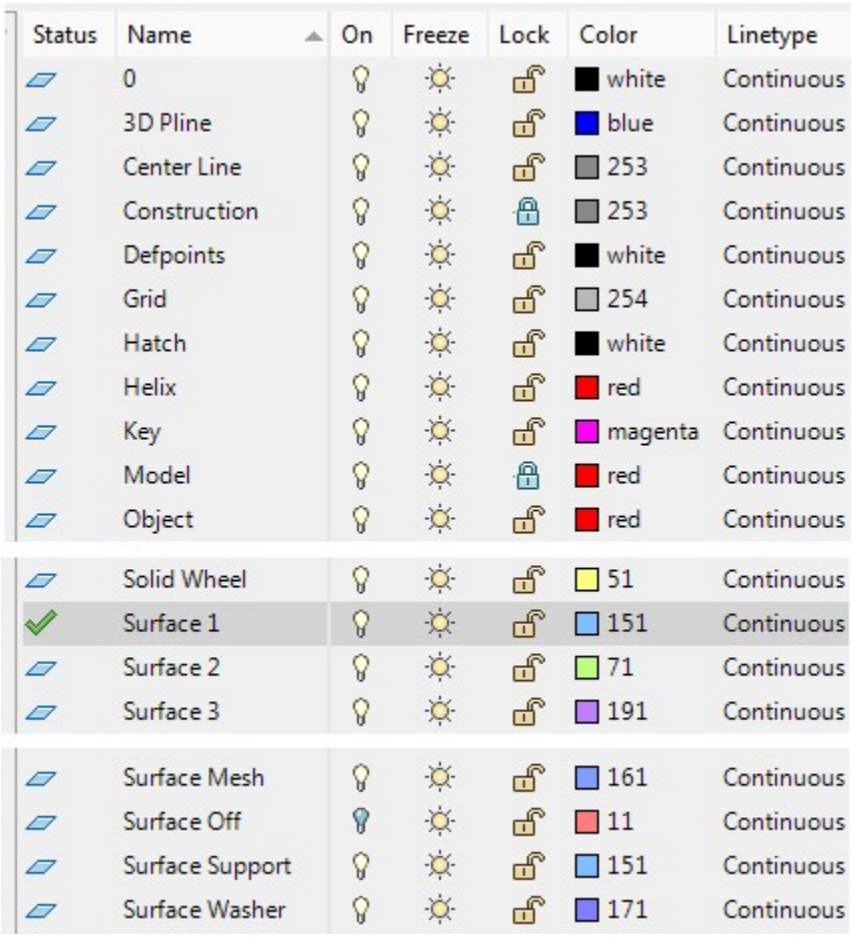


Figure Step 9

AUTHOR'S COMMENTS: I use a little trick in Step 9 to help you when surfacing the model. I temporarily change the layers of newly created surfaces to a layer that is turned off. That way, the surfaces do not display and get in the way while you are inserted additional surfaces on the model. When all of the surfaces are inserted to complete the model, turn layer Surface Off on and change the layer of all surfaces back to layer Surface 1. The reason I locked layers Model and Construction is to make is easier to select only the surfaces using windows when changing their layer property.

Step 10

Using what was just taught, insert ruled surfaces around the top of the object and inside of the hole as shown in the figure. (Figure Step 10).

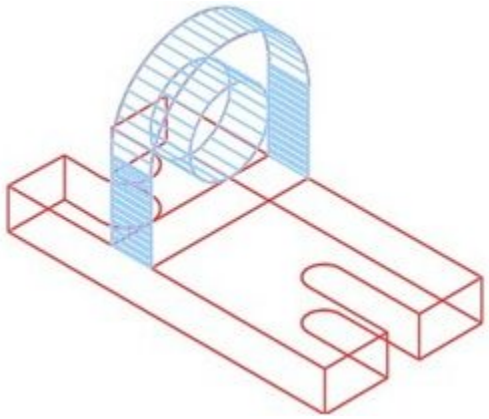


Figure Step 10

Step 11

Change the layer of the surfaces inserted in Step 10 to layer: Surface Off.

Step 12

On layer: Construction, draw the construction lines on the top surface as shown in the figure. Ensure to snap to the endpoints of the lines and arcs. (Figure Step 12)

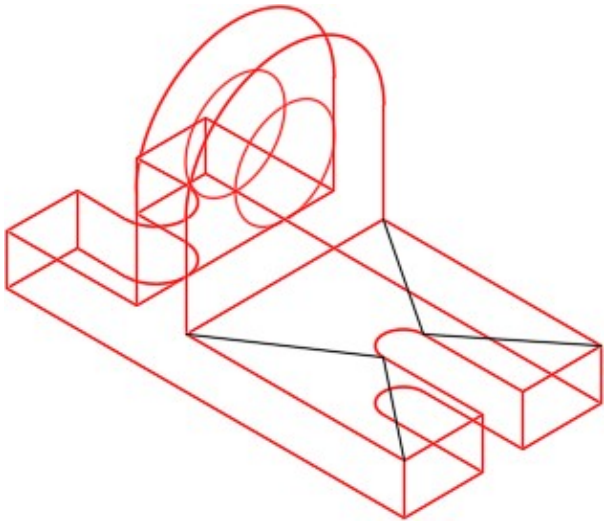
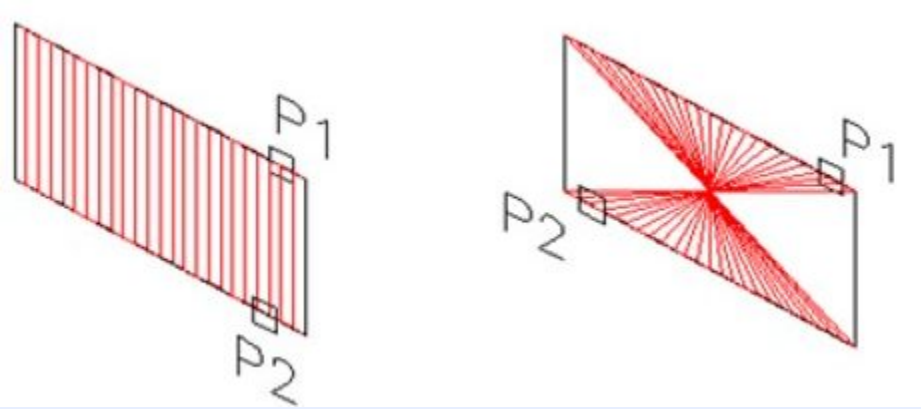


Figure Step 12

MUST KNOW: When using the RULESURF command, ensure that you select the objects closest to

matching ends. If opposite ends are selected, the mesh will twist as shown in the figure.



Step 13

On layer: Surface 1, use the RULESURF command to insert the surfaces as shown in the figure. Change the surfaces the layer: Surface off. (Figure Step 13)

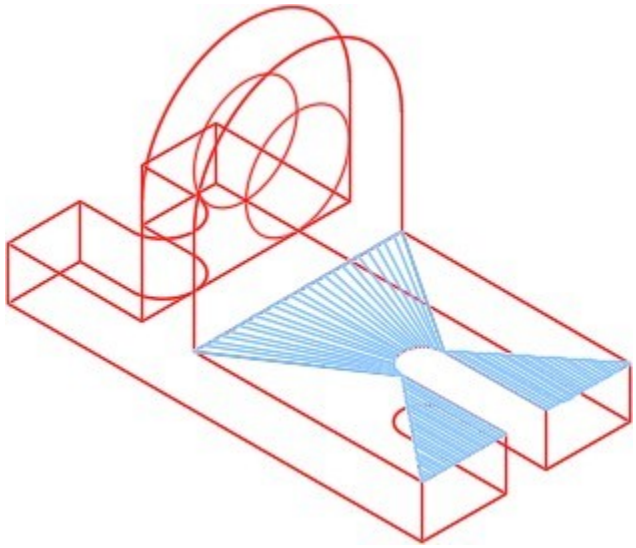


Figure Step 13

Step 14

On layer: Surface 1, use the RULESURF command to draw surfaces as shown in the figure. Change the layer of the surfaces to layer: Surface off. (Figure Step 14)

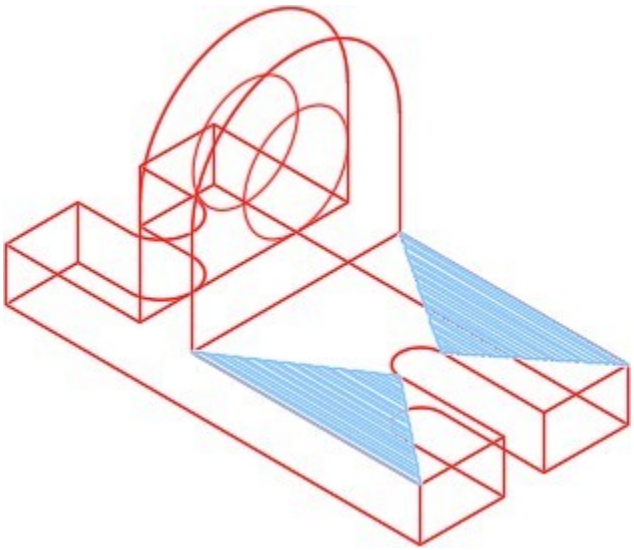


Figure Step 14

Step 15

On layer: Construction, draw a 180 degree arc on the top half of the circle. Ensure that you snap to the quads of the circle. To do that, ensure to locate the UCS on the same plane as the circle. (Figure Step 15)

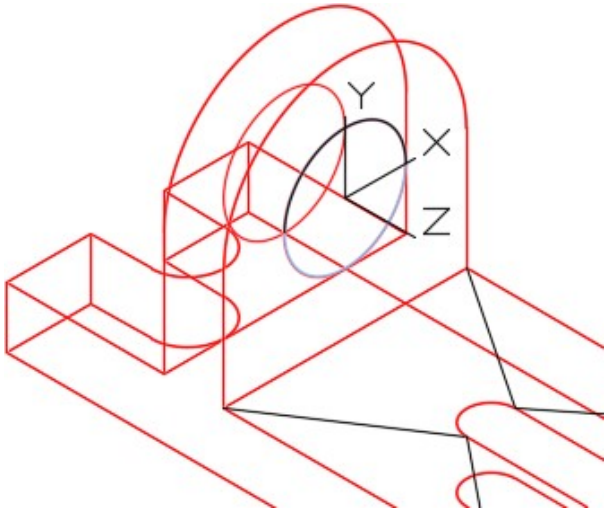


Figure Step 15

Step 16

On layer: Surface 1, insert a ruled surface between the arcs. (Figure Step 16)

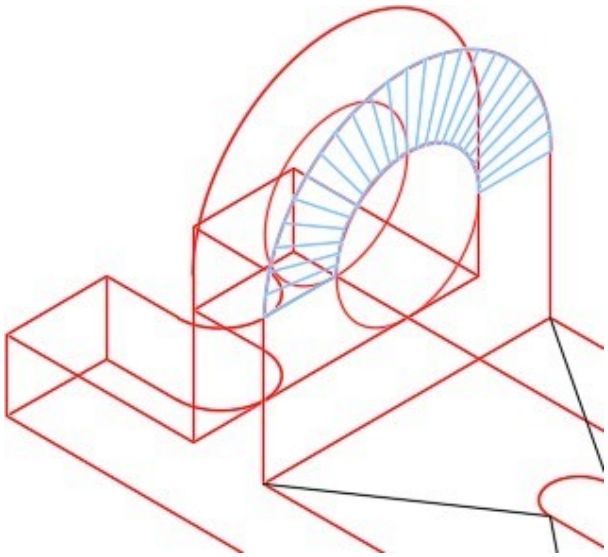


Figure Step 16

AUTHOR'S COMMENTS: Ensure that you select the arc instead of the circle under it. Sometimes, this can be tricky.

Step 17

Change the layer of the surface that you just drew to layer: Surface Off.

Step 18

Draw two construction lines and a construction arc on the bottom half of the circle. (Figure Step 18)

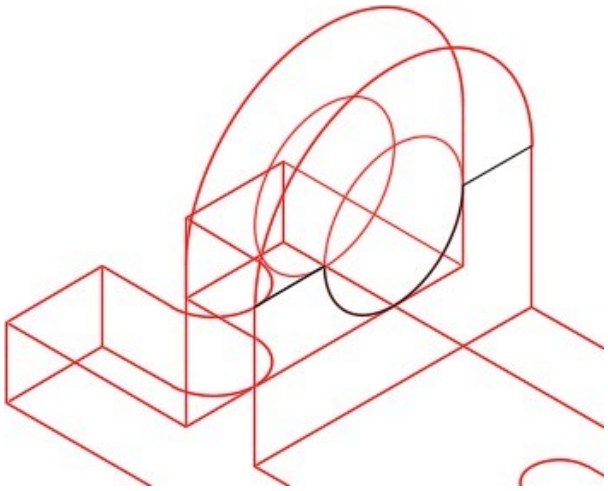


Figure Step 18

Step 19

On layer: Surface 1, insert a ruled surface. (Figure Step 19)

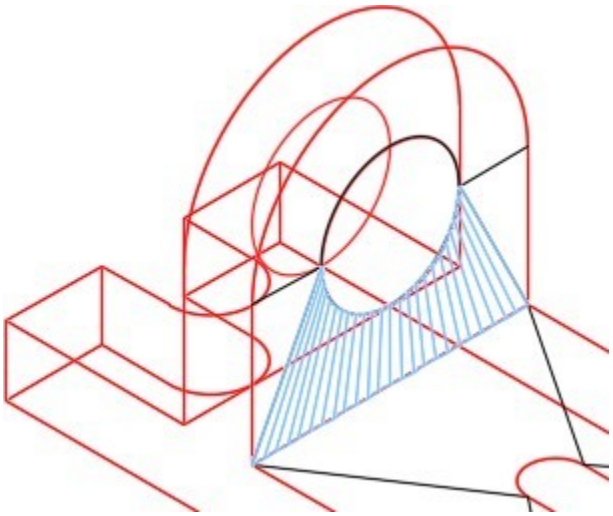


Figure Step 19

Step 20

On layer: Surface 1, insert the ruled surfaces as shown in the figure. (Figure Step 20)

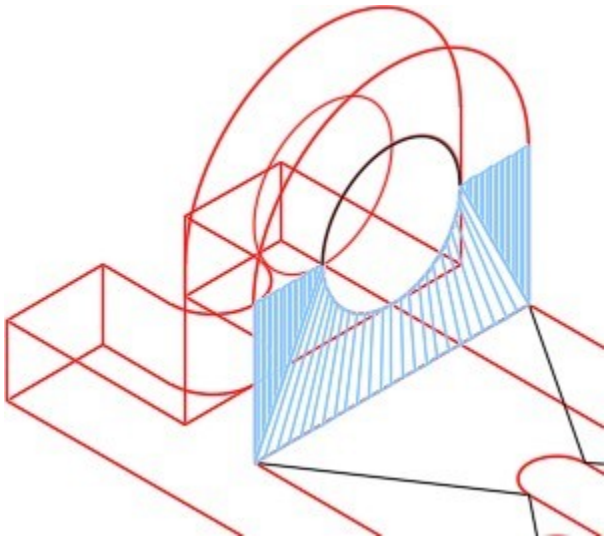


Figure Step 20

Step 21

Change the layer of the existing surfaces to layer: Surface 1 until your model matches the figure. (Figure Step 21)

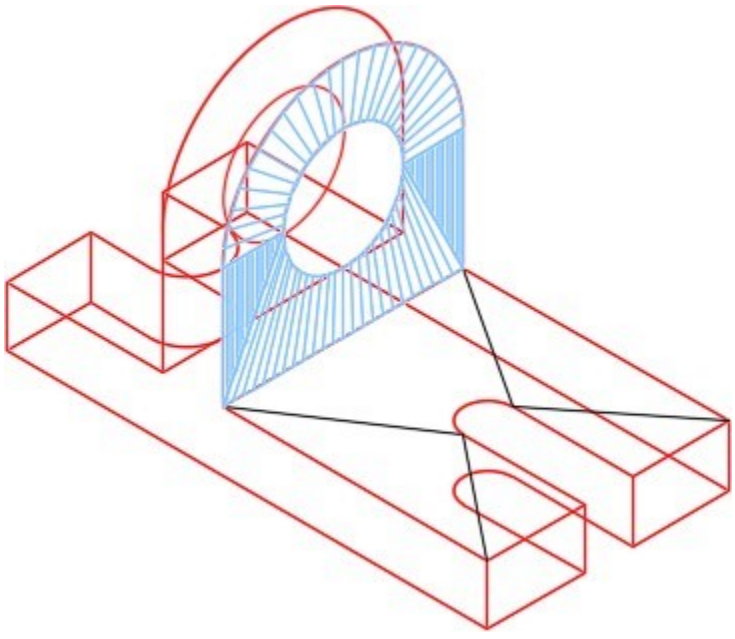


Figure Step 21

Step 22

Copy the surfaces from the front side of the model to back side. (Figure Step 22)

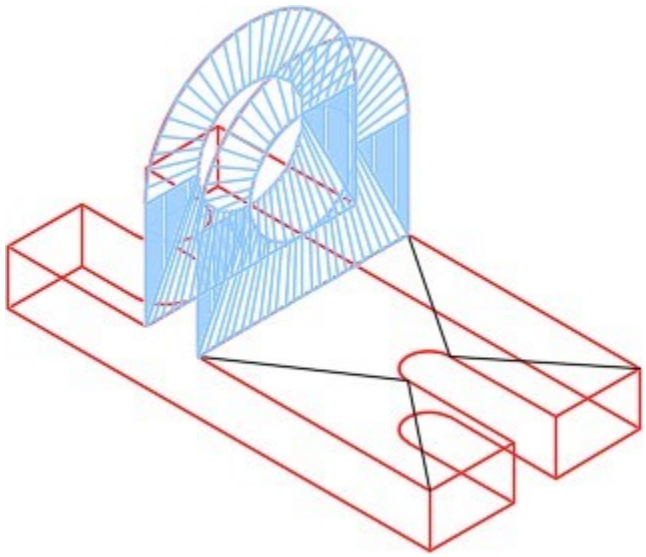


Figure Step 22

AUTHOR'S COMMENTS: To help you surface a model faster, you can copy surfaces from one side of a model to another, if they are identical.

Step 23

Using what was just taught, complete surfacing the model. Ensure to surface the bottom of the model. When complete, change all the surfaces from layers: Surface off to Surface 1.

Step 24

Turn off all the layers except layer: Surface 1 and the model should appear as shown in the figure. (Figure Step 24)

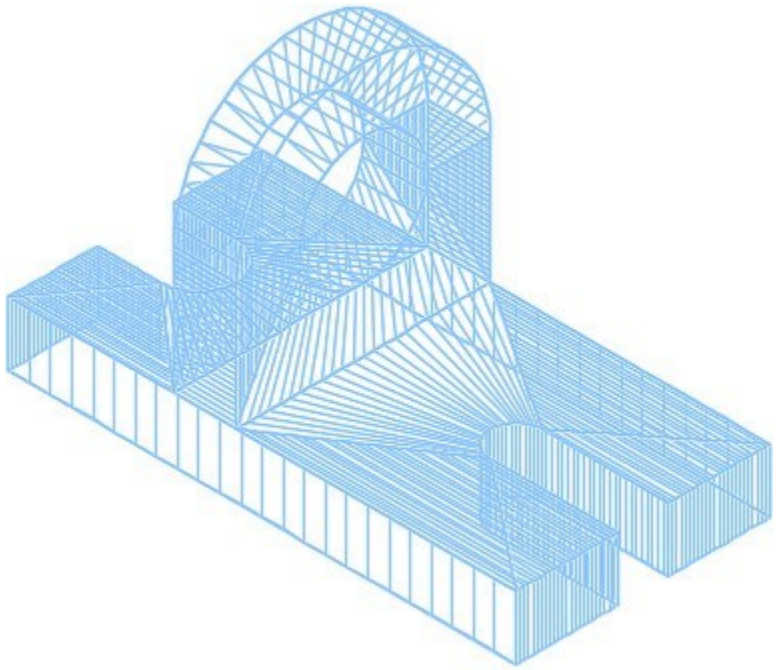


Figure Step 24

Step 25

Set the current visual style to Realistic. (Figure Step 25)

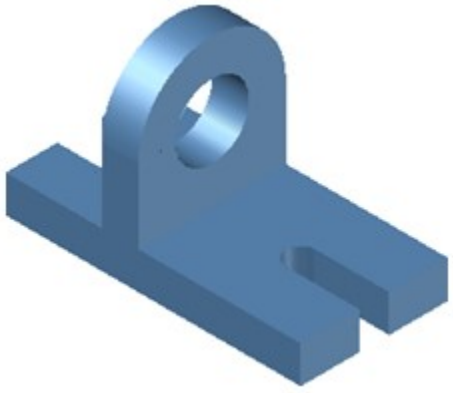


Figure Step 25

Step 26

Save and close the drawing.

AUTHOR'S COMMENTS: Do not be concerned if your shaded model does not match the figures in the module exactly. There are many visual style settings that can be set to change the look of the shaded model. For now, all you should be concerned about is being able to surface the model.

The advanced features of customizing visual styles are taught in the AutoCAD 3D Advanced book.

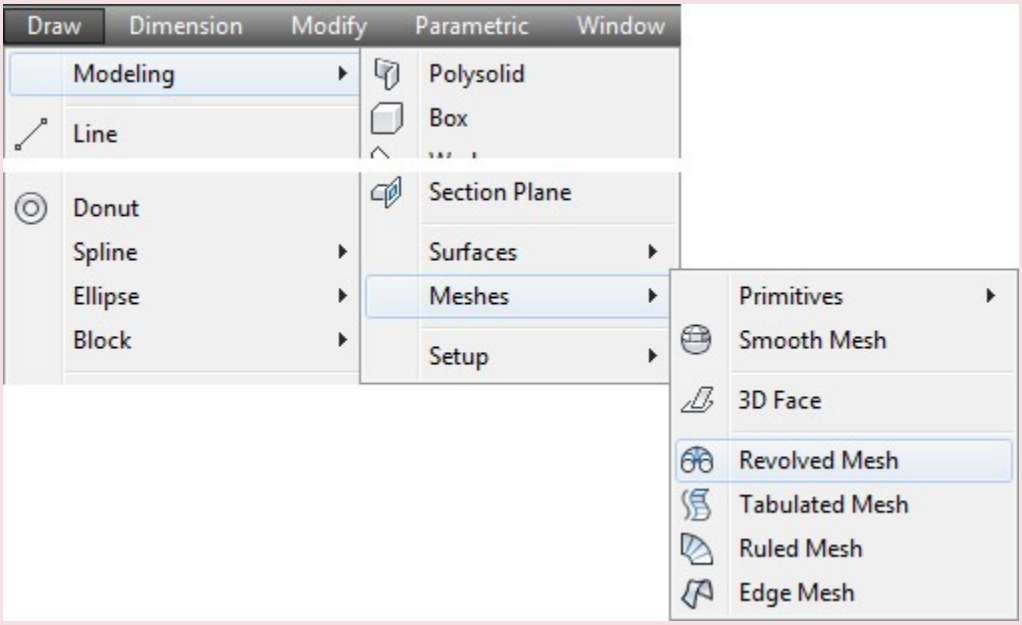
AutoCAD Command: REVSURF

The REVSURF command is used to create a surface of revolution by revolving an existing profile around an axis.

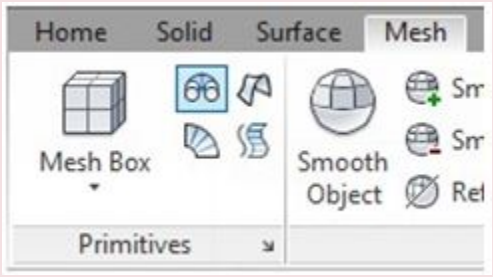
Shortcut: none

SURFTAB1 = YES

SURFTAB2 = YES



Draw Pull-down



Mesh Ribbon

WORK ALONG: Inserting Revolved Surfaces

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 13-2. (Figure Step 2)

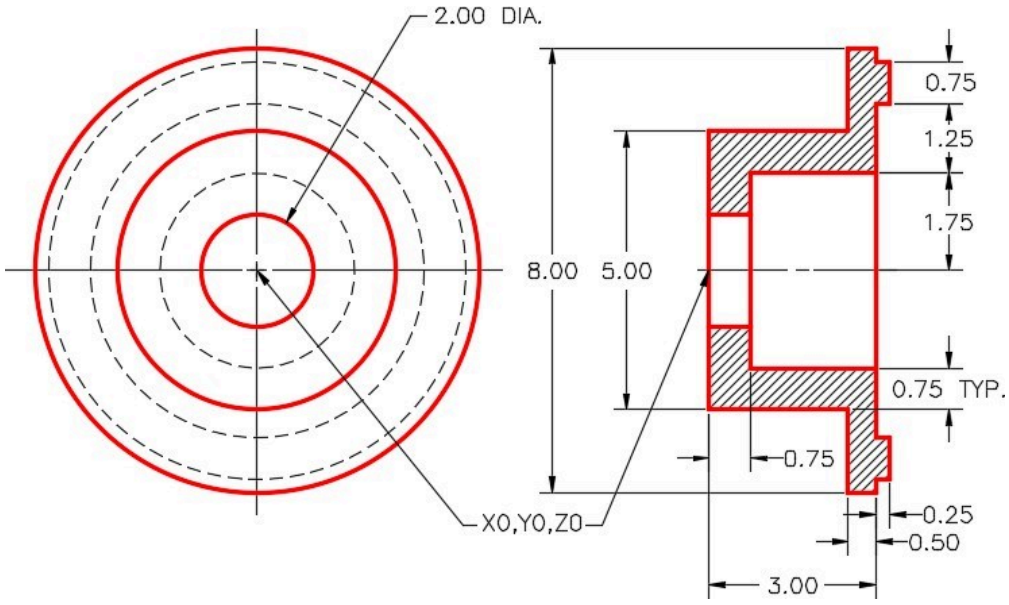


Figure Step 2
Dimensioned Multiview Drawing

Step 3

Set the current layer to: Pline and the current visual style to 2D Wireframe.

Step 4

Change the current view to SE Isometric and the current UCS to Right.

Step 5

Using the multiview drawing as a reference, draw the right side cross section of the solid part of the object. (Figure Step 5)

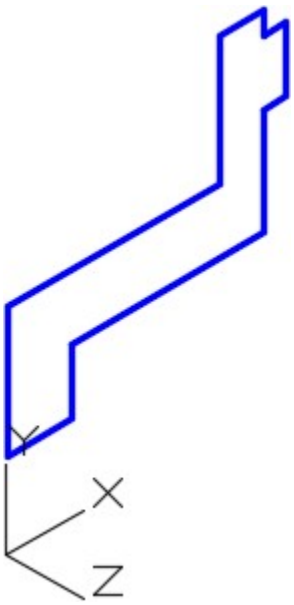


Figure Step 5

AUTHOR'S COMMENTS: You will be revolving this cross section 360 degrees to create a surface as it revolves. Therefore, only one-half of the section view is drawn.

Step 6

On layer: Construction, draw a line, of any length, from 0,0,0 along the X axis. Enable ortho mode to draw it faster. This is the centre of the model and will be used as the axis for the revolution. (Figure Step 6)

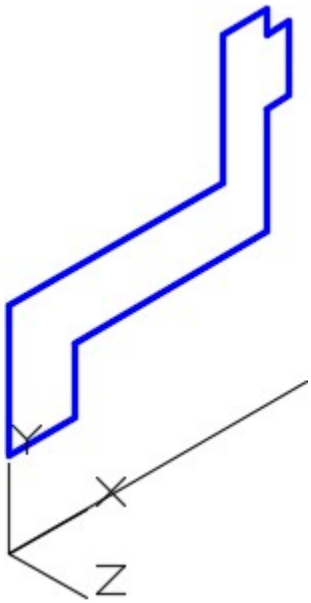


Figure Step 6

AUTHOR'S COMMENTS: To convert the model objects to a closed polyline, the drawing must be perfectly drawn with the endpoints joining exactly. If you have problems using the PEDIT command to create a closed polygon, correct the drawing or redraw the objects from scratch.

Step 7

Using the PEDIT command, as shown below, convert the lines of the model to a closed polyline.

Command: **PE**

Select polyline or [Multiple]:

(Select one line of the model.)

Object selected is not a polyline

Do you want to turn it into one? <Y>

(Press Enter to accept the default.)

Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: **J**

(J for Join.)

Select objects: **ALL**

13 found

Select objects:

11 segments added to polyline

Enter an option [Open/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]:

Command:

Step 8

To ensure that you have successfully created a closed polyline, open the Properties window and select the polyline. The object type should read Polyline, all the objects should show as part of the polyline and the Closed property should indicate Yes. (Figure Step 8)

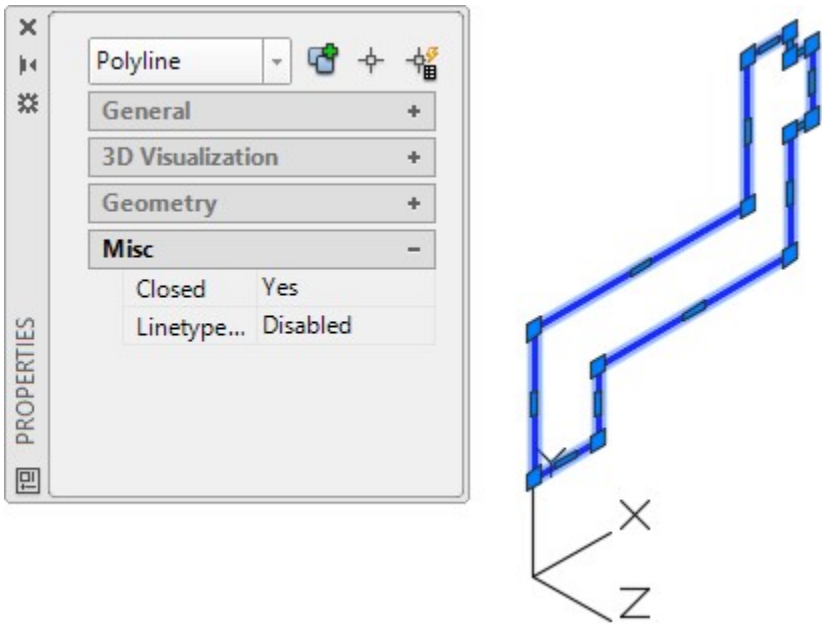


Figure Step 8

Step 9

Set the system variable SURFTAB1 to 64 and SURFTAB2 to 24 as shown below.

Command: **SURFTAB1**

Enter new value for SURFTAB1 <6>: **64**

Command: **SURFTAB2**

Enter new value for SURFTAB2 <6>: **24**

Command:

Step 10

Change the current layer to layer: Surface 1 and enter the REVSURF command as shown below. After you complete the command, your model should appear as shown in the figure. (Figure Step 10)

Command: **REVSURF**

Current wire frame density: SURFTAB1=64 SURFTAB2=24

Select object to revolve

(Select the closed polyline.)

Select object that defines the axis of revolution:

(Select the axis line.)

Specify start angle <0>:

(Accept the default of 0.)

Specify included angle (+=ccw, -=cw) <360>:

(Accept the default of 360.)

Command:

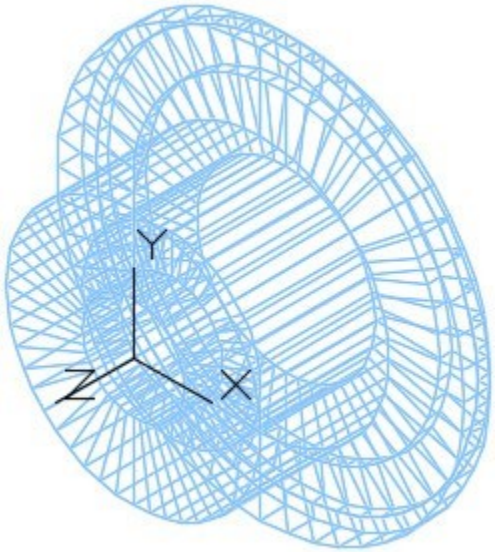


Figure Step 10

Step 11

Turn layers: Construction and Pline off and set the current visual style to Realistic.

Step 12

Using the 3DFORBIT command, orbit the model and look at all sides. (Figure 12A and 12B)

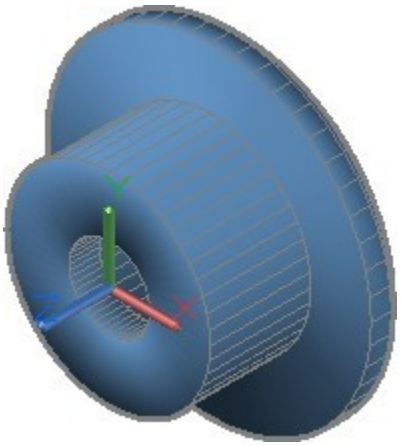


Figure Step 12A
SE Isometric View

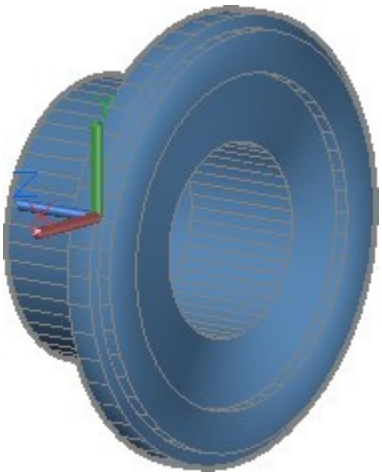


Figure Step 12B
SE Isometric View

Step 13

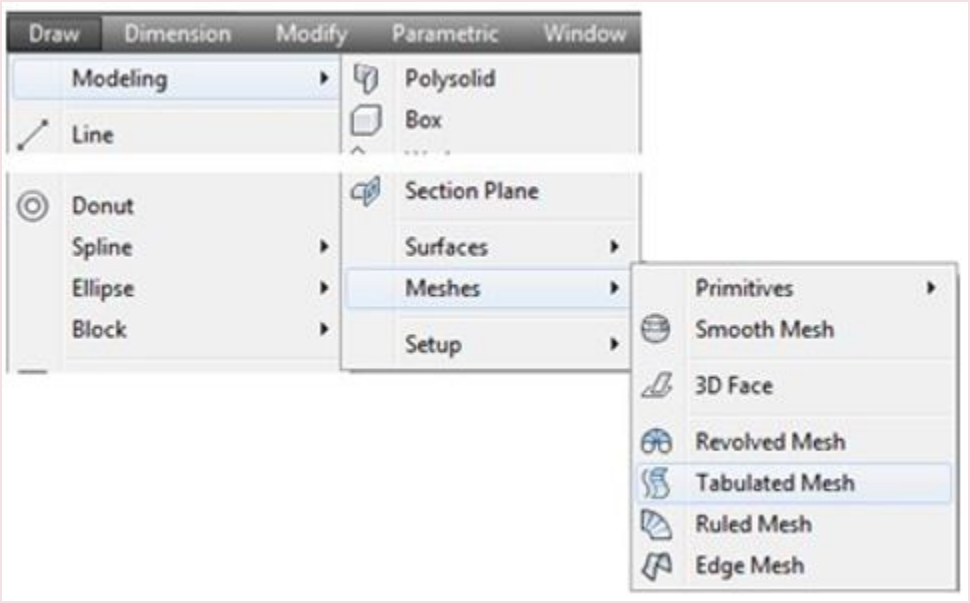
Save and close the drawing.

AutoCAD Command: TABSURF

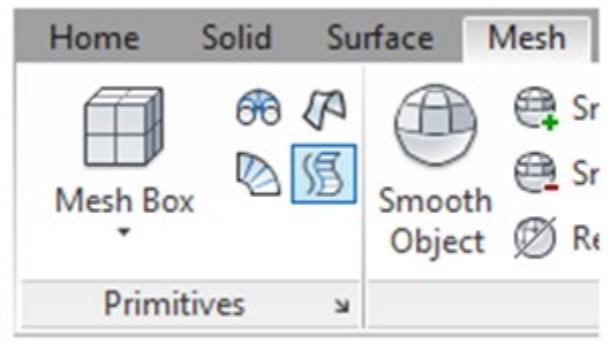
The TABSURF command is used to create a surface mesh moving a profile along a path. Shortcut: none

SURFTAB1 = YES

SURFTAB2 = No Effect



Draw Pull-down



Mesh Ribbon

WORK ALONG: Inserting Tabulated Surfaces

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 13-3.

Step 3

Set the current visual style to 2D Wireframe, the current view to SE Isometric, and the current UCS to World.

Step 4

Set layer: Construction as the current layer. Draw the model shown in the dimensioned model. These are the construction lines and arcs to be used to construct the model. When complete, your model will appear as shown in the figure. The length of the lines that appear dashed in the dimensioned model are not important since they are axis lines only. They are always drawn in the Z direction of the current UCS. They can be drawn any length as continuous linetype. (Figure Step 4A, 4B, and 4C)

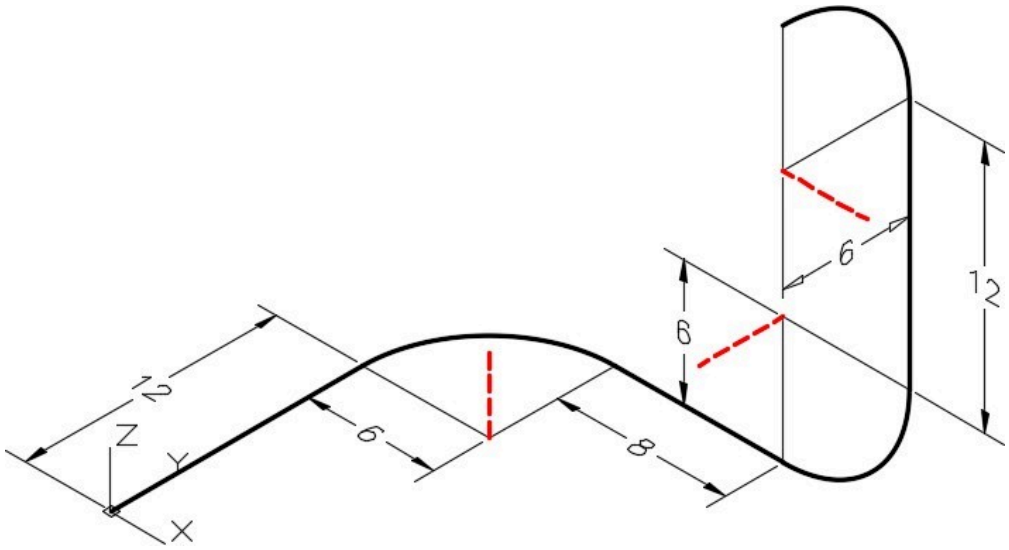


Figure Step 4A Dimensioned Model SE Isometric View

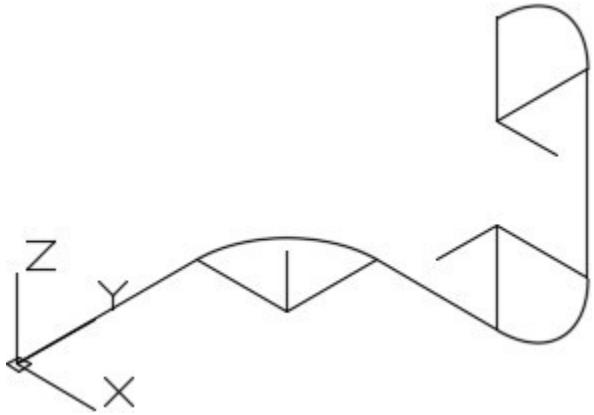


Figure Step 4B
SE Isometric View

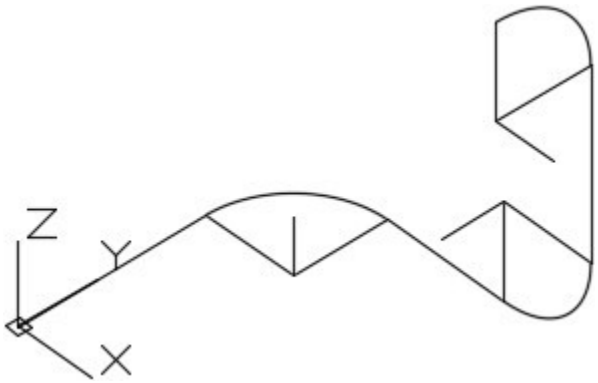


Figure Step 4C
Orbited View

AUTHOR'S COMMENTS: You will have to change the current UCS as you draw the construction lines to ensure that you are drawing on the correct plane.

Step 5

Set the current UCS to Front and then draw a model of the object shown in the dimensioned drawing. The drawing should appear as shown in the figure. Draw the model anywhere in model space. Its location is not important. (Figure Step 5A and 5B)

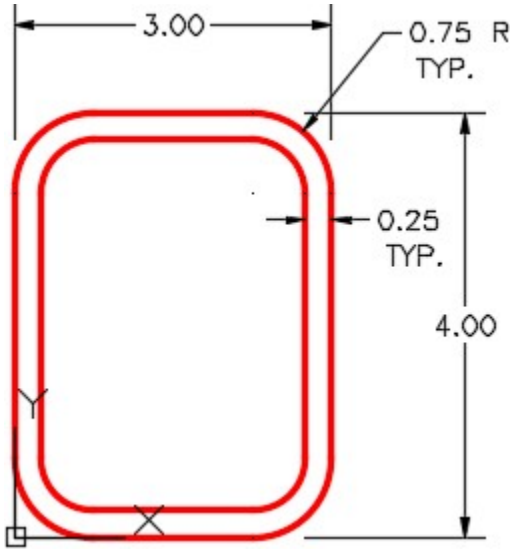


Figure Step 5A

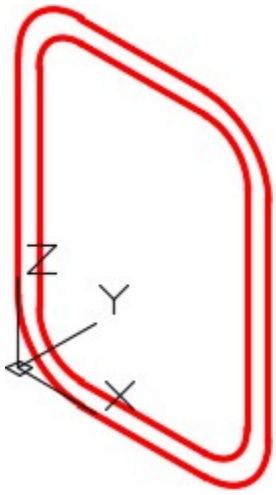


Figure Step 5B

Step 6

Change the lines and arcs into two closed polylines using the PEDIT command. Check them using the Properties window. (Figure Step 6)

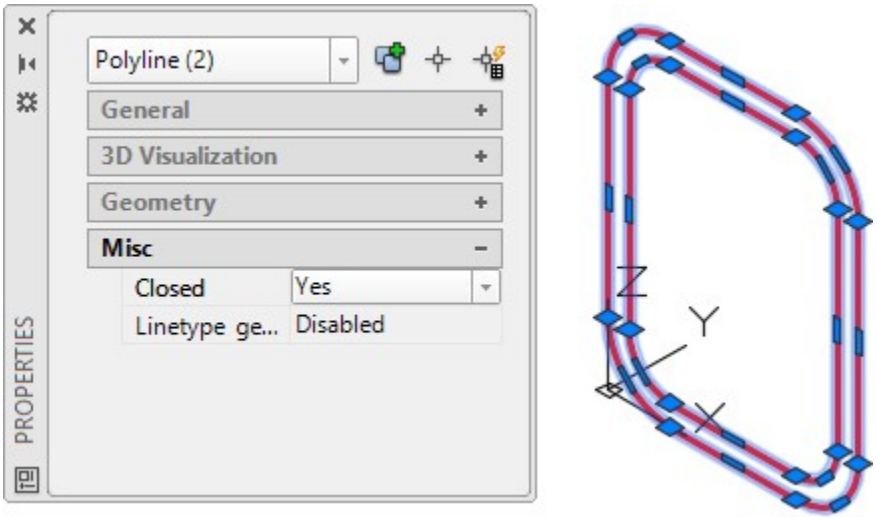


Figure Step 6

Step 7

On layer: Construction, draw in two construction lines from midpoint of the lines to find the exact centre of the object. (Figure Step 7)

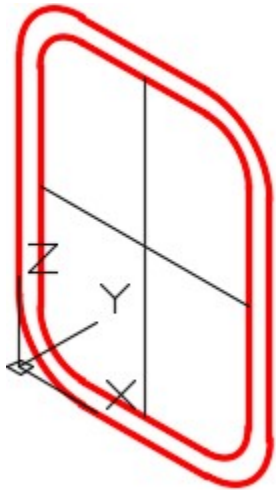


Figure Step 7

Step 8

Make three additional copies of the model and rotate them to lay on different UCS planes. Their location in 3D space is not important. (Figure Step 8)

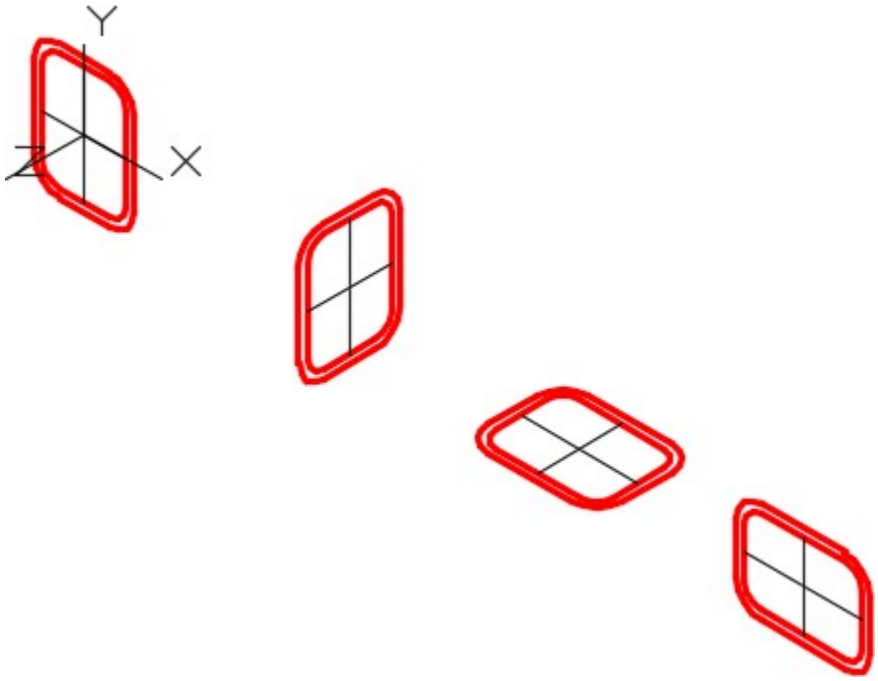


Figure Step 8

Step 9

Your drawing should appear similar to the figure. (Figure Step 9)

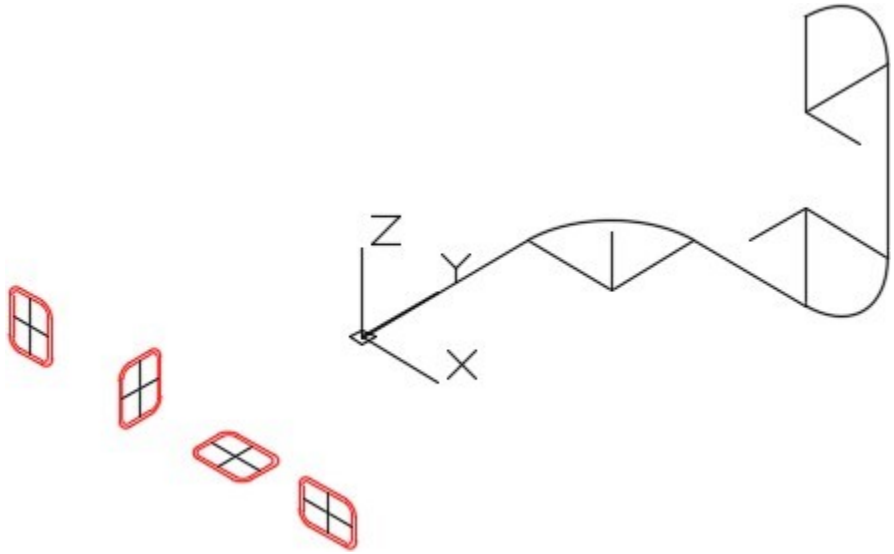


Figure Step 9

Step 10

Copy the objects onto the construction lines. Ensure that the midpoint of the object (the intersections of the construction lines) aligns to the endpoints of the lines and arcs. (Figure Step 10)

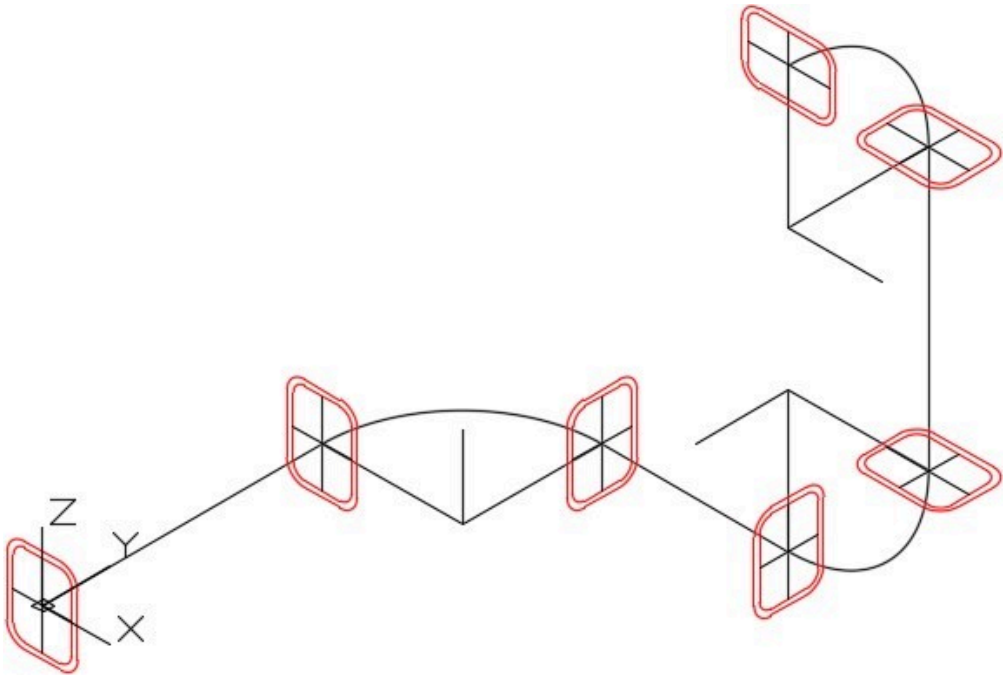


Figure Step 10

Step 11

With the UCS located on the Front, SURFTAB1 set to 48 and layer: Surface 1 as the current layer, enter the TABSURF command, as shown below, to create the surface for the inside of the tube. (Figure Step 11)

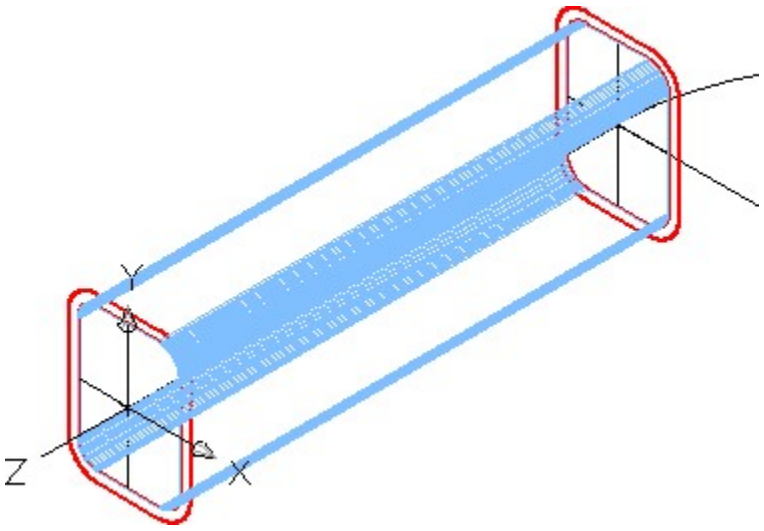


Figure Step 11

Command: **TABSURF**

Current wire frame density: SURFTAB1=48

Select object for path curve:

(Select the inside polyline.)

Select object for direction vector:

(Select the construction axis line. Ensure you select it closer to the end where the object your are tabulating is located.)

Command:

Step 12

Set the current visual style to Realistic. (Figure Step 12)

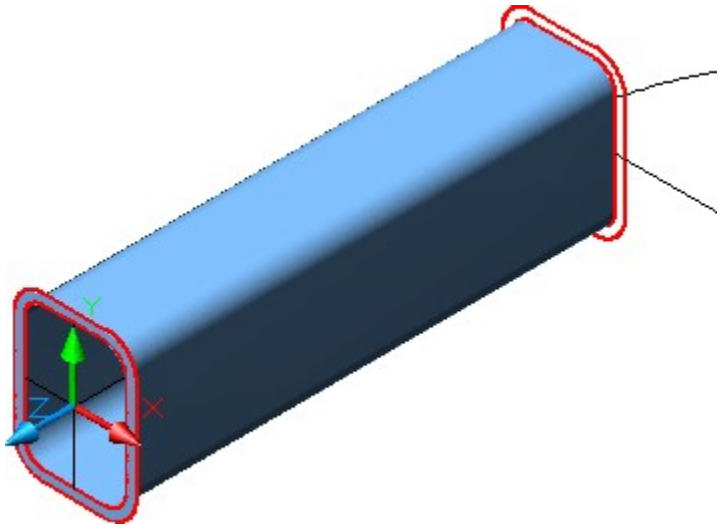


Figure Step 12

Step 13

Using the TABSURF command, create the surface for the outside of the tube. (Figure Step 13)

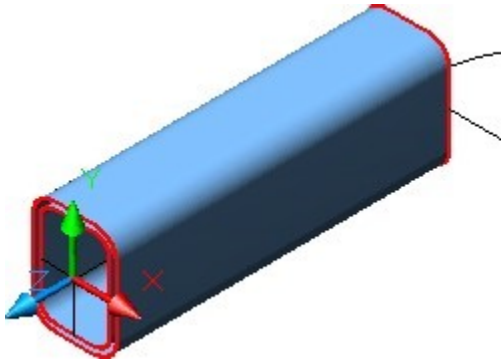


Figure Step 13

Step 14

Enter the REVSURF command, as shown below, to construct the inside of the tube around the first corner. First change the layer of the straight tube surfaces constructed in Steps 11 to 13 to layer: Surface Off. (Figure Step 14)

Command: **REVSURF**

Current wire frame density: SURFTAB1=48 SURFTAB2=6

Select object to revolve:

(Select the inside polyline.)

Select object that defines the axis of revolution

(Select the axis line.)

Specify start angle <0>:

(Accept the default.)

Specify included angle (+=ccw, -=cw) <360>: **-90**

Command:

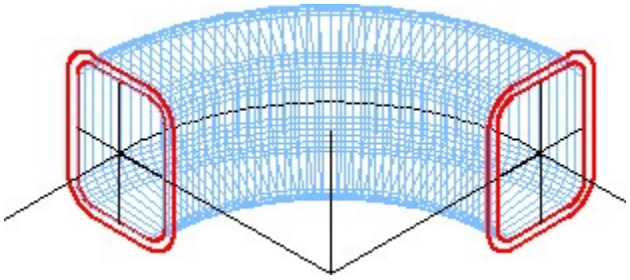


Figure Step 14

Step 15

Using what was just taught, construct the outside of the tube and change the layers of the surfaces until the model appear as shown in the figure. (Figure Step 15)

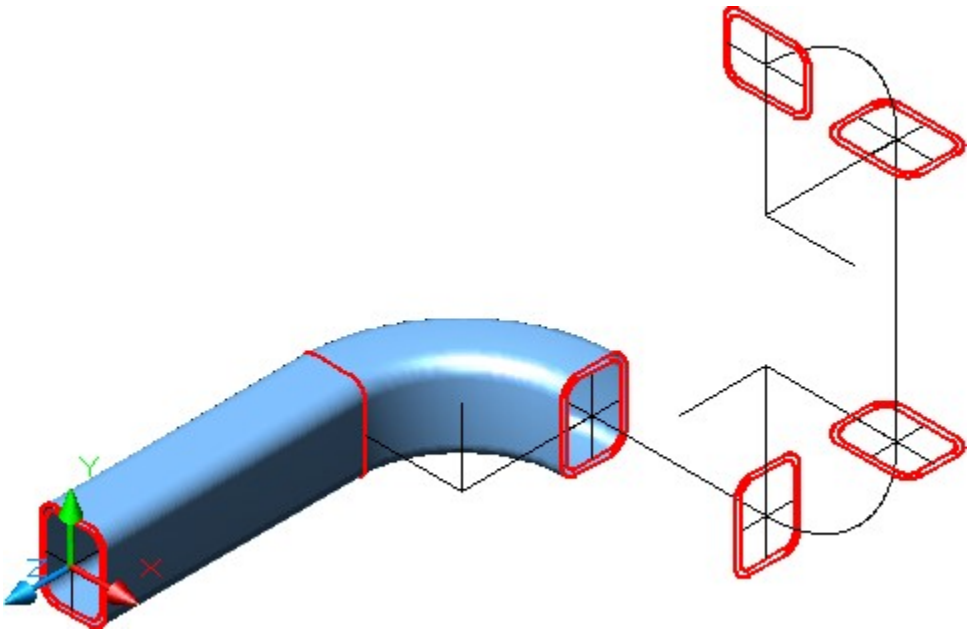


Figure Step 15

Step 16

Using what was just taught, complete the model. (Figure Step 16)

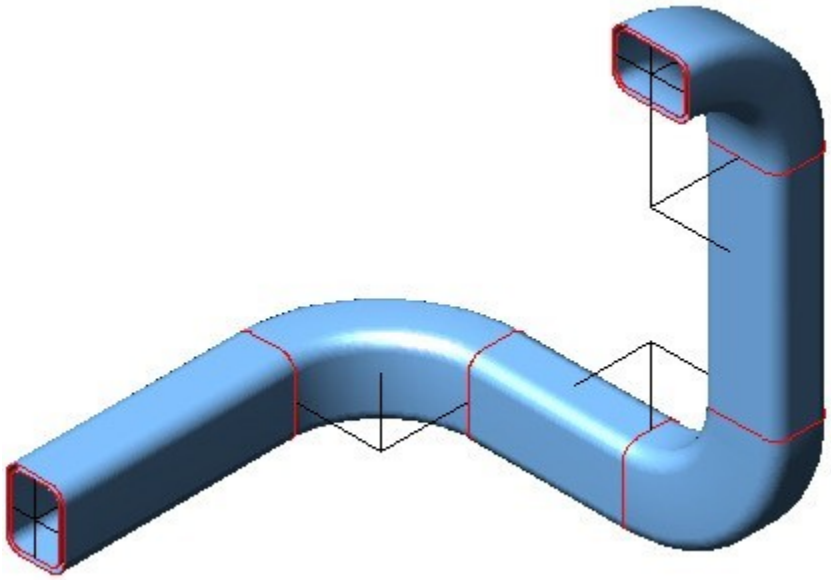


Figure Step 16

Step 17

Turn off layers: Model and Construction and your completed model will appear as shown in the figure. (Figure Step 17)

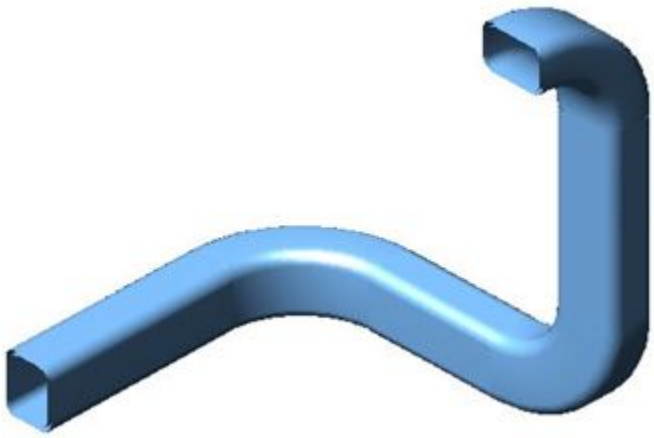


Figure Step 17

Step 18

Save and close the drawing.

AUTHOR'S COMMENTS: Do not be concerned if your shaded model does not match the figures in the module exactly. There are many visual style settings that can be set to change the look of the shaded model. For now, all you should be concerned about is being able to surface the model. The advanced features of customizing visual styles are taught in the AutoCAD 3D Advanced book.

USER TIP: Although any surfacing command can be used to create the surfaces, the RULESURF command is the easiest and fastest to use for most surfaces. To shorten the modeling time, use it whenever possible.

Key Principles

Key Principles in Module 13

1. Geometrically defined surface meshes use existing geometry that must be created before the surfaces. In almost all cases, you will use the wireframe of the model as the existing geometry.
2. Although any surfacing command can be used to create a surface, the RULESURF command is the easiest and fastest to use for most surfaces.
3. When using the RULESURF command, ensure that you select the objects closest to matching If opposite ends are selected, the mesh will twist.
4. The mesh density represented by the lines, circles or arcs that make the rows and columns of the surface mesh are controlled with the system variables SURFTAB1 and SURFTAB2.

Lab Exercise 13-1

Time allowed: 45 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 13-1	3D English	Inches

Step 1

Draw a wireframe of the object shown in the figure. (Figure Step 1)

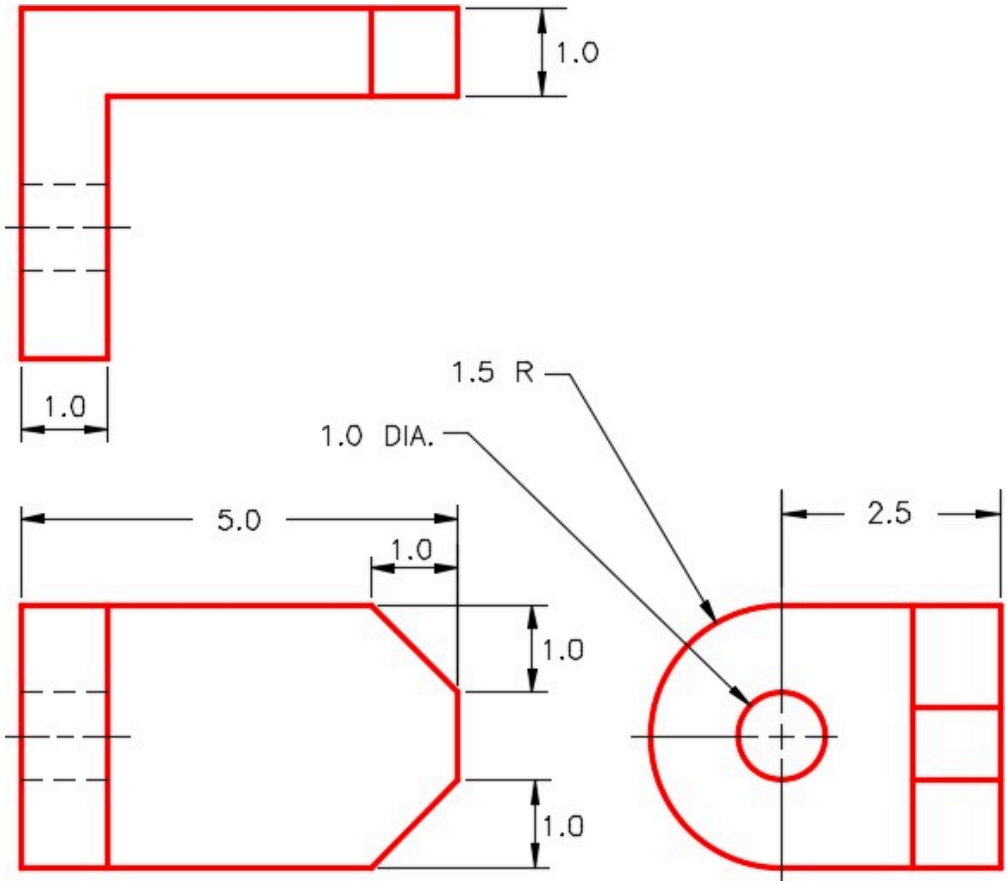


Figure Step 1
Dimensioned Multiview Drawing

Step 2

Pick appropriate mesh densities and on layer: Surface 2, create surface meshes on all surfaces including back and bottom. (Figure Step 2)

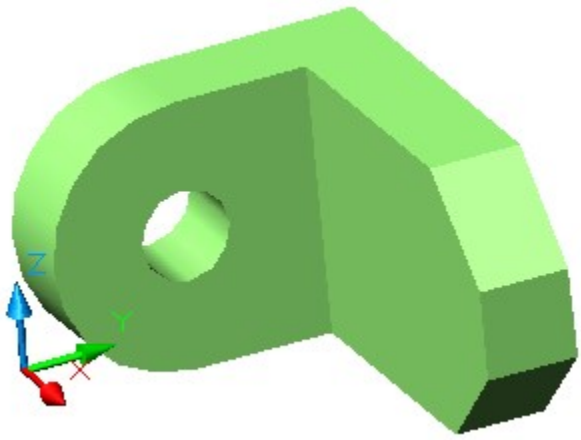


Figure Step 2
Completed Model SE Isometric View

Step 3

Freeze layers: Construction and Model.

Step 4

Set the current visual style to Realistic. Use the 3DORBIT command to check that the model is surfaced on all sides.

Step 5

Set the Insertion Units, change the current UCS to World and check the model with the key.

Step 6

Save and close the drawing.

Lab Exercise 13-2

Time allowed: 45 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 13-2	N/A	Inches

Step 1

Open the drawing: AutoCAD 3D Lab 04-1.

Step 2

Save the drawing with the name: AutoCAD 3D Lab 13-2.

Step 3

On layer: Surface 5, create surface meshes on all surfaces including back and bottom. (Figure Step 3)

Step 4

Freeze layers: Construction and Model.

Step 5

Set the current visual style to Realistic. Use the 3DFORBIT command to check that the model is surfaced on all sides.

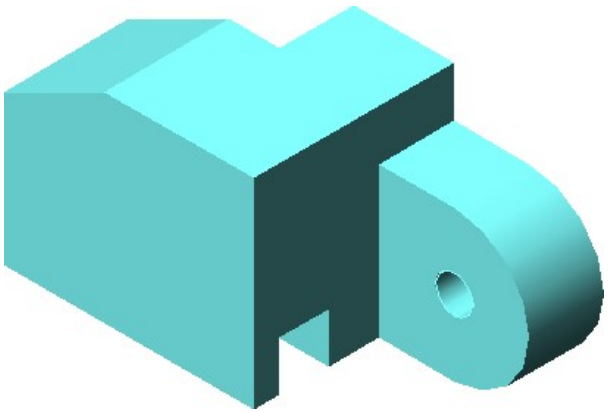


Figure Step 3 Completed Model SE Isometric View

Step 6

Save and close the drawing.

AUTHOR'S COMMENTS: Do not be concerned if your shaded model does not match the figures in the module exactly. There are many visual style settings that can be set to change the look of the shaded model. For now, all you should be concerned about is being able to surface the model.The advanced features of customizing visual styles are taught in the AutoCAD 3D Advanced book.

Lab Exercise 13-3

Time allowed: 45 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 13-3	N/A	Inches

Step 1

Open the drawing: AutoCAD 3D Lab 08-1.

Step 2

Save the drawing with the name: AutoCAD 3D Lab 13-3.

Step 3

On layer: Surface 1, create surfaces on each side including back and bottom. (Figure Step 3)

Step 4

Freeze layers: Construction and Model.

Step 5

Set the current visual style to Realistic. Use the 3DFORBIT command to check that the model is surfaced on all sides.

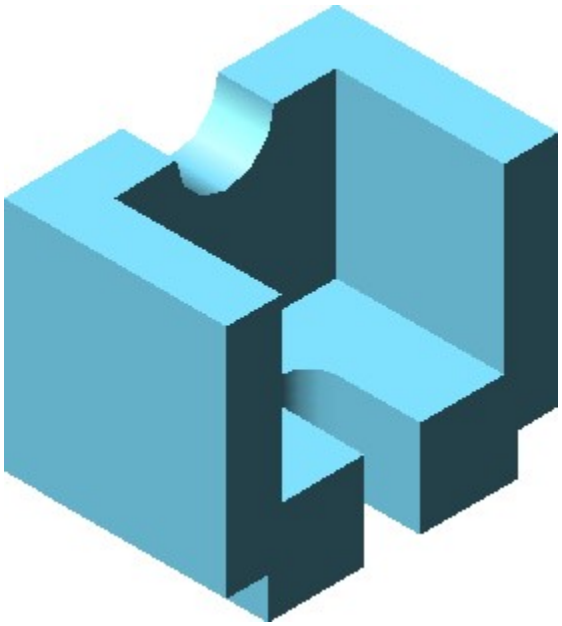


Figure Step 3 Completed Model SE Isometric View

Step 6

Save and close the drawing.

AUTHOR'S COMMENTS: Do not be concerned if your shaded model does not match the figures in the module exactly. There are many visual style settings that can be set to change the look of the shaded model. For now, all you should be concerned about is being able to surface the model.The advanced features of customizing visual styles are taught in the AutoCAD 3D Advanced book.

Lab Exercise 13-4

Time allowed: 45 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 13-4	3D Layout Metric	Millimeters

Step 1

On layer: Pline, draw the cross section of the top half of the model on the right side UCS. (Figure Step 1A and 1B)

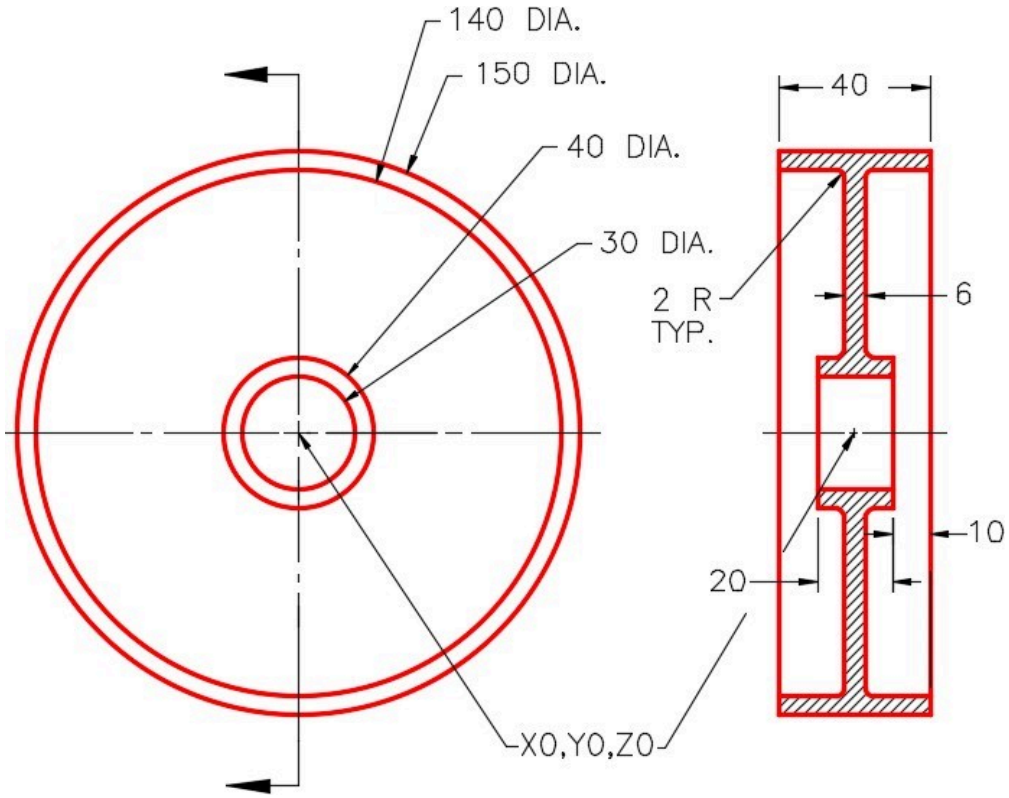


Figure Step 1A
Dimensioned Multiview Drawing

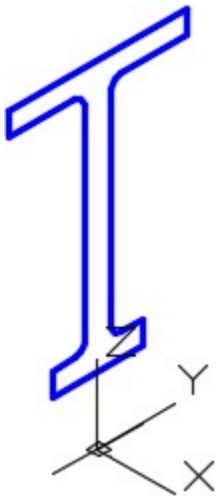


Figure Step 1B
View Of a Cross
Section to Revolve

Step 2

Create a surface revolution on layer: Surface 1. (Figure Step 2)

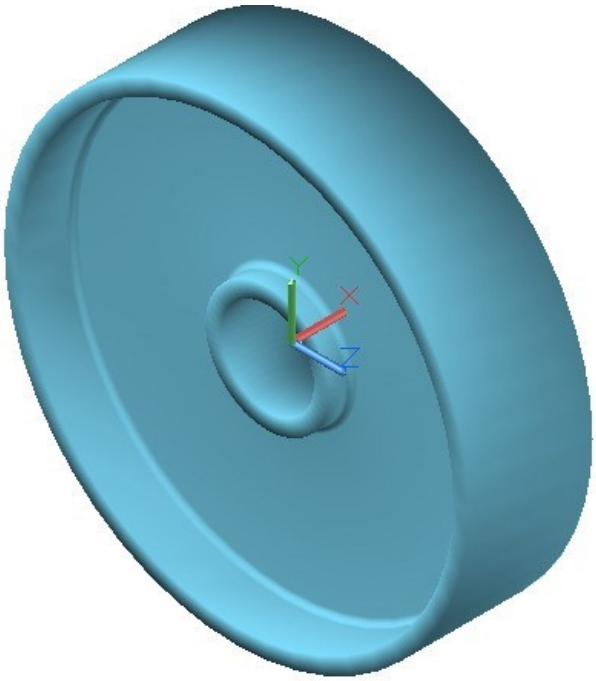


Figure Step 2 Complete Model SE Isometric View

Step 3

Freeze layers: Pline and Construction.

Step 4

Set the current visual style to Realistic. Use the 3DORBIT command to check that the model is surfaced on all sides.

Step 5

Set the Insertion Units, change the current UCS to World and check the model with the key.

Step 6

Save and close the drawing.

AUTHOR'S COMMENTS: Do not be concerned if your shaded model does not match the figures in the module exactly. There are many visual style settings that can be set to change the look of the shaded model. For now, all you should be concerned about is being able to surface the model. The advanced features of customizing visual styles are taught in the AutoCAD 3D Advanced book.

Module 14: Surface Modeling – Part 3

Learning Outcomes

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

1. Describe a complex surface mesh and a region.
2. Apply the 3DMESH, REGION, and SUBTRACT commands.

Drawing Complex Surface Meshes

When an irregular complex mesh cannot be drawn using a geometrically generated surface, the 3DMESH command can be used to create it. The 3DMESH command uses a grid made of XYZ coordinates to generate the surface mesh. This command is used extensively when creating 3D topographical maps. It is also used in automobile and airplane design.

AutoCAD Command: 3DMESH

The 3DMESH command is used to create an irregular complex surface mesh from a grid of XYZ coordinates.

Shortcut: none

WORK ALONG: Drawing 3D Surface Meshes

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 14-1.

Step 3

Set layer: Model as the current layer.

Step 4

Set the current visual style to 2D Wireframe, the current UCS to World, and the current view to SE Isometric. (Figure Step 4)

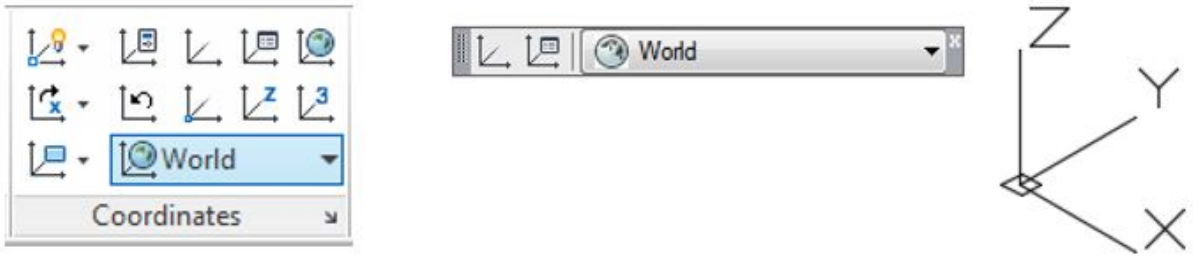


Figure Step 4

Step 5

Draw a 5 X 4 grid as shown in the figure. Draw the grid lines 1 inch apart along both the X and Y axis. (Figure Step 5)

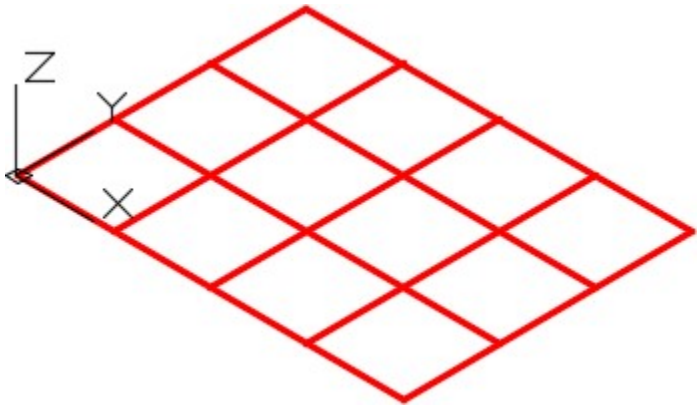


Figure Step 5

AUTHOR'S COMMENTS: The 3DMESH command is used to create an irregular complex surface mesh. It is created from XYZ coordinates created on an XY mesh. The XY mesh can be any shape but must lay on a single plane. You will be prompted for an M and an N value. The M value is the number of columns located on the X axis and the N value is the number of rows located on the Y axis. The tighter the mesh, the smoother the surface will be. Enter the coordinates carefully as there is no undo feature. If you make an input error, you will have to start the command over again.

Step 6

Set layer: Surface 1 as the current layer and using Figure Step 6A as a reference, enter the 3D MESH command, as shown below, to create the 3D mesh. When complete, your drawing should appear as shown in the figure. (Figure Step 6A and 6B)

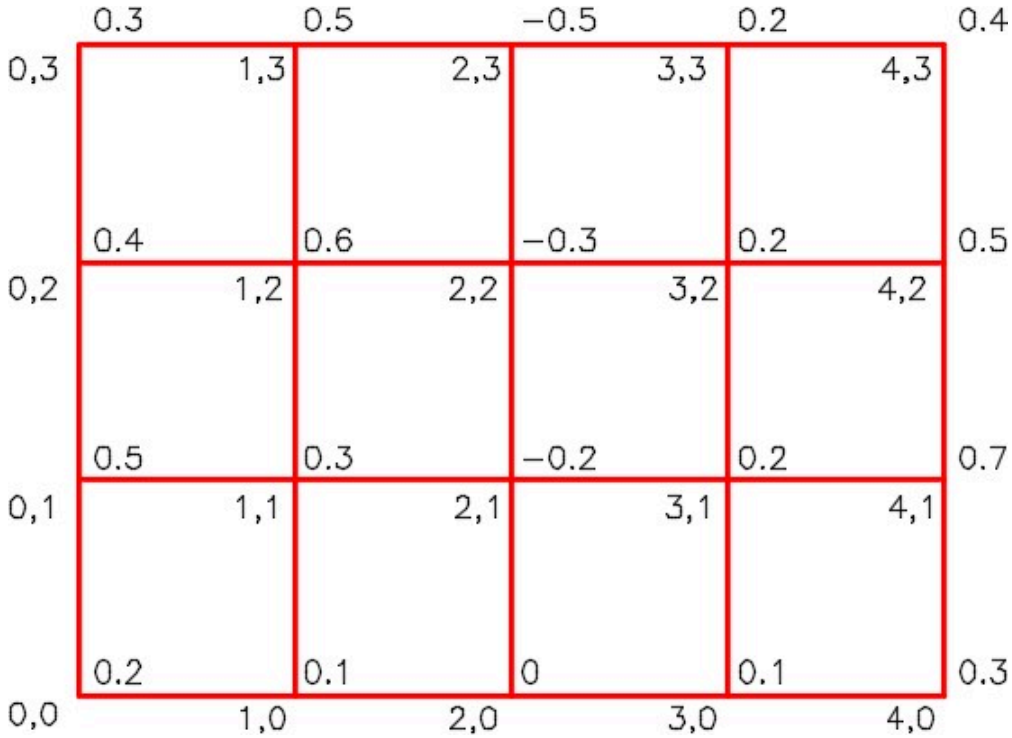


Figure Step 6A

4Figure Step 6BCommand: **3DMESH**

Enter size of mesh in M direction: **5**

(5 columns in the X direction. Count the grid lines not the spaces.)

Enter size of mesh in N direction: **4**

(4 rows in the Y direction. Count the grid lines, not the spaces.)

Specify location for vertex (0, 0): **0,0,0.2**

(Enter the XYZ coordinates stating at bottom left corner which is X0Y0Z0.2. Then work up the Y axis to the top and then jump back down to X1,Y0.)

Specify location for vertex (0, 1): **0,1,0.5**

(The 0.5 is entered here for clarity. If you enter .5 it will do the same thing.)

Specify location for vertex (0, 2): **0,2,0.4**

Specify location for vertex (0, 3): **0,3,0.3**

Specify location for vertex (1, 0): **1,0,0.1**

Specify location for vertex (1, 1): **1,1,0.3**

Specify location for vertex (1, 2): **1,2,0.6**

Specify location for vertex (1, 3): **1,3,0.5**

Specify location for vertex (2, 0): **2,0,0**

Specify location for vertex (2, 1): **2,1,-0.2**

Specify location for vertex (2, 2): **2,2,-0.3**

Specify location for vertex (2, 3): **2,3,-0.5**

Specify location for vertex (3, 0): **3,0,0.1**

Specify location for vertex (3, 1): **3,1,0.2**

Specify location for vertex (3, 2): **3,2,0.2**

Specify location for vertex (3, 3): **3,3,0.2**

Specify location for vertex (4, 0): **4,0,0.3**

Specify location for vertex (4, 1): **4,1,0.7**

Specify location for vertex (4, 2): **4,2,0.5**

Specify location for vertex (4, 3): **4,3,0.4**

Command:

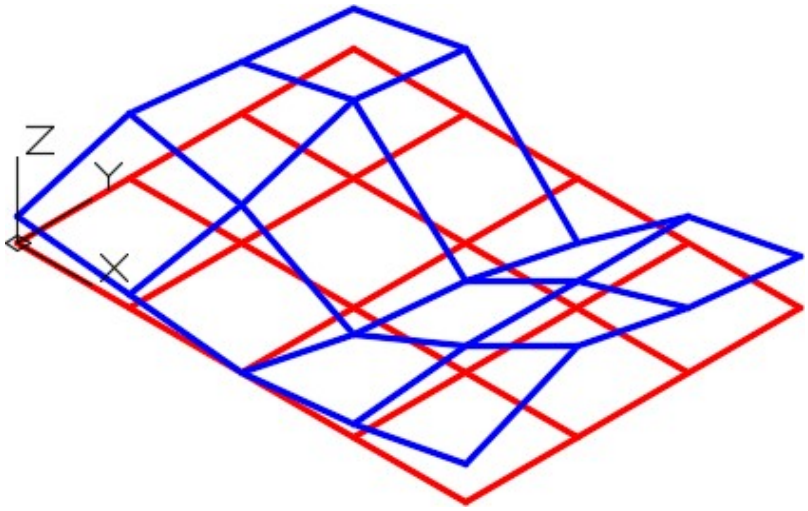


Figure Step 6B

Step 7

Turn layer Model off and set the current visual style to: Realistic. Your model should appear similar to the figure. (Figure Step 7)

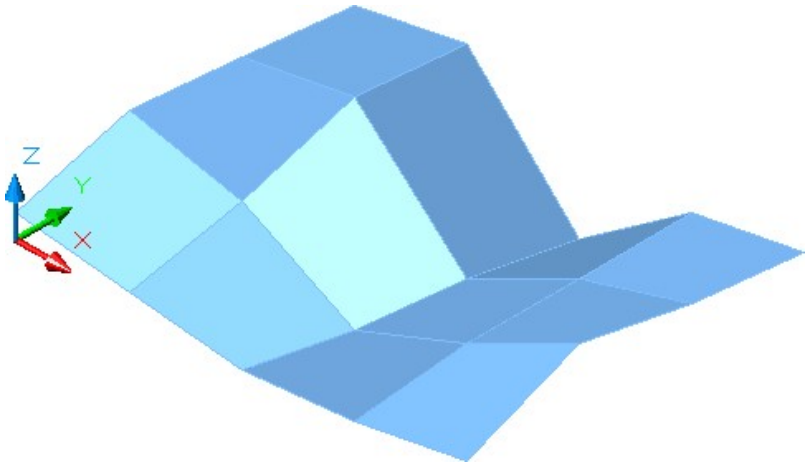


Figure Step 7

AUTHOR'S COMMENTS: Since the 3DMESH is difficult to work with, it is sometimes easier to draw lines representing the Z coordinate. You can then snap to Z coordinate points rather then entering them on the keyboard.

Step 8

Turn layer: Surface 1 off and layer: Model on. Set the current layer to: Model. With the UCS set to World, draw lines from the each grid vertex in the Z distance. See Figure Step 6A for the Z value. Be careful as some are positive and some are negative. When the Z is 0, you do not have to draw anything. For example, the line at X0Y0 should be drawn 0.2 inches in the positive Z direction. When complete, the model should appear as shown in figure. (Figure Step 8)

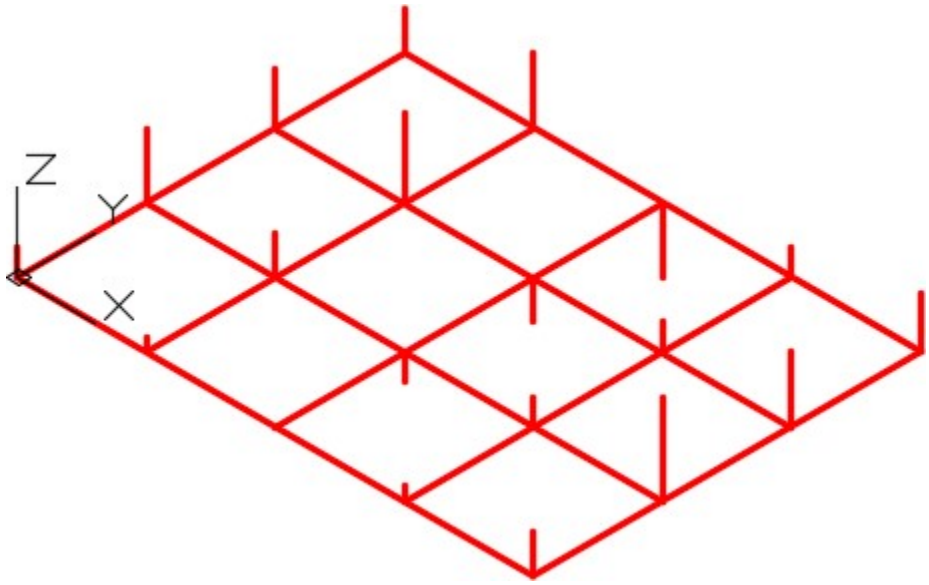


Figure Step 8

Step 9

Set layer: Surface 2 as the current layer and enter the 3DMESH as shown below. (Figure Step 9)

Command: **3DMESH**

Enter size of mesh in M direction: **5**

Enter size of mesh in N direction: **4**

Specify location for vertex (0, 0): *(end)* **P1**

(With osnap enabled, snap to the end of the Z line. You may have to zoom in to ensure you are snapping to the correct end of the line.)

Specify location for vertex (0, 1): *(end)* **P2**

Specify location for vertex (0, 2):*(end)* **P3**

Specify location for vertex (0, 3):*(end)* **P4**

Specify location for vertex (1, 0):*(end)* **P5**

Specify location for vertex (1, 1):*(end)* **P6**

(Continue until you complete the grid.)

Command:

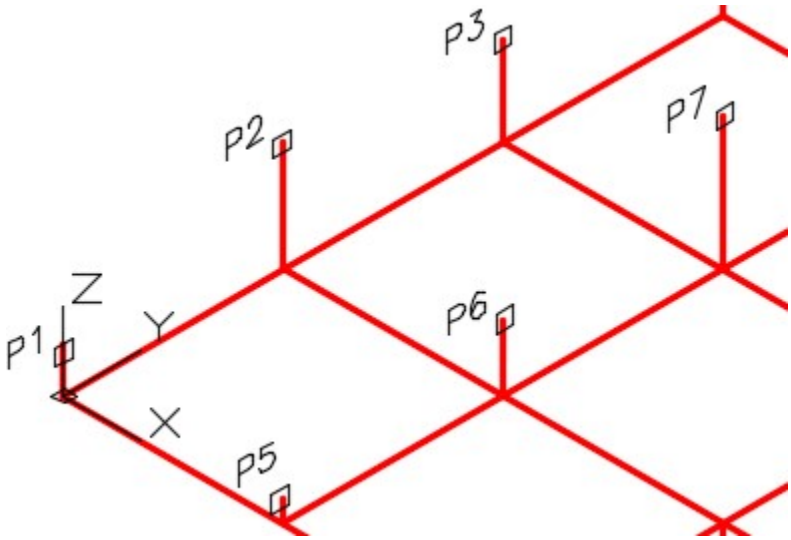


Figure Step 9

Step 10

Set the current visual style to: Realistic. Your model should appear similar to the figure. (Figure Step 10)

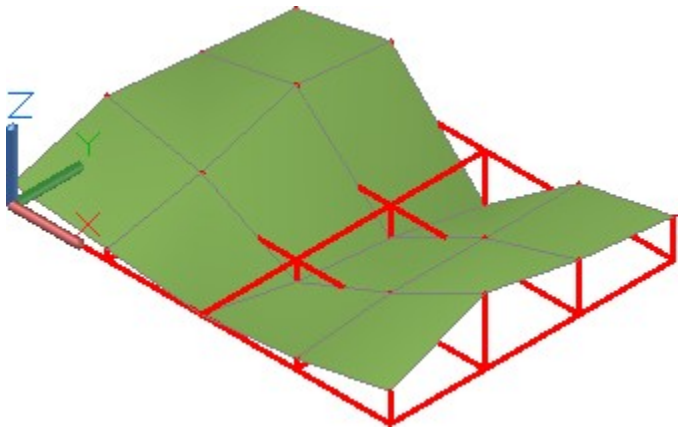


Figure Step 10

Step 11

Save and close the drawing.

Regions

A *region* is a 2D solid and is created using a closed polyline, circle, ellipse or spline as the boundary. A region must lie on a 2D plane. See Figure 14-1.

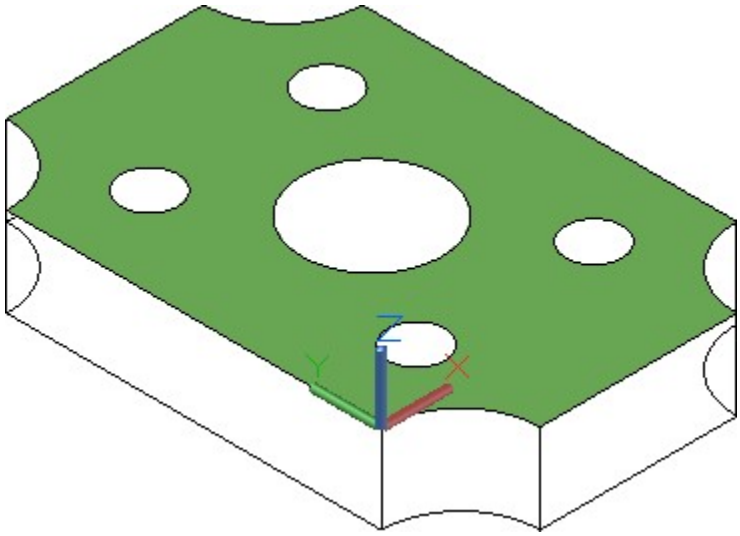


Figure 14-1
A Region

While regions are not technically surfaces, they will shade and can be rendered, therefore, can be used to surface a model.

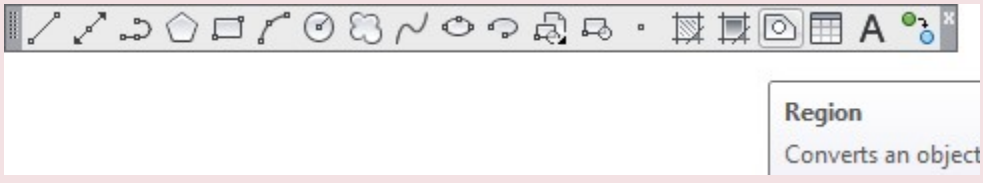
If the system variable DELOBJ is set to 1, AutoCAD will delete the closed object that was used as the boundary to create the region. In most cases, it is important for you to set this variable to 0 to retain the original object used to create the region.

MUST KNOW: If the system variable DELOBJ is set to 1, the REGION command will delete the closed object that was used as the boundary to create the region. In most cases, it is important to set this variable to 0 to retain the original object(s).

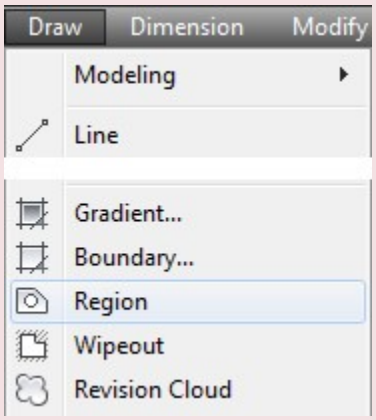
AutoCAD Command: REGION

The REGION command is used to create a 2D solid within a closed object.

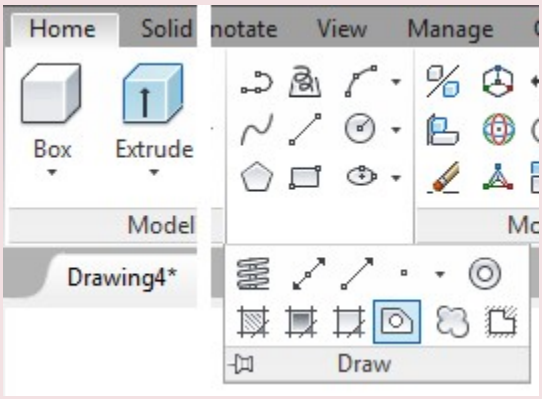
Shortcut: **REG**



Draw Toolbar



Draw Pull-down

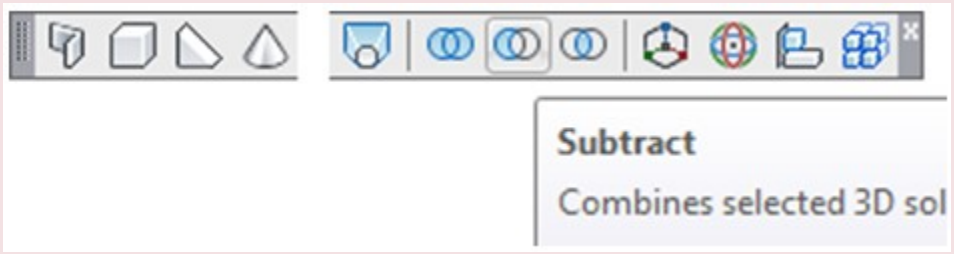


Home Ribbon

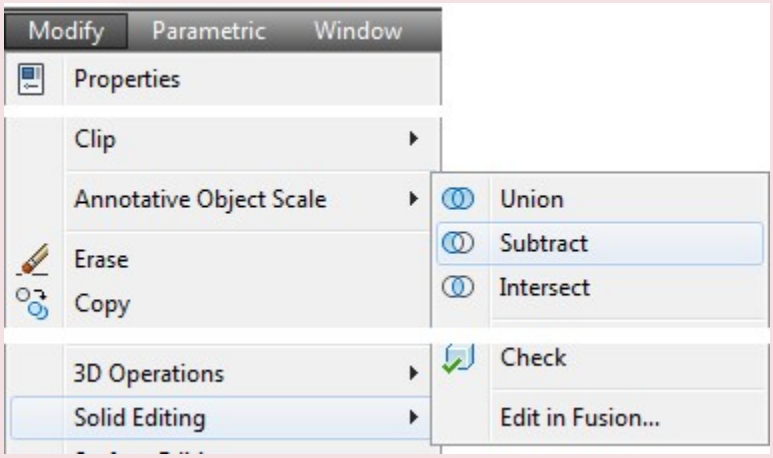
AutoCAD Command: SUBTRACT

The SUBTRACT command is used to subtract one solid from another.

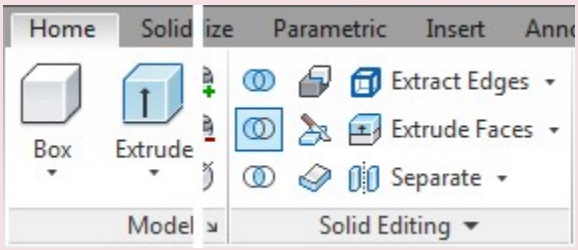
Shortcut: **SU**



Modeling Toolbar



Modify Pulldown



Home Ribbon

WORKALONG: Inserting and Subtracting Regions

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 14-2. (Figure Step 2)

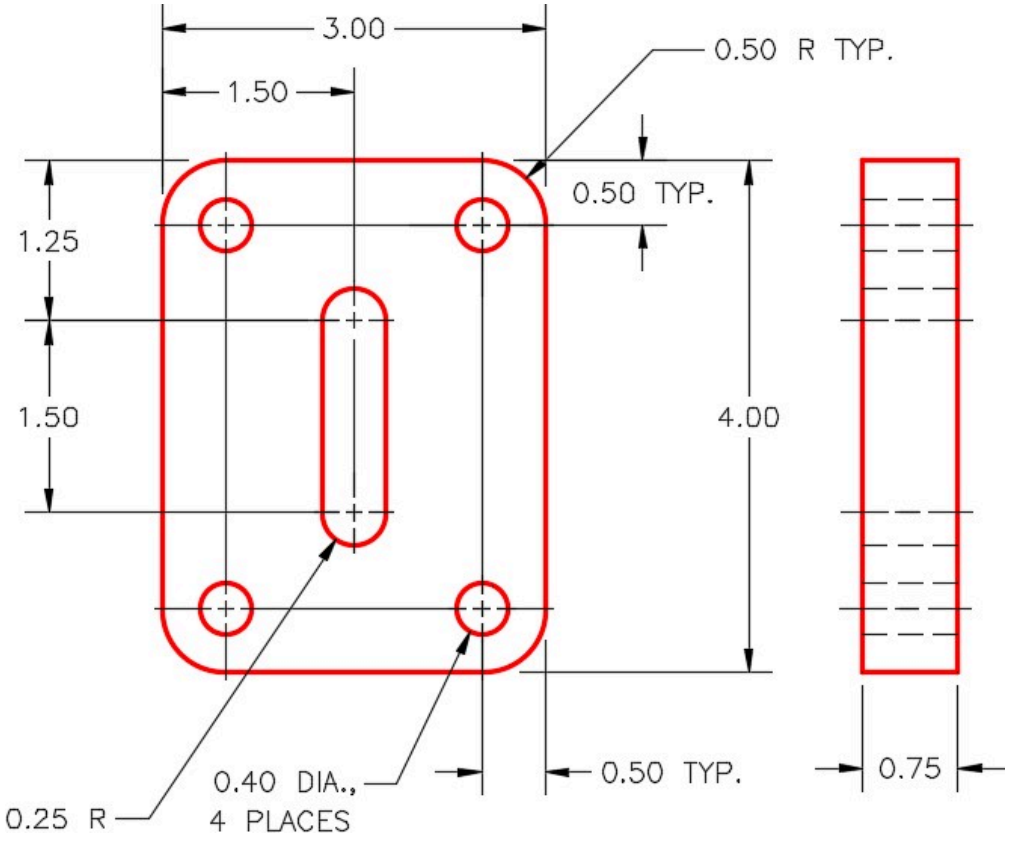


Figure Step 2
Dimensioned Multiview Drawing

Step 3

Set layer: Model as the current layer, the current view to SE Isometric, the current UCS to Top, and the current visual style to 2D Wireframe.

Step 4

Draw the wireframe model of the multiview drawing. (Figure Step 4)

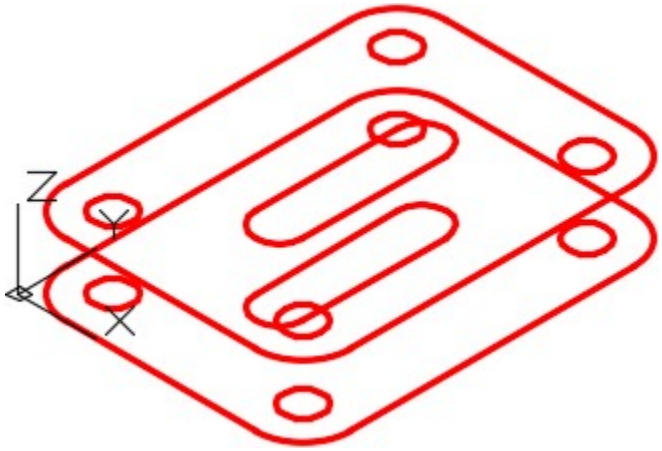


Figure Step 4 Wireframe of Model – SE Isometric View

Step 5

Enter the DELOBJ system variable, as shown below, and set it 0. Command: **DELOBJ**

Enter new value for DELOBJ <1>: **0**

Command: **DELOBJ**

Enter new value for DELOBJ <0>:

Command:

Step 6

Move the UCS to lie on top of the model as shown in the figure. Ensure that the positive Z axis is up and it is located at the centre of the circle. (Figure Step 6)

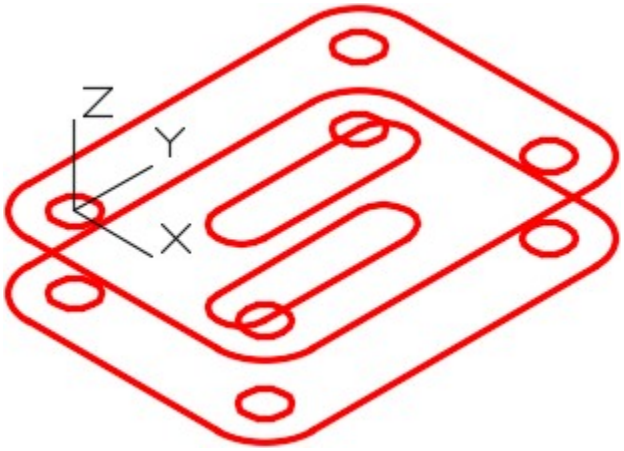


Figure Step 6

Step 7

Using what you learned earlier, use the PEDIT command to convert the lines and arcs to closed polylines. Check the objects with the Properties window to ensure that they are closed. (Figure Step 7)

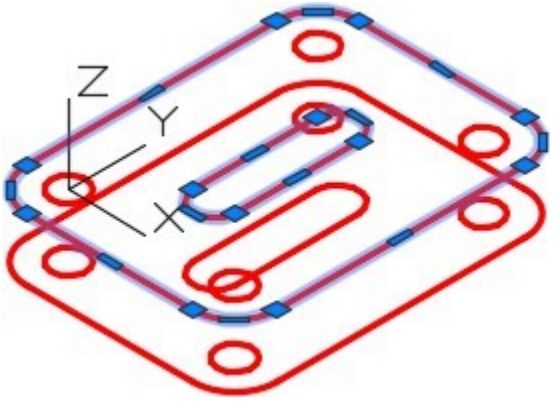


Figure Step 7

AUTHOR'S COMMENTS: If the bottom of the object is the same as the top it is easier to copy the closed plines rather then recreating them. You should delete the model lines and arcs that exist on the bottom first before copying the plines so that you do not duplicate the objects.

Step 8

Delete all of the objects except the circles on the bottom of the model and then copy the plines you just created to the bottom. (Figure Step 8)

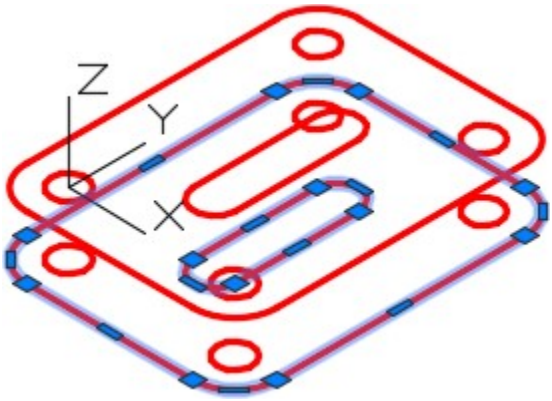


Figure Step 8

Step 9

Set layer: Surface 1 as the current layer and enter the REGION command, as shown below, to create regions for the five closed objects on top of the model.

Command: **REGION**

Select objects: 1 found

(Select each object, by picking them one at a time.)

Select objects: 1 found, 2 total

Select objects: 1 found, 3 total

Select objects: 1 found, 4 total

Select objects: 1 found, 5 total

Select objects:

5 loops extracted.

5 Regions created.

Command:

Step 10

Set the current visual style to: Realistic. Your model should appear as shown in the figure. (Figure Step 10)

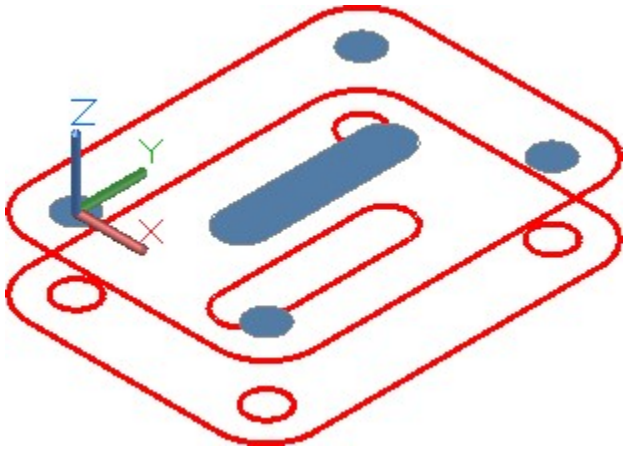


Figure Step 10

Step 11

Using the REGION command, insert a region using the outside closed polyline. (Figure Step 11)

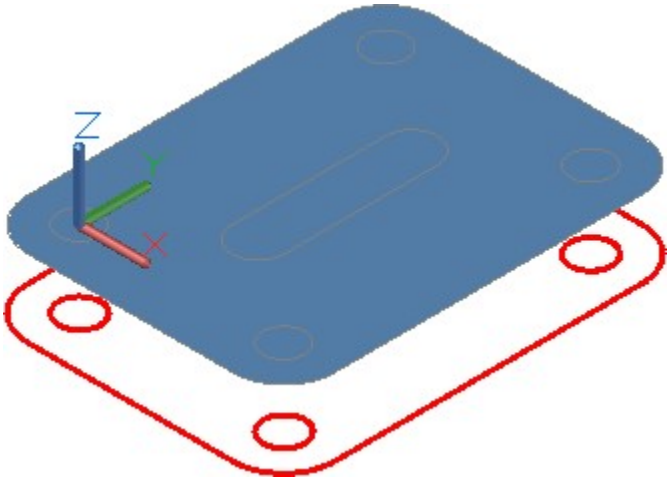


Figure Step 11

Step 12

Using the SUBTRACT command, subtract the five inner regions that you created in Step 9 from the region you created in Step 11. (Figure Step 12)

Command: **SUBTRACT**

Select solids and regions to subtract from ..

Select objects: 1 found

(Select the large region.)

Select objects:

(Press Enter to indicate you have finished selecting.)

Select solids and regions to subtract ..

Select objects: 1 found

(Select each of the five smaller regions one at a time.)

Select objects: 1 found, 2 total

Select objects: 1 found, 3 total

Select objects: 1 found, 4 total

Select objects: 1 found, 5 total

Select objects:

Command:

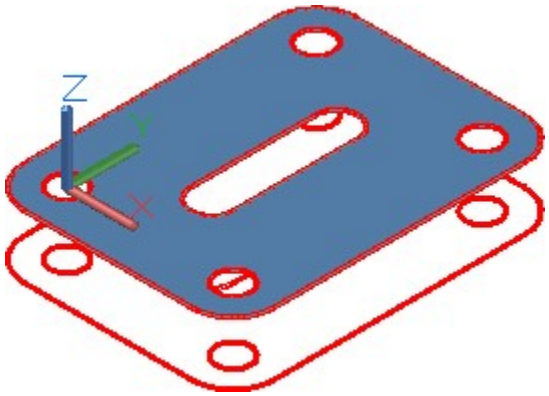


Figure Step 12

AUTHOR'S COMMENTS: When using the SUBTRACT command, first select the region you want to subtract from and then press Enter. Then select the region(s) you want to subtract from it.

Step 13

Ensure that layer: Surface Off is off and then change the layer of the region you just created to layer: Surface Off. Your model will now appear as shown in the figure. (Figure Step 13)

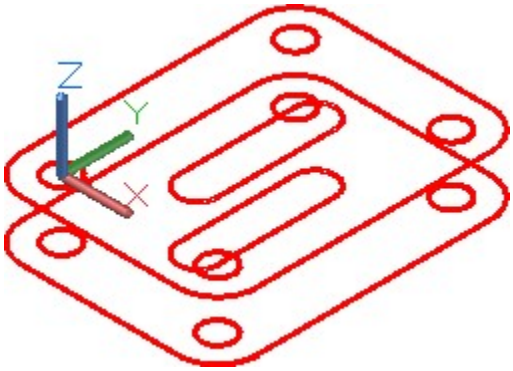


Figure Step 13

Step 14

Click the UCS Origin icon and then snap to the centre of the circle on the bottom of the object. (Figure Step 14)

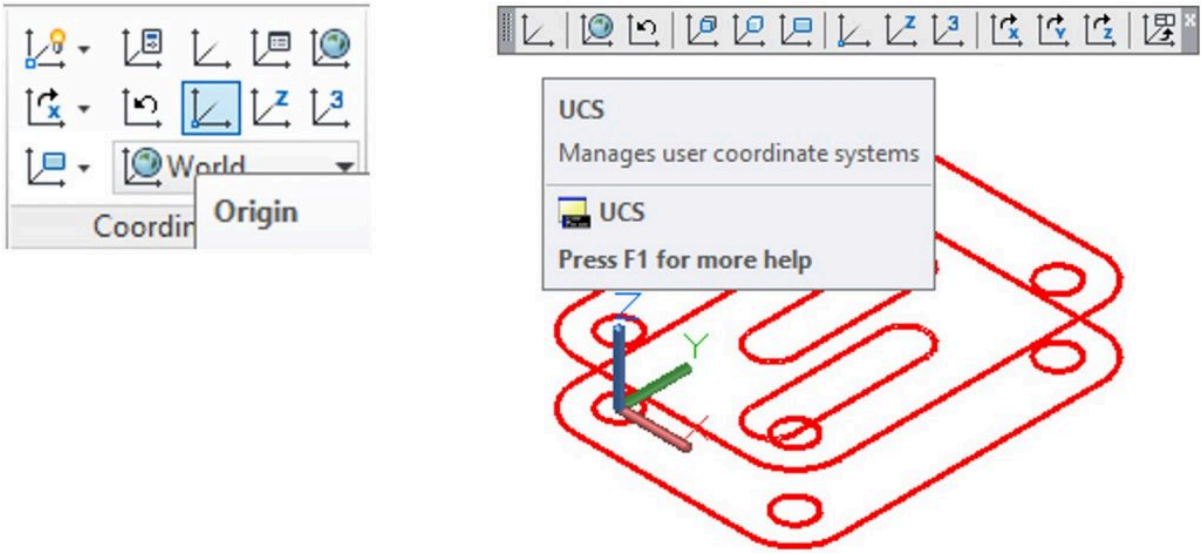


Figure Step 14

Step 15

Click the Rotate X Axis UCS icon on the UCS toolbar and rotate it 180 degree rotation. This will rotate the UCS so that the positive Z direction is pointing down. (Figure Step 15)

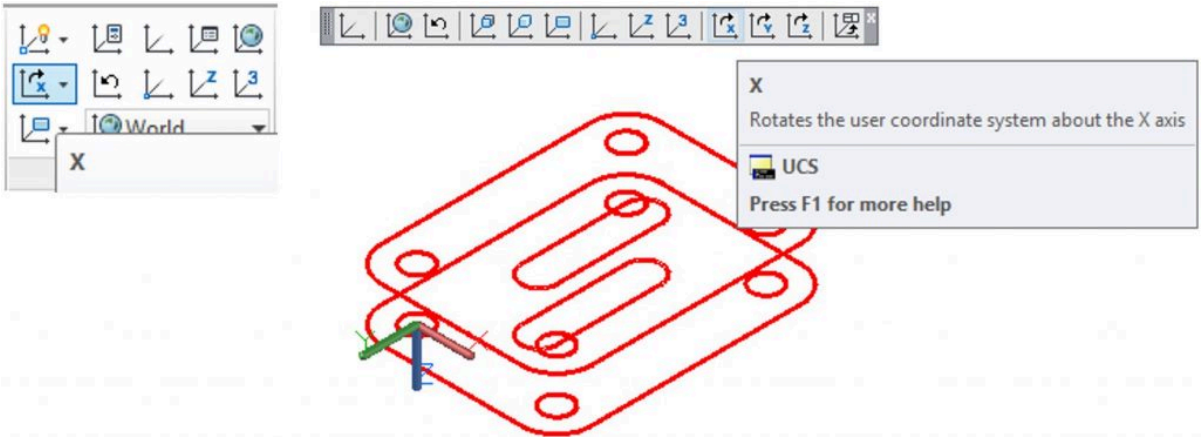


Figure Step 15

Step 16

Using the 3DORBIT command, rotate the model around about 180 degrees and then using what you learned in Steps 10 to 12, insert the regions on the bottom. Subtract the inner regions as you did before. (Figure Step 16)

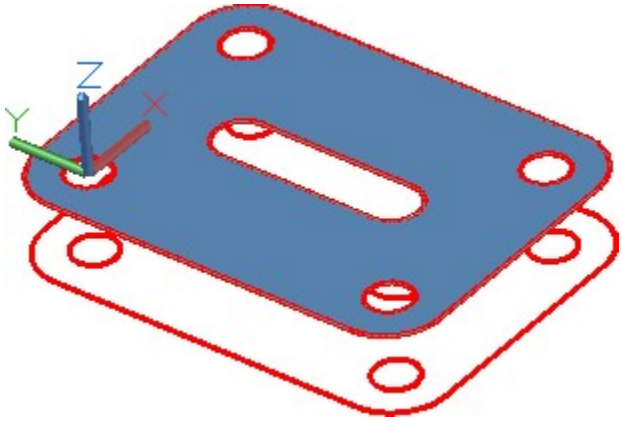


Figure Step 16

Step 17

Change the layer of the regions to layer: Surface Off and then using the RULESURF command, create surfaces on the edges and holes in the model. Create the surfaces on layer: Surface 1. (Figure Step 17)

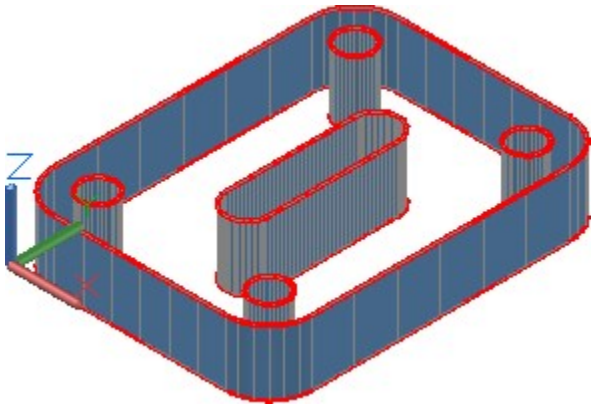


Figure Step 17

Step 18

Turn layers: Model and Construction off and change the layer of the regions to layer: Surface 1. Your completed model should appear as shown in the figure. Use the 3DORBIT command to check that the complete model is surfaced. (Figure Step 18)

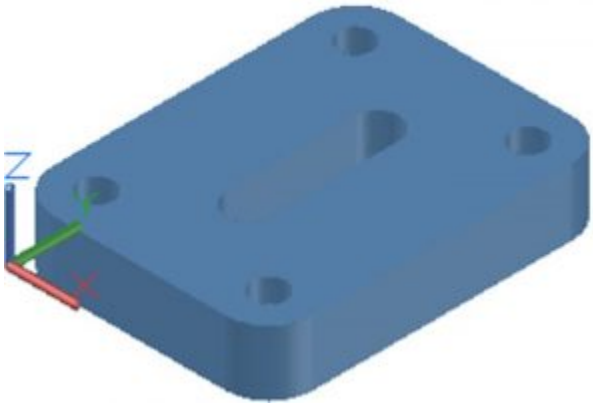
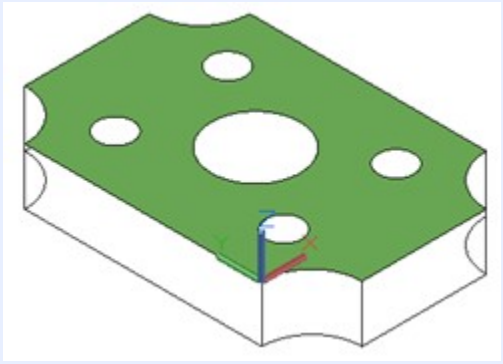


Figure Step 18

MUST KNOW: A region is a two-dimensional solid created within a closed object. Regions can only be created within closed plines, circles, ellipses and splines. The complete region must lay on one plane. A region is actually a 2D solid.While regions are not technically surfaces, they will shade and can be rendered, therefore, can be used to surface a model.If the system variable DELOBJ is set to 1, AutoCAD will delete the closed object that was used to create the region. In most cases, it is important for you to set this variable to 0 to retain your original wireframe object used to create the region.



Key Principles

Key Principles in Module 14

1. A region is a two-dimensional solid created using a closed polyline, circle, ellipse, or spline as the Each region must lie on one plane.
2. While regions are not technically surfaces, they will shade and can be rendered, therefore, can be used to surface a A shaded region will only be visible when viewed from the positive Z direction of the UCS that was current when the region was created.

Lab Exercise 14-1

Time allowed: 60 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 14-1	3D Layout Metric	Millimeters

Step 1

On layer: Model, draw the grid shown below. The grid lines are an equal distance apart. (Figure Step 1A and 1B)

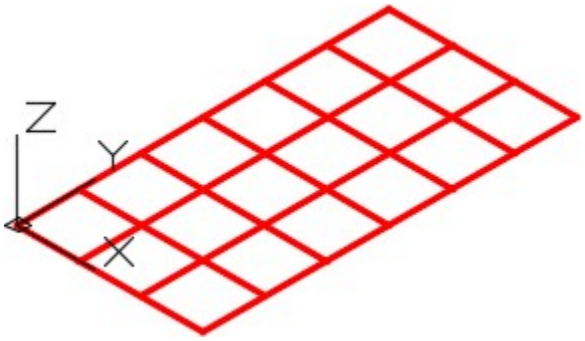


Figure Step 1A
Grid in SE Isometric View

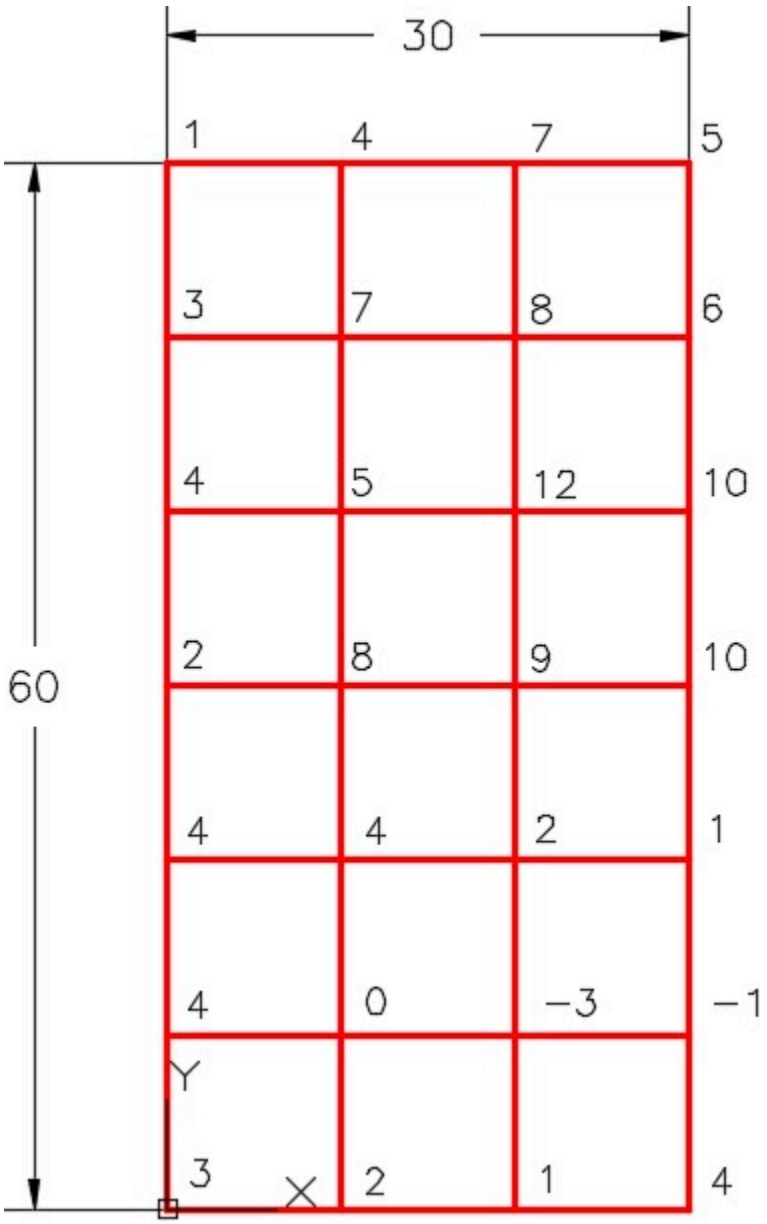


Figure Step 1B
Grid in Top View

Step 2

On layer: Surface 2, draw a 3D mesh on the grid using the 3D MESH command. The Z values are displayed at each vertex in Figure Step 1B. Set the current visual style to Realistic. It should appear as shown in the figure. (Figure Step 2)

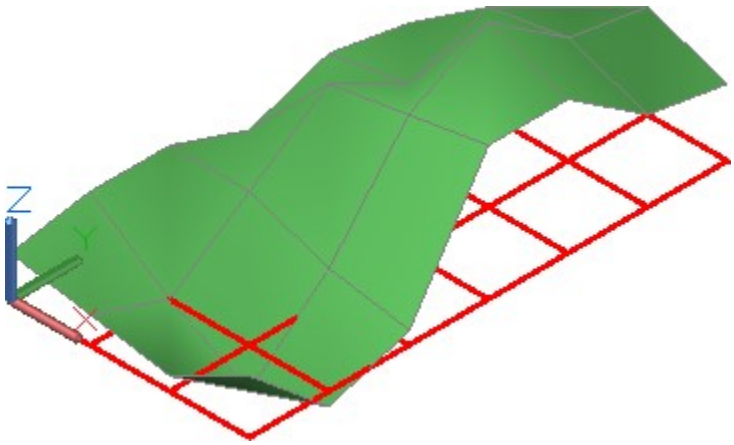


Figure Step 2

Step 3

Set the current visual style to 3D Wireframe. Turn layer Surface 2 off and on layer: Model, draw lines representing the Z coordinates. (Figure Step 3)

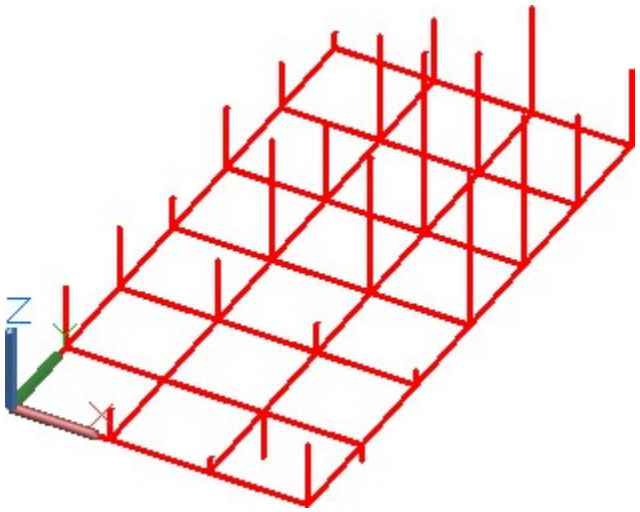


Figure Step 3

Step 4

On layer: Surface 3, draw a 3D mesh on the grid using the 3D MESH command and snapping to ends of the Z coordinates. Set the current visual style to Realistic. (Figure Step 4)

Figure Step 4

Step 5

Turn layer: Model off and the completed mesh should appear as shown in the figure. (Figure Step 5)

Figure Step 5

Step 6

Set the Insertion Units, change the current UCS to World and check the model with the key.

Step 7

Save and close the drawing.

Lab Exercise 14-2

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 14-2	3D Layout Metric	Millimeters

Step 1

Draw a wireframe model of the object. (Figure Step 1A and 1B)

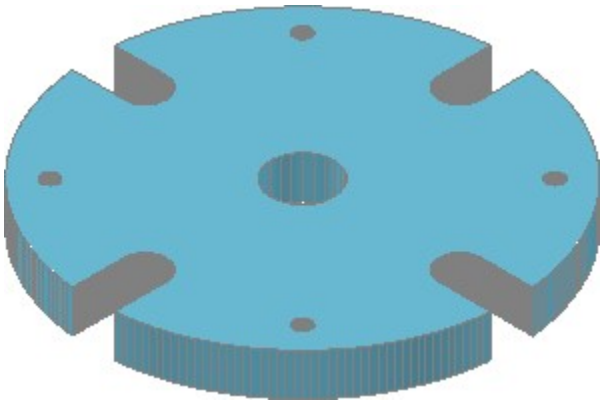


Figure Step 1A
Completed Surfaced Model

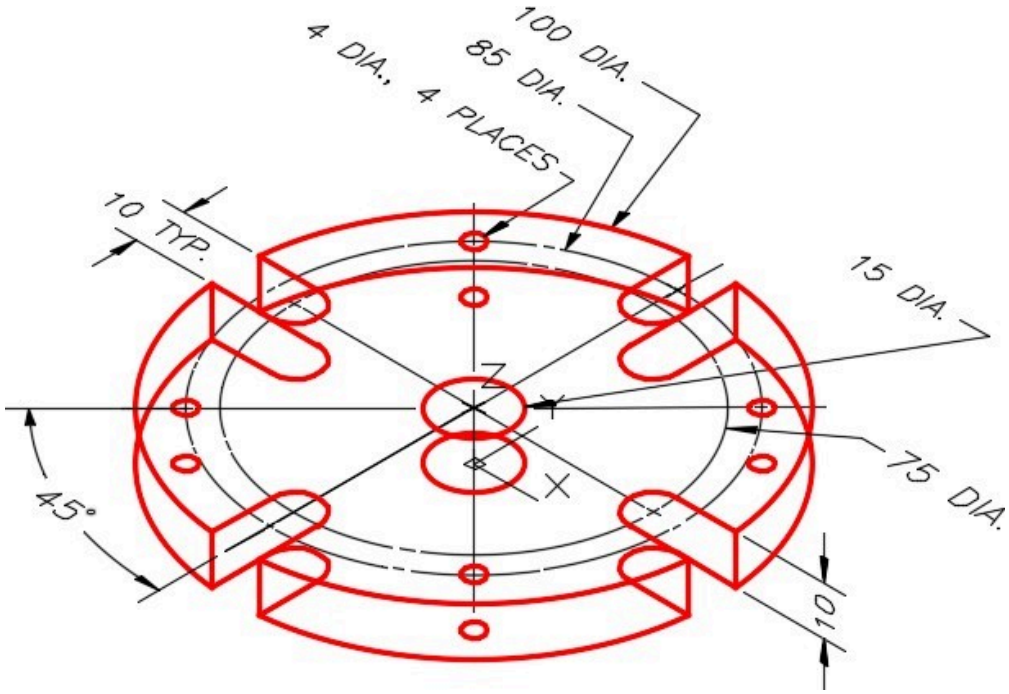


Figure Step 1B
Dimensioned Wireframe Model

Step 2

Use the 2D ARRAY command to speed the model construction.

Step 3

On layer: Surface 1, create regions for the top and bottom of the model.

Step 4

On layer: Surface 1, create ruled surfaces for the remainder of the surfaces. Ensure that the complete model is surfaced.

Step 5

Set the visual style to: Realistic.

Step 6

Freeze layers: Construction and Model.

Step 7

Using the 3DORBIT command, check the model for completeness

Step 8

Set the Insertion Units, change the current UCS to World and check the model with the key.

Lab Exercise 14-3

Time allowed: 30 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 14-3	N/A	Millimeters

Step 1

Open the drawing: AutoCAD 3D Lab 03-2.

Step 2

Save the drawing with the name: AutoCAD 3D Lab 14-3.

Step 3

On layer: Surface 1, create regions and surfaces on all sides and holes. (Figure Step 3)

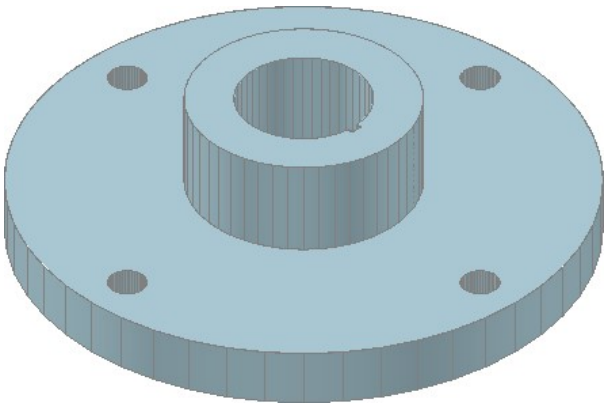


Figure Step 3
Completed Surfaced Model

Step 4

Set the visual style to Realistic.

Step 5

Freeze layers: Construction and Model.

Step 6

Using the 3DORBIT command, check the model for completeness

Step 7

Save and close the drawing.

AUTHOR'S COMMENTS: Do not be concerned if your shaded model does not match the figures in the module exactly. There are many visual style settings that can be set to change the look of the shaded model. For now, all you should be concerned about is being able to surface the model.The advanced features of customizing visual styles are taught in the AutoCAD 3D Advanced book.

Lab Exercise 14-4

Time allowed: 50 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 14-4	N/A	Inches

Step 1

Open the drawing: AutoCAD 3D Lab 09-3.

Step 2

Save the drawing with the name: AutoCAD 3D Lab 14-4.

Step 3

On layer: Surface 2, create regions and surfaces on all sides and holes. (Figure Step 3)

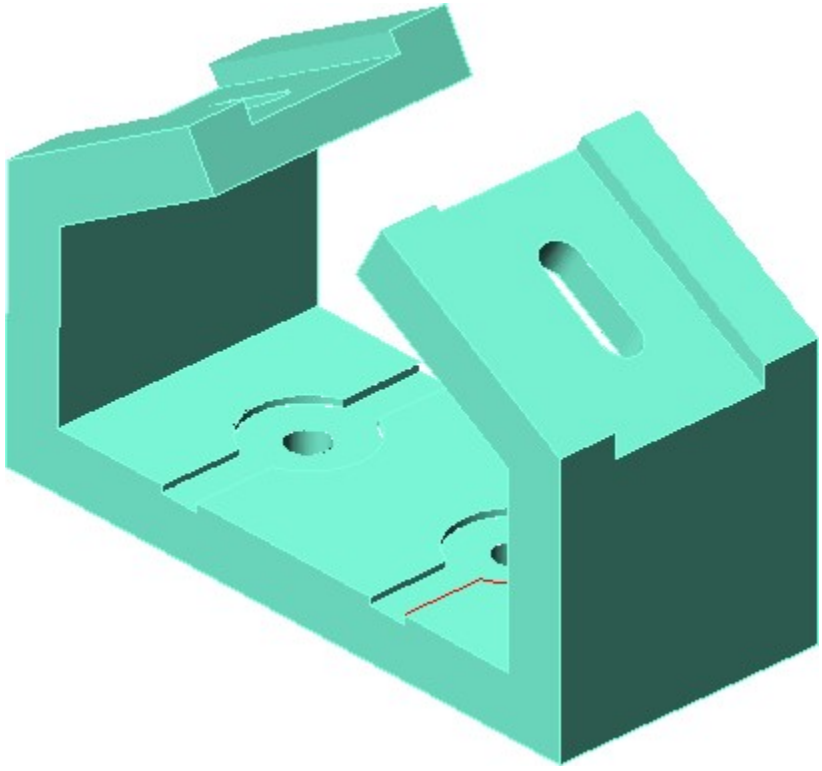


Figure Step 3
Completed Surfaced Model

Step 4

Set the visual style to: Realistic.

Step 5

Freeze layers: Construction and Model.

Step 6

Using the 3DORBIT command, check the model for completeness

Step 7

Save and close the drawing.

AUTHOR'S COMMENTS: Do not be concerned if your shaded model does not match the figures in the module exactly. There are many visual style settings that can be set to change the look of the shaded model. For now, all you should be concerned about is being able to surface the model. The advanced features of customizing visual styles are taught in the AutoCAD 3D Advanced book.

Module 15: Surface Modeling – Part 4

Learning Outcomes

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

1. Describe a 3D polyline and how it compares to a 2D polyline.
2. Apply the 3DPOLY command and draw 3D polylines.
3. Apply the EDGESURF command.
4. Apply the PEDIT, SURFTYPE, SPLFRAME, SURFV, and SURFU commands and system variables to modify the appearance of 3D meshes.

3D Polylines

Since the release of AutoCAD 14, polylines created with the PLINE command have been created as lightweight polylines. Sometimes called a 2D polylines, shown in Figure 15-1, lightweight polyline are greatly improved over the older heavyweight or 3D polylines. A 2D polyline must lay on a single plane.

A *3D polyline* is created with the 3DPOLY command using XYZ coordinates to locate each of the vertices. See Figure 15-2. A 3D polyline is not restricted to lay on the same plane. It requires a larger part of the drawings database compared to a 2D polyline. For that reason, use 3D polylines only when absolutely required in 3D modeling. 3D polylines must be constructed with lines only and cannot contain arcs or be given a width.

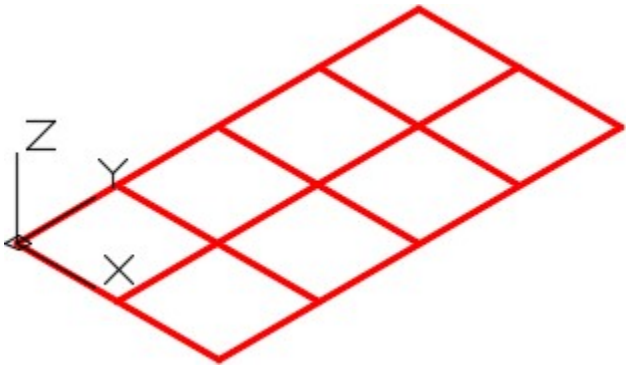


Figure 15-1
2D polylines

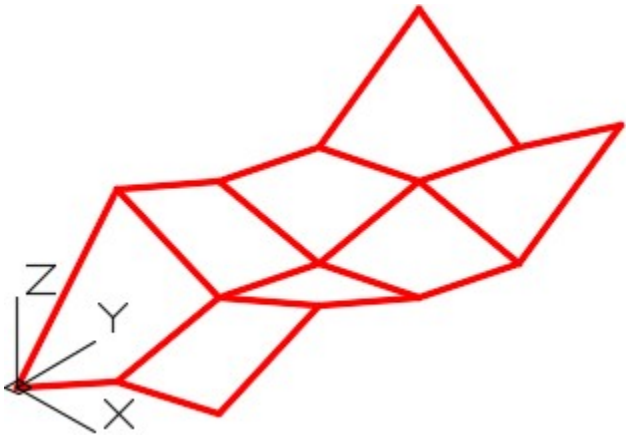
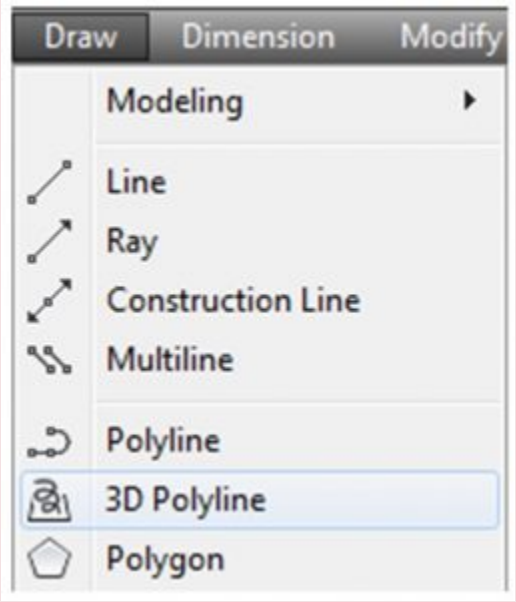


Figure 15-2
3D polylines

AutoCAD Command: 3DPOLY

The 3DPOLY command is used to create 3D polylines.

Shortcut: none

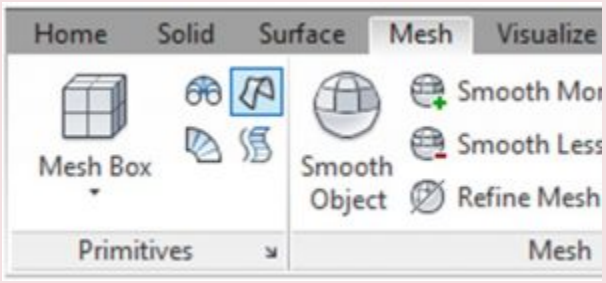


Draw Pull-down

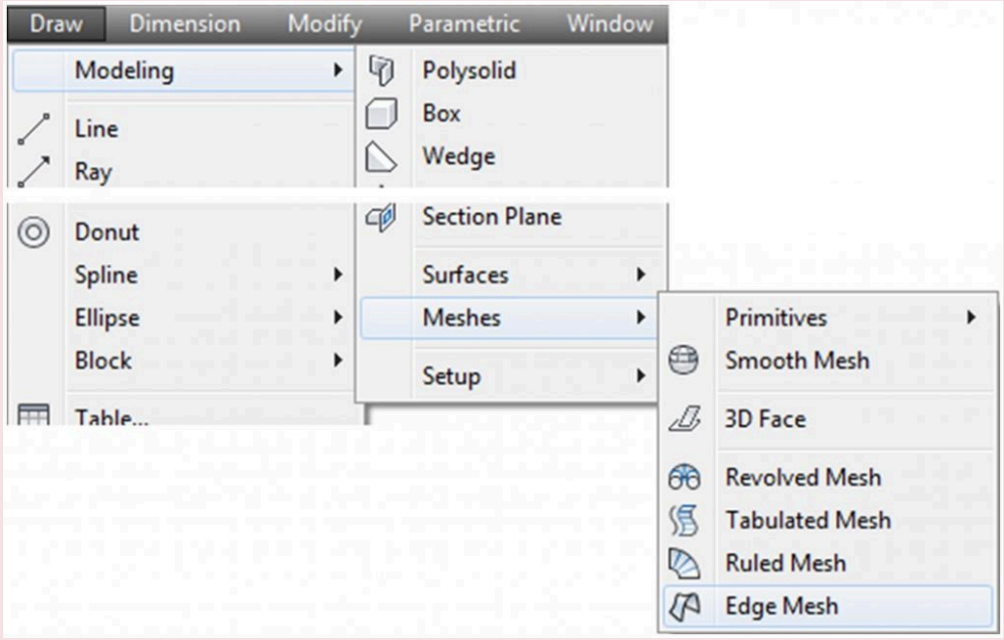
AutoCAD Command: EDGESURF

The EDGESURF command is used to create a surface mesh by selecting four drawing objects. The drawing objects can be lines, arcs, 2D polylines, and 3D polylines.

Shortcut: none



Mesh Ribbon



Draw Pull-down

WORK ALONG: Surfacing Models Using 3D Polylines and Edge Surfaces

Step 1

Open the drawing: AutoCAD 3D Workalong 14-1.

Step 2

Using the SAVEAS command, save the drawing with the name: AutoCAD 3D Workalong 15-1.

Step 3

Turn off all layers except layer: Model.

Step 4

Set the current visual style to 2D Wireframe, the current UCS to World and the current view to SE Isometric. Your model should appear as shown in the figure. (Figure Step 4)

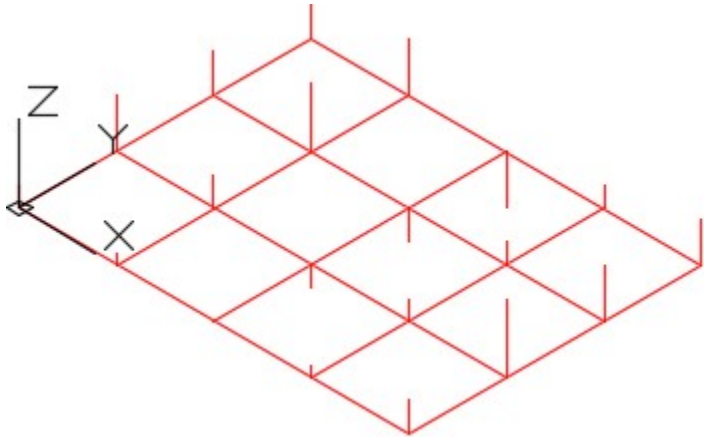


Figure Step 4

Step 5

Set layer: 3D Pline as the current layer. Enter the 3DPOLY command, as shown below, to draw 3D polylines between all of the Z coordinates on the grid in both the X and the Y direction. (Figure Step 5)

Command: **3DPOLY**

Specify start point of polyline: *(end)* **P1**

Specify endpoint of line or [Undo]: *(end)* **P2**

Specify endpoint of line or [Undo]: *(end)* **P3**

Specify endpoint of line or [Undo]: *(end)* **P4**

Specify endpoint of line or [Close/Undo]:

Command: **3DPOLY**

Specify start point of polyline: *(end)* **P5**

Specify endpoint of line or [Undo]: *(end)* **P6**

Specify endpoint of line or [Undo]: *(end)* **P7**

Specify endpoint of line or [Undo]: *(end)* **P8**

Specify endpoint of line or [Close/Undo]:

Command:

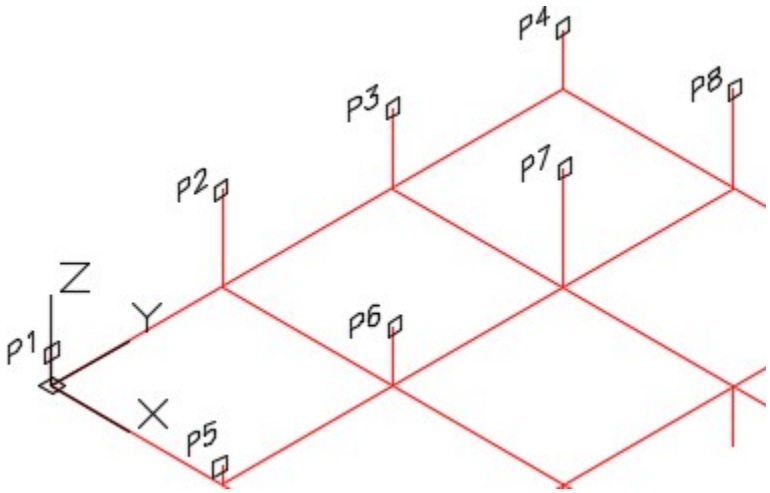


Figure Step 5

Step 6

Continue drawing the remainder of the 3D polylines on the Y axis. (Figure Step 6)

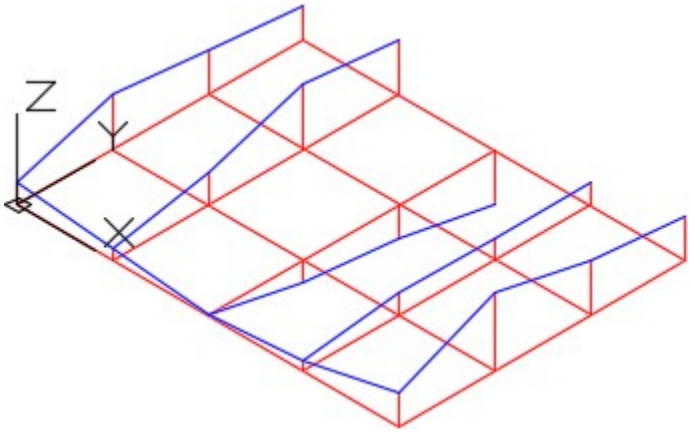


Figure Step 6

Step 7

Draw 3D polylines between all the Z coordinates in the X direction until you complete the grid. (Figure Step 7)

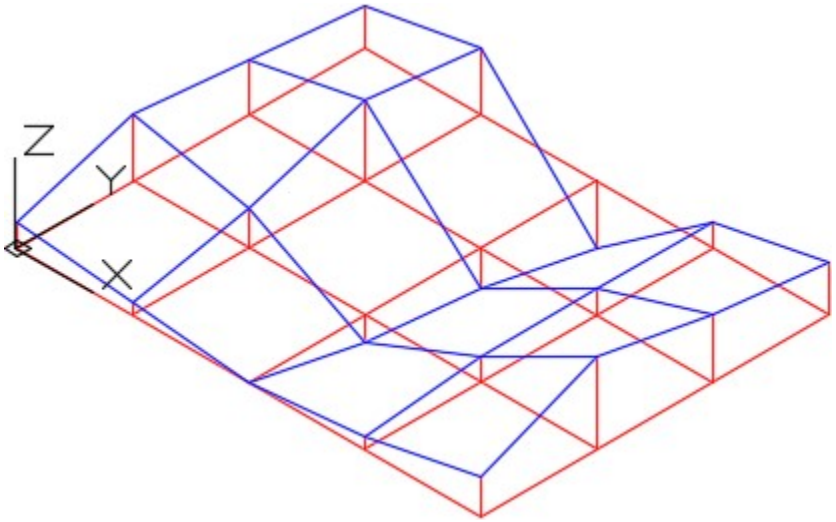


Figure Step 7

Step 8

Turn layer: Model off and your model should appear as shown in the figure. Using the EXPLODE command and a window, explode the 3D plines to convert them into lines. (Figure Step 8)

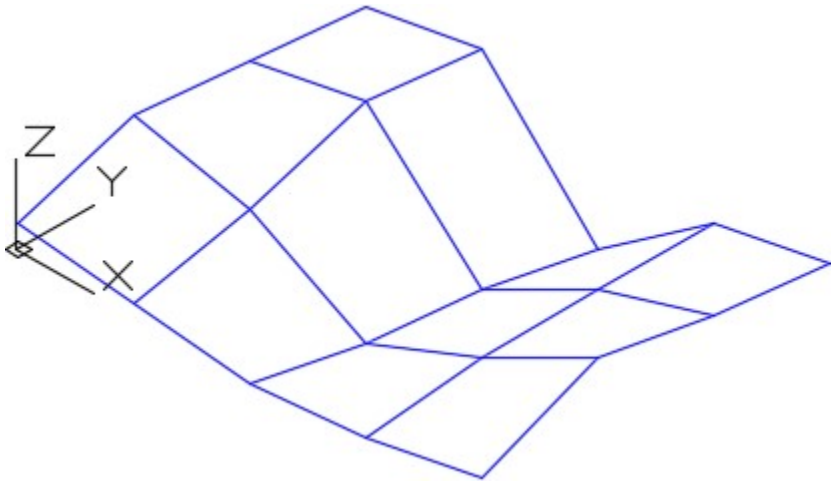


Figure Step 8

Step 9

Change the view to Top. (Figure Step 9)

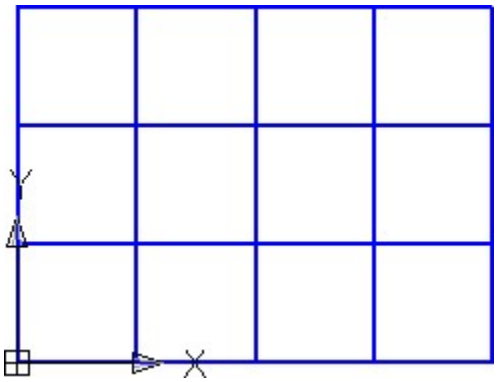


Figure Step 9

AUTHOR'S COMMENTS: With the model in the top view, it is now easy to create the 3D mesh using the 3DMESH command and snapping to the vertices of the lines.

Step 10

Set layer: Surface 1 as the current layer. Enter the 3DMESH command, as shown below, to create the mesh. (Figure Step 10)

Command: **3DMESH**

Enter size of mesh in M direction: **5**

Enter size of mesh in N direction: **4**

Specify location for vertex (0, 0): *(end)* **P1**

Specify location for vertex (0, 1): *(end)* **P2**

Specify location for vertex (0, 2): *(end)* **P3**

Specify location for vertex (0, 3): *(end)* **P4**

Specify location for vertex (1, 0): *(end)* **P5**

Specify location for vertex (1, 1): *(end)* **P6**

Specify location for vertex (1, 2): *(end)* **P7**

Specify location for vertex (1, 3): *(end)* **P8**

(Continue to cover the complete mesh.)

Command:

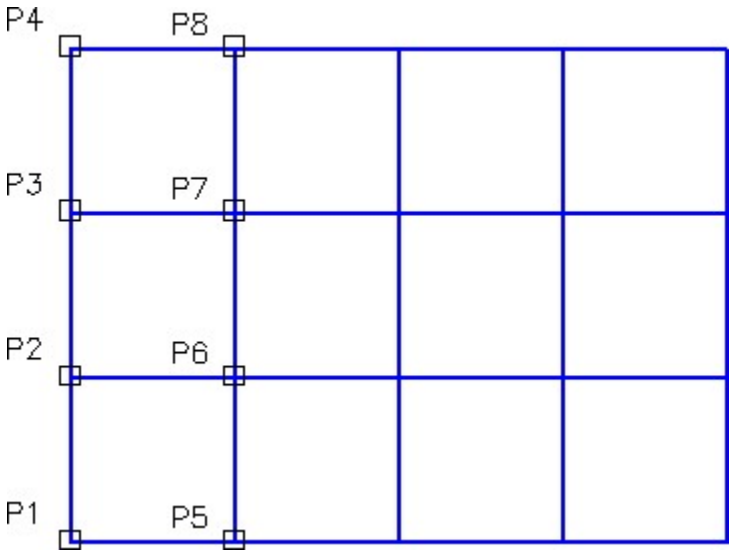


Figure Step 10

Step 11

Set the current visual style to Realistic and the current view to SE Isometric. (Figure Step 11)

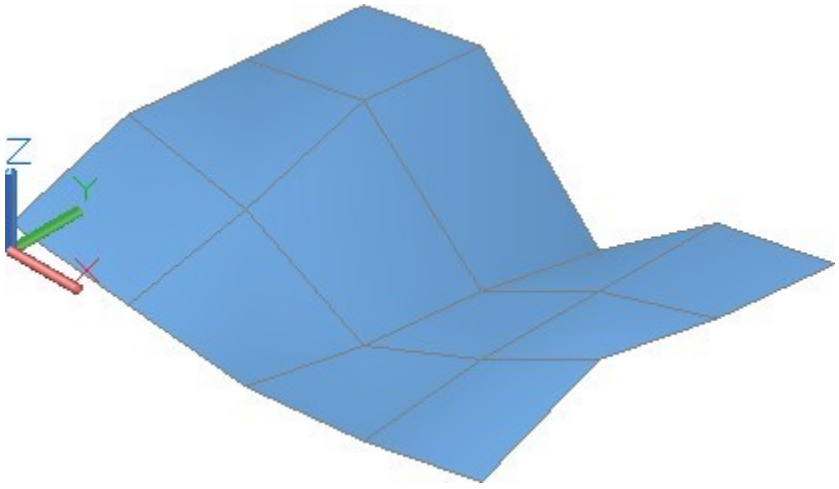


Figure Step 11

Step 12

Set the current visual style to 2D Wireframe and turn layer: Surface 1 off. You will only see the 3D polylines. (Figure Step 12)

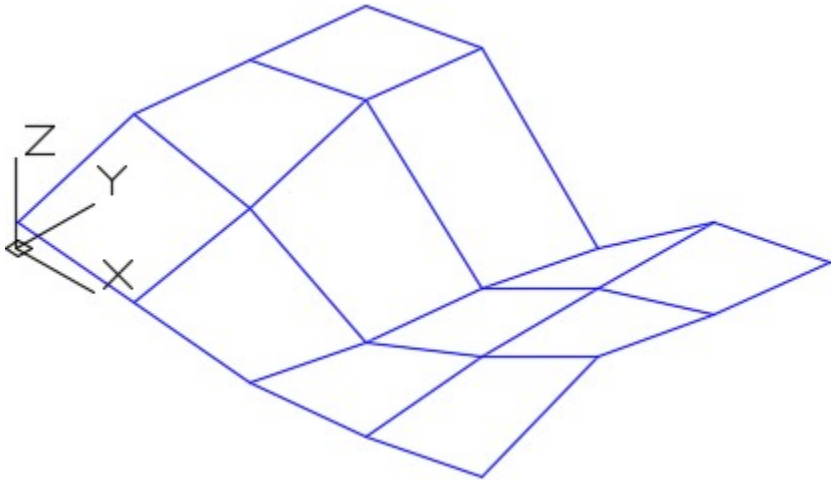


Figure Step 12

Step 13

Set the system variables SURFTAB1 to 6 and SURFTAB2 to 6.

AUTHOR'S COMMENTS: Another command that can be used to draw surface meshes is the EDGESURF. It requires you to select four lines, arcs, 2D polylines and 3D polylines or a combination of them. You must always select four drawing objects as edges.

Step 14

On layer: Surface 4, use the EDGESURF command, as shown below, to create a surface mesh on one grid square. (Figure Step 14)

Command: **EDGESURF**

Current wire frame density: SURFTAB1=6 SURFTAB2=6

Select object 1 for surface edge: **P1**

Select object 2 for surface edge: **P2**

Select object 3 for surface edge: **P3**

Select object 4 for surface edge: **P4**

Command:

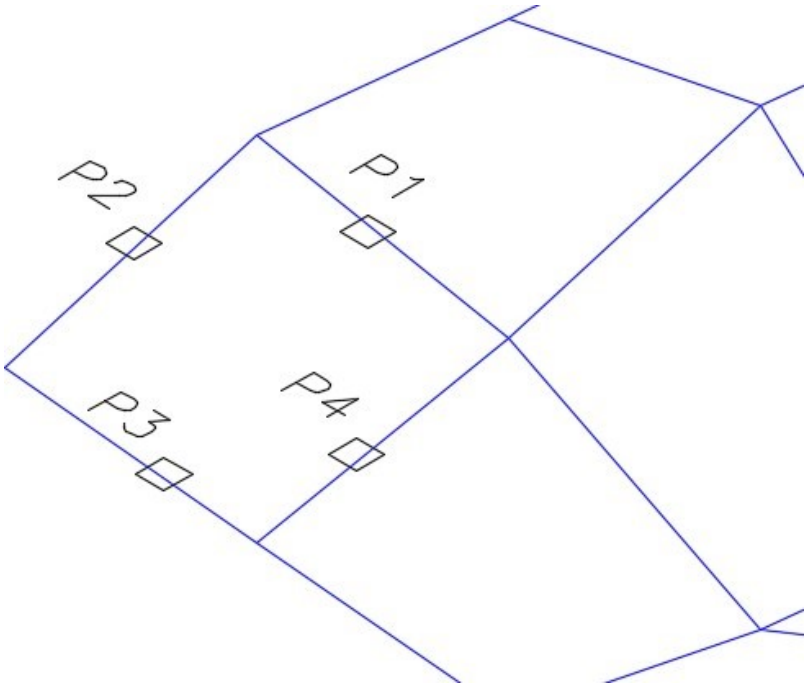


Figure Step 14

Step 15

Set the current visual style to Realistic. (Figure Step 15)

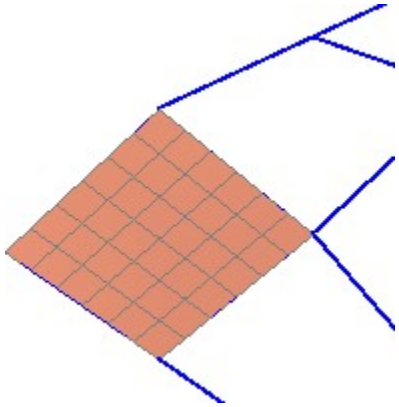


Figure Step 15

Step 16

On layer: Surface 4, using the EDGESURF command, create surfaces in all grid squares to cover the model. When complete, the model will appear as shown in the figure. (Figure Step 16)

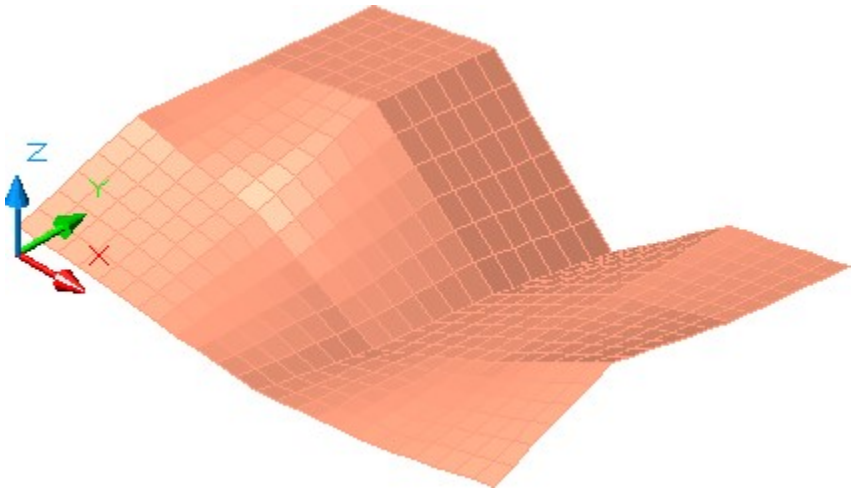


Figure Step 16

AUTHOR'S COMMENTS: You are going to have to use the Surface Off layer here to temporarily remove the display of some of the surface meshes as you insert others. Try to do it in a checker board pattern so you can easily keep track of the grids you completed.

Step 17

Save and close the drawing.

Modifying the Appearance of Surface Meshes

There are several system variables that can be used to modify or change the appearance of existing surface meshes or changing their mesh density. The PEDIT command and SURFV, SURFU, SURFTYPE, and SPLFRAME system variables are used to smooth and change the setting for the appearance of and mesh density of 3D meshes. If the desired results cannot be obtained using these system variables, it is sometime easier and faster to erase the surface and construct it again using a smaller base grid.

System Variable: SURFV and SURFU

The SURFV and SURFU system variables are used to set the surface mesh density of the M and N directions when the Smooth option is selected in the PEDIT command.

Shortcut: **none**

SURFU sets the M direction.

SURFV sets the N direction.

System Variable: SURFTYPE

The SURFTYPE system variable is used to control the method AutoCAD fits the surface when the Smooth option is selected in the PEDIT command.

Shortcut: none

It has the following settings:

5 – Quadratic B-spline surface

6 – Cubic B-spline surface

8 – Bezier surface

System Variable: SPLFRAME

The SPLFRAME command is used to control the display of splines and meshes that have been smoothed.

Shortcut: none

It has the following settings:

0 – Enables the display of the smoothed meshes and spline-fit

1 – Disables the display of the smoothed meshes or spline-fit It will display the original mesh. It does not display the invisible edges of 3D faces or polyface meshes.

WORK ALONG: Modifying the Appearance of Surface Meshes

Step 1

Open drawing: AutoCAD 3D Workalong 15-1. Using the SAVEAS command, save the drawing with the name: AutoCAD 3D Workalong 15-2.

Step 2

Turn off all layers except layer: Surface 1. Set the current visual style to Realistic, the current UCS to World and the current view to SE Isometric. (Figure Step 2)

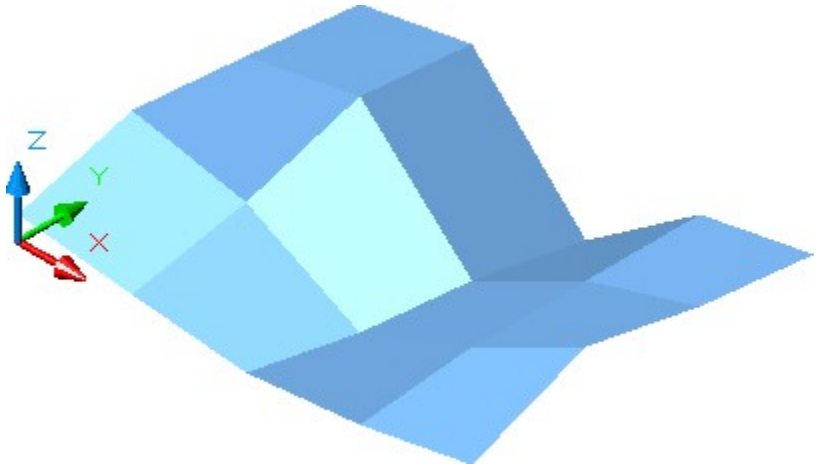


Figure Step 2

Step 3

Set the current visual style to 3D Wireframe. (Figure Step 3)

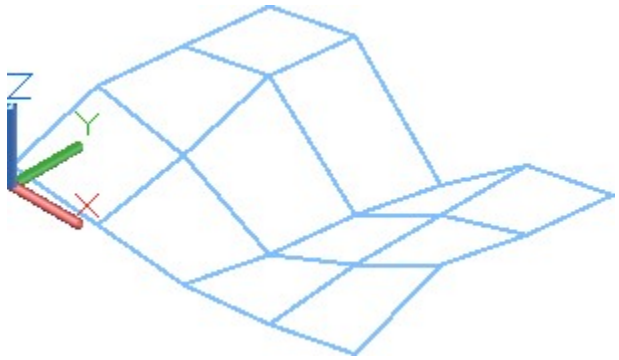


Figure Step 3

Step 4

Enter the commands shown below and your model will appear as shown in the figure. (Figure Step 4)

Command: **SURFV**

Enter new value for SURFV <6>: **12**

Command: **SURFU**

Enter new value for SURFU <6>: **24**

Command: **PEDIT**

Select polyline or [Multiple]:

(Select the mesh.)

Enter an option [Edit vertex/Smooth surface/Desmooth/Mclose/Nclose/Undo]: **S**

Generating segment 2… Command: **REGEN**

REGEN Regenerating model.

Command:

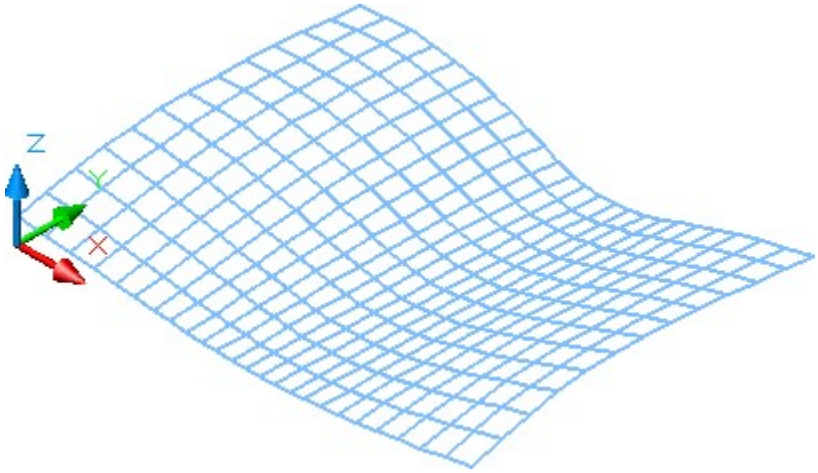


Figure Step 4

Step 5

Enter the commands shown below. The model should display as it was when you started this exercise. (Figure Step 5)

Command: **SPLFRAME**

Enter new value for SPLFRAME <0>: **1**

Command: **REGEN** Regenerating model.

Command:

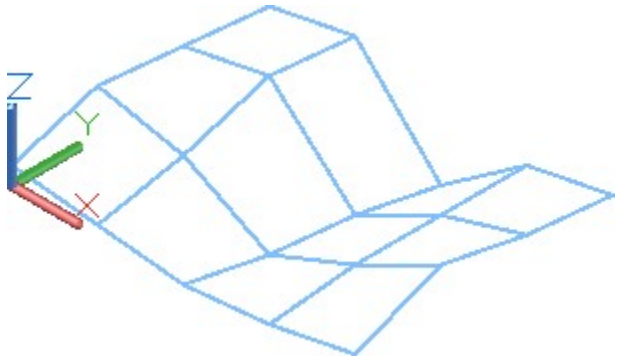


Figure Step 5

Step 6

Enter the commands shown below and your model will display as shown in the figure. (Figure Step 6)

Command: **SPLFRAME**

Enter new value for SPLFRAME <1>: **0**

Command: **REGEN**

Regenerating model

Command:

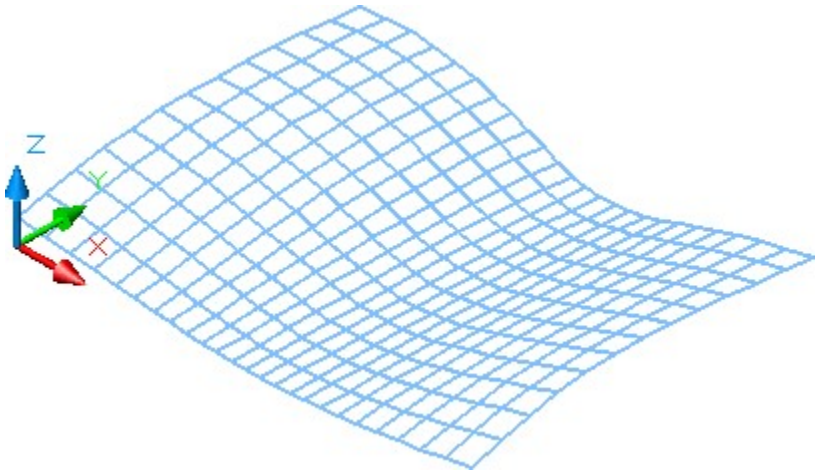


Figure Step 6

Step 7

Enter the commands shown below and your model will display as shown in the figure. (Figure Step 7)

Command: **SURFTYPE**

Enter new value for SURFTYPE <6>: **5**

Command: **PEDIT**

Select polyline or [Multiple]:

(Select the mesh.)

Enter an option [Smooth surface/Desmooth/MOpen/MClose/NOpen/NClose/Undo]: **S**

Command:

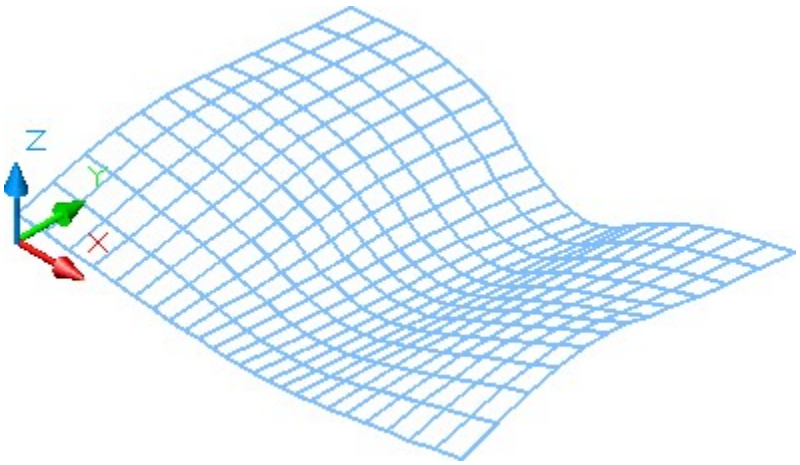


Figure Step 7

Step 8

Enter the commands shown below and your model will display as shown in the figure. (Figure Step 8)

Command: **SURFTYPE**

Enter new value for SURFTYPE <5: **8**

Command: **PEDIT**

Select polyline or [Multiple]:

(Select the mesh.)

Enter an option [Smooth surface/Desmooth/MOpen/MClose/NOpen/NClose/Undo]: **S**

Command:

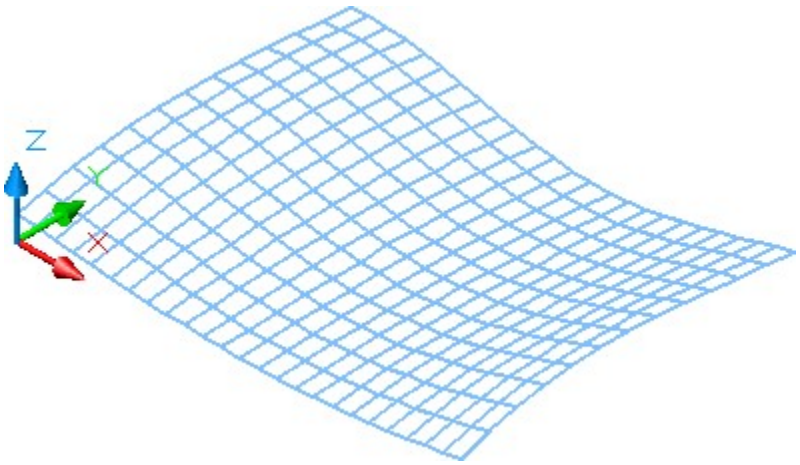


Figure Step 8

Step 9

Set the current visual style to Realistic. (Figure Step 9)

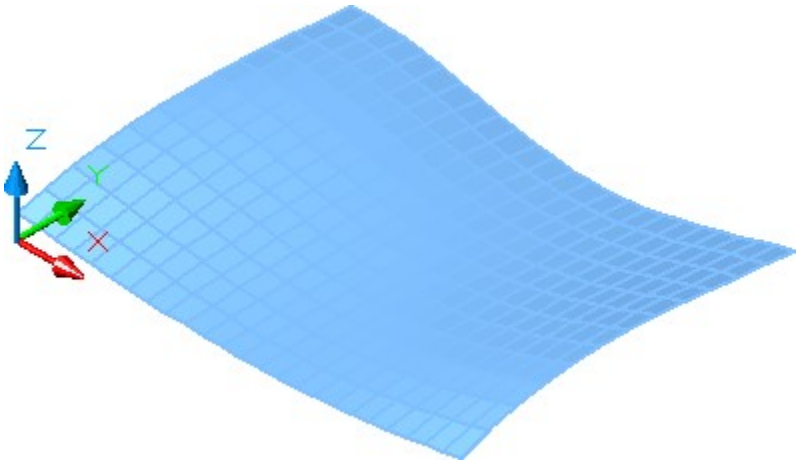


Figure Step 9

Step 10

Save and close the drawing.

Key Principles

The Key Principles in Module 15

1. Sometimes called a 2D polyline, a lightweight polyline is greatly improved over the older heavyweight or 3D A 2D polyline must lay on a single plane.
2. A 3D polyline is created with the 3DPOLY command using XYZ coordinates to locate each A 3D polyline is not restricted to lay on the same plane.
3. The command PEDIT and system variables SURFV, SURFU, SURFTYPE, and SPLFRAME are used to smooth and change the setting for the appearance of and mesh density of 3D meshes.

Lab Exercise 15-1

Time allowed: 3 hours.

Drawing Name	Template	Units
AutoCAD 3D Lab 15-1	3D Layout English	Feet

Step 1

In this lab exercise, you will be creating a surface mesh on a plot of land that is 2500 X 2500 feet. There is an exiting 100 ft. road on the south side of the property. (Figure Step 1)

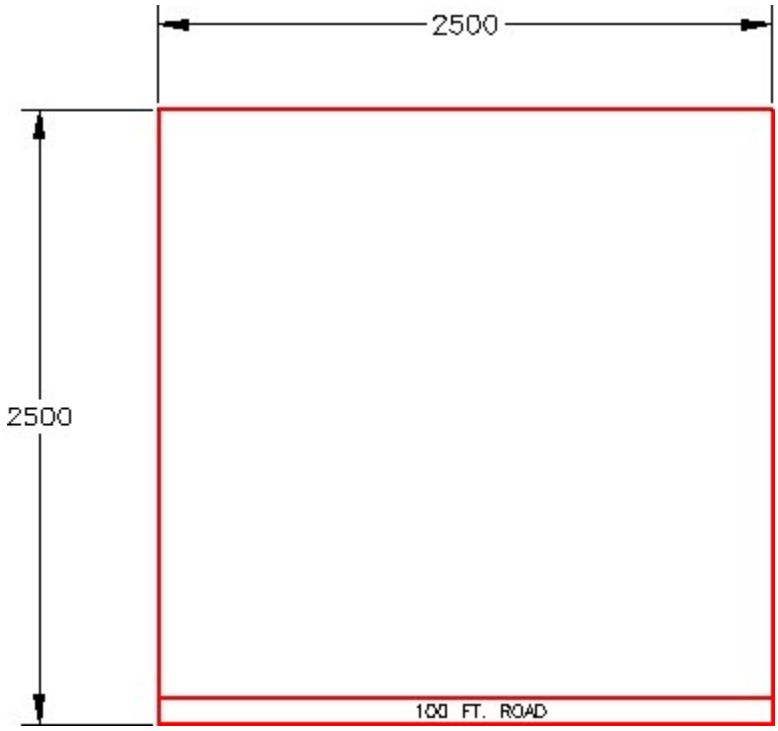


Figure Step 1

Step 2

Start a new drawing, using the template 3D Layout English. Insert the block: AutoCAD 3D Lab 15-1. Explode the block.

Step 3

Change the layer of all objects to layer: Model. (Figure Step 3)

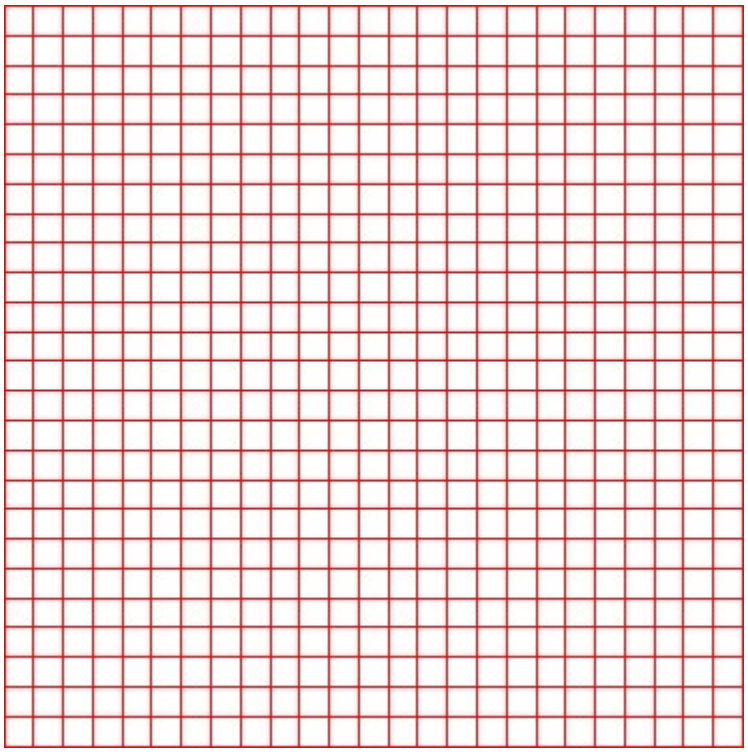


Figure Step 3

Step 4

Change the current view to SE Isometric. (Figure Step 4)

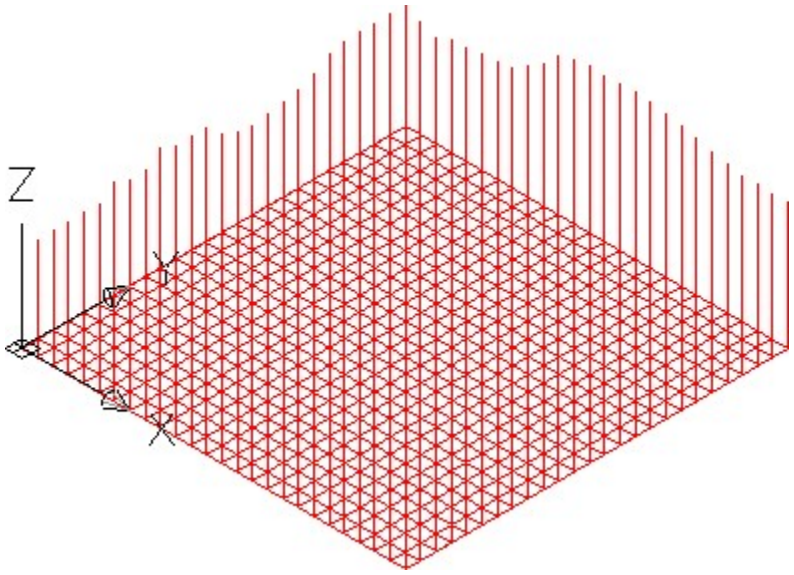


Figure Step 4

Step 5

The 2D grid is located at sea level or Z=zero and is divided equally every 100 ft.

Step 6

On the layer: 3D Pline, draw 3D polylines between the top of each of the Z coordinate lines to construct a digital terrain model of the property as shown below. Try rotating the grid and lines to different positions to find the best view to insert the 3D polylines. (Figure Step 6)

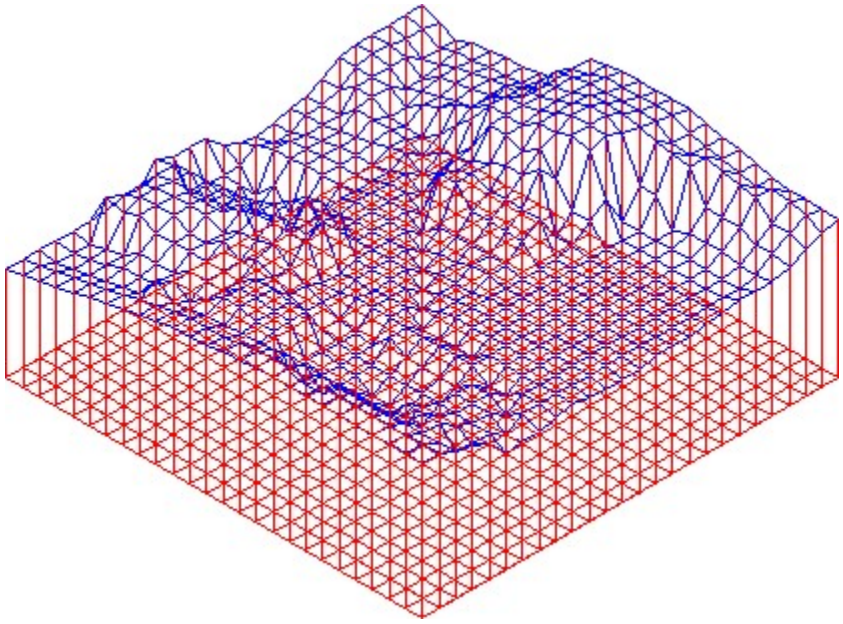


Figure Step 6

Step 7

Turn off all layers except layers: 3D Pline and Surface 2.

Step 8

Change the view to Top. (Figure Step 8)

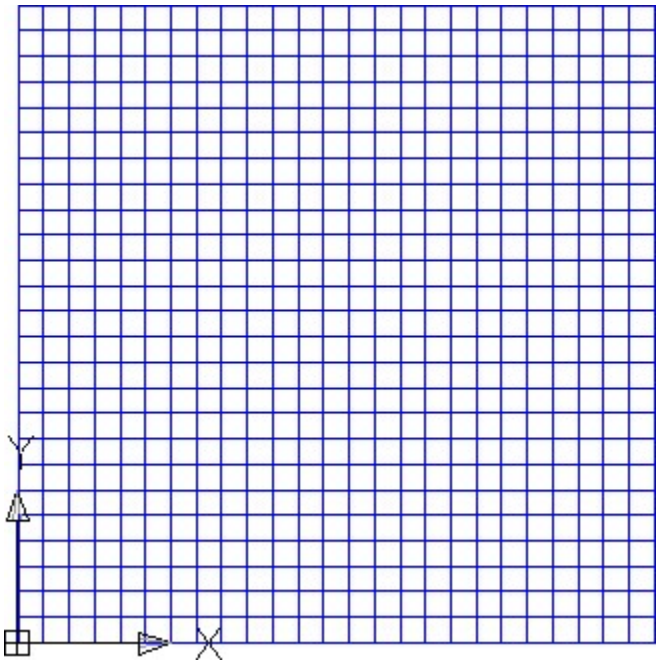


Figure Step 8

Step 9

Set layer: Surface 2 as the current layer. Using the EXPLODE command and a window, explode the 3D plines to convert them into lines.

Step 10

Using the 3DMESH command, draw a mesh by snapping to each of the 625 endpoints. Ensure to select the points in the correct sequence and zoom in to select very accurately. If an error is made, start over again. This will require about 30 minutes of uninterrupted work. Turn layer: 3D Pline off. Set the current view to SE Isometric and the current visual style to Realistic. (Figure Step 10)

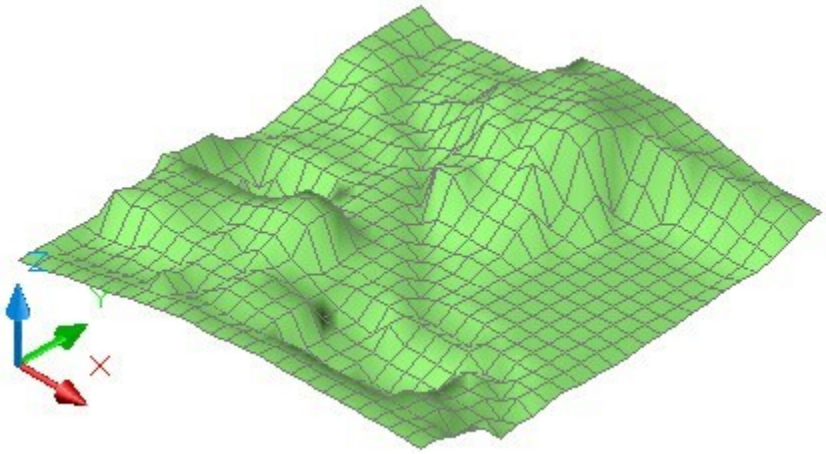


Figure Step 10

Step 11

Save and close the drawing.

Module 16 Competency Test No.3 Open Book

Learning Outcomes

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

1. Within a five hour time limit, complete a written exam and the lab exercises without the aid of a key.

The AutoCAD 3D 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. In a surface mesh, what is the intersection of a column and a row called?
    A. Crossing
    B. Point
    C. Junction
    D. Corner
    E. Vertex
2. What command creates a surface mesh by selecting 4 edges?
    A. EDGE

B. EDGESURF

C. REVSURF

D. TABSURF

E. RULESURF

3. What system variable(s) controls the surface mesh density of the M and N directions when a surface mesh is smoothed with the PEDIT command?

A. SURFMESH

B. SURFTAB1 and SURTAB2

C. SPLFRAME

D. SURFV and SURFU

E. SURFTYPE

4. What type of AutoCAD object is a two-dimensional solid created within a closed object? Choose the BEST answer.

A. Wireframe

B. Mesh

C. Region

D. Surface

E. Solid

5. A mesh is defined by rows and columns. What are the columns called?

A. M

B. S

C. R

D. C

E. N

6. What command creates a mesh from a path curve along a direction vector?

A. 3DMESH

B. EDGE

C. 3DFACE

D. REVSURF

E. TABSURF

7. What system variable(s) controls the mesh density of a surface mesh when the mesh is created?

A. SURFMESH

B. SURFTAB1 and SURTAB2

C. SPLFRAME

D. SURFV and SURFU

E. SURFTYPE

8. What command creates a surface face bounded by a minimum of three and a maximum of four edges?

A. 3DMESH

B. EDGE

C. 3DFACE

D. REVSURF

E. TABSURF

9. What system variable(s) controls the display of splines and meshes that have been smoothed with the PEDIT command?.

A. SURFMESH

B. SURFTAB1 and SURTAB2

C. SPLFRAME

D. SURFV and SURFU

E. SURFTYPE

10. What command creates an irregular complex surface mesh from a grid of XYZ coordinates?

A. 3DMESH

B. EDGE

C. 3DFACE

D. REVSURF

E. TABSURF

Lab Exercise 16-1A OPEN BOOK

Drawing Name	Template	Units
AutoCAD 3D Lab 16-1A	3D Layout English	Inches

Step 1

On layer: Model, draw a wireframe model of the Base. (Figure Step 1)

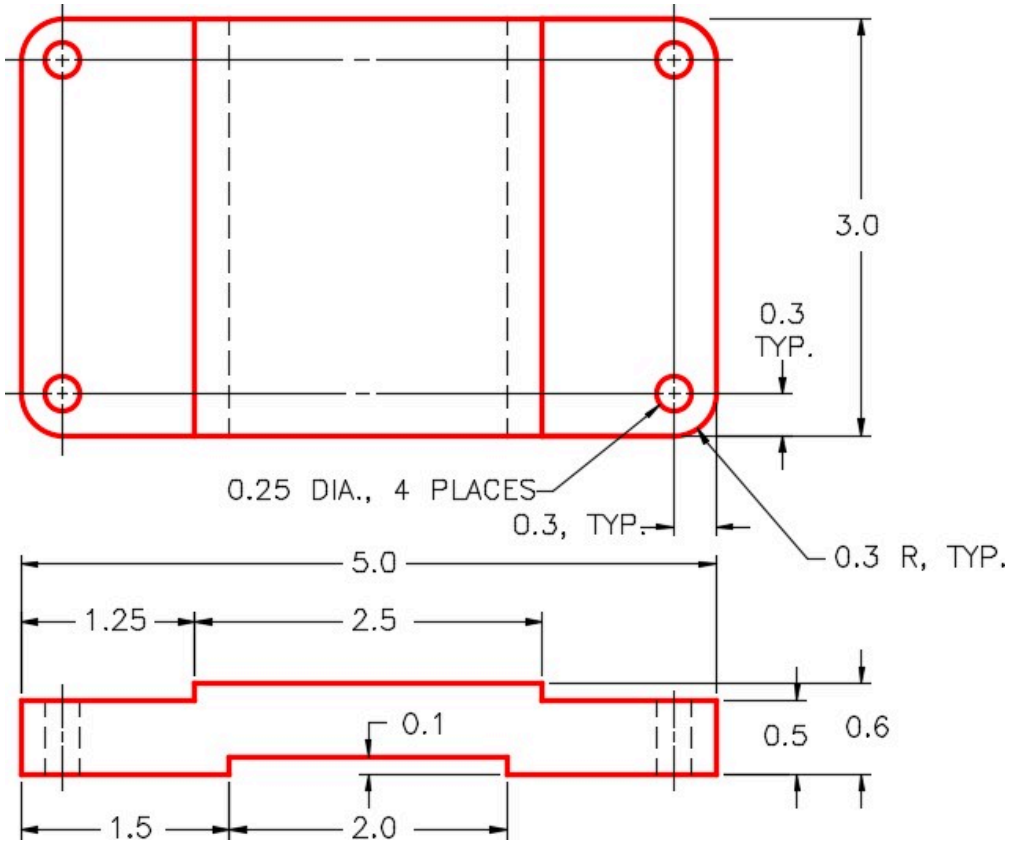


Figure Step 1
Dimensioned Multiview Drawing of the Base

Step 2

On layer: Surface Base, surface the complete model. Use faces, meshes, regions, or a mixture of them. (Figure Step 2)

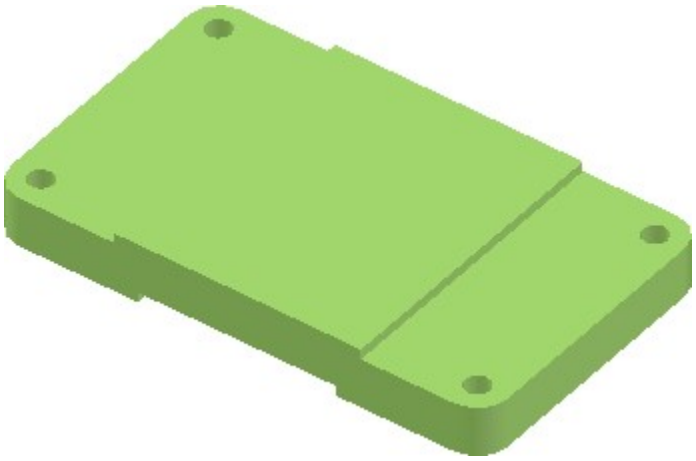


Figure Step 2
Surfaced Model of the Base

Step 3

Create a block of the model. Name it: AutoCAD 3D Lab 16-1A. Pick an appropriate location for 0,0,0. Make 0,0,0 as the insert point for the block. Ensure that the current UCS is located at the World when creating the block.

Lab Exercise 16-1B OPEN BOOK

Drawing Name	Template	Units
AutoCAD 3D Lab 16-1B	3D Layout English	Inches

Step 1

On layer: Model, draw a wireframe model of the Support. (Figure Step 1)

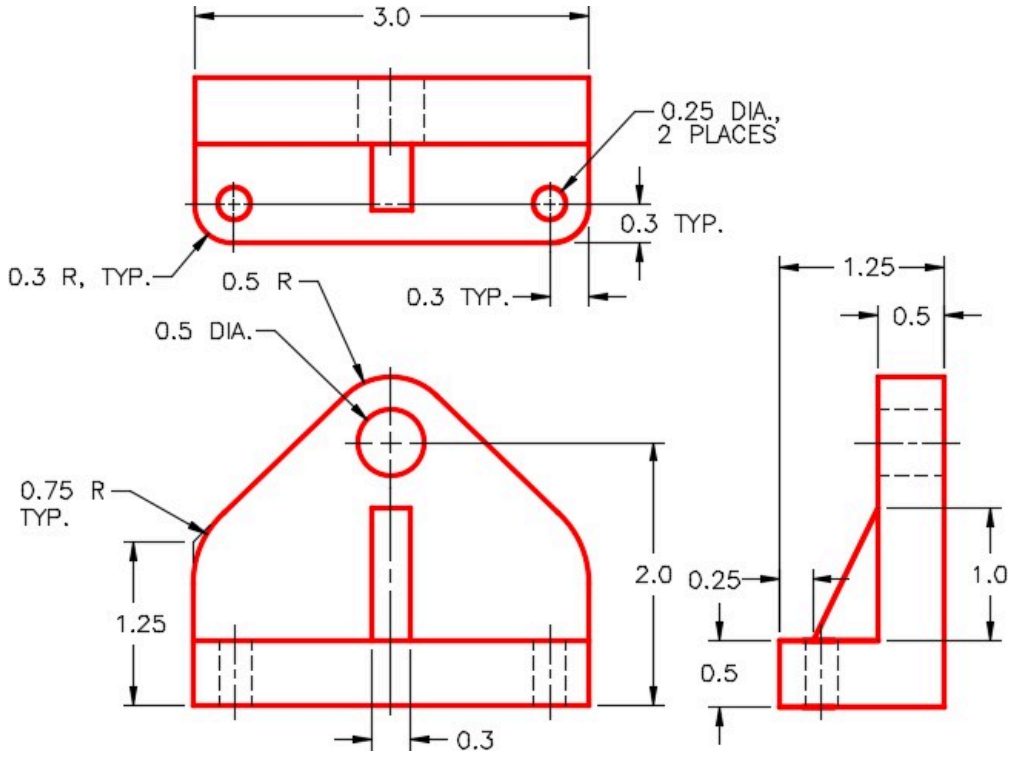


Figure Step 1
Dimensioned Multiview Drawing of the Support

Step 2

On layer: Construction, add any necessary construction objects that may help insert the model into the assembly.

Step 3

On layer: Surface Support, surface the complete model. Use faces, meshes, regions, or a mixture of them. (Figure Step 3)

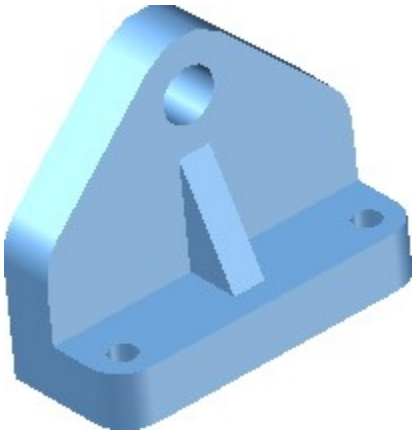


Figure Step 3 Surfaced Model of the Support

Step 4

Create a block of the model. Name it:

AutoCAD 3D Lab 16-1B. Pick an appropriate location for 0,0,0. Make 0,0,0 as the insert point for the block. Ensure that the current UCS is located at the World when creating the block.

Lab Exercise 16-1C OPEN BOOK

Drawing Name	Template	Units
AutoCAD 3D Lab 16-1C	3D Layout English	Inches

Step 1

On layer: Model, draw a wireframe model of the Axel. (Figure Step 1)

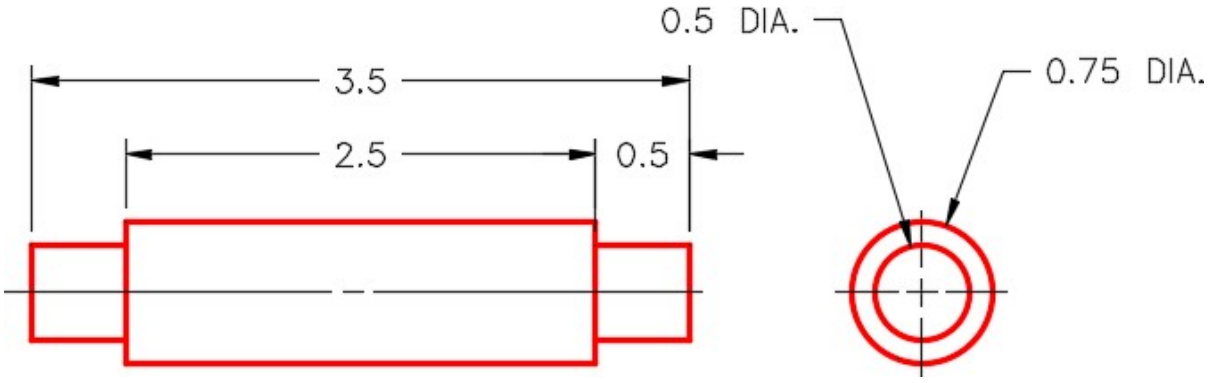


Figure Step 1
Dimensioned Multiview Drawing of the Axle

Step 2

On layer: Construction, add any necessary construction objects that may help insert the model into the assembly.

Step 3

On layer: Surface Axel, surface the complete model. Use faces, meshes, regions, or a mixture of them. (Figure Step 3)

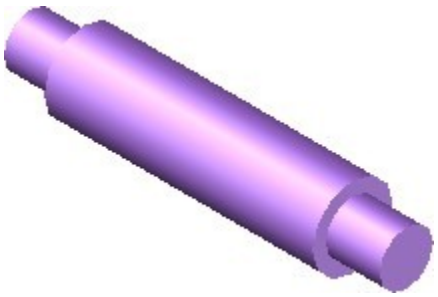


Figure Step 3
Surfaced Model of the Axle

Step 4

Create a block of the model. Name it:

AutoCAD 3D Lab 16-1C. Pick an appropriate location for 0,0,0. Make 0,0,0 as the insert point for the block. Ensure that the current UCS is located at the World when creating the block.

Lab Exercise 16-1D OPEN BOOK

Drawing Name	Template	Units
AutoCAD 3D Lab 16-1D	3D Layout English	Inches

Step 1

on layer Model, draw a wireframe model of the Wheel. (Figure Step 1)

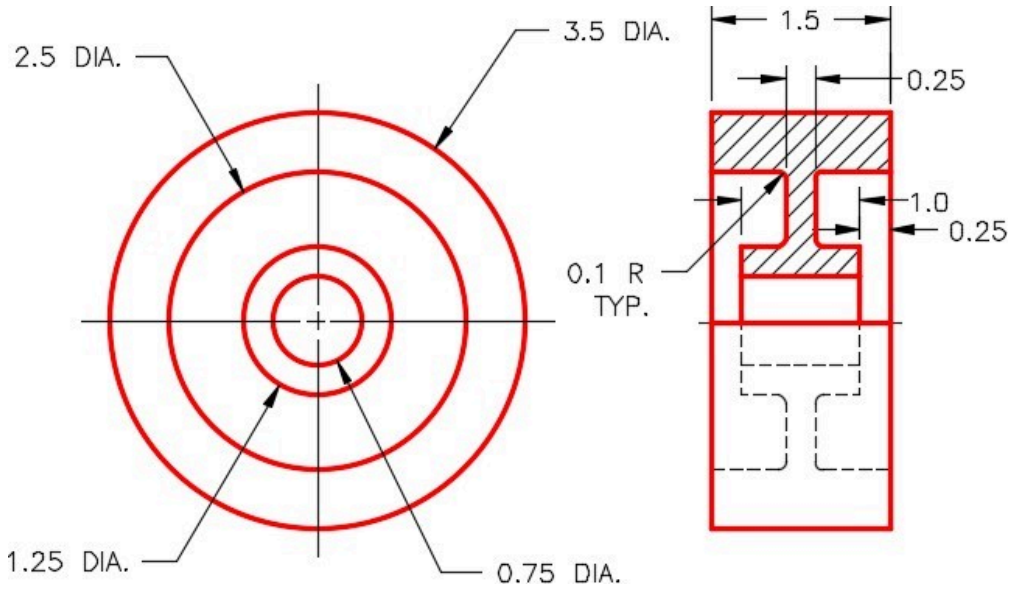


Figure Step 1
Dimensioned Multiview Drawing of the Wheel

Step 2

On layer Construction, add any necessary construction objects that may help insert the model into the assembly.

Step 3

On layer Surface Wheel, surface the complete model. Use faces, meshes, regions, or a mixture of them. (Figure Step 3)

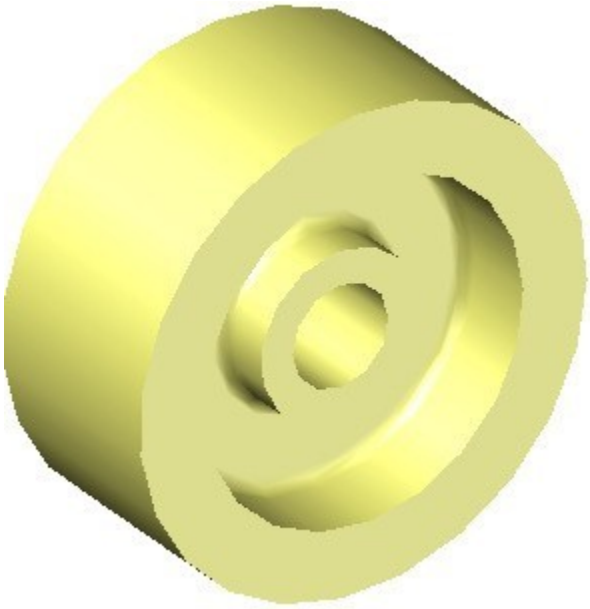


Figure Step 3
Surfaced Model of the Wheel

Step 4

Create a block of the model. Name it:

AutoCAD 3D Lab 16-1D. Pick an appropriate location for 0,0,0. Make 0,0,0 as the insert point for the block. Ensure that the current UCS is located at the World when creating the block.

Lab Exercise 16-1E OPEN BOOK

Drawing Name	Template	Units
AutoCAD 3D Lab 16-1E	3D Layout English	Inches

Step 1

On layer: Model, draw a wireframe model of the Washer. (Figure Step 1)

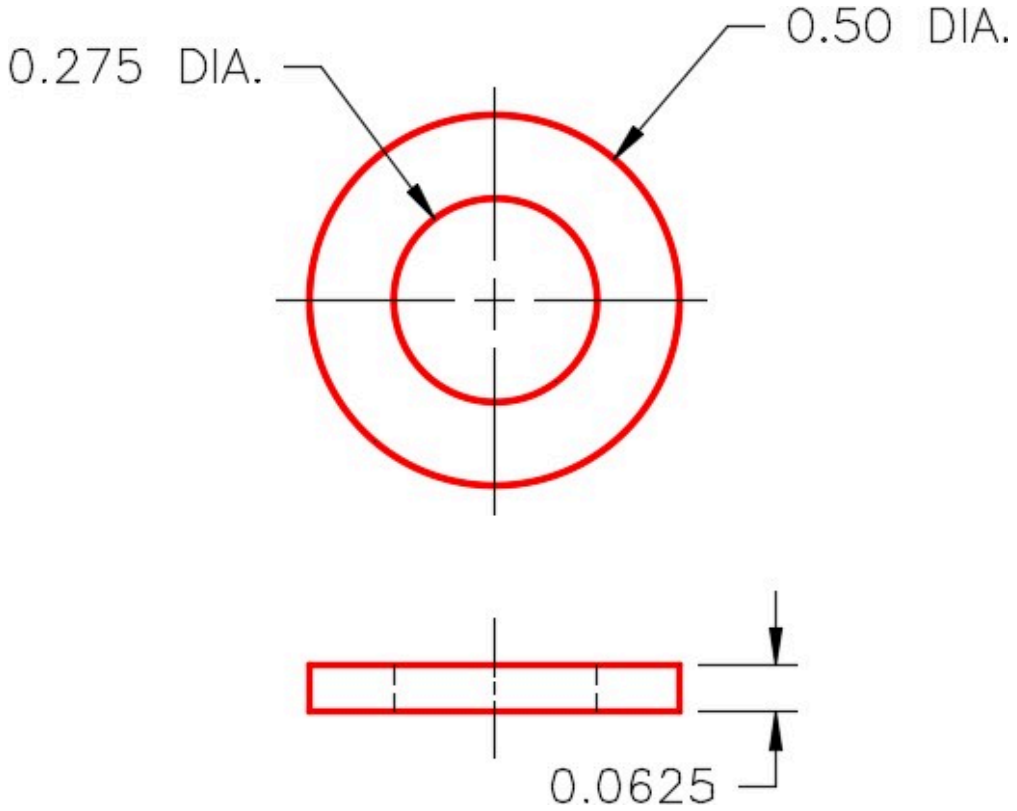


Figure Step 1
Dimensioned Multiview Drawing of the Washer

Step 2

On layer: Construction, add any necessary construction objects that may help insert the model into the assembly.

Step 3

On layer: Surface Washer, surface the complete model. Use faces, meshes, regions, or a mixture of them. (Figure Step 3)

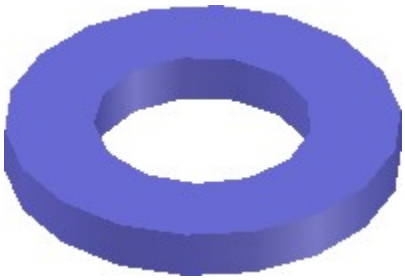


Figure Step 3 Surfaced Model of the Washer

Step 4

Create a block of the model. Name it:

AutoCAD 3D Lab 16-1E. Pick an appropriate location for 0,0,0. Make 0,0,0 as the insert point for the block. Ensure that the current UCS is located at the World when creating the block.

Lab Exercise 16-1F OPEN BOOK

Drawing Name	Template	Units
AutoCAD 3D Lab 16-1F	3D Layout English	Inches

Step 1

On layer: Model, draw a wireframe model of the Bolt. (Figure Step 1)

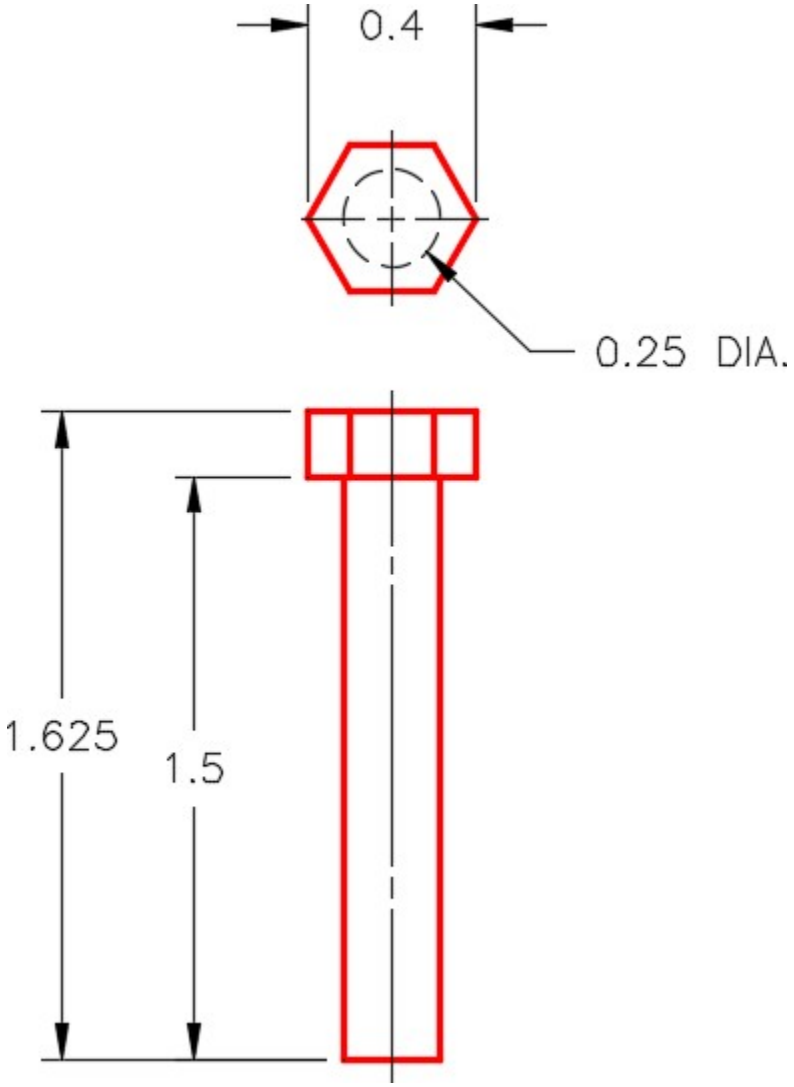


Figure Step 1
Dimensioned Multiview Drawing of the Bolt

Step 2

On layer: Construction, add any necessary construction objects that may help to insert the model into the assembly.

Step 3

On layer: Surface Bolt, surface the complete model. Use faces, meshes, regions, or a mixture of them. (Figure Step 3)

Figure Step 3 Surfaced Model of the Bolt

Step 4

Create a block of the model. Name it:

AutoCAD 3D Lab 16-1F. Pick an appropriate location for 0,0,0. Make 0,0,0 as the insert point for the block. Ensure that the current UCS is located at the World when creating the block.

Lab Exercise 16-1 OPEN BOOK

Drawing Name	Template	Units
AutoCAD 3D Lab 16-1	3D Layout English	Inches

Step 1

Start a new drawing and save it with the name: AutoCAD 3D Lab 16-1.

Step 2

Open the six drawings that were created earlier in this lab exercise.

Step 3

Set the UCS to World and using DesignCenter, insert the following blocks into the drawing: AutoCAD 3D Lab 16-1.

1- AutoCAD 3D Lab 16-1A

2- AutoCAD 3D Lab 16-1B

1- AutoCAD 3D Lab 16-1C

1- AutoCAD 3D Lab 16-1D

4- AutoCAD 3D Lab 16-1E

4- AutoCAD 3D Lab 16-1F

Step 4

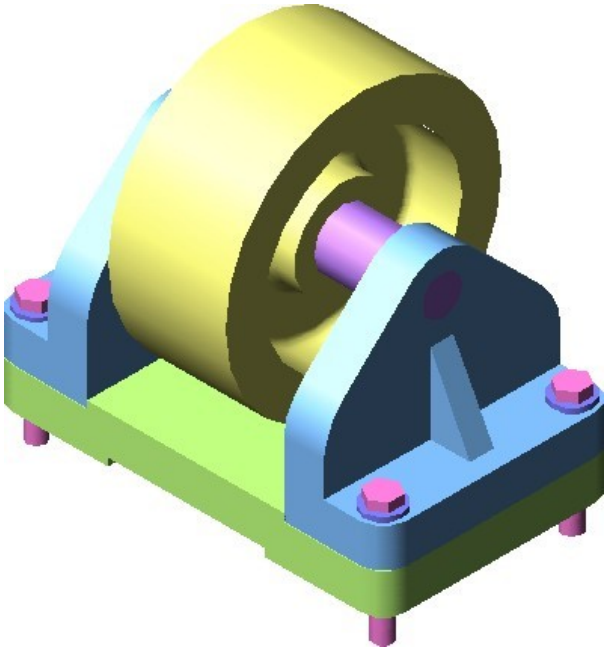
Close all drawings except for: AutoCAD 3D Lab 16-1

Step 5

Ensure that layers: Model and Construction are on and all surface layers are off. Move the blocks into their exact location to create the assembly using the model or construction objects and object snap to help locate them accurately.

Step 6

Turn on all surface layers and turn off the layers: Model and Construction. (Figure Step 6)



Step 7

With UCS set to World, copy the complete Caster Assembly 10 inches in the X direction. Rotate the copy of the assembly 180 degrees around the X axis. (Figure Step 7)

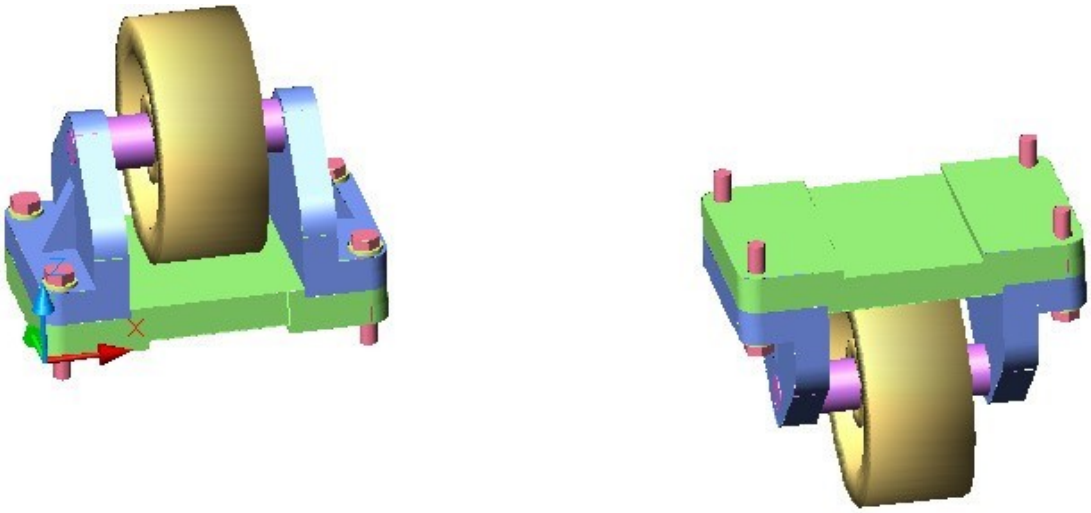


Figure Step 7

Part 4

Module 17: Solid Modeling – Part 1

Learning Outcomes

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

1. List the six solid primitives.
2. Apply the BOX, WEDGE, CYLINDER, UNION, and SUBTRACT commands to draw solid models using solid models using solid primitives.

Solid Modeling

A *solid model* is the best possible computerized representation of a real object. A solid model is one AutoCAD object. It is a much more complete model then a wireframe or a surfaced model. A solid model, unlike a hollow wireframe or surfaced model, is solid. A solid model can be rendered or shaded plus the mass properties can be obtained from it. Displaying the mass properties is taught in Module 25.

In a lot of ways, constructing a solid model is simpler than constructing a surfaced model. It is important to understand that when you construct solid models, you must still use all of the 3D construction techniques that were taught in the first 16 modules.

Solid Primitives

The most basic method of constructing solid models is to create a series of solid primitives and then add them together or subtract them from one another to complete the solid model. Using solid primitives is not the best method of solid modeling but at times they can be a very useful modeling tool. The six primitives are the box, wedge, cylinder, sphere, cone and torus. See Figure 17-1. In this module, you will be constructing solid boxes, wedges, and cylinders.

The next four modules teaches many different commands and techniques used to construct solid models. You can pick and choose how and when to use them to construct each model.

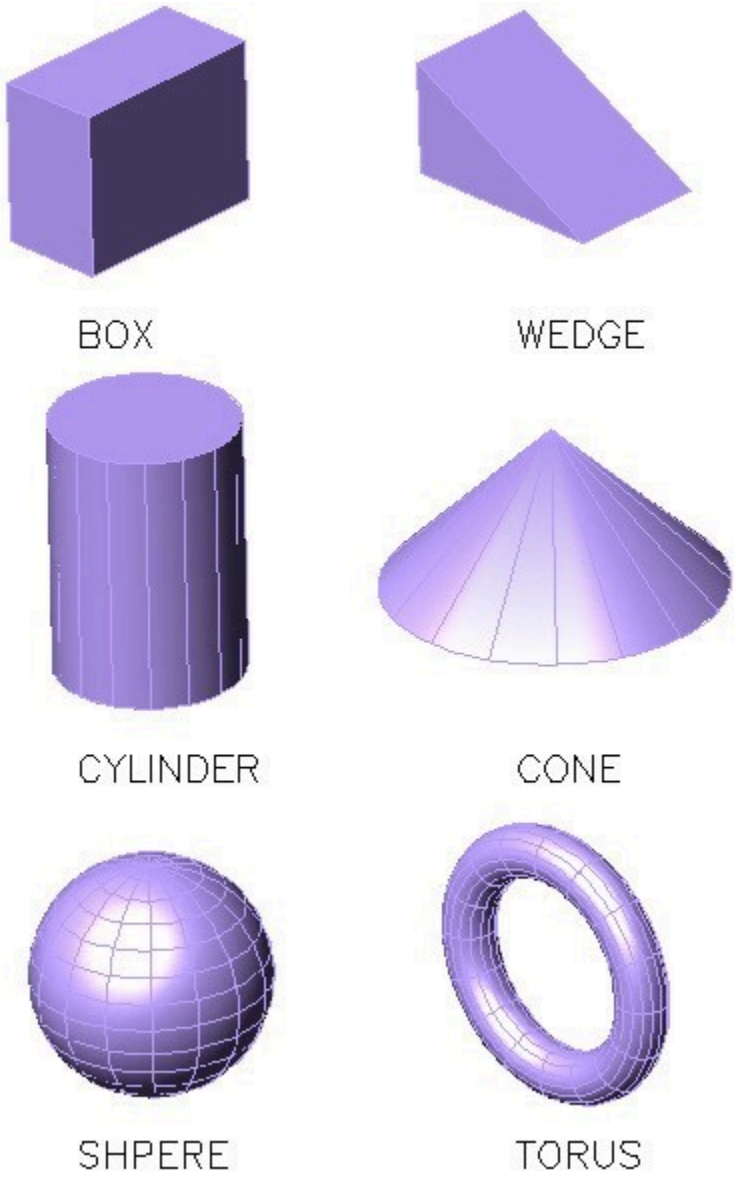
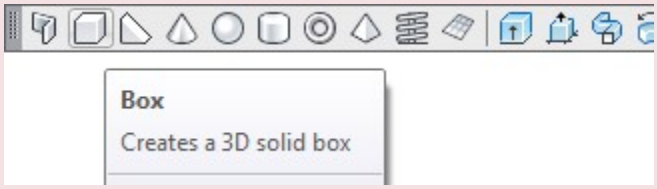


Figure 17-1
Solid Primitives

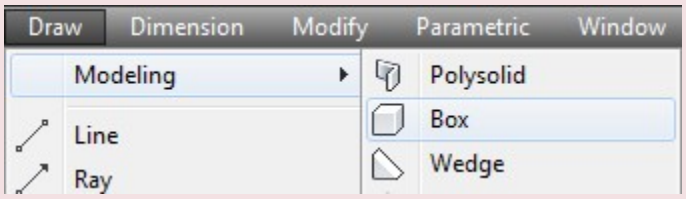
AutoCAD Command: BOX

The BOX command is used to create a solid box primitive.

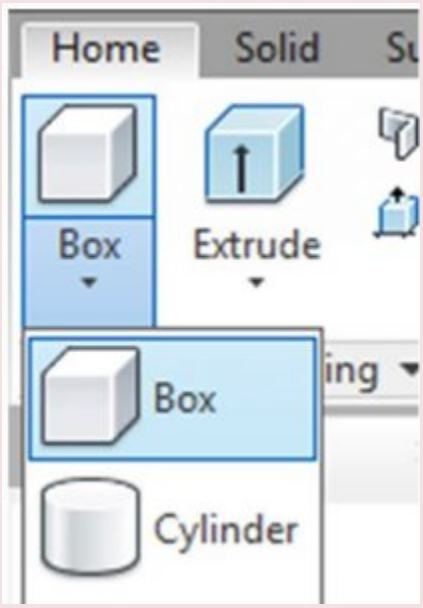
Shortcut: none



Modeling Toolbar



Draw Pull-down

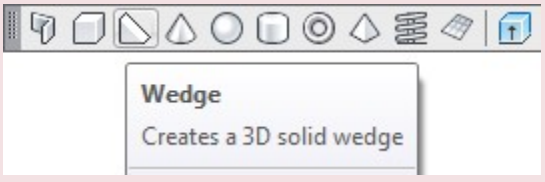


Home Ribbon

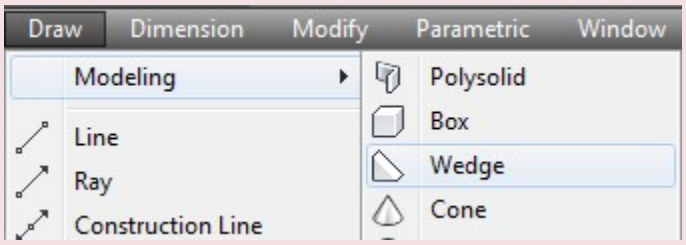
AutoCAD Command: WEDGE

The WEDGE command is used to create a solid wedge primitive.

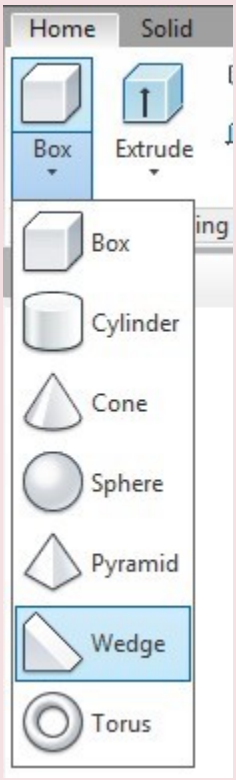
Shortcut: none



Modeling Toolbar



Draw Pull-down



Home Ribbon

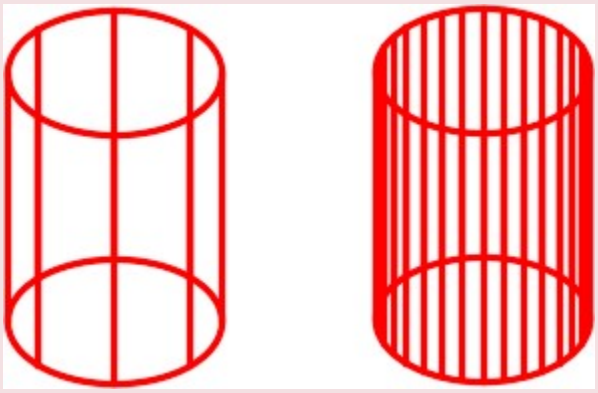
AutoCAD Command: ISOLINES

The ISOLINES system variable is used set the number of contour lines that a curved surface solid model will be constructed with. The valid settings are from 0 to 2047. After you change the setting with this variable, the drawing must be regenerated to display the revised setting.

Shortcut: none Command: **ISOLINES**

Enter new value for ISOLINES <8>: **32**

Command:

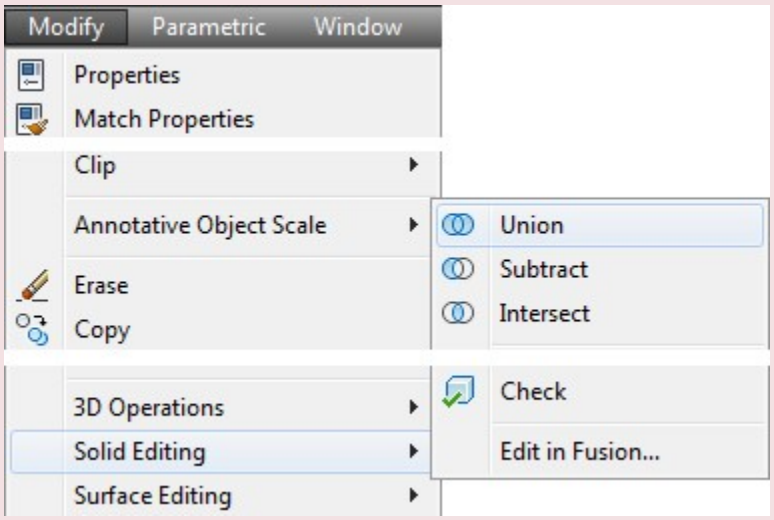


Isolines = 8 *Isolines = 32*

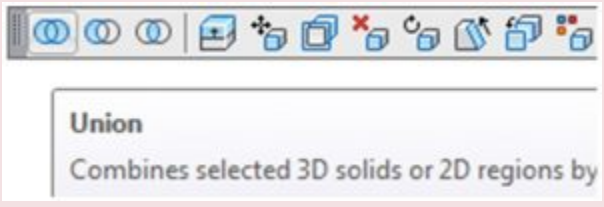
AutoCAD Command: UNION

The UNION command is used to join two or more solids together to form one solid object.

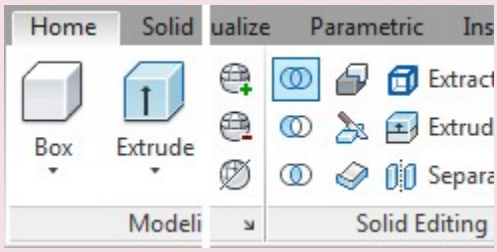
Shortcut: none



Modify Pull-down



Solid Editing Toolbar

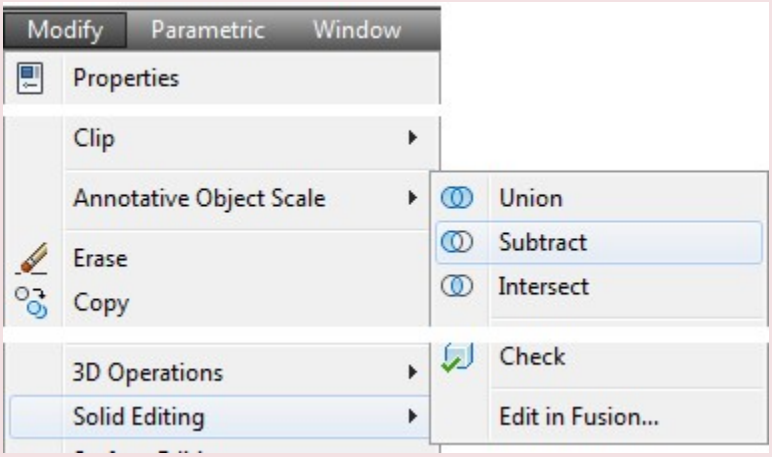


Home Ribbon

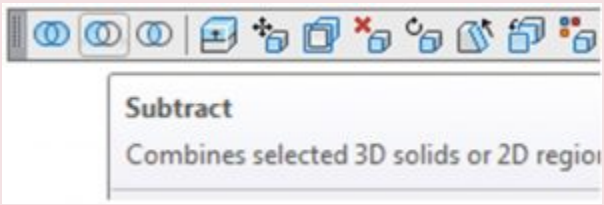
AutoCAD Command: SUBTRACT

The SUBTRACT command is used to subtract one or more solids from another solid.

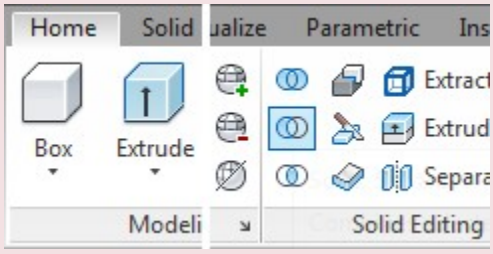
Shortcut: **SU**



Modify Pull-down



Solid Editing Toolbar

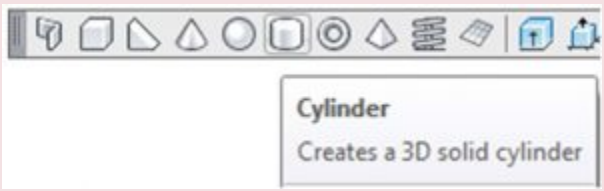


Home Ribbon

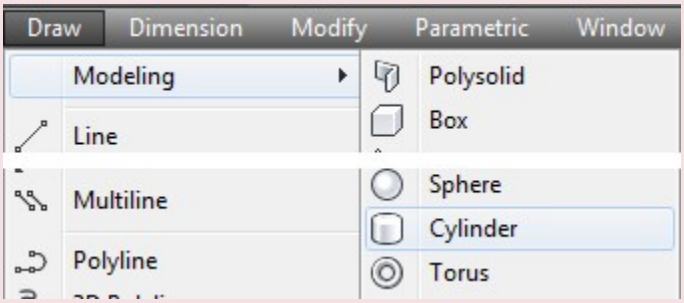
AutoCAD Command: CYLINDER

The CYLINDER command is used to create a solid cylinder primitive.

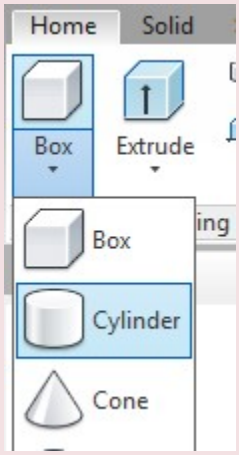
Shortcut: none



Modeling Toolbar



Raw Pull-down



Home Ribbon

WORK ALONG: Creating Solid Models Using Boxes, Wedges and Cylinders

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 17-1. (Figure Step 2A and 2B)

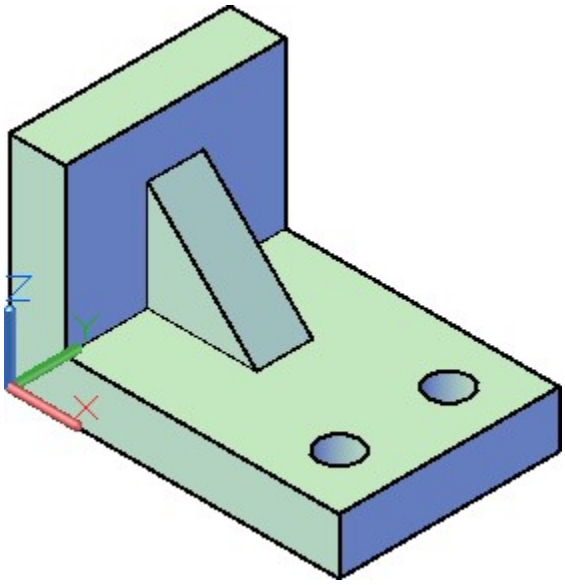


Figure Step 2A
SE Isometric View – Solid Model

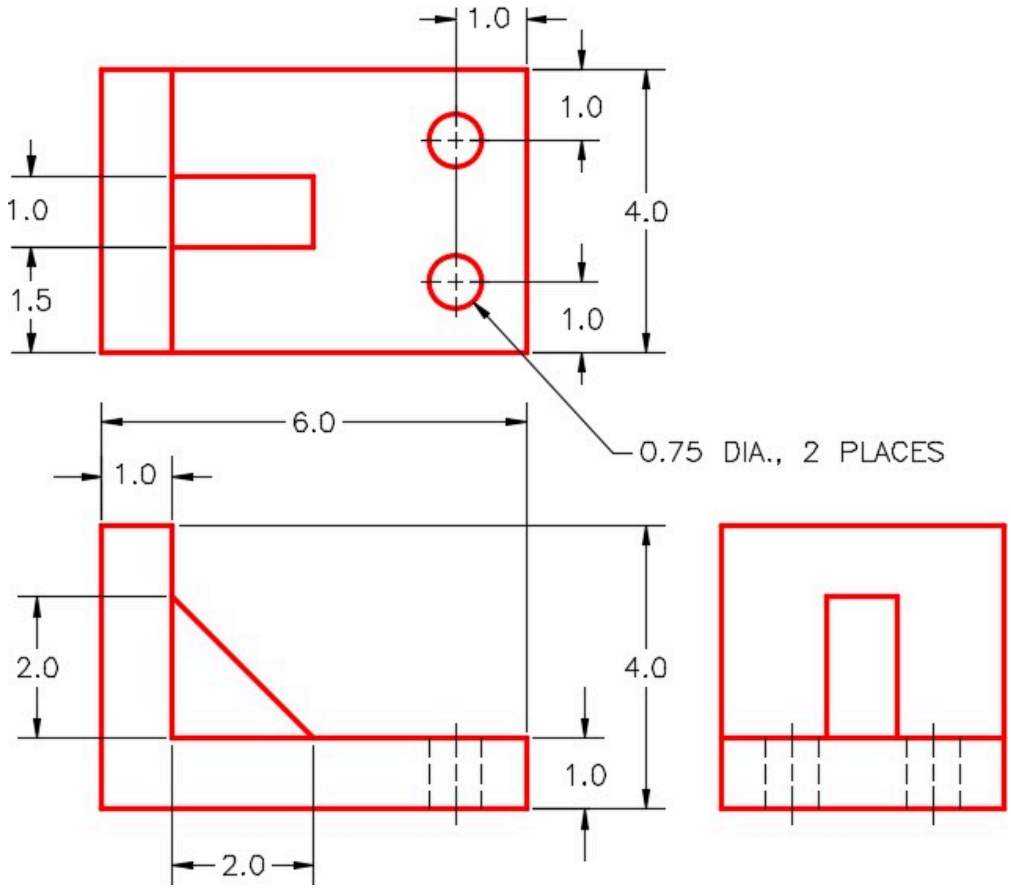


Figure Step 2B
Dimensioned Multiview Drawing

Step 3

Set the current view to SE Isometric, the current UCS to World and the current visual style to 2D Wireframe.

Step 4

Set the current layer to Solid 7.

Step 5

Enter the BOX command, as shown below, to draw the solid box. (Figure Step 5).

Command: **BOX**

Specify first corner or [Center]: 0,0,0 Specify other corner or [Cube/Length]: **@6,4**

(6 units in the positive X and 4 units in the positive Y.)

Specify height or [2Point] <3.1668>: **1**

(1 unit in the positive Z.)

Command:

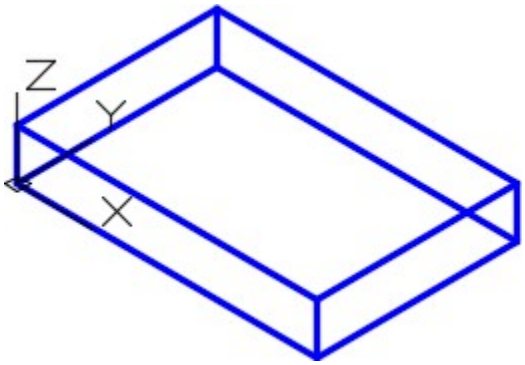


Figure Step 5

AUTHOR'S COMMENTS: Ensure that you disable osnap mode when you are not using it. In 3D, an enabled osnap mode can cause a lot of problems. In the BOX command in Step 6, as soon as you snap to the first corner, disable osnap for the remainder of the command.

Step 6

Enter the BOX command, as shown below, to add a box to the model. (Figure Step 6A and 6B).

Command: **BOX**

Specify first corner or [Center]: *(end)* **P1**

Specify other corner or [Cube/Length]: **@1,4**

Specify height or [2Point] <1.0000>: **3**

Command:

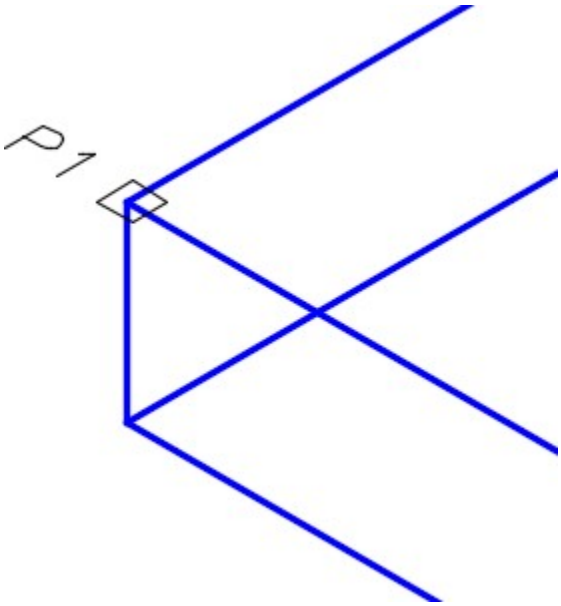


Figure Step 6A

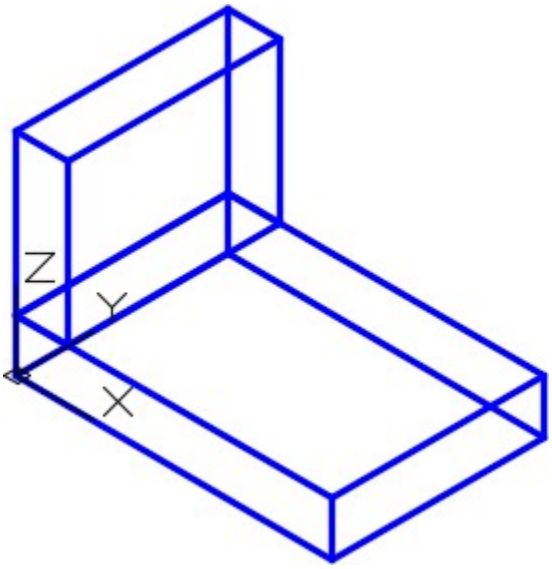


Figure Step 6B

Step 7

Using on of the orbit commands, orbit the model slightly. Enter the WEDGE command, as shown below, to create a solid wedge primitive as shown in the figure. Using the MOVE command, move the

wedge to centre it. Move the wedge by snapping to the midpoint of the wedge to the midpoint on the box. (Figure Step 7A, 7B, and 7C)

Command: **WEDGE**

Specify first corner or [Center]: *(mid)*

(See Figure Step 7A)

Specify other corner or [Cube/Length]: **@2,1**

Specify height or [2Point] <3.0000>: **2**

Command:

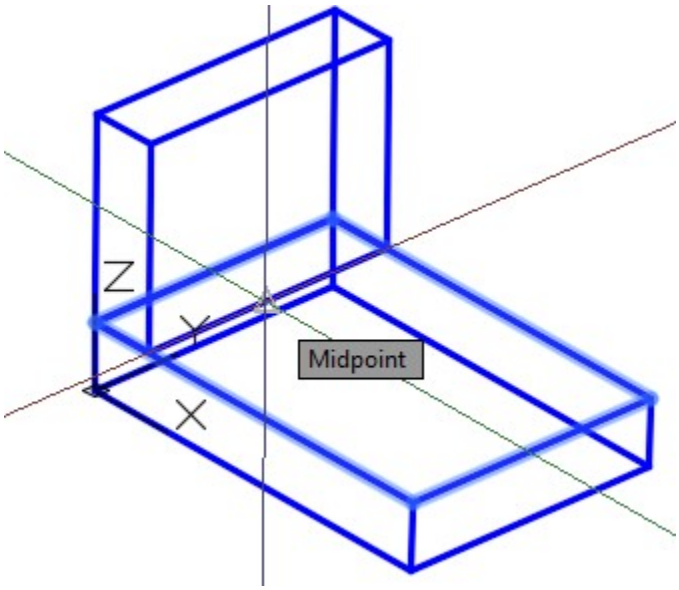


Figure Step 7A

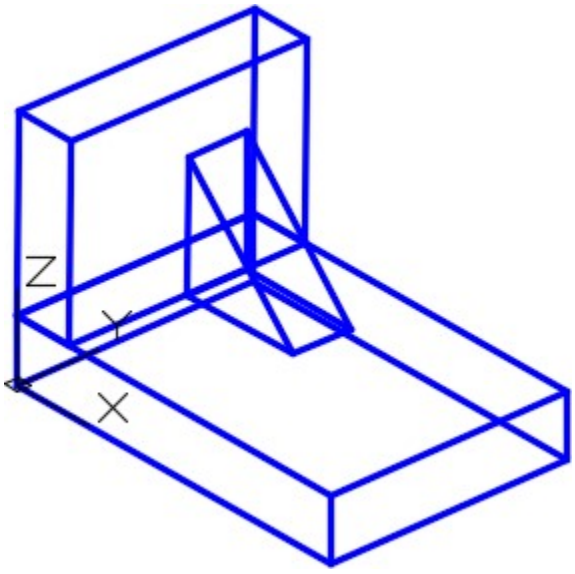


Figure Step 7B

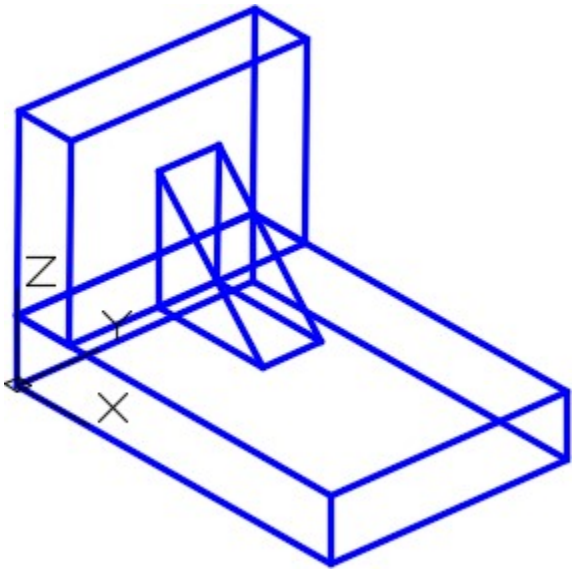


Figure Step 7C

AUTHOR'S COMMENTS: The wedge was drawn in the wrong location to save doing the math and then moved to its correct location. The less math you do, the fewer mistakes you will make.

AUTHOR'S COMMENTS: Unlike a wireframe, you cannot offset an edge of a solid model as a construction technique. What you have to do is draw construction lines on the model first. You can snap to the corners of the edges of the model so it is relatively simple to draw the construction lines. After you draw the construction lines, you can offset them.

Step 8

Draw three construction lines on the edges of the box. (Figure Step 8)

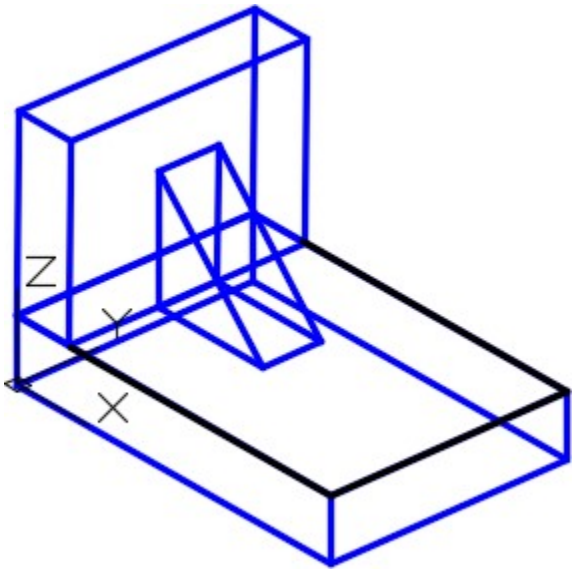


Figure Step 8

AUTHOR'S COMMENTS: Keep in mind that the OFFSET command is a 2D command so you must locate the current UCS in the correct orientation before you use it. In this case, World or Top will work.

Step 9

Offset the construction lines to locate the centre of the holes. (Figure Step 9)

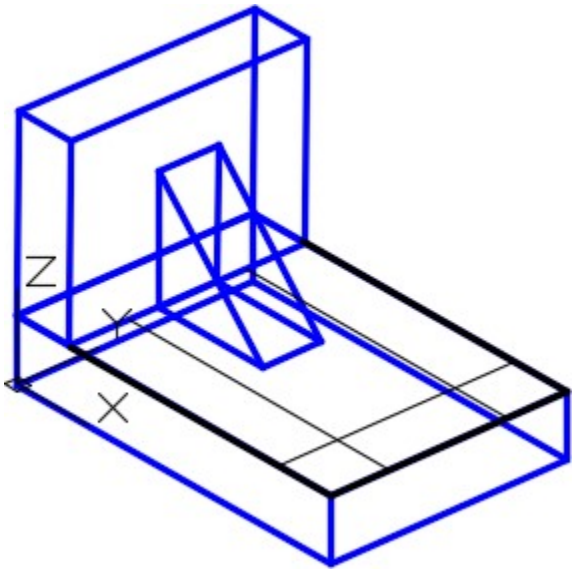


Figure Step 9

AUTHOR'S COMMENTS: The ISOLINES system variable is used set the number of contour lines that a curved surface solid model will be constructed with. They appear in the wireframe visual style of the model. The higher the setting, the smoother the solid will appear when it is shaded or rendered. The down side is that the higher the number of isolines used, the more calculations are required when AutoCAD regenerates the model, which in turn slows you down. The valid settings are from 0 to 2047. After you change the setting with this variable, you must regenerate the drawing to display the revised setting on the model.

Step 10

Enter the ISOLINES system variable, as shown below, setting it to 16. Command: **ISOLINES**

Enter new value for ISOLINES <4>: **16**

Command:

Step 11

Enter the CYLINDER command, as shown below, to construct a cylinder for the left hole. After you do that, construct the cylinder on the right side. (Figure Step 11A and 11B)

Command: **CYLINDER**

Current wire frame density: ISOLINES=16

Specify centre point for base of cylinder or [Elliptical] <0,0,0>: *(int)* **P3**

Specify radius for base of cylinder or [Diameter]: **D**

Specify diameter for base of cylinder: **0.75**

Specify height of cylinder or [Center of other end]: **-1**

(Since the cylinder is constructed 1 unit in the -Z direction -1 is used here.)

Command:

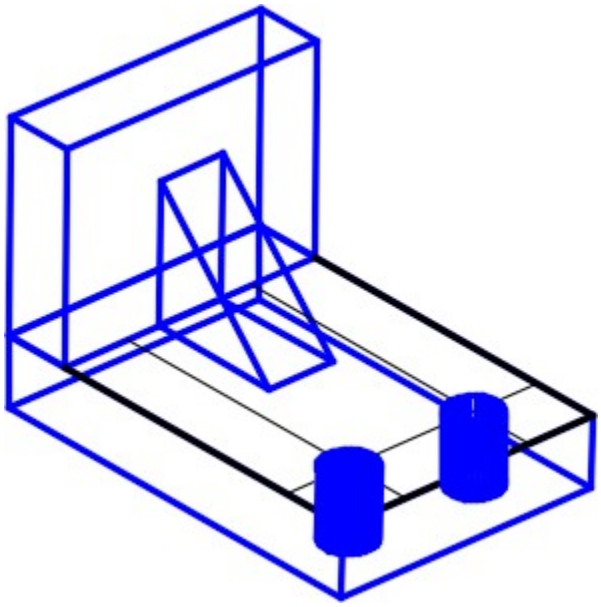


Figure Step 11A

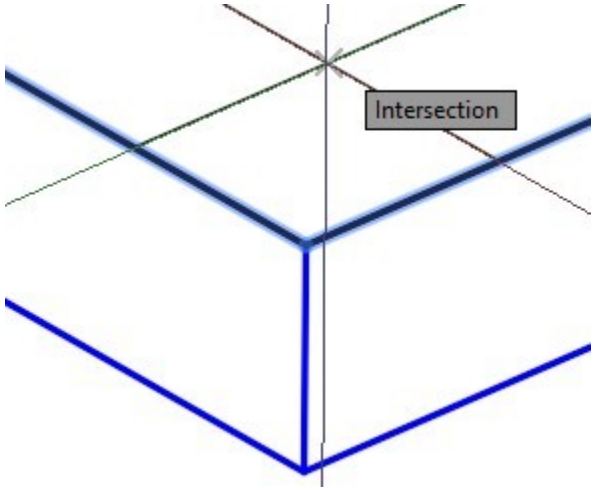


Figure Step 11B

Step 12

Turn layer Construction off and your model should appear as shown in the figure. (Figure Step 12)

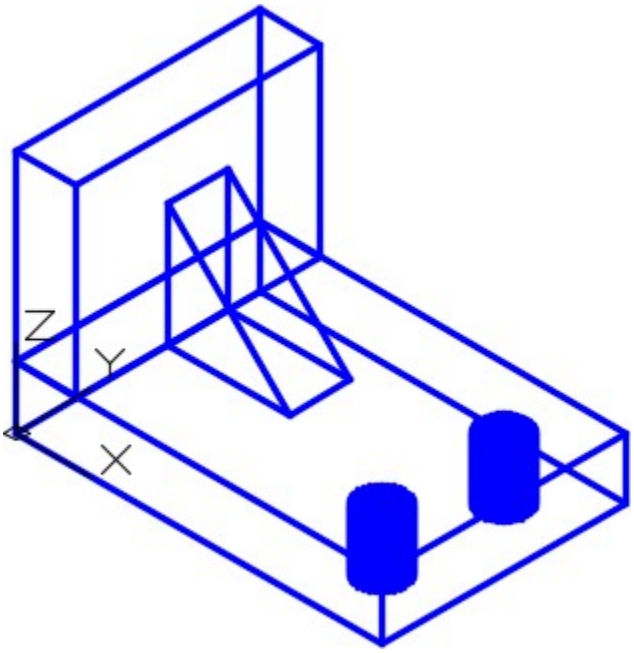


Figure Step 12

Step 13

Enter the UNION command, as shown below, to create one solid from the two boxes and the wedge. (Figure Step 13)

Command: **UNION**

Select objects: 1 found

Select objects: 1 found, 2 total

Select objects: 1 found, 3 total

(Select one solid at a time by picking them.)

Select objects:

Command:

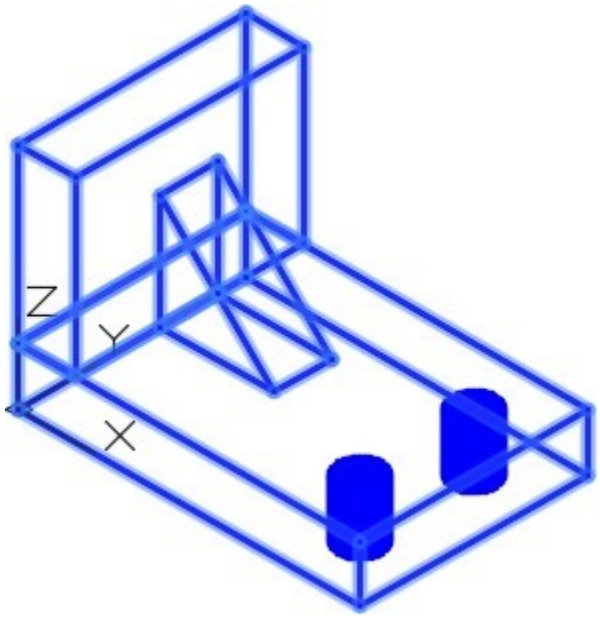


Figure Step 13

Step 14

Your model is now one solid and should appear as shown in the figure. (Figure Step 14)

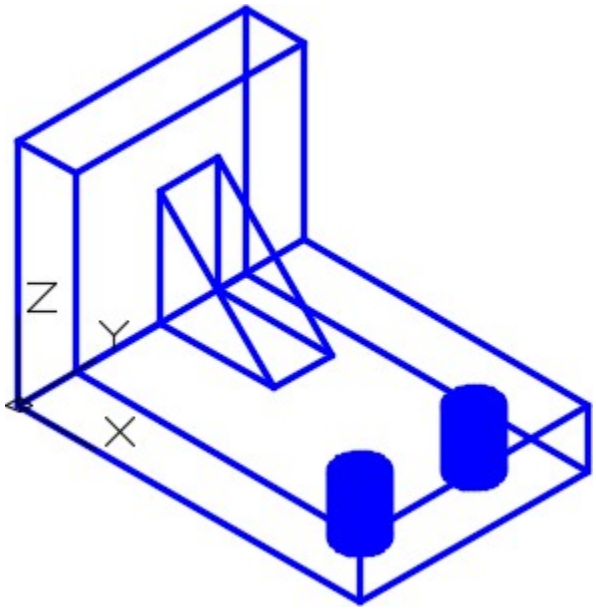


Figure Step 14

AUTHOR'S COMMENTS: When complete, a solid model must be one object. It cannot be a combination of solids. Use the UNION and/or the SUBTRACT commands to do this.

Step 15

Enter the SUBTRACT command, as shown below, to subtract the two cylinders from the model to create the holes. (Figure Step 15A and 15B)

Command: **SUBTRACT**

Select solids and regions to subtract from ..

Select objects: 1 found

(Select the solid model. See Figure Step 15A)

Select objects:

(Press Enter to change to subtract mode.)

Select solids and regions to subtract .. Select objects: 1 found

Select objects: 1 found, 2 total

(Select each cylinder by picking them one at a time See Figure Step 15B.)

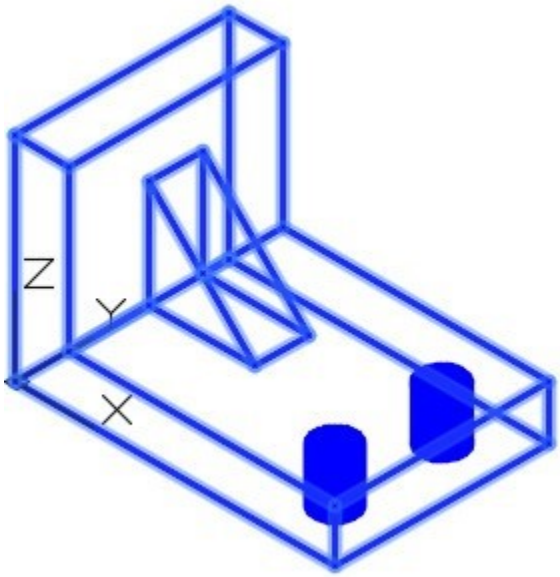


Figure Step 15A

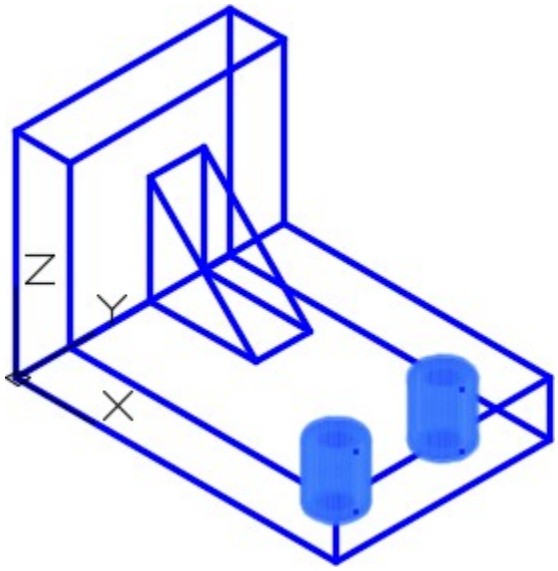


Figure Step 15B

Step 16

Open the Properties window and select the solid model. Ensure it is one AutoCAD object only. (Figure Step 16)

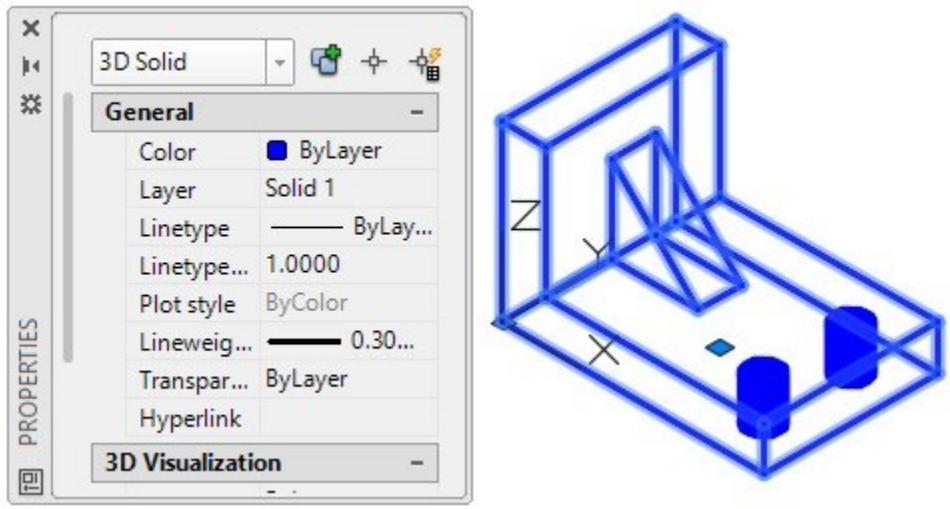


Figure Step 16

AUTHOR'S COMMENTS: When the solid is selected, it should all highlight and display dashed indicating it is one solid object. Another way to check if the solid is one object is to select the solid and delete it. It should totally disappear. Undo the command to undelete the solid.

Step 17

Set the current visual style to Realistic. (Figure Step 17)

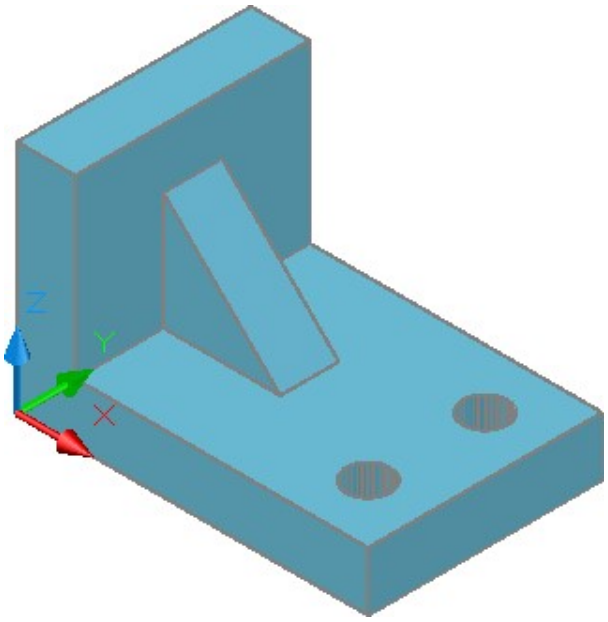
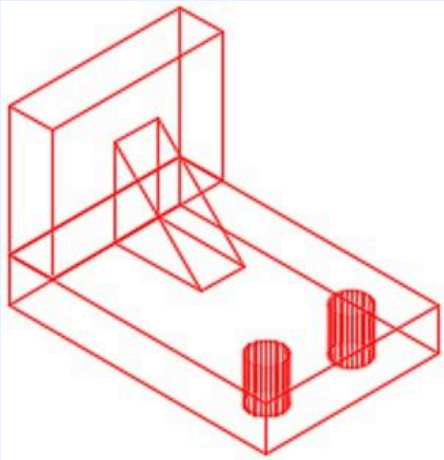


Figure Step 17

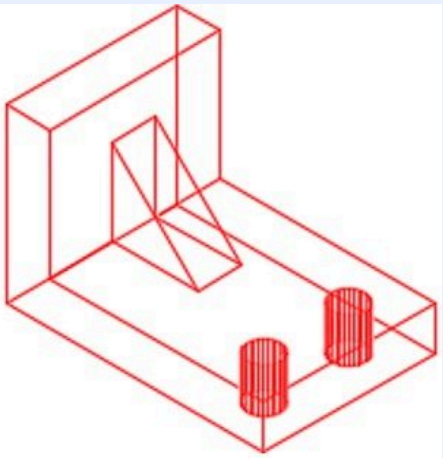
Step 18

Save and close the drawing.

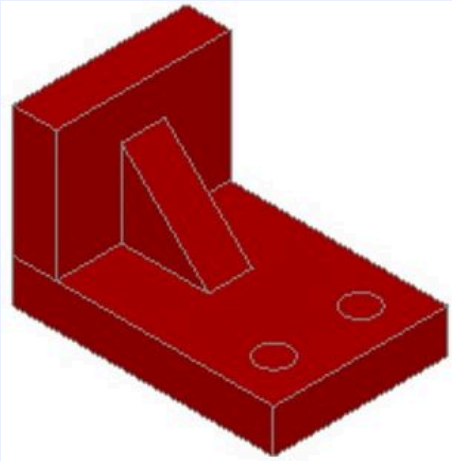
MUST KNOW: When two or more solid primitives are used to construct a solid model, the UNION and/or SUBTRACT commands must be used to create one solid object which is one AutoCAD object. When complete, a solid model must be only one object.



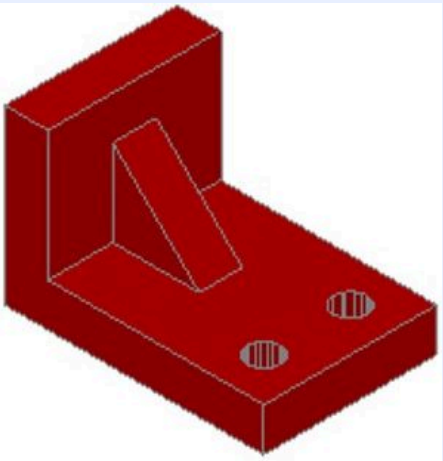
Five Solids that Appear as One Solid. Object in AutoCAD



One Solid Model or One



Five Solids that Appear as One Solid Shaded



The Solid Model Shaded

Key Principles

Key Principles in Module 17

1. A solid model is the best possible computerized representation of an object. It is a much more complete model then a wireframe or a surfaced model. A solid model can be rendered or shaded plus the mass properties can be obtained from it.
2. Using solid primitives is not the best method of solid modeling, but at times they can be a very useful modeling tool.
3. Ensure that you disable osnap mode when you are not using When working in 3D, an enabled osnap mode can cause a lot of problems.

4. The ISOLINES system variable is used set the number of contour lines that are used to construct curved surfaces in the solid model.

Lab Exercise 17-1

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 17-1	3D Layout English	Inches

Step 1

On layer Solid 3, draw a solid model of the object. (Figure Step 1A, 1B, and 1C)

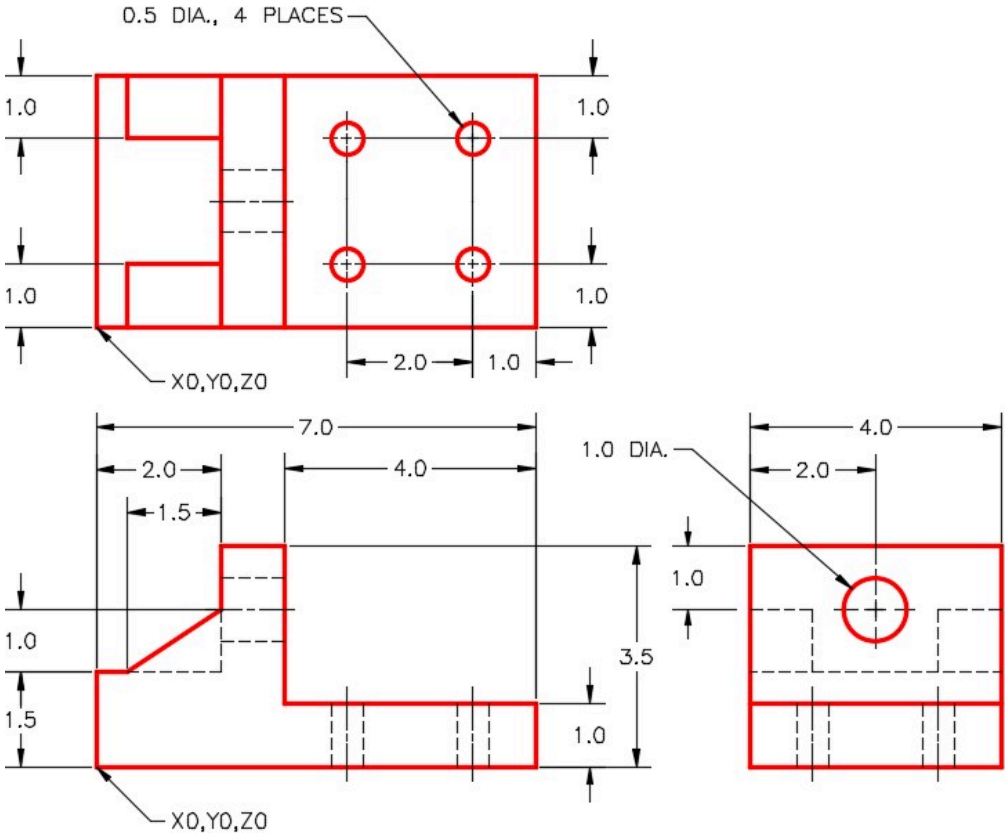


Figure Step 1A
Dimensioned Multiview Drawing

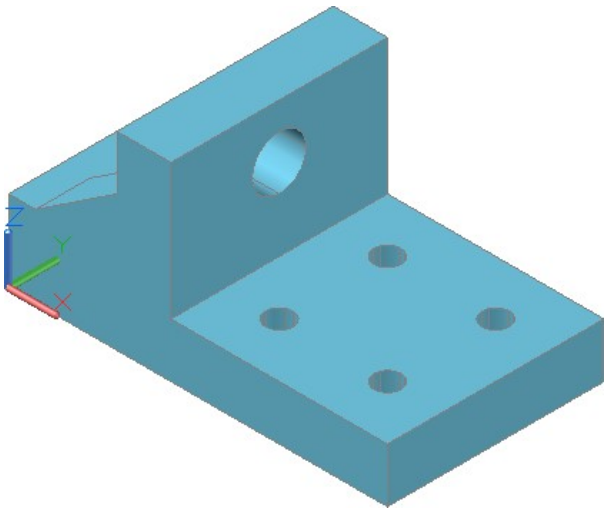


Figure Step 1B Completed Solid Model SE Isometric View

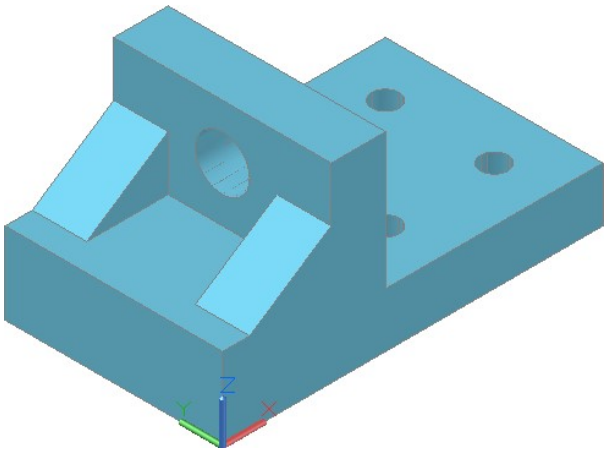


Figure Step 1C Completed Solid Model SW Isometric View

Step 2

Set the Isolines to 16.

Step 3

Use the UNION and SUBTRACT commands to create one solid object only.

Step 4

Set the current visual style to Realistic and check the model by orbiting it with the with the 3DFORBIT command.

Step 5

Check to ensure that the solid is one object only. See Step 16 in WORKALONG: Creating Solid Models Using Boxes, Wedges and Cylinders.

Step 6

Save and close the drawing.

Lab Exercise 17-2

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 17-2	3D Layout Metric	Millimeters

Step 1

On layer Solid 1, draw a solid model of the object. (Figure Step 1A, 1B, and 1C)

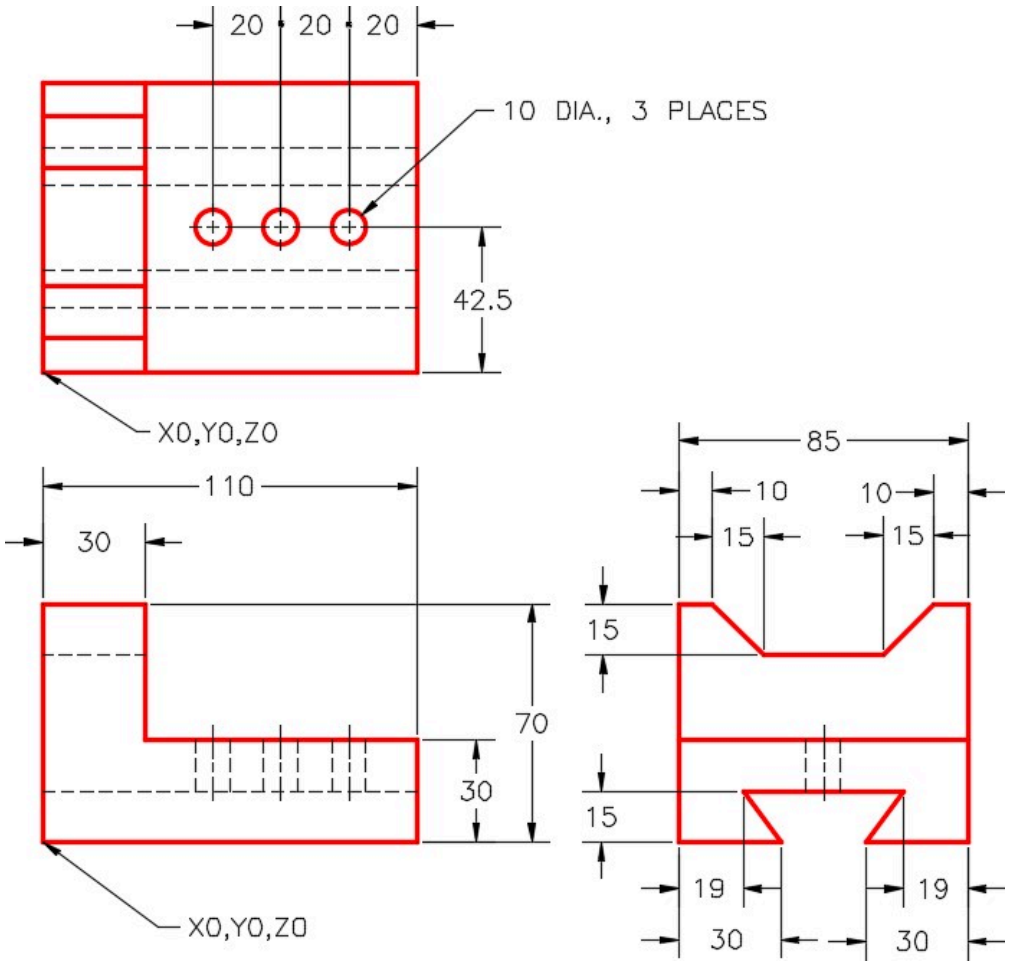


Figure Step 1A
Dimensioned Multiview Drawing

Step 2

Set the Isolines to 20.

Step 3

If necessary, use the UNION and SUBTRACT commands to create one solid object only.

Step 4

Set the current visual style to Realistic and check the model by orbiting it with the with the 3DFORBIT command.

Step 5

Check to ensure that the solid is one object only. See Step 16 in WORKALONG: Creating Solid Models Using Boxes, Wedges and Cylinders.

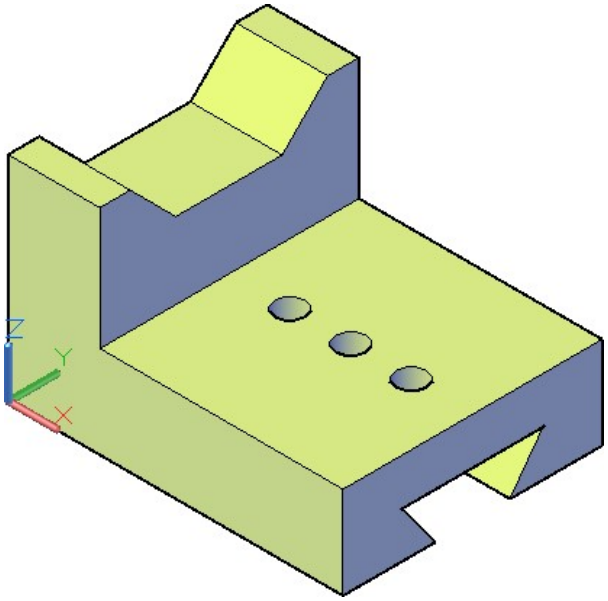


Figure Step 1B Completed Solid Model SE Isometric View

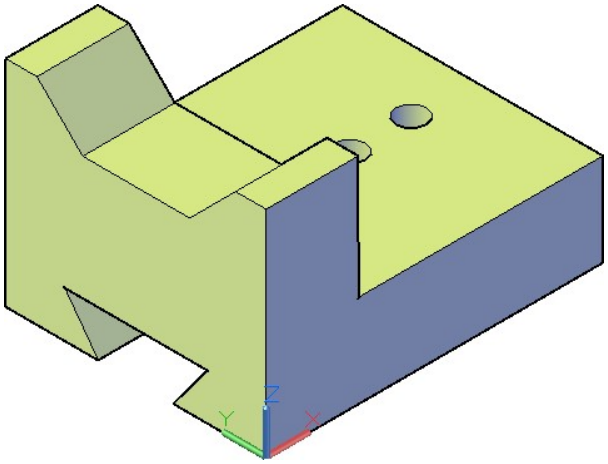


Figure Step 1C Completed Solid Model SW Isometric View

Step 6

Save and close the drawing.

Module 18 Solid Modeling – Part 2

Learning Outcomes

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

1. Describe how solid models are created by extrusion or revolving.
2. Apply the JOIN command to create a single 2D or 3D object from existing objects.
3. Apply the EXTRUDE, PRESSPULL, and REVOLVE commands to draw solid models.
4. Set the DELOBJ system variable to control objects used to create solid models.

Constructing Solid Models Without Using Solid Primitives

Constructing most solid models using solid primitives would be to difficult and slow. It is much faster and simpler to construct most solid models using the EXTRUDE and/or the REVOLVE commands.

Extruding

Extruding involves drawing a closed object and, using the EXTRUDE or the PUSHPULL command, project it in the Z direction at a given distance. See Figure 18-1. The closed object can be a 2D polygon, a circle or an ellipse. Extruded solids can then be joined with the UNION command or subtracted using the SUBTRACT command to form the final solid model.

Revolving

Not all solid models can be extruded. A solid model that is symmetrical can be created by *revolving* a closed object. See Figure 18-2. The closed 2D object can be a polygon, a circle or an ellipse. It is then revolved around an axis. The contour of the object will create the solid as it is revolved around the axis. It can be revolved any angle from 1 degree to 360 degrees.

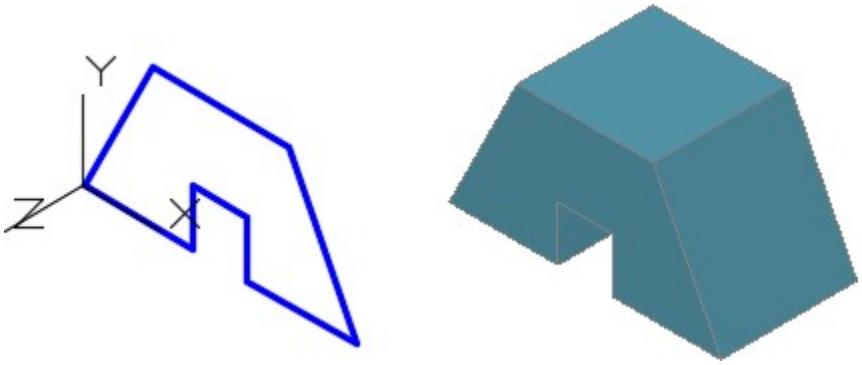


Figure 18-1
Extruding a Solid Model

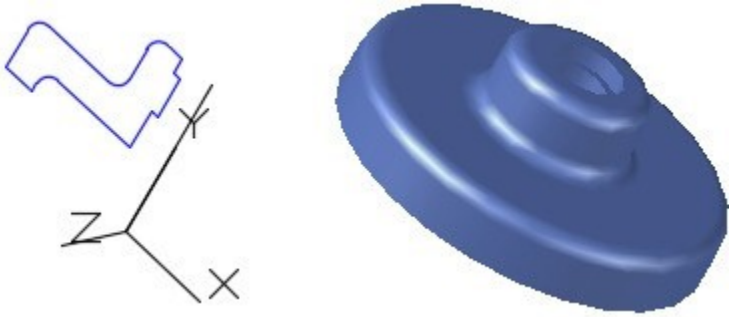
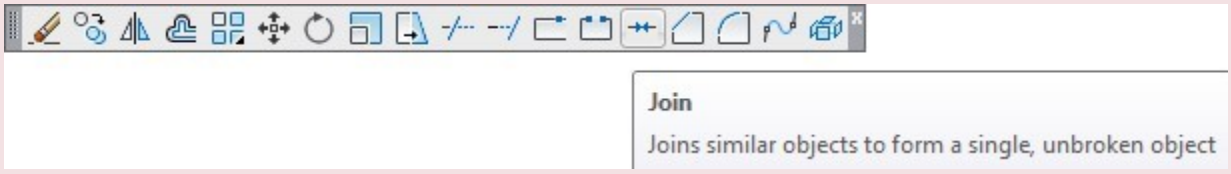


Figure 18-2
Revolving a Solid Model

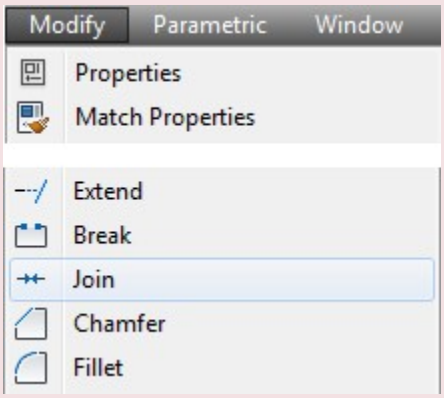
AutoCAD Command: JOIN

The JOIN combines a series of finite linear and open curved objects at their common endpoints to create a single 2D or 3D object.

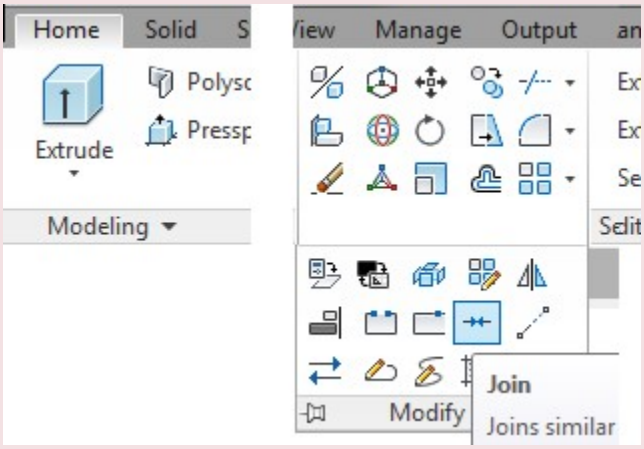
Shortcut: **J**



Modify Toolbar



Home Ribbon

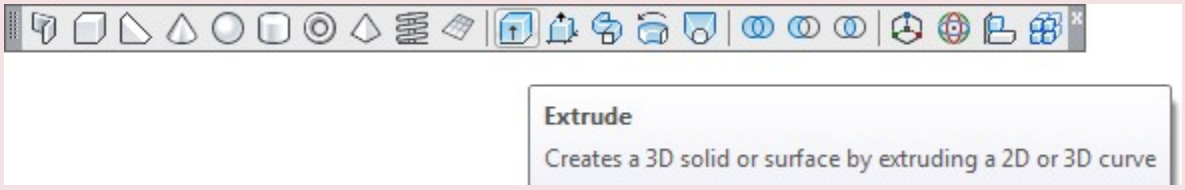


Modify Pull-down

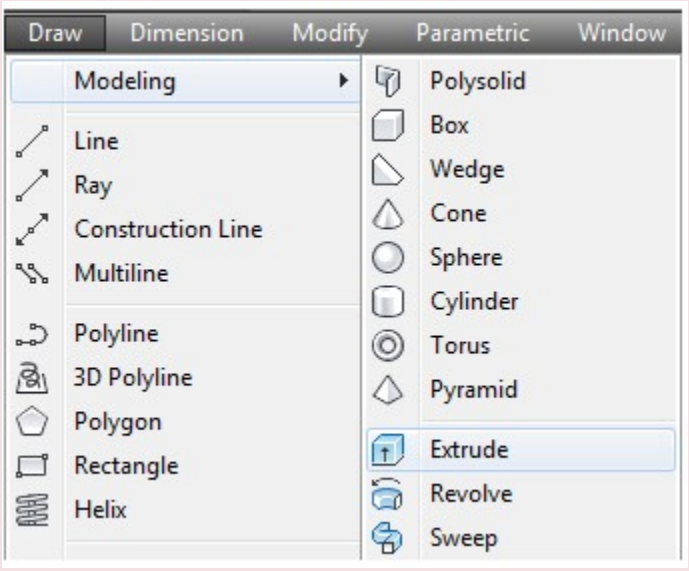
AutoCAD Command: EXTRUDE

The EXTRUDE command is used to create a solid by projecting a closed 2D object along the Z axis of the current UCS.

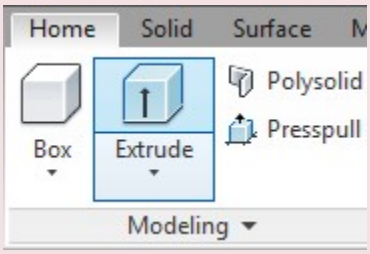
Shortcut: **EXT**



Modeling Toolbar



Draw Pull-down



Home Ribbon

AutoCAD Command: DELOBJ

The DELOBJ system variable controls whether the EXTRUDE or REVOLVE command deletes or retains the closed object when the command is executed.

Command: **DELOBJ**

Enter new value for DELOBJ <0>: Command:

Set to:

0 – Will retain the closed polygon

1 – Will delete the closed polygon

WORK ALONG: Creating a Extruded Solid Model Using the EXTRUDE Command

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 18-1.

Step 3

Set the current visual style to 2D Wireframe, set the layer: Pline as the current layer, the current view to SE Isometric, and the current UCS to World.

Step 4

Draw the top contour of the wireframe model only. Use the multiview drawing as a reference. (Figure Step 4A and 4B)

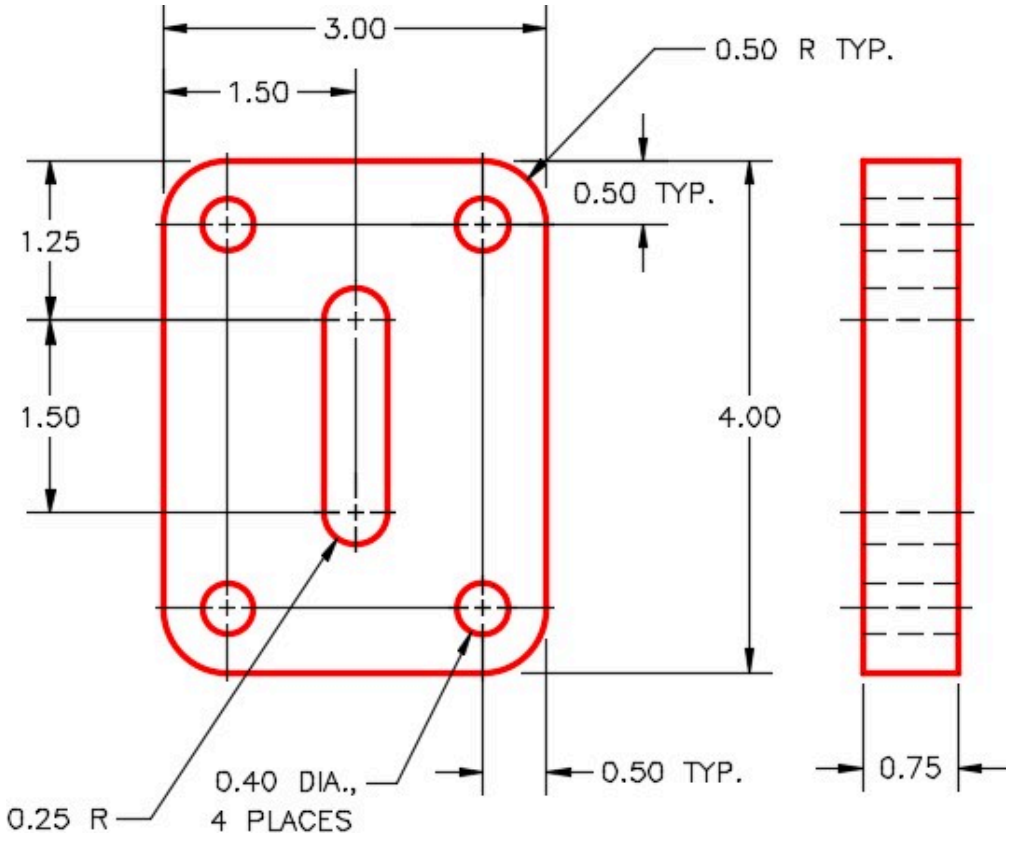


Figure Step 4A

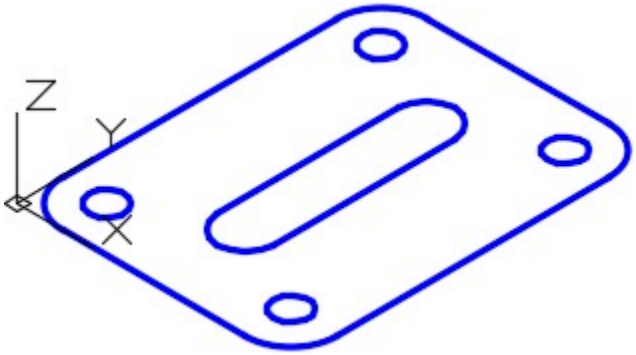


Figure Step 4B

Step 5

Set the current UCS to Top and locate it at the centre of the circle as shown in the figure. (Figure Step 5)

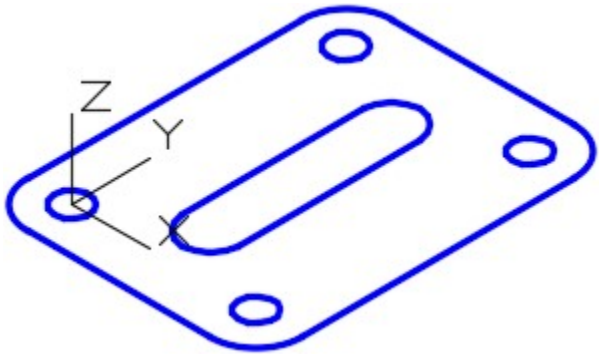


Figure Step 5

Step 6

Enter the DELOBJ system variable as shown below. Ensure that it is set to 0. Command: **DELOBJ**

Enter new value for DELOBJ <1>: **0**

Command:

AUTHOR'S COMMENTS: When the DELOBJ system variable is set to 0, the closed polyline that is used in the EXTRUDE and REVOLVE commands will be not be deleted.

Step 7

Set the system variable ISOLINES to 32, as shown below. Command: **ISOLINES**

Enter new value for ISOLINES <4>: **32**

Command:

AUTHOR'S COMMENTS: The ISOLINES system variable is used set the number of contour lines that will be displayed on a curved surface of the solid model.

Step 8

Enter the JOIN command, as shown below, to create a single 2D polyline from the 8 lines and arcs. (Figure Step 8)

Command: **JOIN**

Select source object or multiple objects to join at once:

1 found

Select objects to join: 1 found, 2 total

Select objects to join: 1 found, 3 total

Select objects to join: 1 found, 4 total

Select objects to join: 1 found, 5 total

Select objects to join: 1 found, 6 total

Select objects to join: 1 found, 7 total

Select objects to join: 1 found, 8 total

Select objects to join:

8 objects converted to 1 polyline

Command:

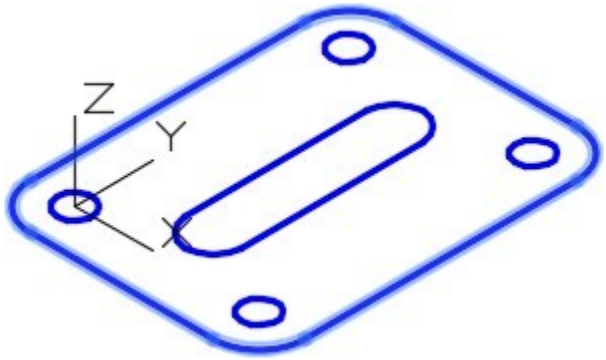


Figure Step 8

AUTHOR'S COMMENTS: Although the PEDIT command, that was taught earlier, can be used to join objects the JOIN command is much easier to use.

Step 9

Click the newly created pline and insure it is closed using the Properties window. (Figure Step 9)

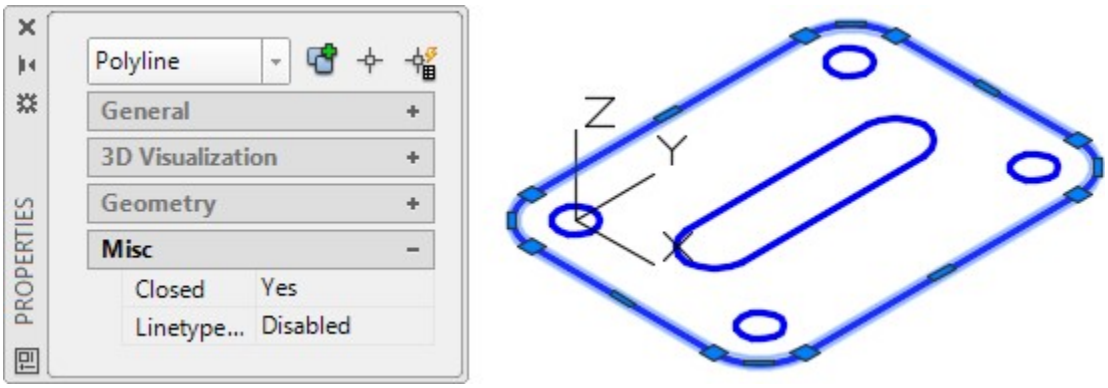


Figure Step 9

Step 10

Using what you learned in the last two steps join the inside 4 lines and arcs to create on 2D polyline. Check to ensure it is closed using the Properties window. (Figure Step 10)

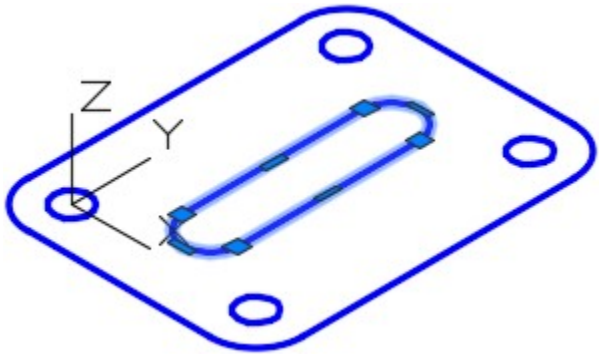


Figure Step 10

Step 11

Set layer: Solid 3 as the current layer. Enter the EXTRUDE command, as shown below, to create the solid model.

Command: **EXTRUDE**

Current wire frame density: ISOLINES=32

Select objects: 6 found

(Select all of the objects in a window or pick then individually.)

Select objects:

Specify height of extrusion or [Path]: **-0.75**

(Use -0.75 since the extrusion in the negative Z direction.)

Specify angle of taper for extrusion <0>:

(Press Enter to select the default.)

Command:

Step 12

Turn layer: Pline off and your model should appear as shown in the figure. (Figure Step 12)

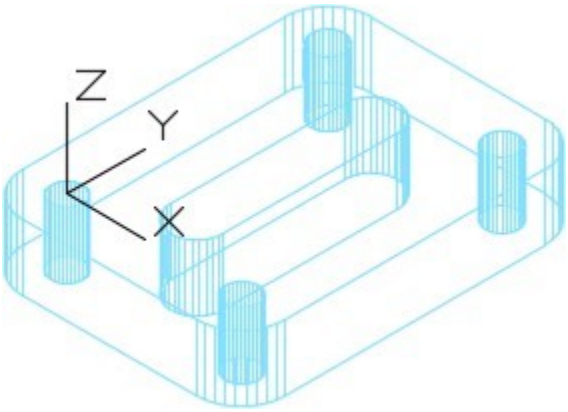


Figure Step 12

Step 13

Set the current visual style to Realistic. (Figure Step 13)

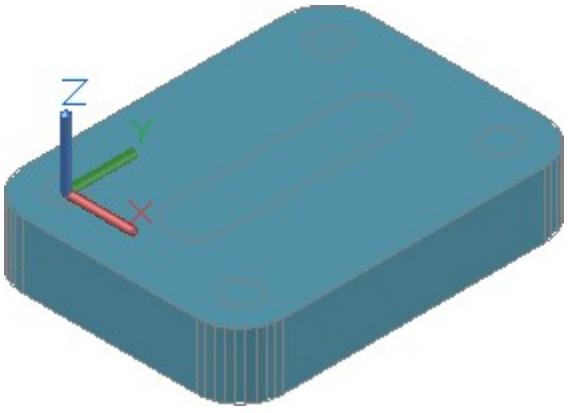


Figure Step 13

AUTHOR'S COMMENTS: You now have to subtract the inner solids from the overall solid to complete the

model with its holes. To use the SUBTRACT command, first select the solid you want to subtract from, press Enter to change modes and then select the solids you want to subtract from it. In this model, it is easiest to subtract solids when the current visual style is set to 2D Wireframe. You may have to practice subtracting the solids a few times before you get good at doing it.

Step 14

Set the current visual style to 2D Wireframe. Enter the SUBTRACT command to subtract the five inner solids from the larger solid. (Figure Step 14)

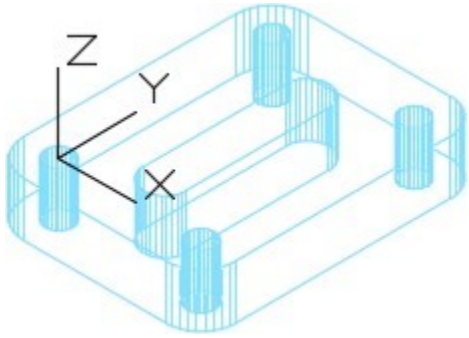


Figure Step 14

Step 15

Set the current visual style to Realistic. Your model should now appear as shown in the figures. Using the ORBIT command, orbit the model to enure the holes go through the model. (Figure Step 15)

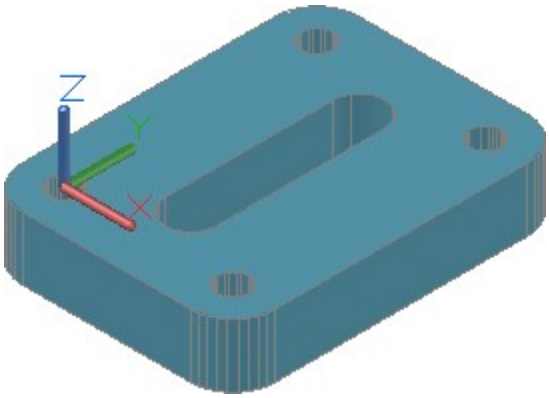


Figure Step 15 SE Isometric View

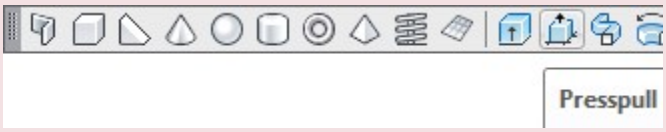
Step 16

Save and close the drawing.

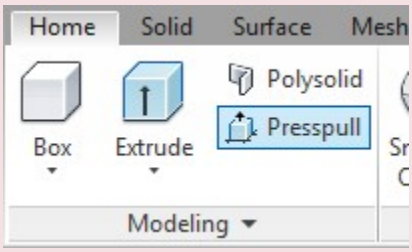
AutoCAD Command: PRESSPULL

The PRESSPULL command is used to create a solid by either selecting an area formed by a closed boundary (or boundaries) or a closed 2D object.

Shortcut: **none**



Modeling Toolbar



Home Ribbon

WORK ALONG: Creating a Extruded Solid Model Using the PRESSPULL Command

Step 1

Using the NEW command, start a new drawing using template: 3D Layout Metric.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 18-2.

Step 3

Set the current visual style to 3D Wireframe, set layer: Pline, the current view to SE Isometric, and the current UCS to World.

Step 4

Draw the top contour of the wireframe model only. Use the multiview drawing as a reference. (Figure Step 4A and 4B)

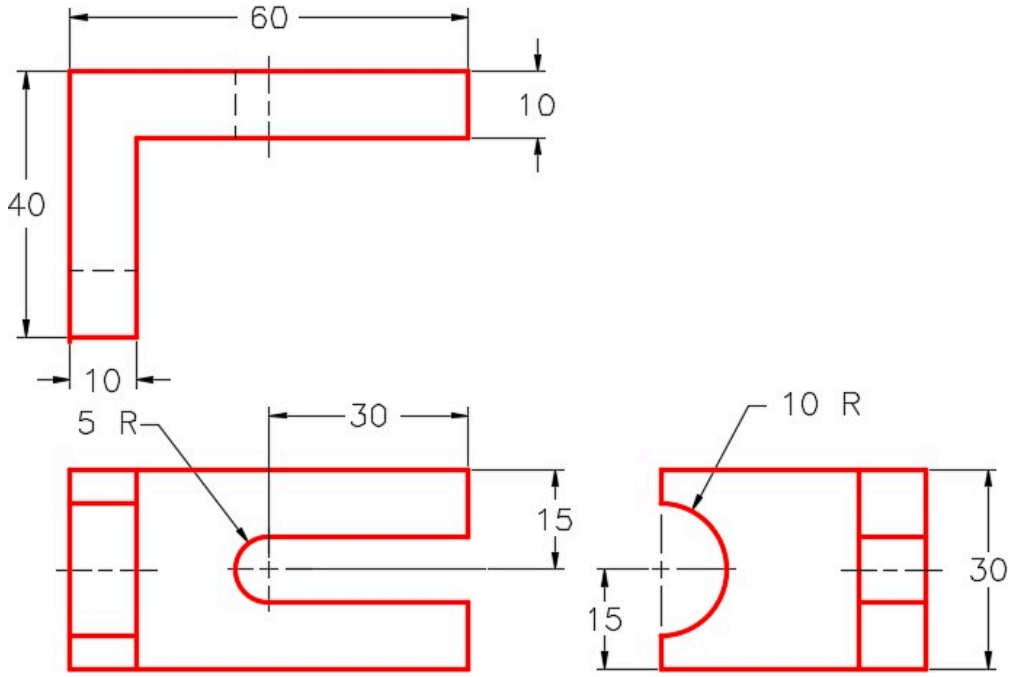


Figure Step 4A

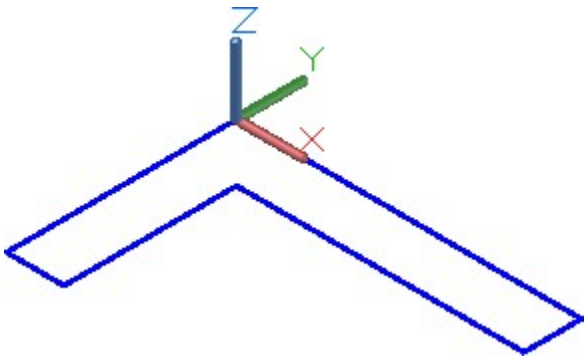


Figure Step 4B

Step 5

Use the JOIN command to create a single 2D polyline. Use the Properties window to ensure that the polyline is closed.

Step 6

Set layer: Solid 2 as the current layer.

Step 7

Enter the DELOBJ system variable as shown below and ensure that it is set to 0.

Command: **DELOBJ**

Enter new value for DELOBJ <1>: **0**

Command:

Step 8

Enter the PRESSPULL command. When prompted, select the polyline. Move the cursor in the positive Z direction and the solid will extrude with it. Enter 30 for the height. (Figure Step 8A and 8B)

Command: **PRESSPULL**

Select object or bounded area:

Specify extrusion height or [Multiple]:

Specify extrusion height or [Multiple]: **30**

1 extrusion(s) created

Select object or bounded area:

Command:

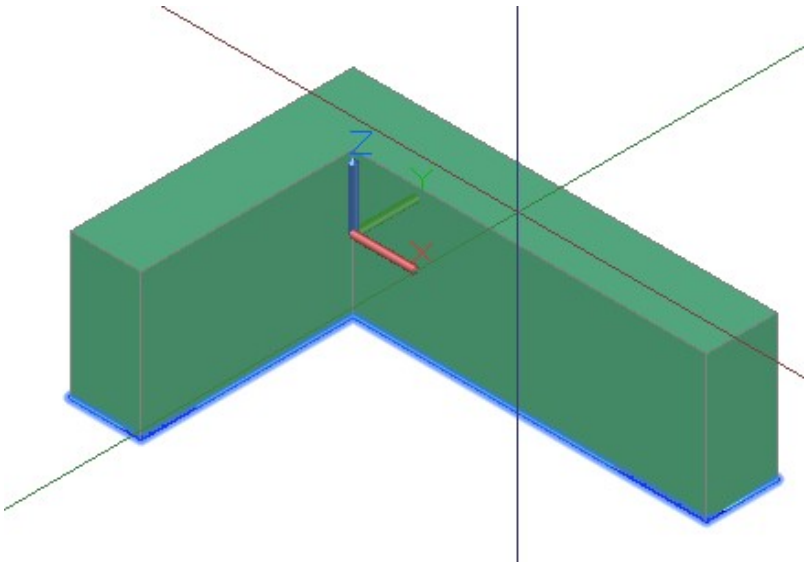


Figure Step 8A

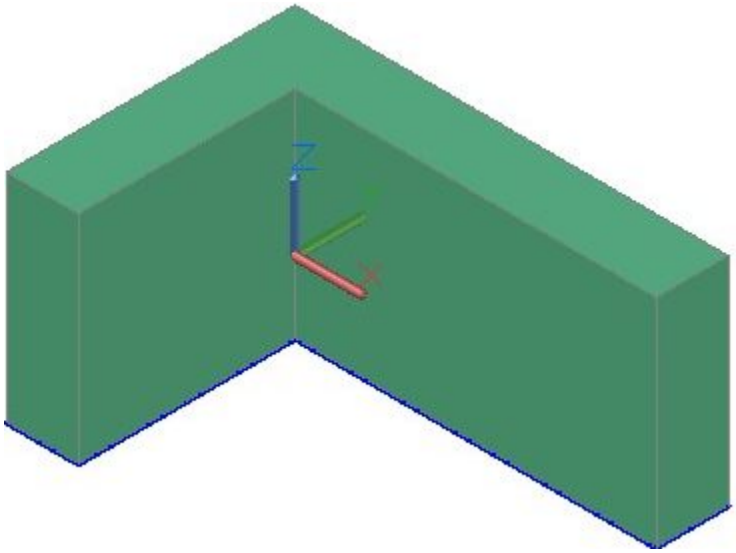


Figure Step 8B

Step 9

Using the multiview drawing for the dimensions, draw the lines and arcs in the Front and Right Side of the model. Use the JOIN command to join the plines and ensure that they are closed. (Figure Step 9)

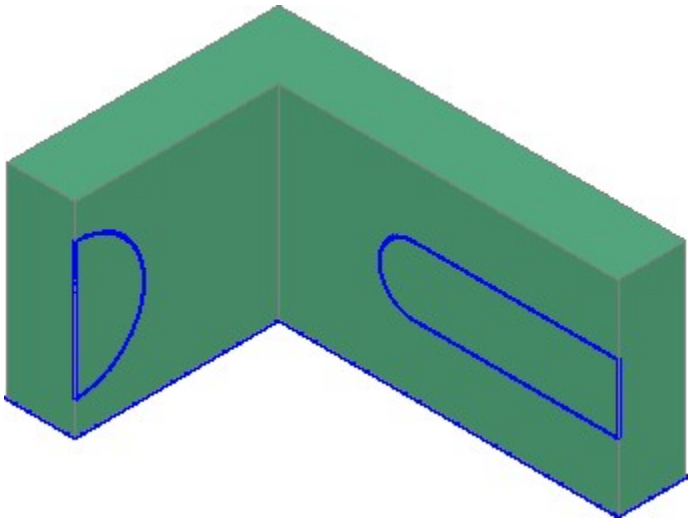


Figure Step 9

Step 10

Enter the PRESSPULL command and when prompted, select inside the closed pline as shown in the figure. Snap to the back corner to indicate the depth of the extrusion. (Figure Step 10A and 10B)

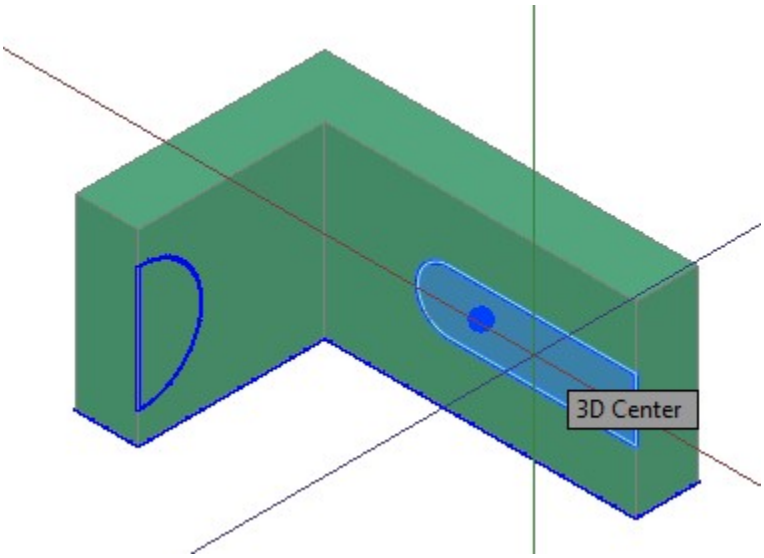


Figure Step 10A

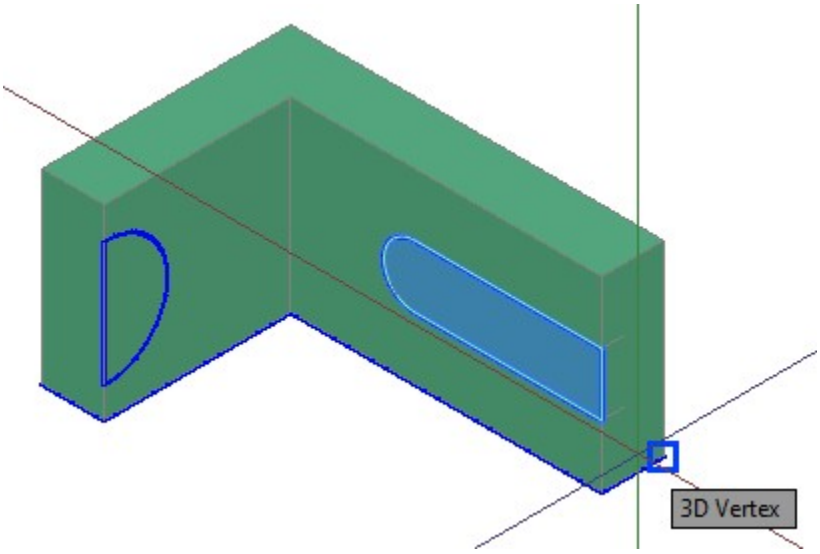


Figure Step 10B

Step 11

Using the same principle as Step 10, use the PRESSPULL command to extrude the other pline. (Figure Step 11A, 11B, and 11C)

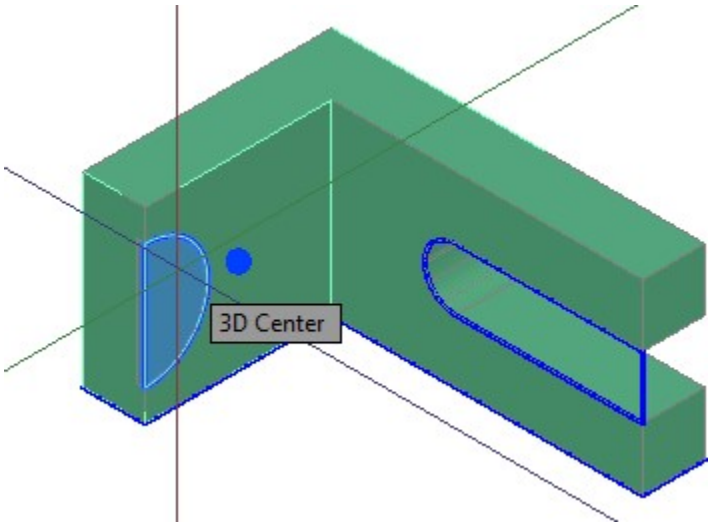


Figure Step 11A

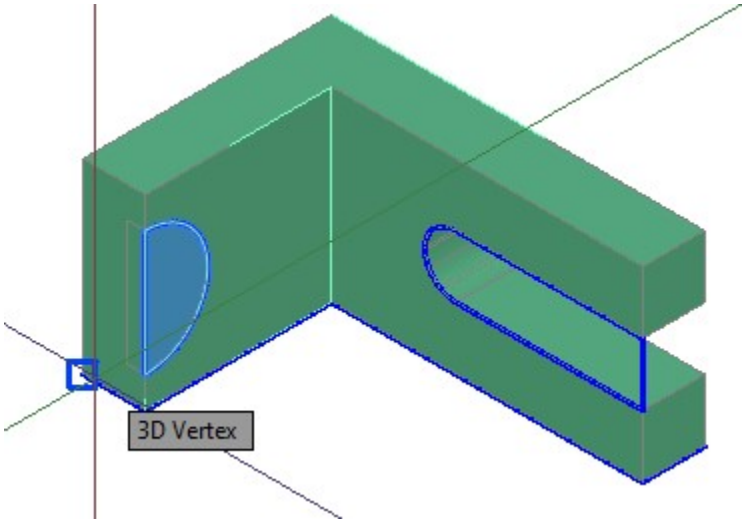


Figure Step 11B

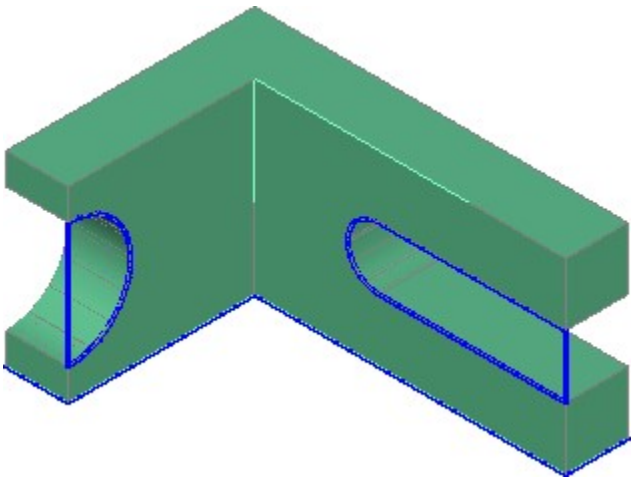


Figure Step 11C

Step 12

Turn layer: Pline off.

Step 13

Your completed model should appear as shown on the figure. (Figure Step 13)

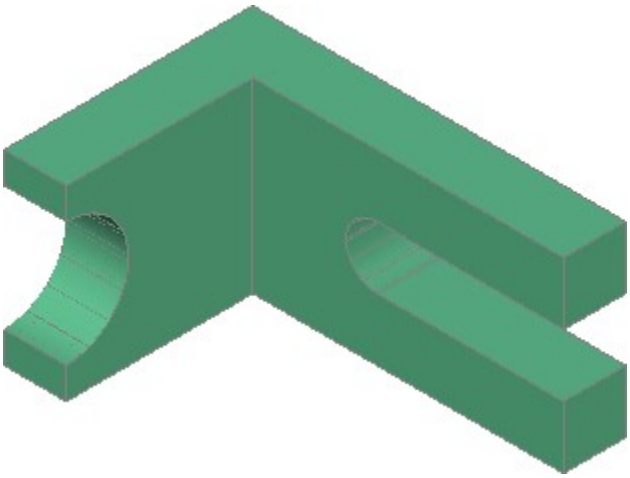


Figure Step 13

Step 14

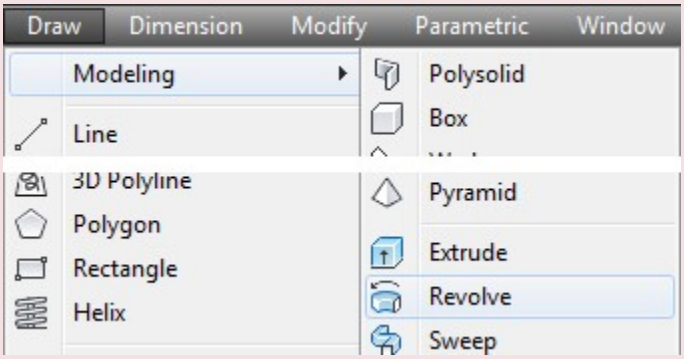
Save and close the drawing

AutoCAD Command: REVOLVE

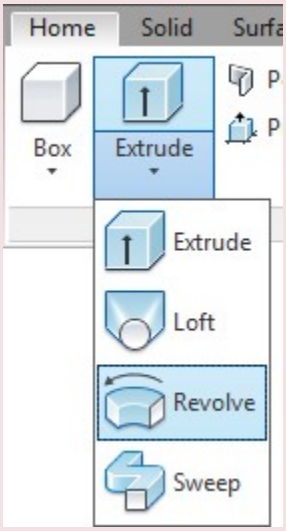
The REVOLVE command is used to create a solid model by revolving a 2D object around an axis.

Shortcut: **REV**

Modeling Toolbar



Draw Pull-down



Home Ribbon

WORK ALONG: Creating a Revolved Solid Model

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 18-3. (Figure Step 2)

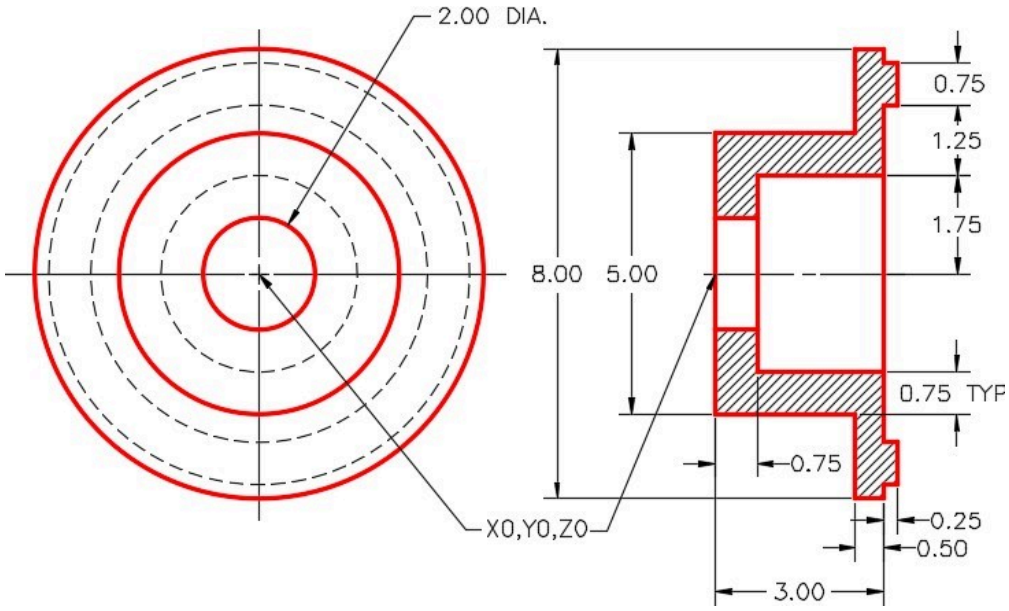


Figure Step 2
Dimensioned Multiview Drawing

Step 3

Set layer Pline as the current layer and current visual style to 2D Wireframe.

Step 4

Set the current view to SE Isometric and the current UCS to Right. (Figure Step 4)

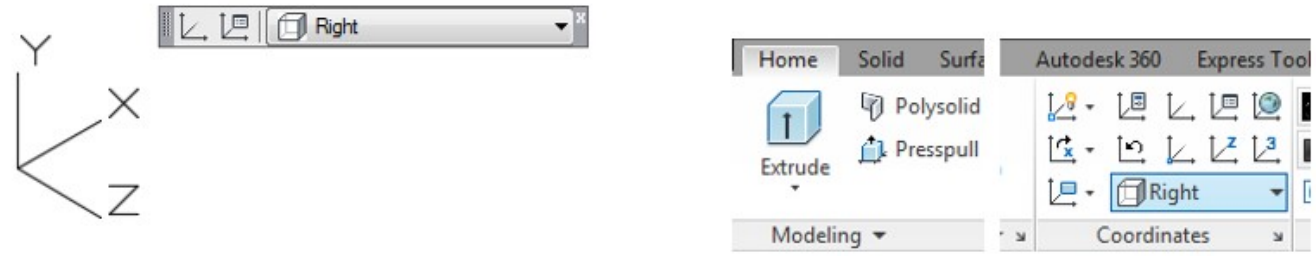


Figure Step 4

Step 5

Using the multiview drawing as a reference, draw one-half of the right side cross section of the solid part of the object. (Figure Step 5)

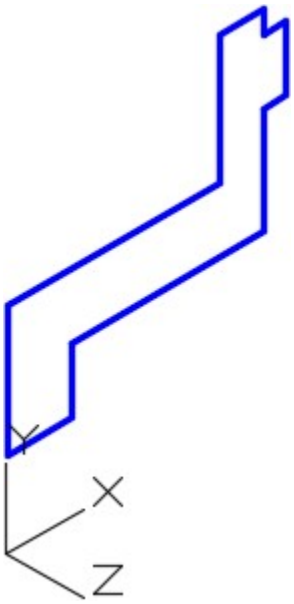


Figure Step 5

AUTHOR'S COMMENTS: You will be revolving this cross section 360 degrees to create a solid as it revolves. Therefore, only one-half of the section view is drawn.

Step 6

On layer: Construction, from 0,0,0 draw a line, of any length, along the X axis. Use ortho mode to draw it quicker. This will be the axis for the revolution and is the centre line of the solid. (Figure Step 6)

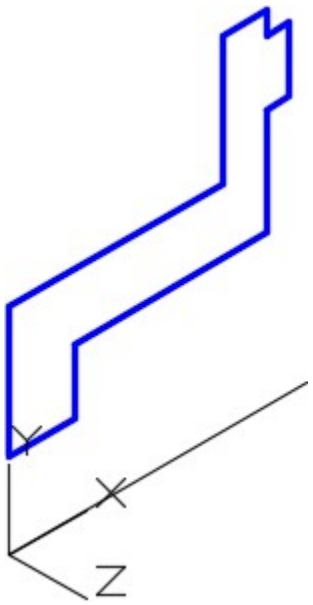


Figure Step 6

Step 7

Using the JOIN command, create a closed polygon from the lines. Ensure that it is closed. (Figure Step 7)

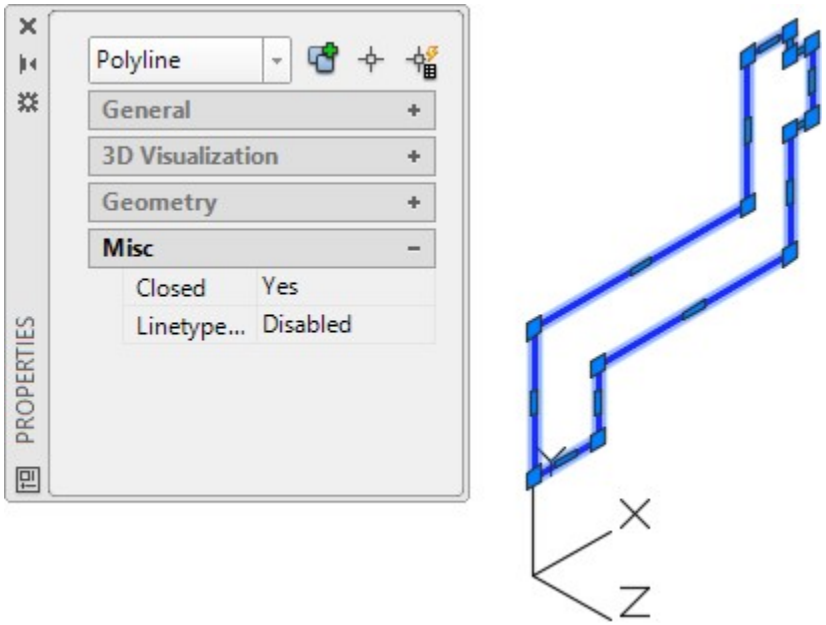


Figure Step 7

Step 8

Set the ISOLINES system variable to 48 as shown below. Command: **ISOLINES**

Enter new value for ISOLINES <4>: **48**

Command:

Step 9

Set layer Solid 3 as the current layer. Enter the REVOLVE command, as shown below. After completing the command, the model should appear as shown in the figure. (Figure Step 9)

Command: **REVOLVE**

Current wire frame density: ISOLINES=48

Select objects: 1 found

(Select the closed polyline.)

Select objects:

(Press Enter.)

Specify start point for axis of revolution or define axis by

[Object/X (axis)/Y (axis)]: **O**

Select an object:

(Select the axis (the construction line.)

Specify angle of revolution <360>:

(Press Enter to select the default.)

Command:

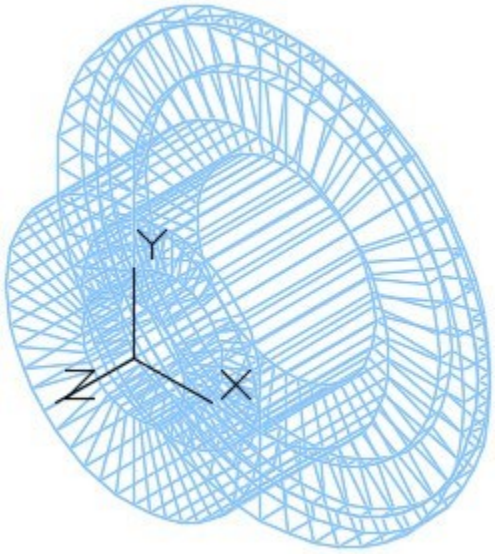


Figure Step 9

Step 10

Turn off layers: Construction and Pline and set the current visual style to Realistic. (Figure Step 10)

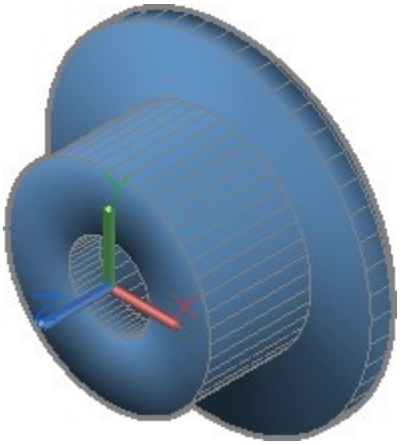


Figure Step 10
SE Isometric View

Step 11

Using 3D Orbit, orbit the model as shown in the figure. (Figure Step 11).

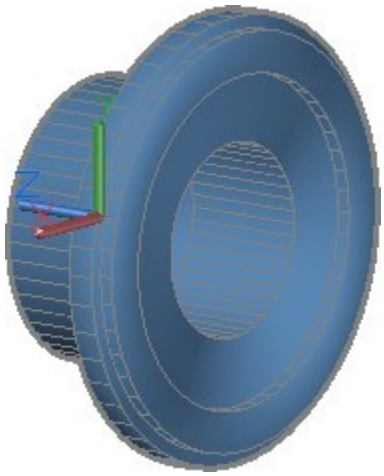


Figure Step 11

Step 12

Save and close the drawing.

Key Principles

Key Principles in Module 18

1. The object being extruded or revolved with the EXTRUDE, PRESSPULL, and REVOLVE commands must be closed polyline, a circle, or an ellipse.
2. Before entering the EXTRUDE command, ensure that UCS is located with the Z axis going in the direction of the extrusion.

Lab Exercise 18-1

Time allowed: 45 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 18-1	3D Layout Metric	Millimeters

Step 1

Set the system variable DELOBJ to 0.

Step 2

Draw the closed plines on layer: Pline.

Step 3

On layer: Solid 1, draw a solid model of the object. (Figure Step 3A and 3B)

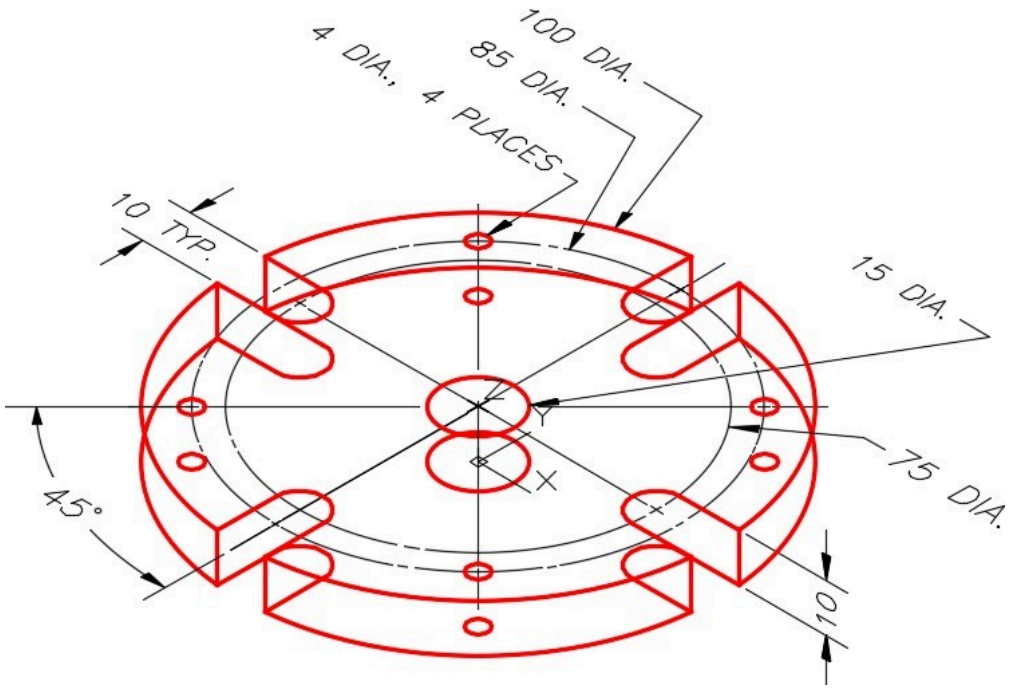


Figure Step 3A Dimensioned Wireframe SE Isometric View

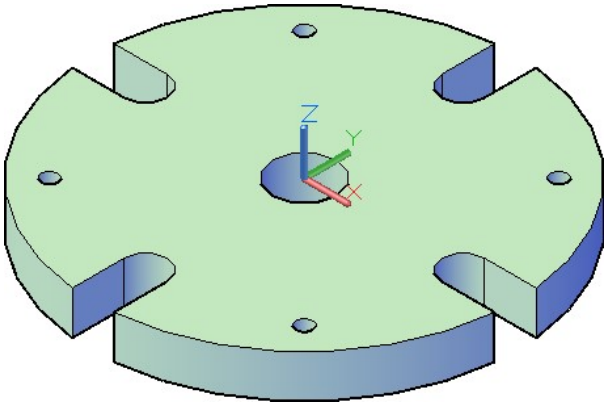


Figure Step 3B
Solid Model SE Isometric View

Step 4

Use the UNION and SUBTRACT commands to complete the solid model. When complete, the solid must be one object.

Step 5

Turn layers: Construction and Pline off and set the current visual style to Realistic.

Step 6

Set the Insertion Units, change the current UCS to World and check the model with the key.

Step 7

Save and close the drawing.

Lab Exercise 18-2

Time allowed: 45 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 18-2	3D Layout Metric	Millimeters

Step 1

Set the system variable DELOBJ to 0.

Step 2

On layer: Solid 3, draw a solid model of the object. (Figure Step 2A, 2B, 2C, and 2D)

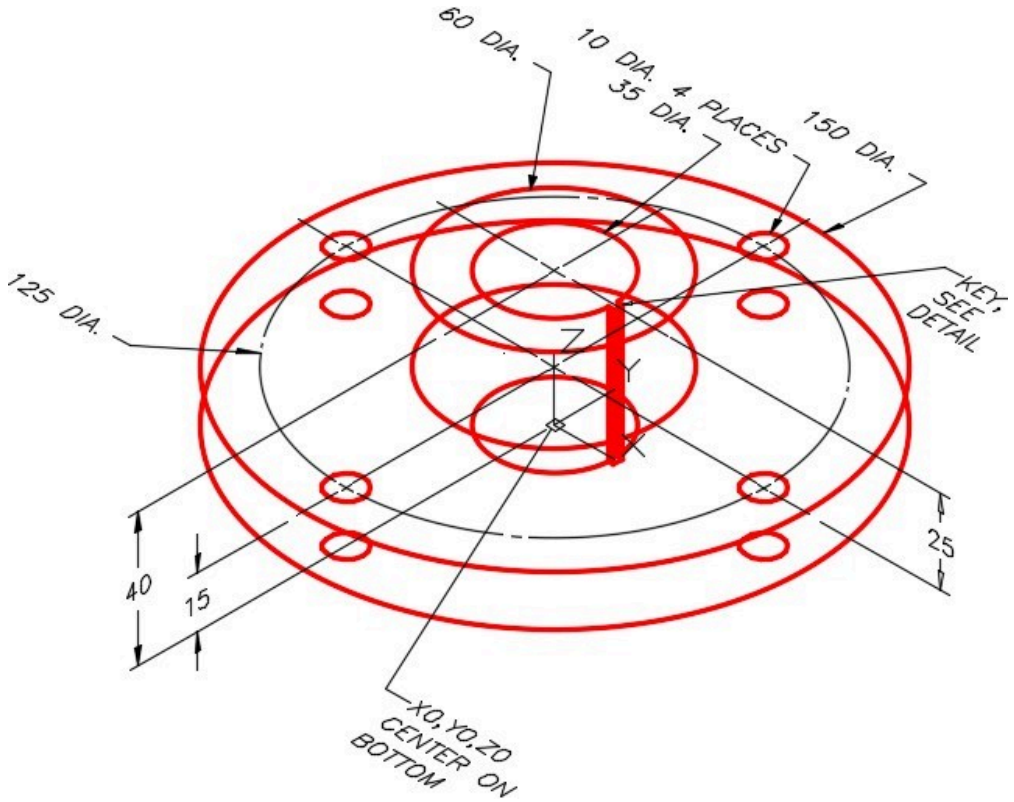


Figure Step 2A Dimensioned Wireframe SE Isometric View

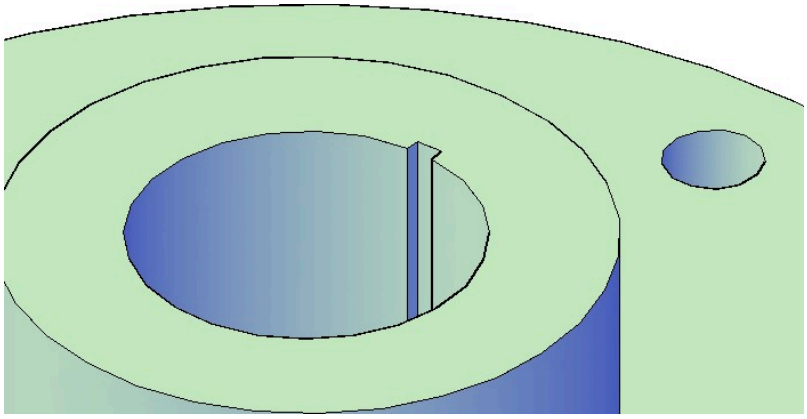


Figure Step 2B
View of Keyway – Rotated

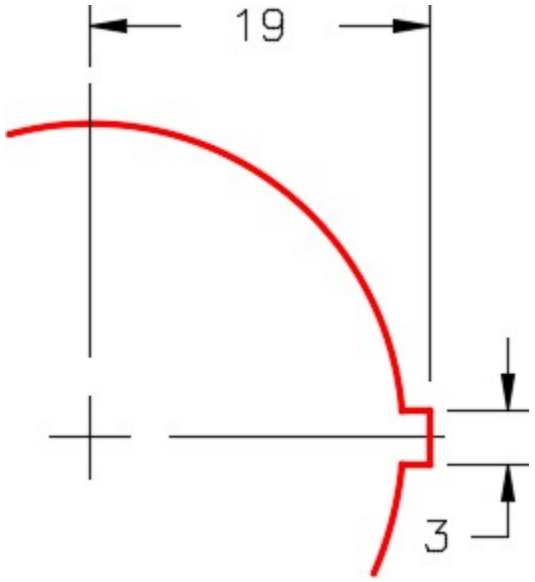


Figure Step 2C
Key Detail

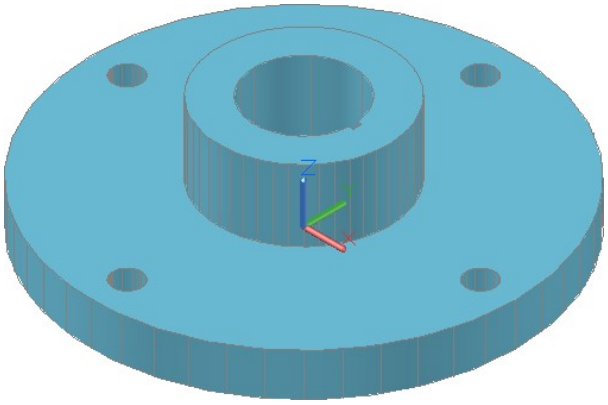


Figure Step 2D
Solid Model SE Isometric View

Step 3

On layer: Pline, draw the closed plines.

Step 4

Use the UNION and SUBTRACT commands to complete the solid model. When complete, the solid must be one object.

Step 5

Turn layers: Construction and Pline off and set the current visual style to Realistic.

Step 6

Set the Insertion Units, change the current UCS to World, and check the model with the key.

Step 7

Save and close the drawing.

Lab Exercise 18-3

Time allowed: 60 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 18-3	3D Layout English	Inches

Step 1

Set the system variable DELOBJ to 0.

Step 2

On layer: Solid 4, draw a solid model of the object. (Figure Step 2A, 2B, and 2C)

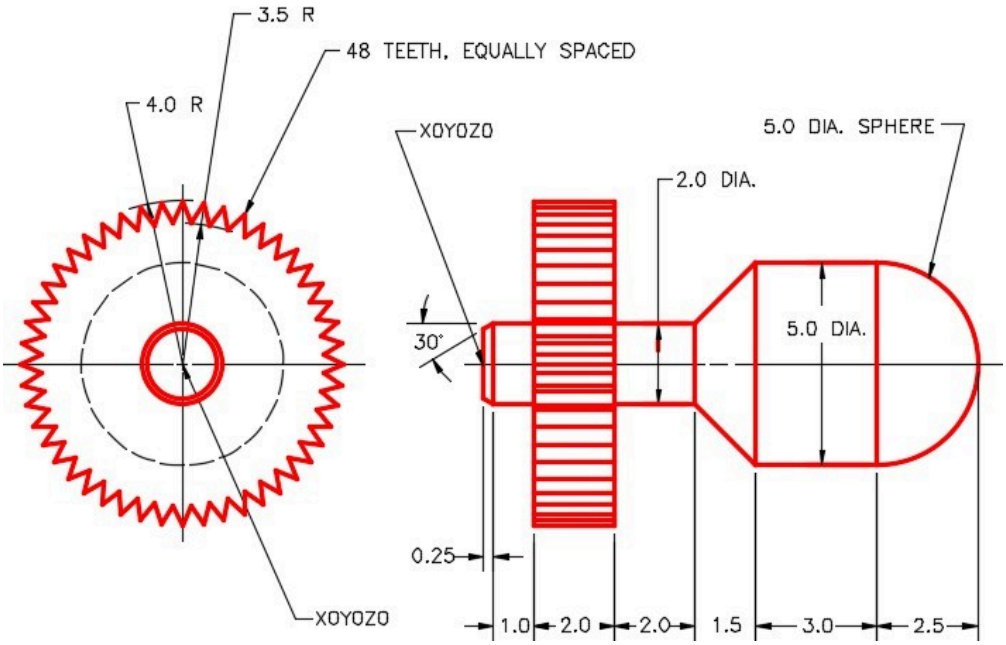


Figure Step 2A
Dimensioned Multiview Drawing

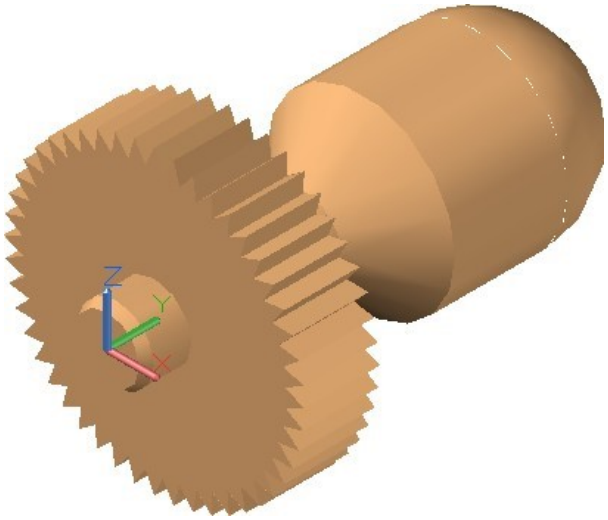


Figure Step 2B Completed Solid Model SE Isometric View

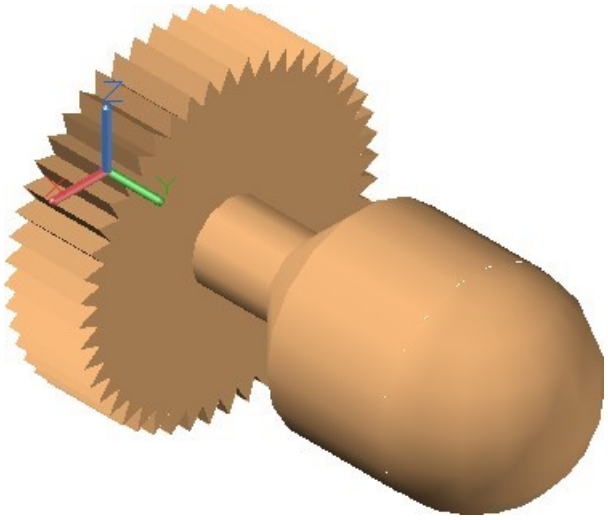


Figure Step 2C Completed Solid Model NE Isometric View

Step 3

On layer: Pline, draw the closed plines.

> **AUTHOR'S COMMENTS:** I suggest that you construct the model in two parts. One extrusion and one revolution. UNION them together to form one solid model.

Step 4

Use the UNION command to complete the solid model. When complete, the solid must be one object.

Step 5

Turn layers: Construction and Pline off and set the current visual style to Realistic.

Step 6

Set the Insertion Units, change the current UCS to World, and check the model with the key.

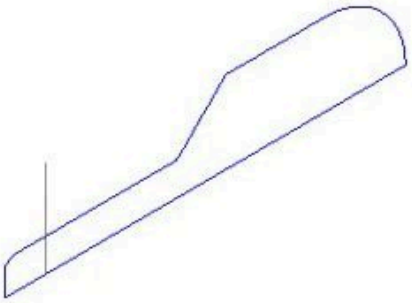
Step 7

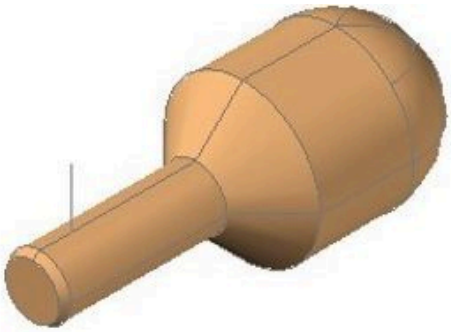
Save and close the drawing.

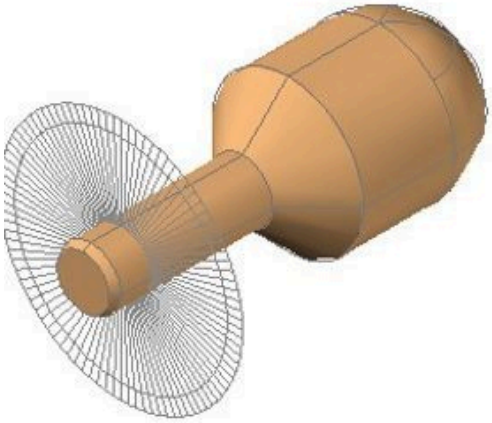
AUTHOR'S CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hint. If you get stuck and can't complete it on your own, use the following hint to help you.

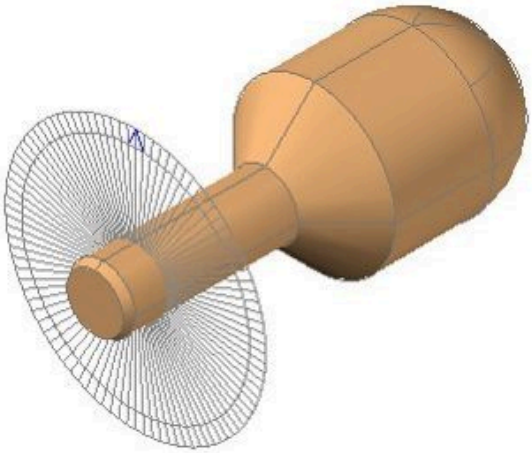
Hint 1

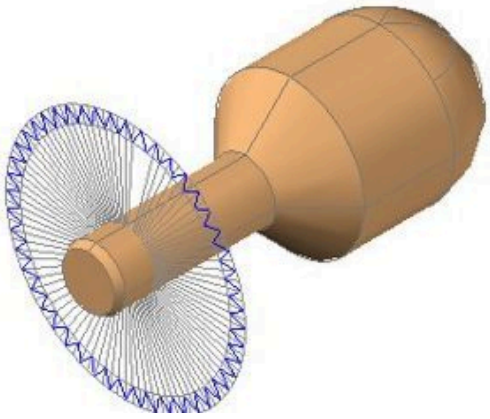
See steps below:

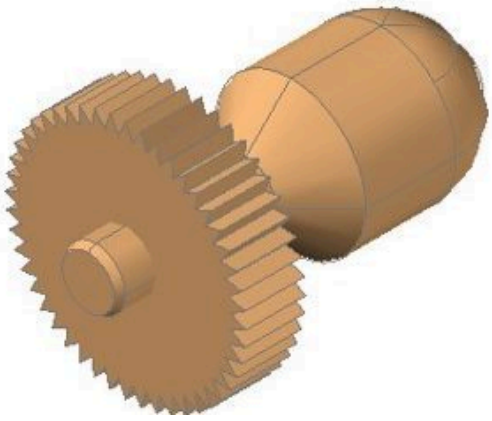












Lab Exercise 18-4

Time allowed: 45 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 18-4	3D Layout Metric	Millimeters

Step 1

Set the system variable DELOBJ to 0.

Step 2

On layer: Solid 5, draw a solid model of the object. (Figure Step 2A, 2B, and 2C)

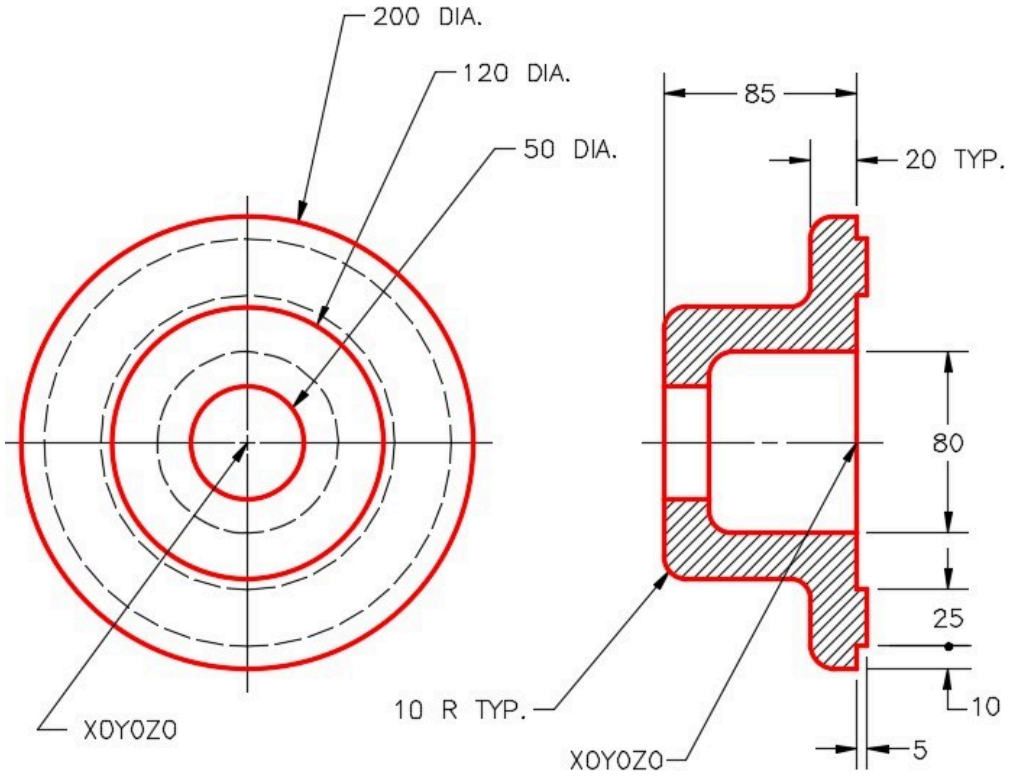


Figure Step 2A
Dimensioned Multiview Drawing

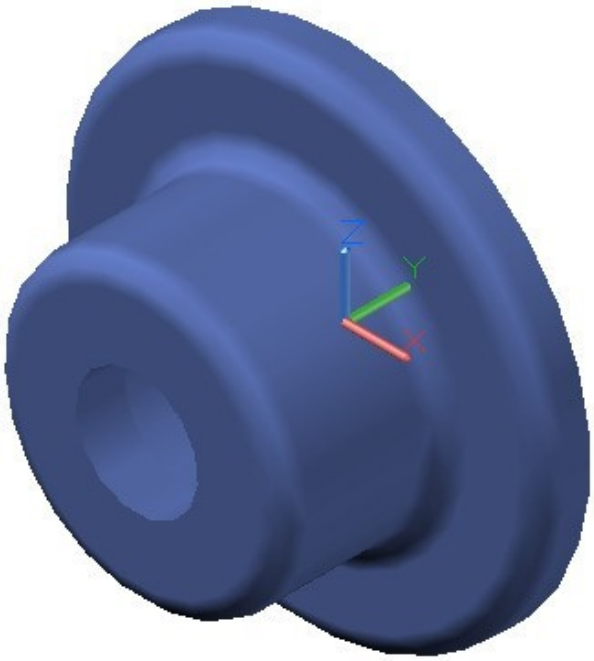


Figure Step 2B Completed Solid Model SE Isometric View

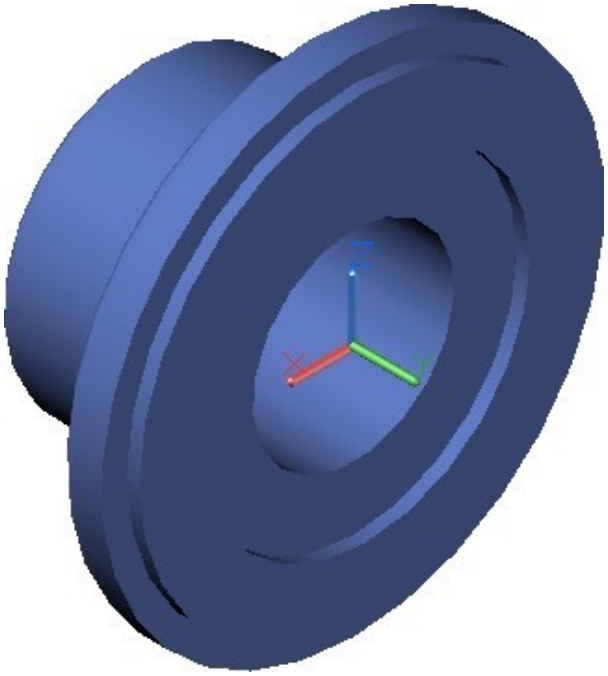


Figure Step 2C Completed Solid Model NE Isometric View

Step 3

On layer: Pline, draw the closed pline.

Step 4

When complete, the solid model must be one object.

Step 5

Turn layers: Construction and Pline off and set the current visual style to Realistic.

Step 6

Set the Insertion Units, change the current UCS to World, and check the model with the key.

Step 7

Save and close the drawing.

Lab Exercise 18-5

Time allowed: 45 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 18-5	3D Layout English	Inches

Step 1

Set the system variable DELOBJ to 0.

Step 2

On layer: Solid 3, draw a solid model of the object. (Figure Step 2A, 2B, and 2C)

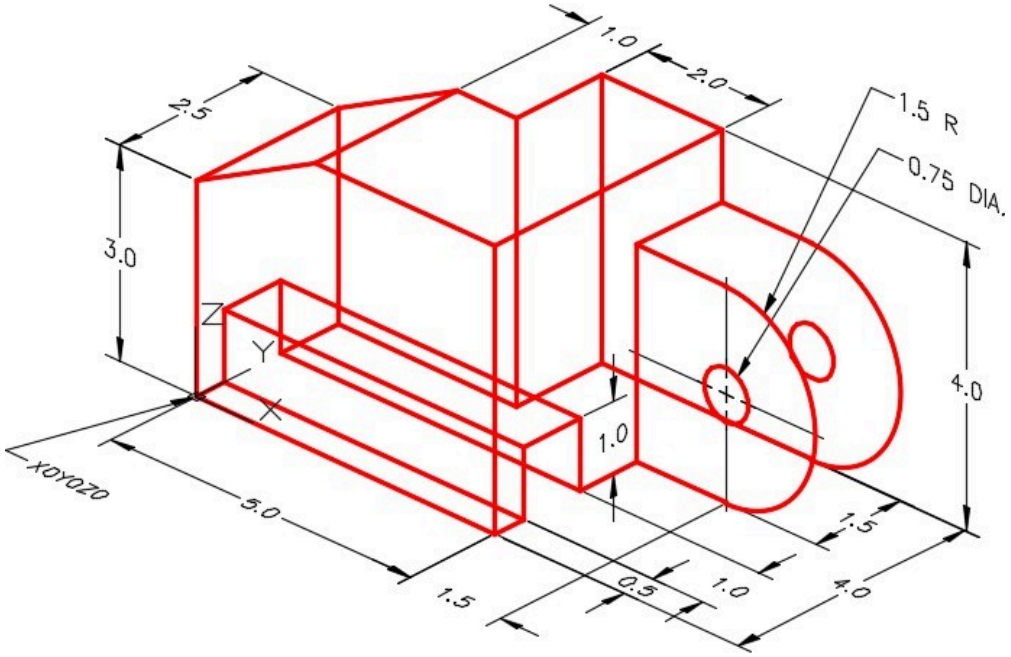


Figure Step 2A
Dimensioned Wireframe Model

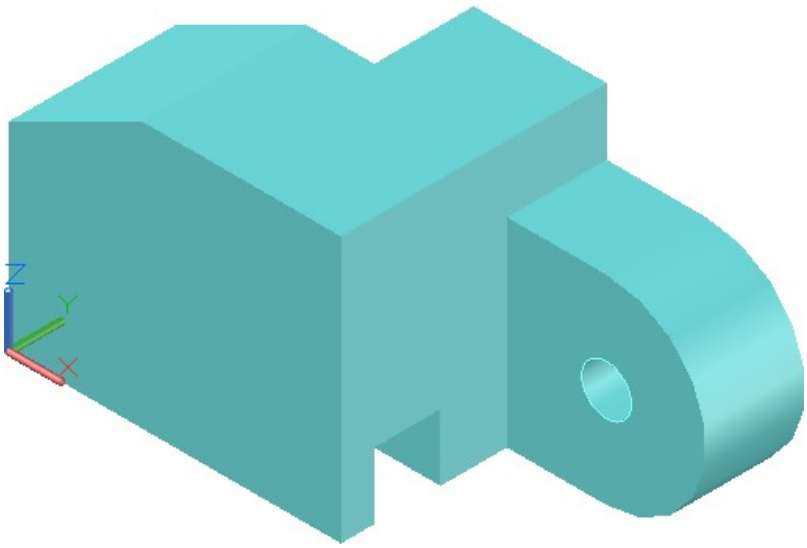


Figure Step 2B
Completed Solid Model SE Isometric View

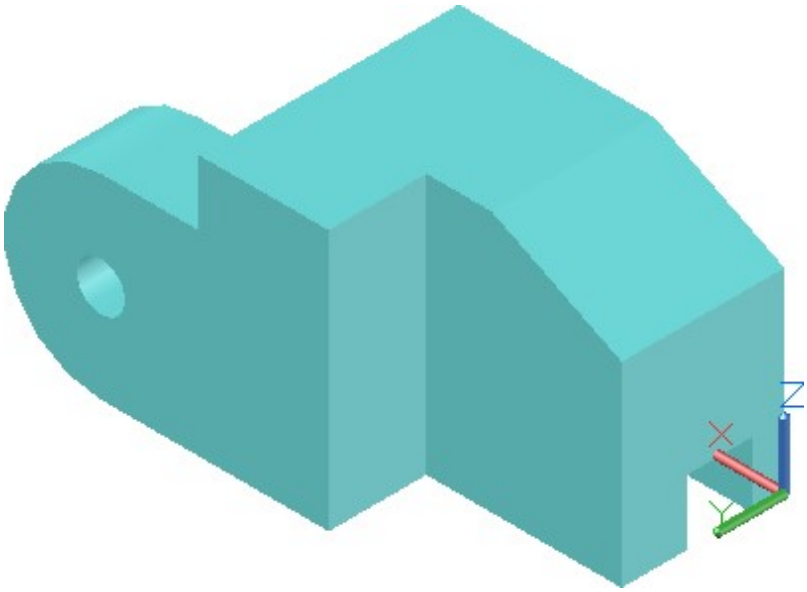


Figure Step 2C
Completed Solid Model NW Isometric View

Step 3

On layer: Pline, draw the closed plines.

Step 4

Use the UNION and SUBTRACT commands to complete the solid model. When complete, the solid must be one object.

Step 5

Turn layers: Construction and Pline off and set the current visual style to Realistic.

Step 6

Set the Insertion Units, change the current UCS to World, and check the model with the key.

Step 7

Save and close the drawing.

Module 19 Solid Modeling – Part 3

Learning Outcomes

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

1. Describe how the UNION, SUBTRACT, and INTERSECT commands are used and applied when solid modeling.
2. Apply the INTERSECT, FILLETEDGE, CHAMFEREDGE, and OFFSETEDGE commands to construct and modify solid models.

Working With Composite Solids

When you are constructing solid models, it is important to ensure that the completed solid model consists of only one object. The three commands used to help you to do this are UNION, SUBTRACT and INTERSECT. Study the following figures to help you understand how these commands affect the solid model.

Figure 19-1 shows two solid models in which parts of each one occupies the same 3D space. This, of course, would be impossible in real life.

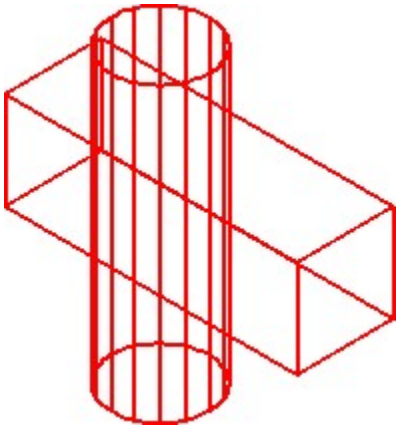


Figure 19-1
Two Solids

Figure 19-2 shows the same model after the UNION command was used to join the two solids into one.

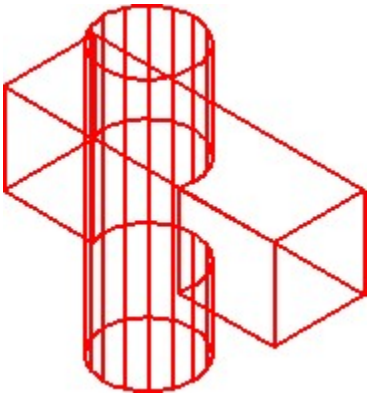


Figure 19-2
One Solid

Figure 19-3 shows a shaded model and how the model in Figure 19-1 will appear when the two solids are joined together with the UNION command.

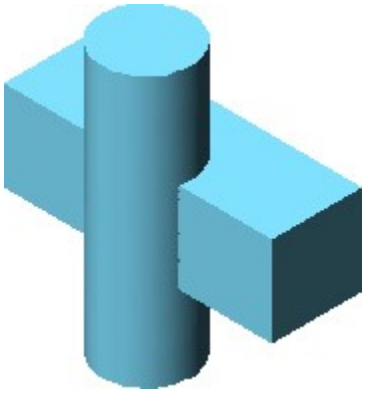


Figure 19-3
One Solid – Joined With UNION Command

Figure 19-4 shows how the two solids in Figure 19-1 will appear when the cylinder is subtracted from the box using the SUBTRACT command.

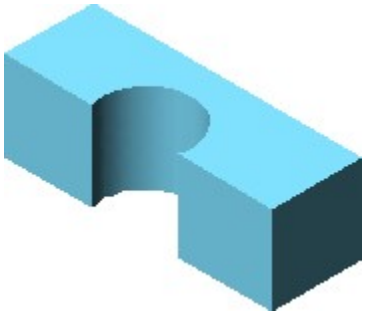


Figure 19-4
One Solid – Subtracted

Figure 19-5 shows how the two models in Figure 19-1 would appear when the box is subtracted from the cylinder using the SUBTRACT command.

Figure 19-5 One Solid Subtracted

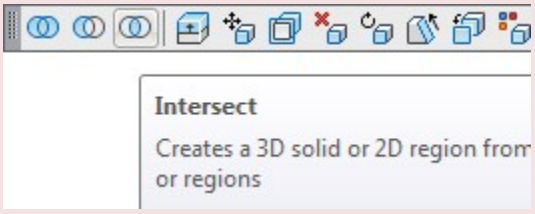
Figure 19-6 shows how the solid models in Figure 19-1 would appear if the INTERSECT command is used. Notice how only the volume shared by the two models remains.

Figure 19-6 One Solid Intersecti on

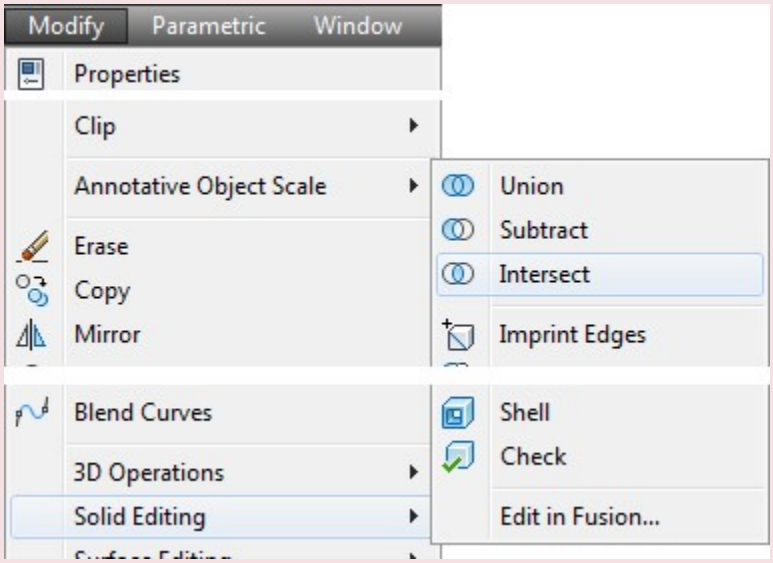
AutoCAD Command: INTERSECT

The INTERSECT command is used to create a solid by finding the volume shared by two or more solids.

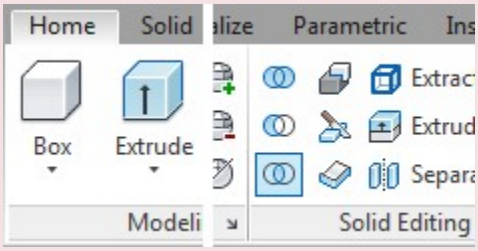
Shortcut: none



Solid Editing Toolbar



Modify Pull-down



Home Ribbon

WORK ALONG: Creating a Solid Model by Intersecting Two or Three Solid Models

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 19-1.

Step 3

Ensure that the DELOBJ system variable is set to 0.

Step 4

Set layer: Pline as the current layer, the current UCS to Front, the current view to SE Isometric, and the current visual style to 2D Wireframe.

Step 5

Set the current UCS to Front. Using the figures as a reference, draw the contour of the Front view. (Figure Step 5A and 5B)

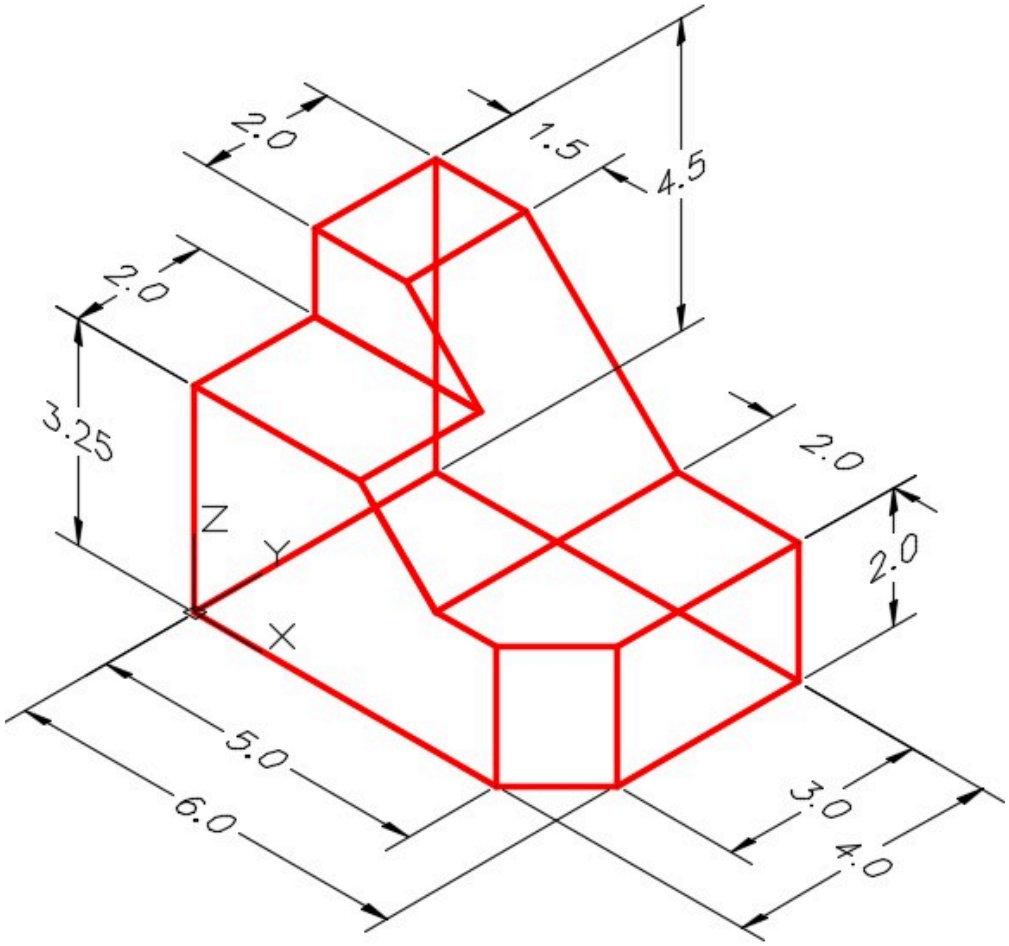


Figure Step 5A
Dimensioned Wireframe Model

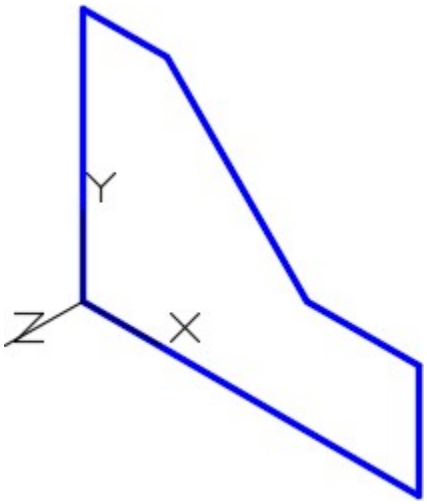


Figure Step 5B

Step 6

Using the JOIN command, create a closed polyline. (Figure Step 6)

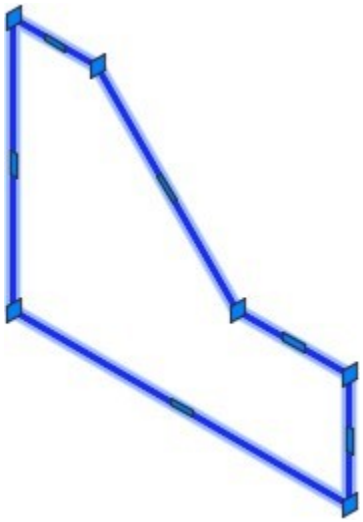


Figure Step 6

Step 7

Set the current UCS to Right and relocate the UCS to the right side as shown in the figure. Using the PLINE command, draw a closed pline of the contour of the right side. (Figure Step 7)

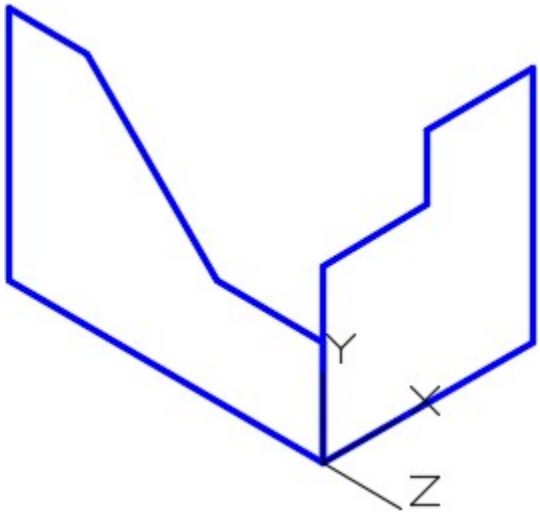


Figure Step 7

Step 8

Set the current UCS to Top and relocate it to the top corner of the model as shown in the figure. Using the PLINE command, draw a closed pline of the contour of the top view. (Figure Step 8)

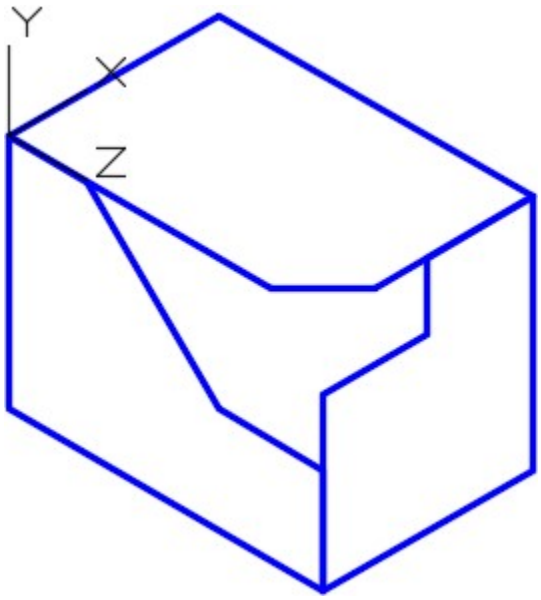


Figure Step 8

AUTHOR'S COMMENTS: The contour of a view is the outside shape of the view as it would appear in a

multiview drawing. Do not include any of the interior objects in the view.

Step 9

Set layer: Solid 3 as the current layer. Extrude the Front view the depth of the model. (Figure Step 9)

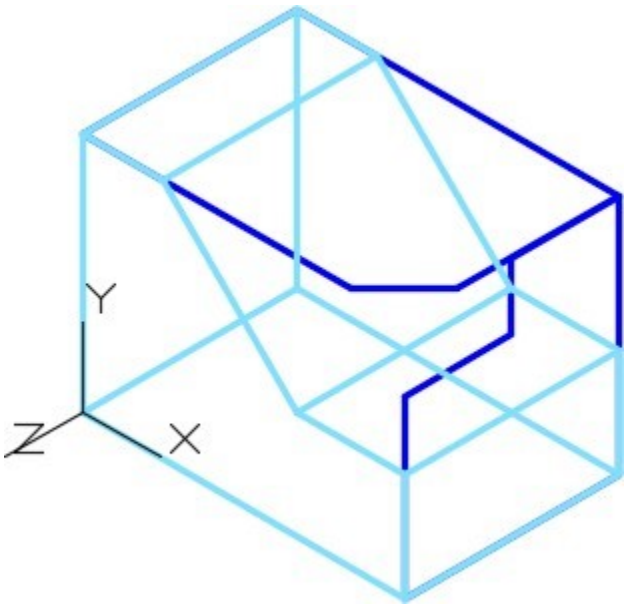


Figure Step 9

Step 10

Set the current visual style to Realistic to see the extruded solid. (Figure Step 10)

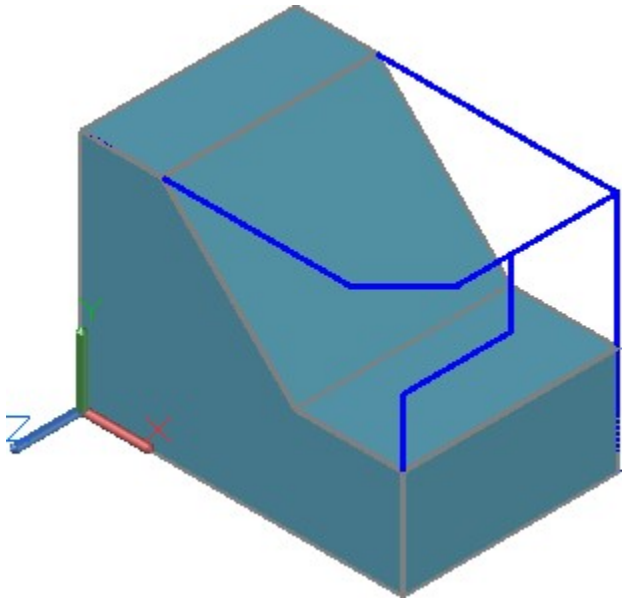


Figure Step 10

Step 11

Set the current visual style to 2D Wireframe. Using PRESSPULL, extrude the Right Side and the Top view. Set the current visual style to Realistic to see all three extruded solids at the same time. (Figure Step 11A and 11B)

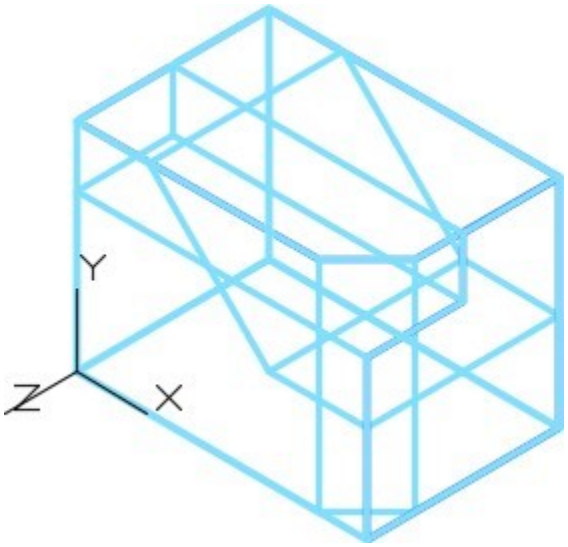


Figure Step 11A

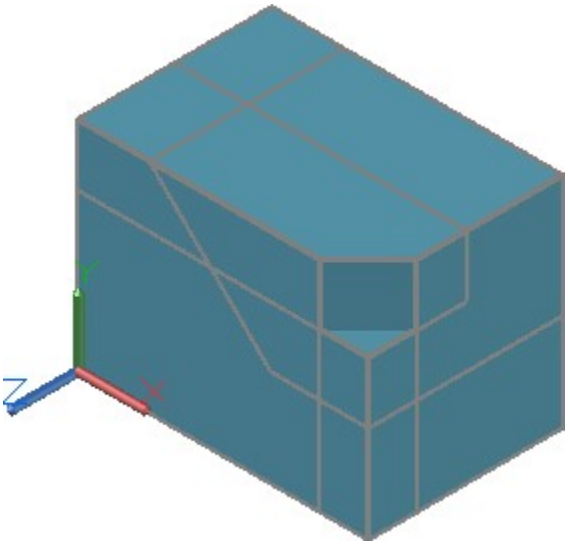


Figure Step 11B

AUTHOR'S COMMENTS: You now have the three solids that occupy a portion of the same 3D space. The next step is to create a single solid model of the 3D space that they all share. To do this, use the INTERSECT command.

Step 12

Save drawing.

Step 13

Set the current visual style to 2D Wireframe. Enter the INTERSECT command, as shown below, to create the common solid from the three intersecting solids. (Figure Step 13)

Command: **INTERSECT**

Select objects: 1 found

Select objects: 1 found, 2 total

Select objects: 1 found, 3 total

(Select each of the solid models.)

Select objects:

Command:

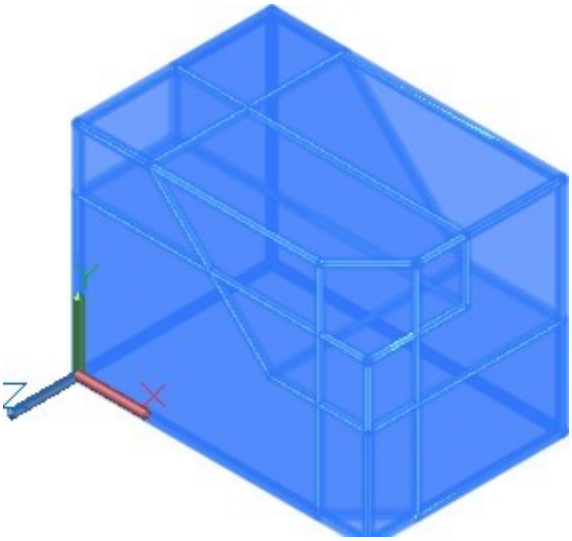


Figure Step 13

Step 14

Set the current visual style to Realistic. (Figure Step 14)

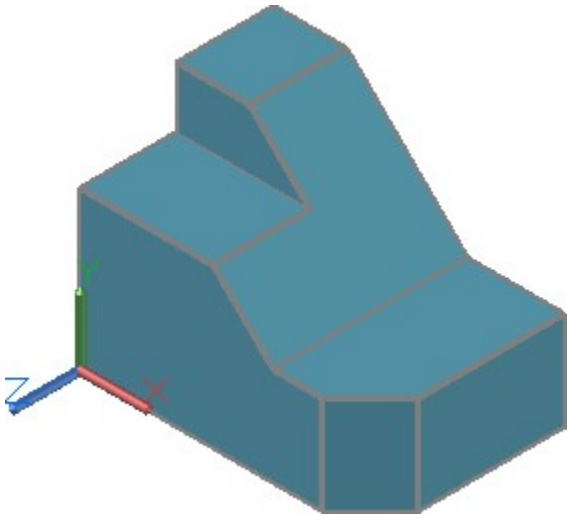


Figure Step 14

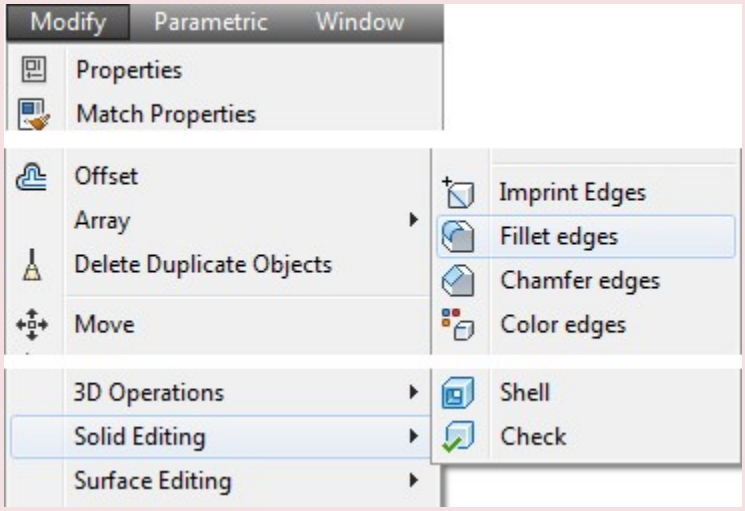
Step 15

Save and close the drawing.

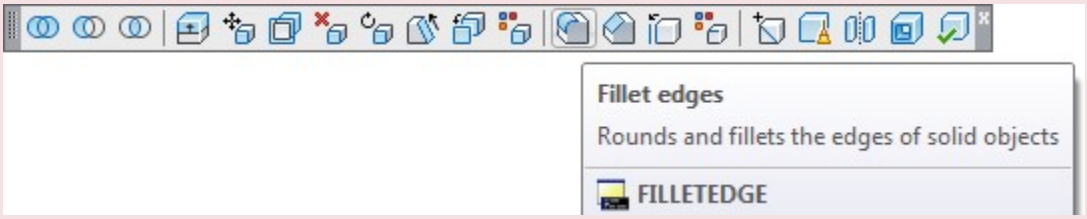
AutoCAD Command: FILLETEDGE

The FILLETEDGE command is used to create a fillet (tangent arc) on the edge or edges of a solid model.

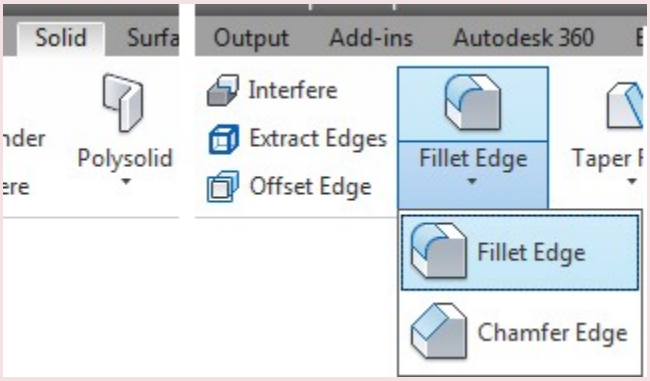
Shortcut: **none**



Modify Pull-Down



Solid Editing Toolbar

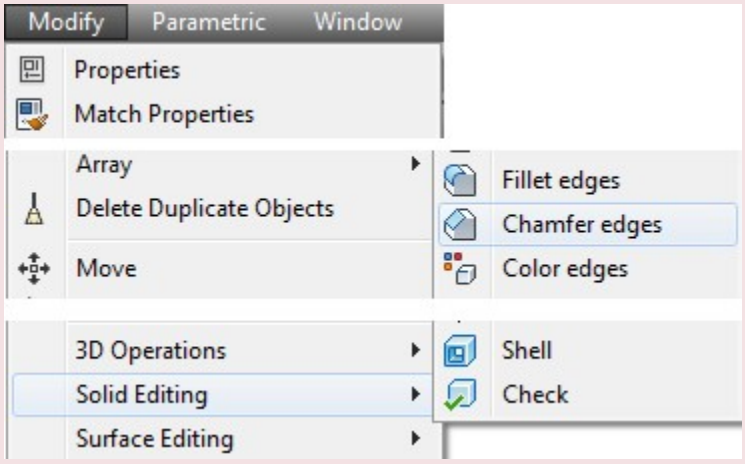


Solid Ribbon

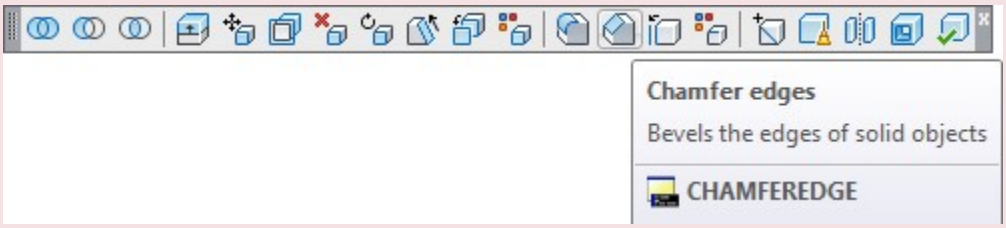
AutoCAD Command: CHAMFEREDGE

The CHAMFEREDGE command is used to create a chamfer on the edge or edges of a solid model.

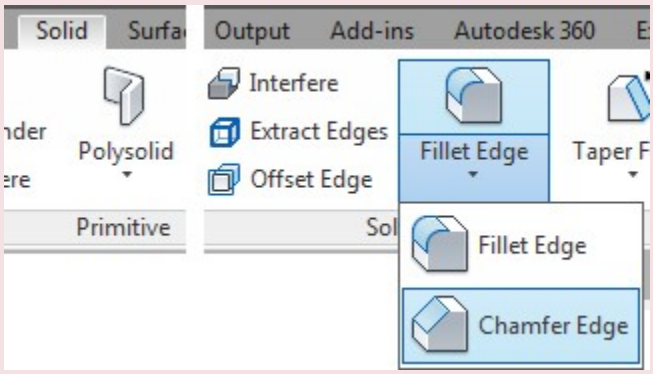
Shortcut: **none**



Modify Pull-down



Solid Editing Toolbar



Solid Ribbon

WORK ALONG: Creating Fillets and Chamfers on Solid Models

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 19-2.

Step 3

Set layer: Pline as the current layer, the current UCS to Front, the current view to SE Isometric, and the current visual style to 2D Wireframe. Create a the closed pline on the Front view using the figures as a reference. Ignore the fillets and rounds. (Figure Step 3A, 3B, and 3C)

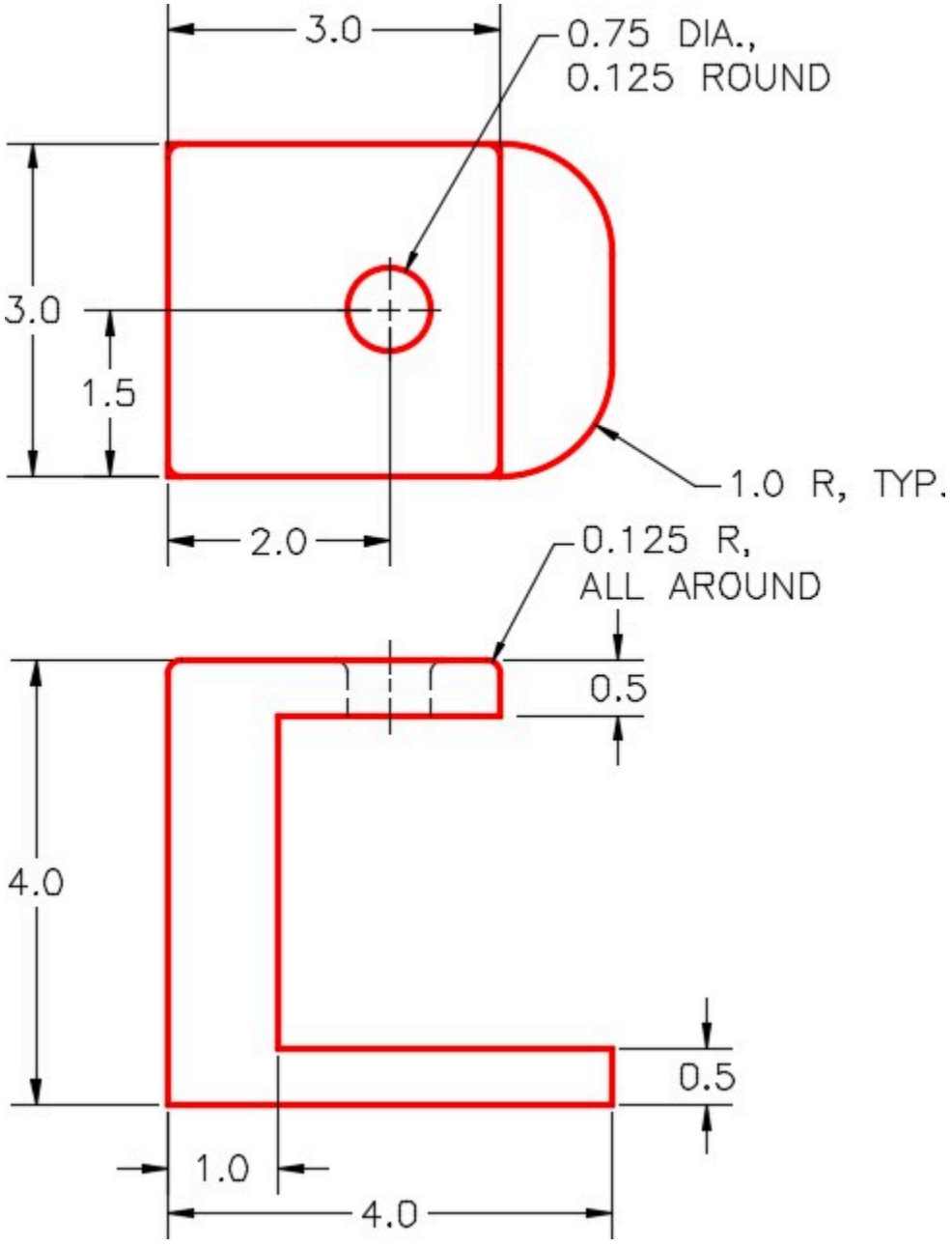


Figure Step 3A
Dimensioned Multiview of Model 1

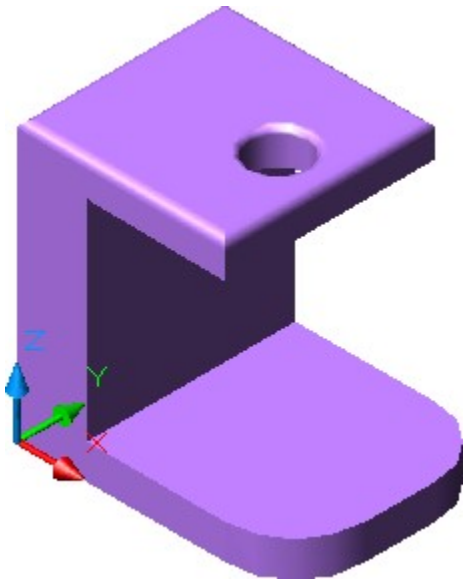


Figure Step 3B Completed Solid Model of Model 1 –
SE Isometric View

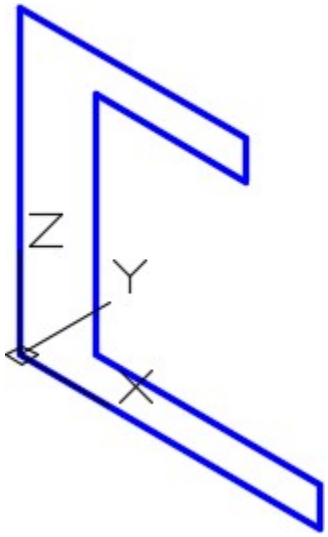


Figure Step 3C
Location of 0,0,0

Step 4

On layer: Solid 2, create a solid model as shown in the figure. Do not include the fillets and rounds. For the hole, extrude a circle. Check to ensure that your model is one solid object only using the Properties window. (Figure Step 4A and 4B)

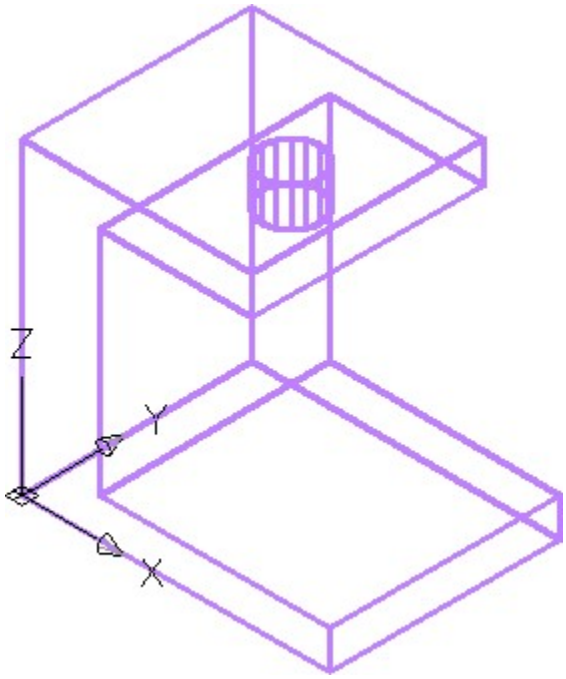


Figure Step 4A

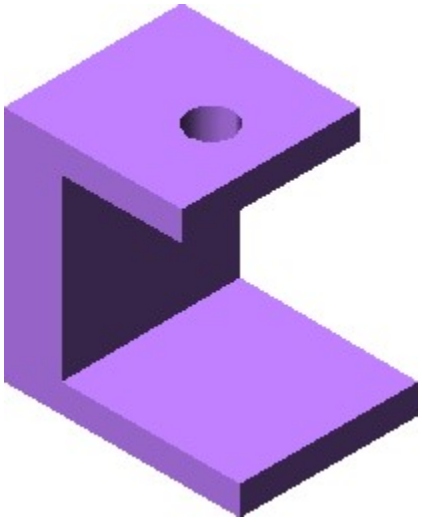


Figure Step 4B

Step 5

Turn layer: Pline off.

Step 6

Set the current UCS to Top. Copy the solid model, that you completed in Step 4, 8 inches in the positive Y direction. The 8 inches is measured from the same point on model to the same point on the copy. The actual distance between the two model will be 5 inches. (Figure Step 6)

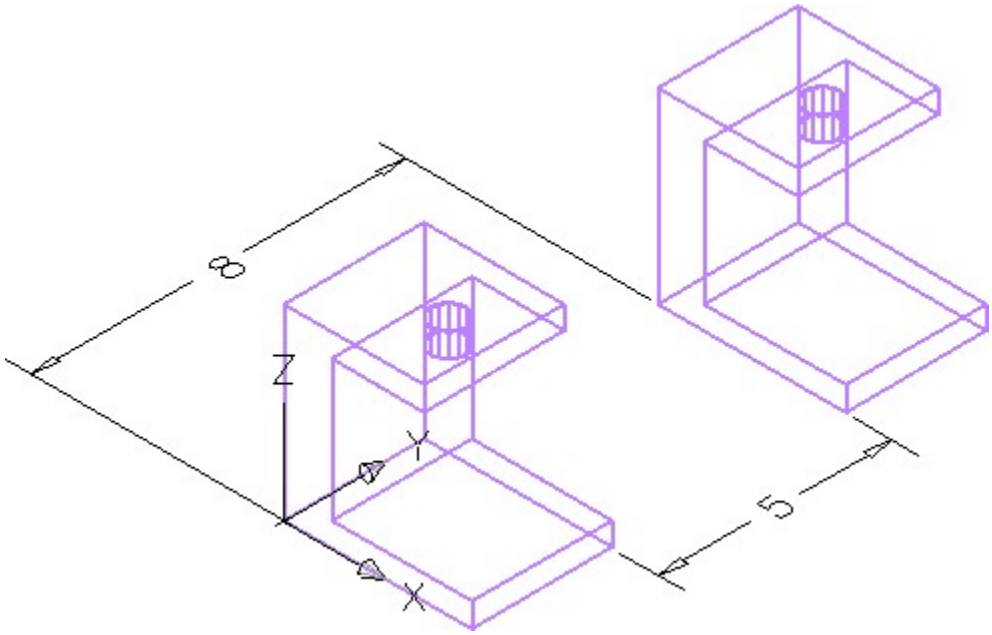


Figure Step 6

Apply Steps 7 to 10 to the model on the left side of Figure Step 6.

Step 7

Turn layer: Pline off. Enter the FILLETEDGE command, as shown below, to fillet the bottom corners. (Figure Step 7)

Command: **FILLETEDGE**

Radius = 1.2500

Select an edge or [Chain/Loop/Radius]: **P1**

Select an edge or [Chain/Loop/Radius]: **P2**

Select an edge or [Chain/Loop/Radius]:

2 edge(s) selected for fillet.

Press Enter to accept the fillet or [Radius]: **R**

Specify Radius or [Expression] <1.0000>: **1**

Press Enter to accept the fillet or [Radius]:

Command:

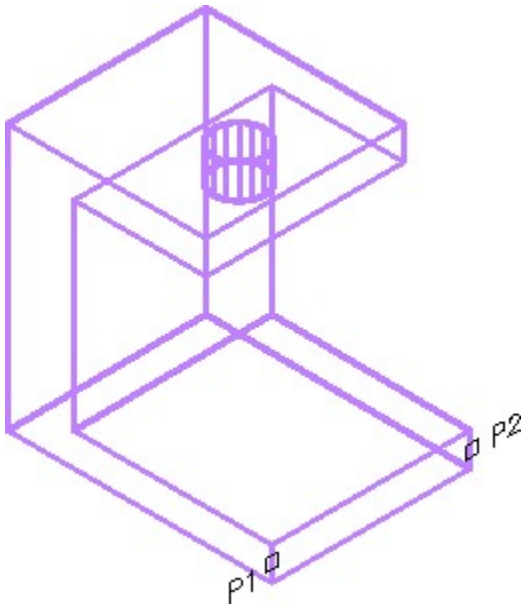


Figure Step 7

AUTHOR'S COMMENTS: I had you turn layer Pline off so that the plines do not get in the way when you are selecting the fillet edges of the model.

Step 8

Enter the FILLETEDGE command, as shown below, to fillet the top edge. (Figure Step 8)

Command: **FILLETEDGE**

Radius = 1.0000

Select an edge or [Chain/Loop/Radius]: **P3**

select an edge or [Chain/Loop/Radius]: **C**

(Use the Chain option.)

Select an edge chain or [Edge/Radius]: **E**

Select an edge or [Chain/Loop/Radius]: **P4**

Select an edge or [Chain/Loop/Radius]: **P5**

Select an edge or [Chain/Loop/Radius]: **P6**

Select an edge or [Chain/Loop/Radius]: **R**

Enter fillet radius or [Expression] <1.0000>: **.125**

Select an edge or [Chain/Loop/Radius]:

4 edge(s) selected for fillet.

Press Enter to accept the fillet or [Radius]:

Command:

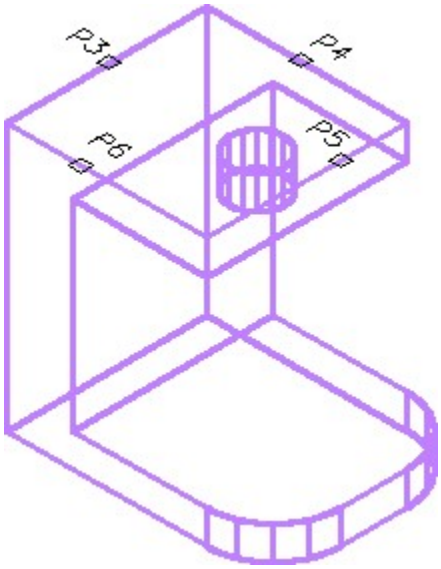


Figure Step 8

AUTHOR'S COMMENTS: It is important to use the Chain option when the fillet starts and ends at the same corner. When a chain is used, all the corners are filleted correctly.

Step 9

Enter the FILLETEDGE command, as shown below, to fillet the top of the hole. (Figure Step 9)

Command: **FILLETEDGE**

Radius = 0.1250

Select an edge or [Chain/Loop/Radius]: **P7**

Select an edge or [Chain/Loop/Radius]:

1 edge(s) selected for fillet.

Press Enter to accept the fillet or [Radius]:

Command:

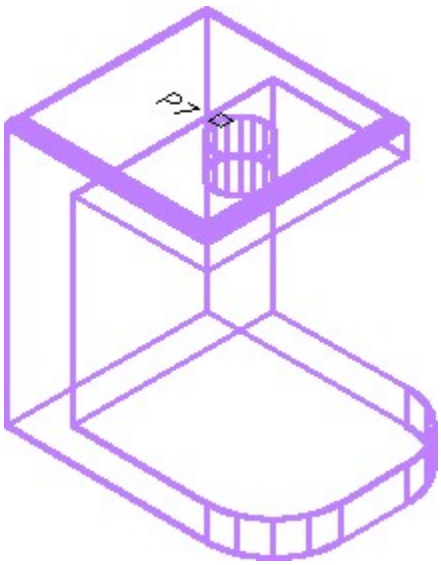


Figure Step 9

Step 10

Set the visual style to Realistic. The drawing should appear as shown in the figure. (Figure Step 10)

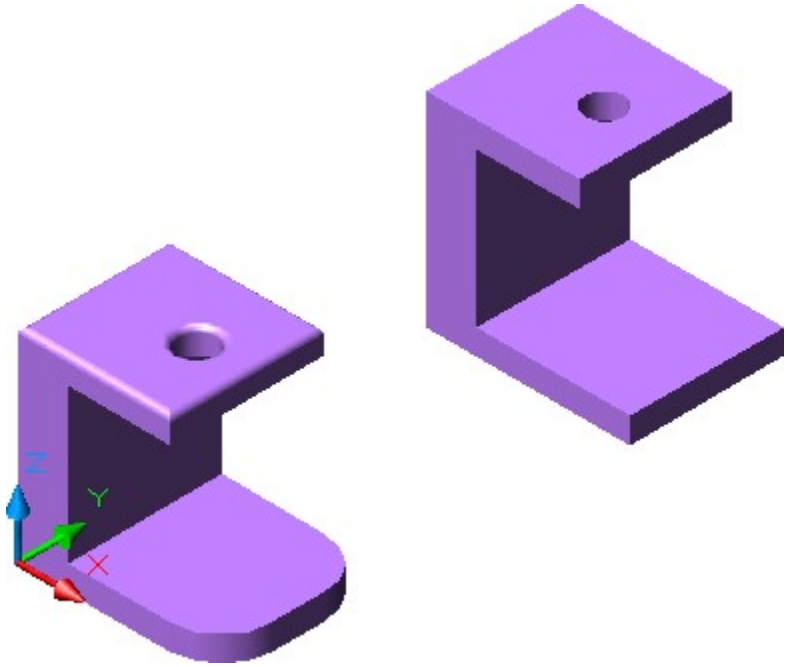


Figure Step 10

Step 11

Referring to the figures, edit the model shown in Steps 11 to 13. (Figure Step 11A and 11B)

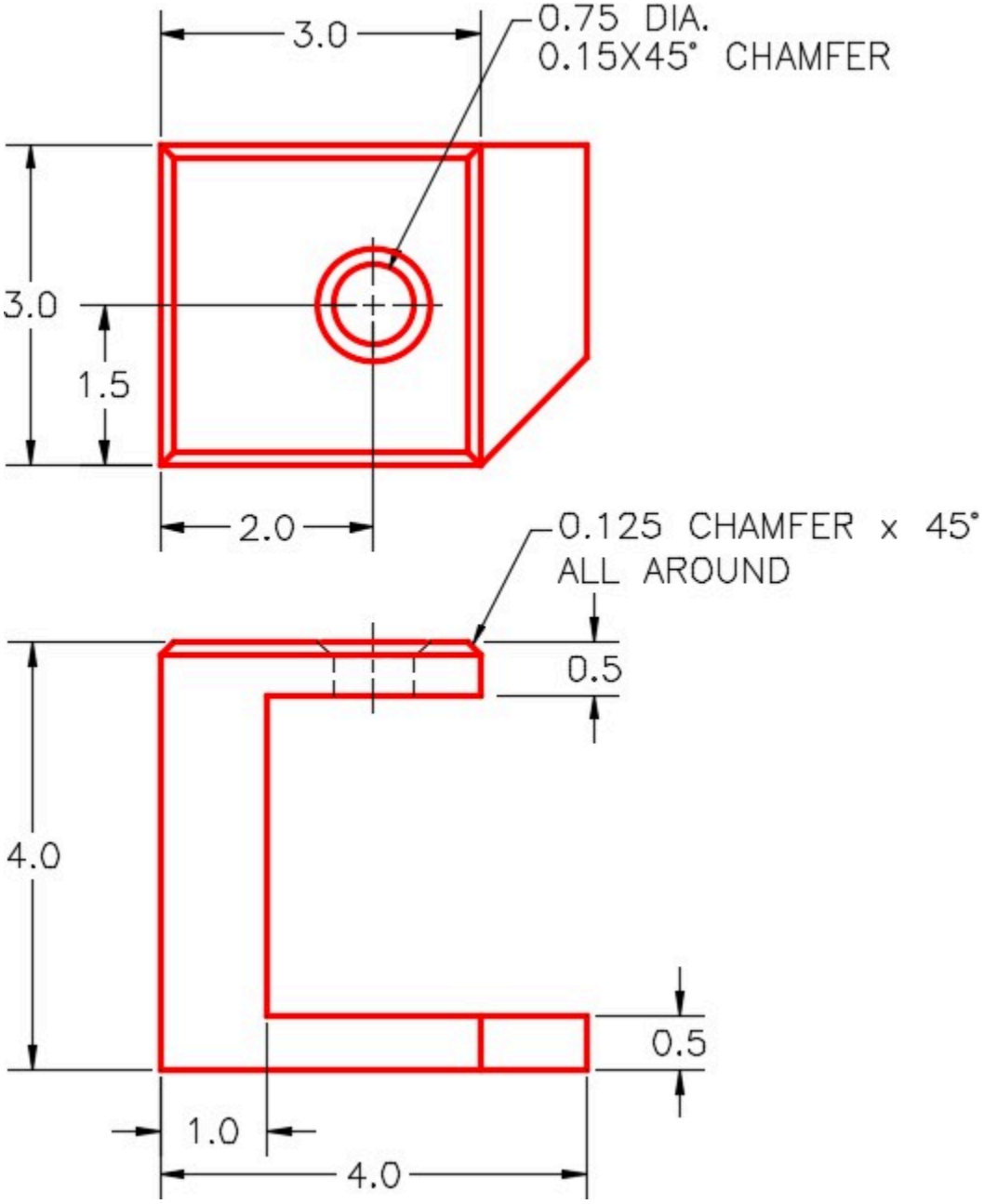


Figure Step 11A
Dimensioned Multiview Of Model 2

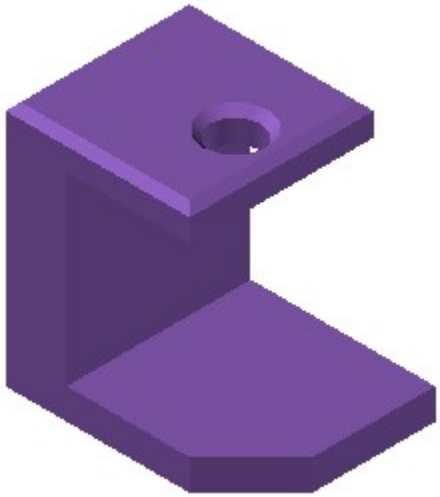


Figure Step 11B
Completed Solid Model of Model 2 – SE Isometric View

Apply Steps 12 to 14 to the model on the right side in Figure Step 6.

Step 12

Enter the CHAMFEREDGE command, as shown below, to chamfer the bottom corner. (Figure Step 12A and 12B)

Command: **CHAMFEREDGE**

Distance1 = 1.2500, Distance2 = 1.2500

Select an edge or [Loop/Distance]: **P1**

Select another edge on the same face or [Loop/Distance]: **D**

Specify Distance1 or [Expression] <1.0000>: **1**

Specify Distance2 or [Expression] <1.0000>: **1**

Select another edge on the same face or [Loop/Distance]:

Press Enter to accept the chamfer or [Distance]:

Command:

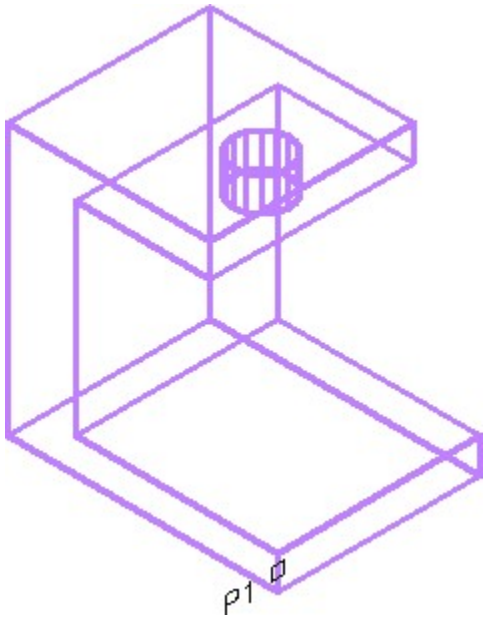


Figure Step 12A

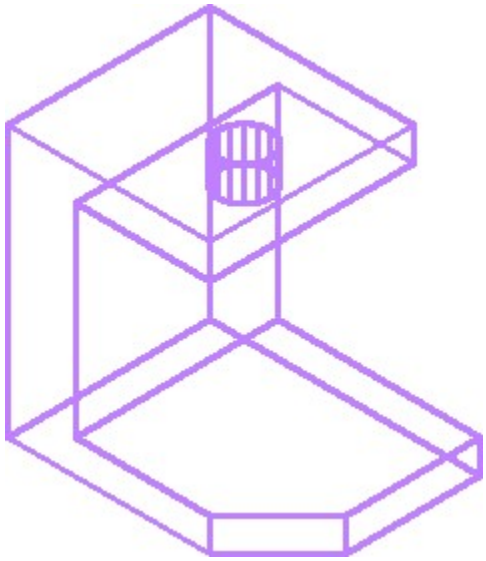


Figure Step 12B

Step 13

Enter the CHAMFEREDGE command, as shown below, to chamfer the hole. (Figure Step 13A and 13B)

Command: **CHAMFEREDGE**

Distance1 = 1.0000, Distance2 = 1.0000

Select an edge or [Loop/Distance]: **P2**

Select another edge on the same face or [Loop/Distance]: **D**

Specify Distance1 or [Expression] <1.0000>: **.15**

Specify Distance2 or [Expression] <1.0000>: **.15**

Select another edge on the same face or [Loop/Distance]:

Press Enter to accept the chamfer or [Distance]:

Command:

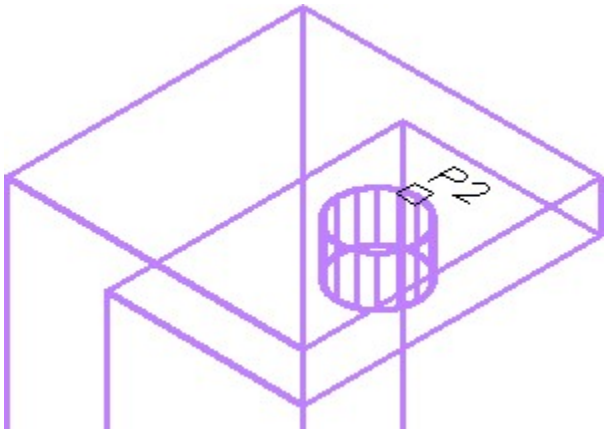


Figure Step 13A

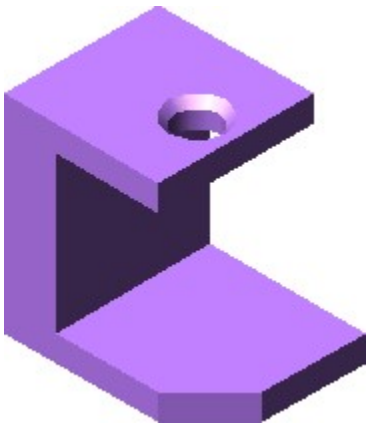


Figure Step 13B

Step 14

Enter the CHAMFEREDGE command, as shown below, to create a loop chamfer around the top of the model. (Figure Step 14)

Command: **CHAMFEREDGE**

Distance1 = 0.1500, Distance2 = 0.1500

Select an edge or [Loop/Distance]: **L**

Select edge of loop or [Edge/Distance]:

Enter an option [Accept/Next] <Accept>: **N**

Enter an option [Accept/Next] <Accept>:

Select edge of loop or [Edge/Distance]: **D**

Specify Distance1 or [Expression] <0.1500>: **0.125**

Specify Distance2 or [Expression] <0.1500>: **0.125**

Select another edge on the same face or [Loop/Distance]:

Press Enter to accept the chamfer or [Distance]:

Command:

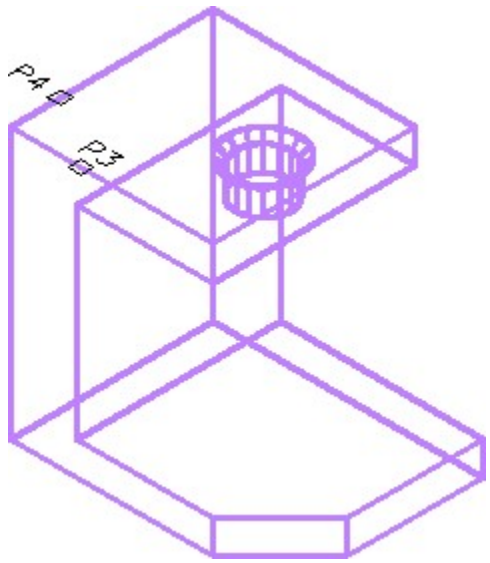


Figure Step 14

AUTHOR'S COMMENTS: The CHAMFEREDGE command is a little tricky. When you select the edge, the command may highlight wrong plane. Continuing entering N until the correct plane highlights.

AUTHOR'S COMMENTS: It is important to use the Loop option when the chamfer starts and ends at the same corner. When a loops is used, all the corners will be chamfered correctly.

Step 15

Set the current UCS to Realistic. The final drawing should appear as shown in the figure. (Figure Step 15)

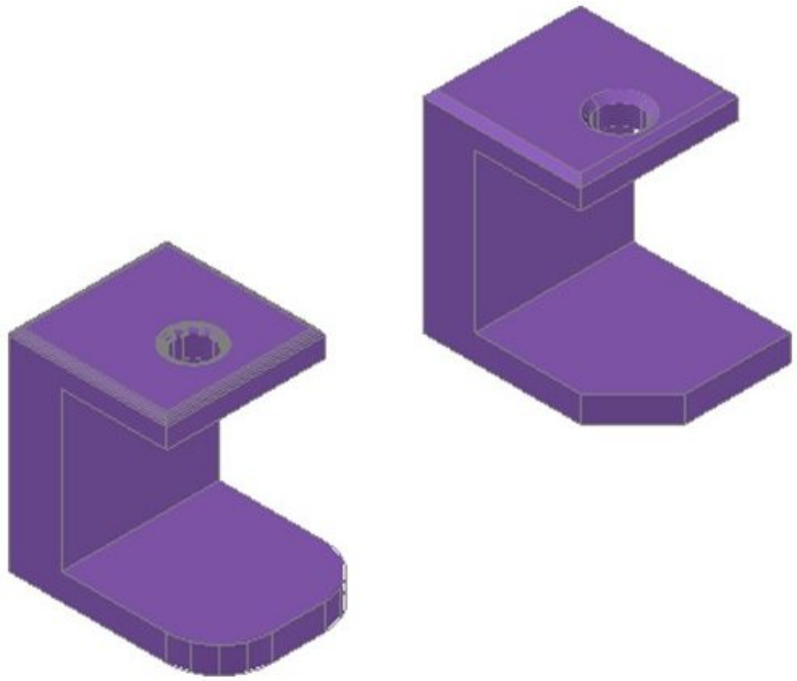


Figure Step 15

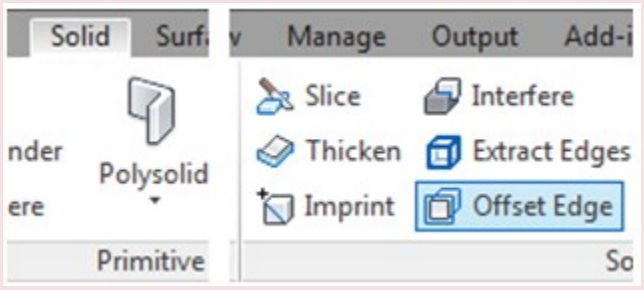
Step 16

Save and close the drawing.

AutoCAD Command: OFFSETEDGE

The OFFSETEDGE command is used to create a closed polyline that is offset at a specified distance from the edges of a selected planar face on a 3D solid or surface.

Shortcut: **none**



Solid Ribbon

WORK ALONG: Creating Fillets and Chamfers on Solid Models

Step 1

Open the drawing: AutoCAD 3D Workalong 19-2. Using SAVEAS, save the drawing with the name: AutoCAD 3D Workalong 19-3 (Figure Step 1)

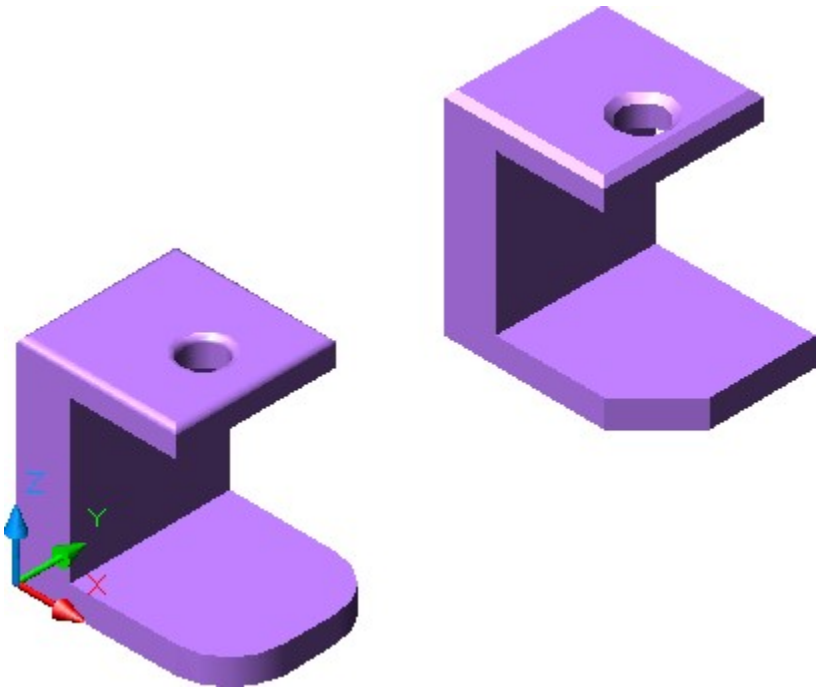


Figure Step 1

Step 2

Set layer: Pline as the current layer.

Step 3

Using the model on the left, enter the OFFSETEDGE command, as shown below, to create a closed polyline 0.25 inches from the edge. (Figure Step 3A and 3B)

Command: **OFFSETEDGE**

Corner = Sharp

Select face:

Specify through point or [Distance/Corner]: **D**

Specify distance <0.1250>: **0.25**

Specify point on side to offset:

Select face:

Command:

Figure Step 3A

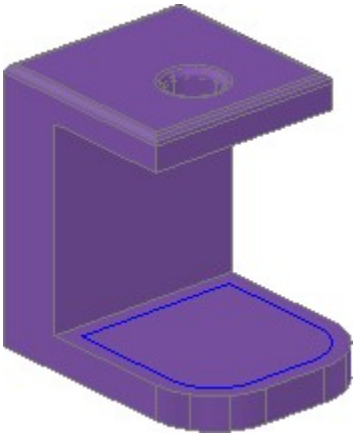


Figure Step 3B

Step 4

Set layer: Solid 2 as the current layer. Enter the PRESSPULL command. When prompted, select inside the polyline and enter the height of -0.35 as shown below. (Figure Step 4A and 4B)

Command: **PRESSPULL**

Select object or bounded area: Specify extrusion height or [Multiple]:

Specify extrusion height or [Multiple]: **-0.35**

1 extrusion(s) created

Select object or bounded area:

Command:

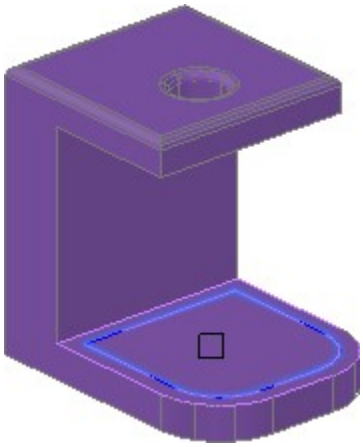


Figure Step 4A

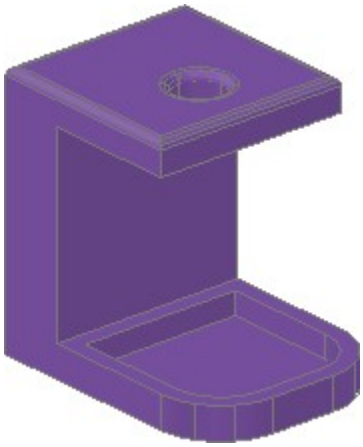


Figure Step 4B

Step 5

Save and close the drawing.

Key Principles

Key Principles in Module 19

1. When constructing solid models, it is important to ensure that the completed solid model consists of one AutoCAD object.
2. The INTERSECT command is used to create a solid by finding the volume shared by two or more solids.
3. It is important to use the Chain option when starting a fillet on a solid model that starts and ends at the same That ensures that all the corners are filleted correctly.
4. It is important to use the Loop option when starting a chamfer on a solid model that starts and ends at the same That ensures that all the corners are chamfered correctly.

Lab Exercise 19-1

Time allowed: 30 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 19-1	3D Layout Metric	Millimeters

Step 1

On layer: Solid 3, draw a solid model of the object. (Figure Step 1A, 1B, and 1C)

Step 2

Draw the closed plines on layer: Pline.

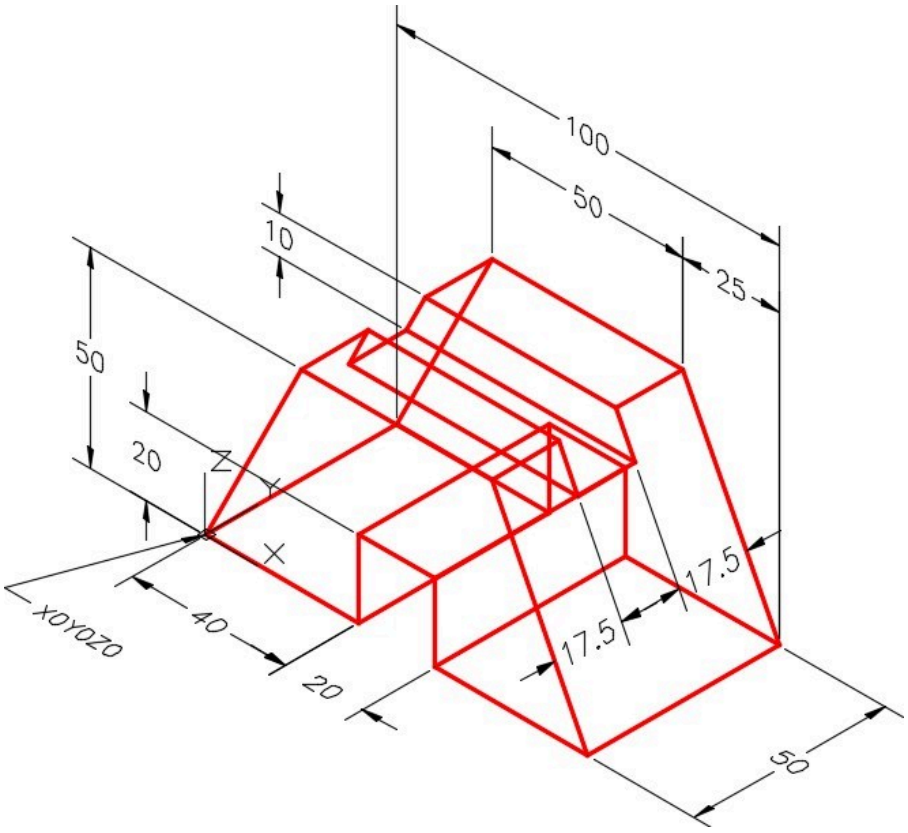


Figure Step 1A
Dimensioned Wireframe Model

AUTHOR'S COMMENTS: Since the Top view is a rectangle, you do not have to include it. The model can be obtained from the intersection of the front and right side views only.

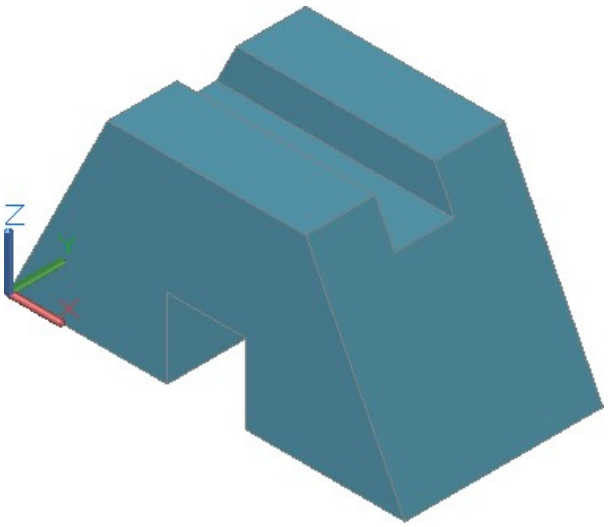


Figure Step 1B Completed Solid Model SE Isometric View

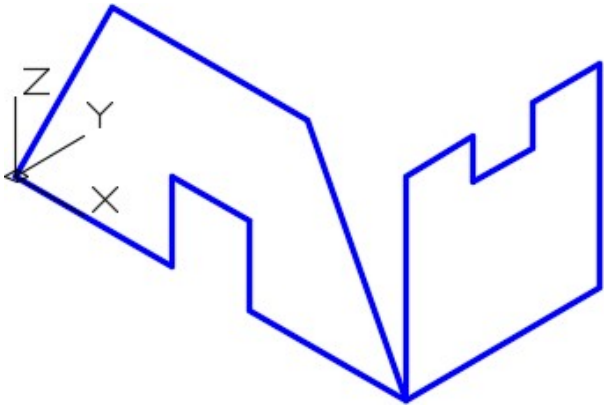


Figure Step 1C Hint

Step 3

Use the INTERSECT command and the two extruded solids to create the solid model.

Step 4

Turn layer: Pline off and set the current visual style to Realistic.

Step 5

Set the Insertion Units, change the current UCS to World, and check the model with the key.

Step 6

Save and close the drawing.

Lab Exercise 19-2

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 19-2	3D Layout Metric	Millimeters

Step 1

Draw the closed plines on layer: Pline.

Step 2

On layer: Solid 3, draw a solid model of the object shown in the figures. (Figure Step 2A, 2B, and 2C)

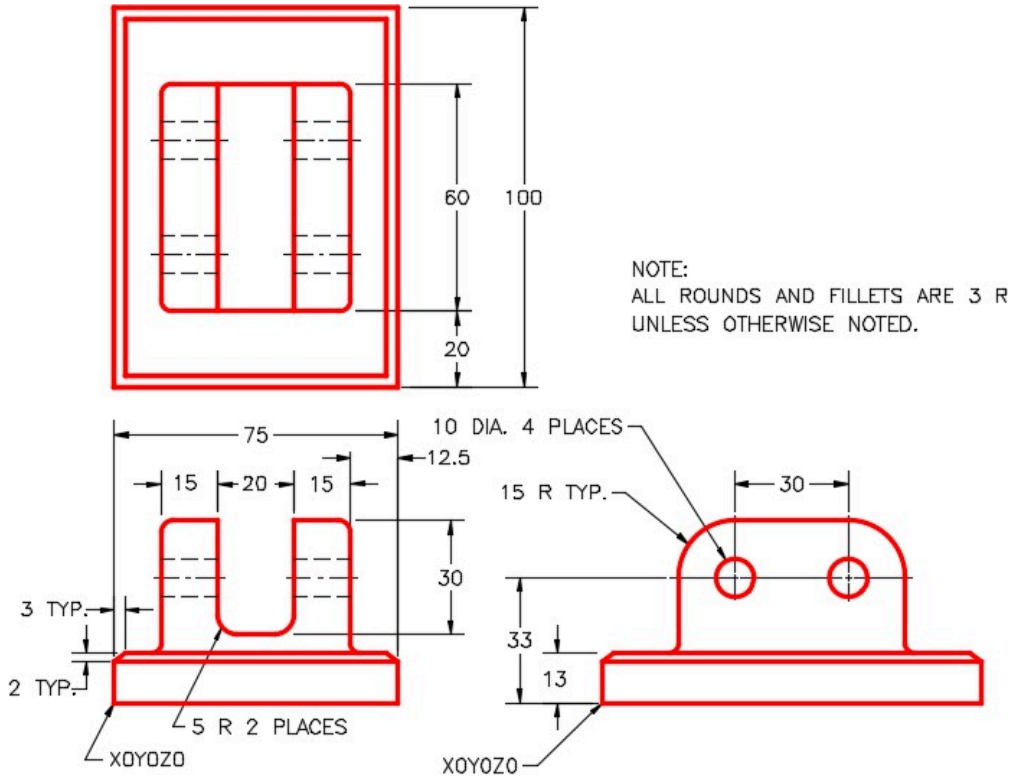


Figure Step 2A
Dimensioned Multiview Drawing

AUTHOR'S COMMENTS: Since the Top view is a rectangle, you do not have to include it. The model can be obtained from the intersection of the front and right side views only.

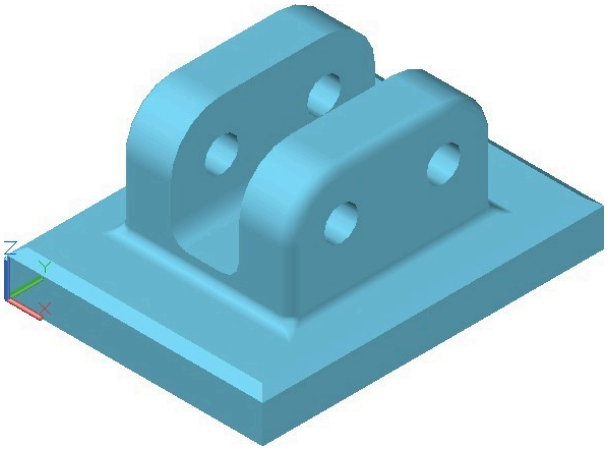


Figure Step 2B Completed Solid Model SE Isometric View

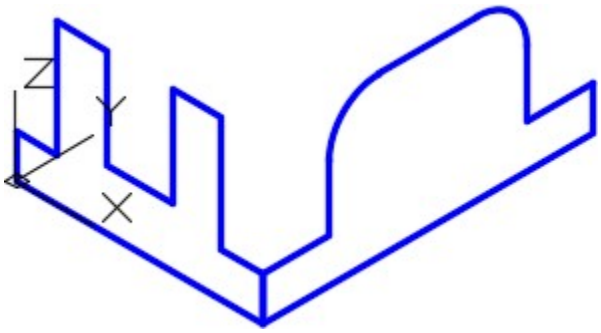


Figure Step 2C Hint

Step 3

Use the INTERSECT command and the two extruded solids to create the solid model.

Step 4

Add the fillets and chamfers to the solid model.

Step 5

Turn layer: Pline off and set the current visual style to Realistic.

Step 6

Set the Insertion Units, change the current UCS to World, and check the model with the key.

Step 7

Save and close the drawing.

Lab Exercise 19-3

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 19-3	3D Layout English	Inches

Step 1

Draw the closed plines on layer Pline.

Step 2

On layer: Solid 3, draw a solid model of the object shown in the figures. (Figure Step 2A, 2B, and 2C)

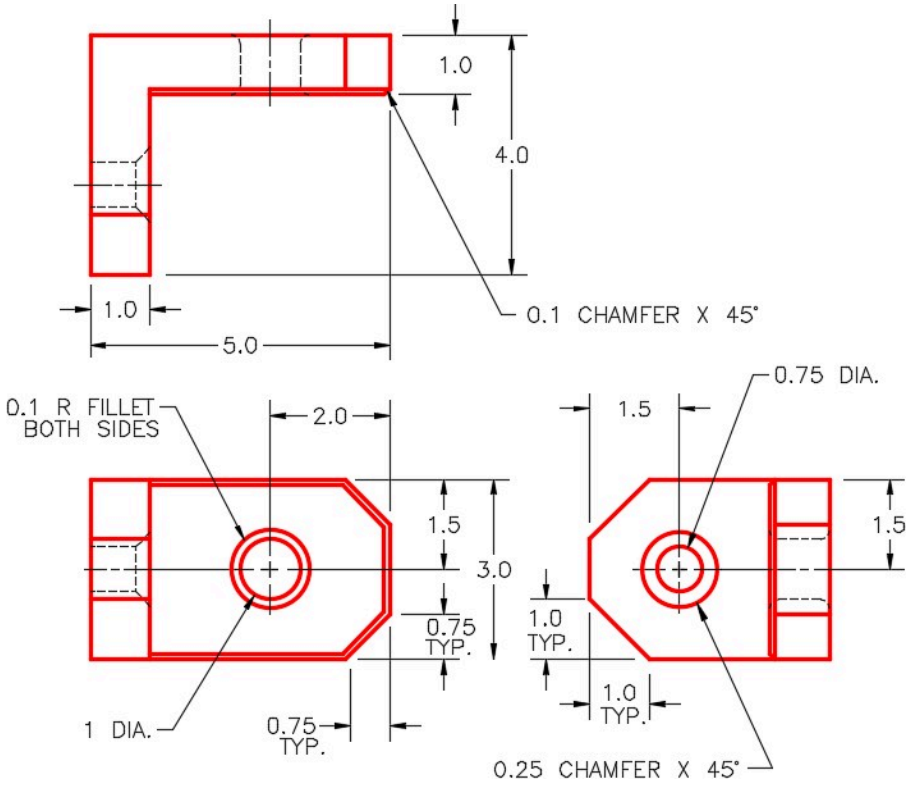


Figure Step 2A
Dimensioned Multiview Drawing

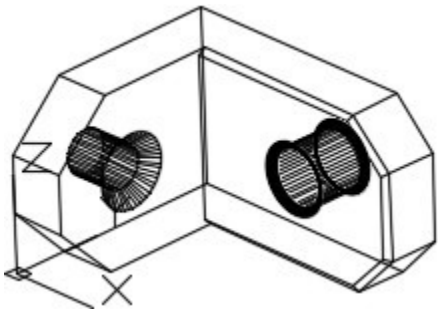


Figure Step 2B
Location of 0,0,0

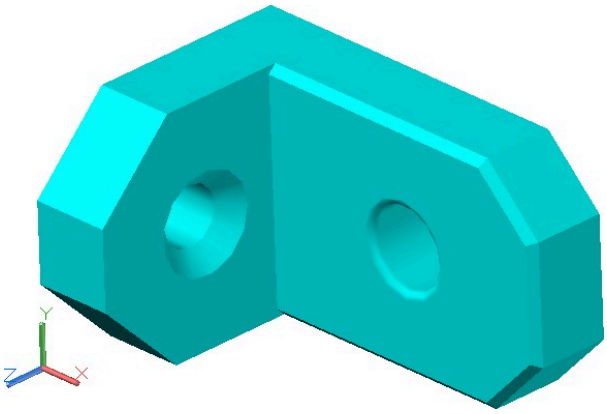


Figure Step 2C Completed Model SE Isometric View

Step 3

Add the fillets and chamfers to the solid model.

Step 4

Turn layer: Pline off and set the current visual style to Realistic.

Step 5

Set the Insertion Units, change the current UCS to World, and check the model with the key.

Step 6

Save and close the drawing.

Lab Exercise 19-4

Time allowed: 40 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 19-4	3D Layout English	Inches

Step 1

Draw the closed plines on layer: Pline.

Step 2

On layer: Solid 4, draw a solid model of the object shown in the figures. Figure Step 2A, 2B, 2C, and 2D)

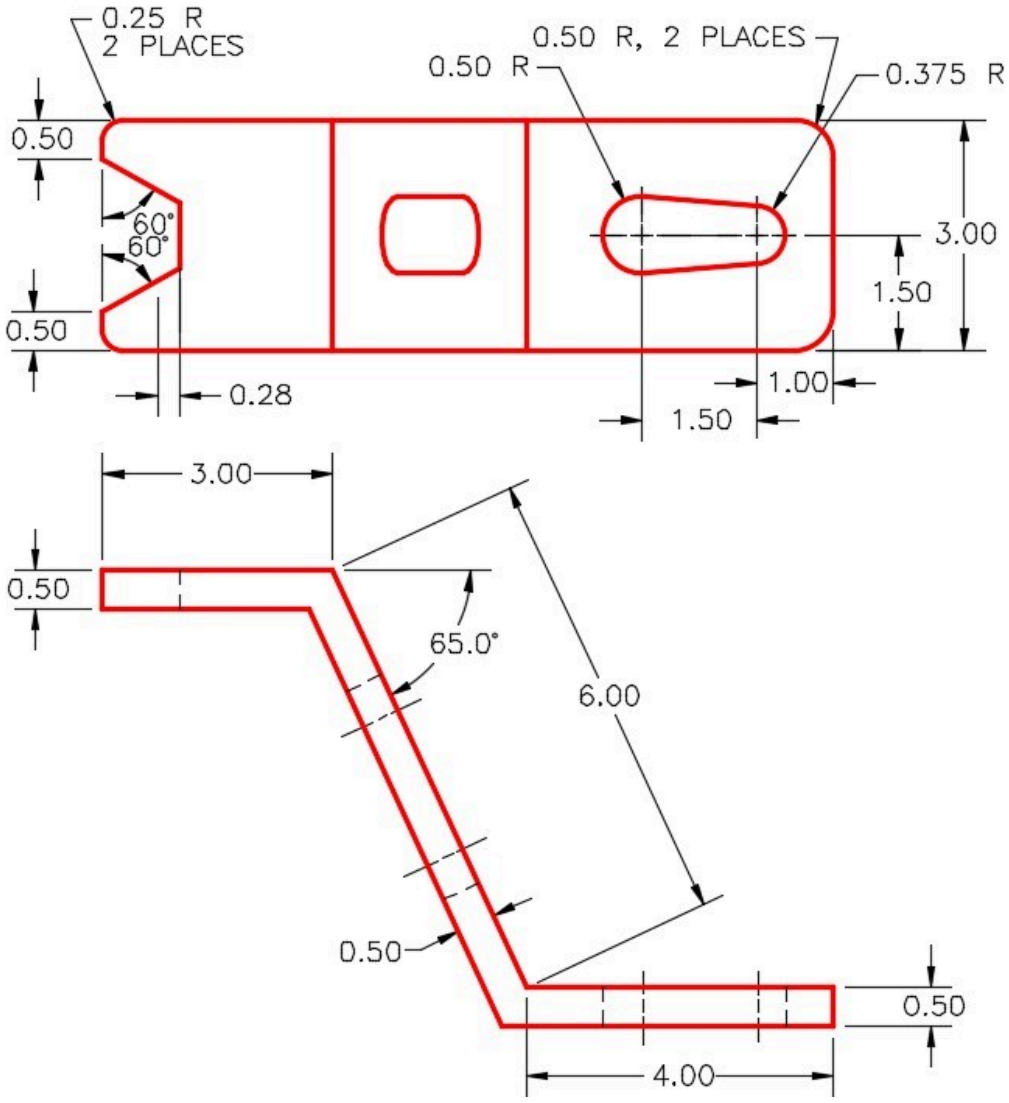


Figure Step 2A
Dimensioned Multiview Drawing

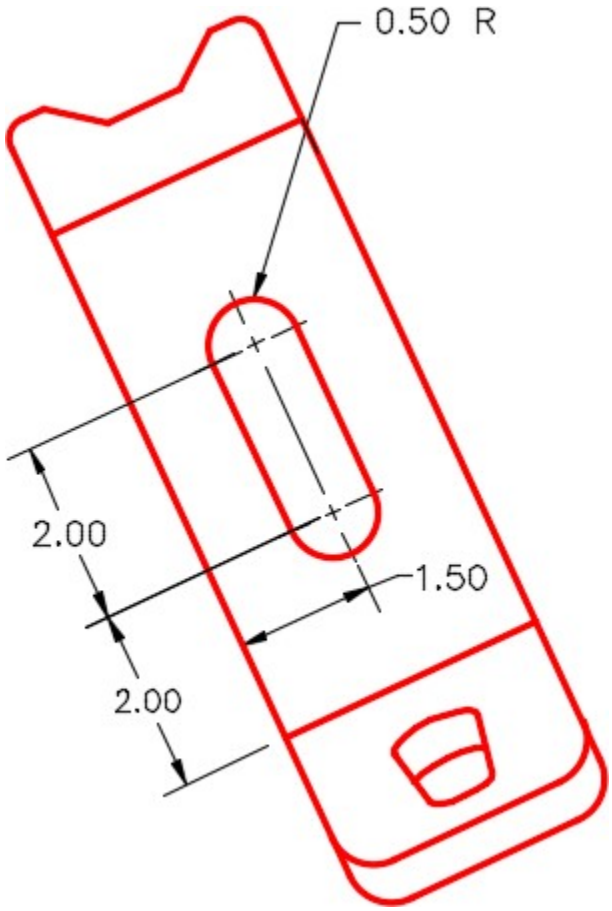


Figure Step 2B
Auxiliary View

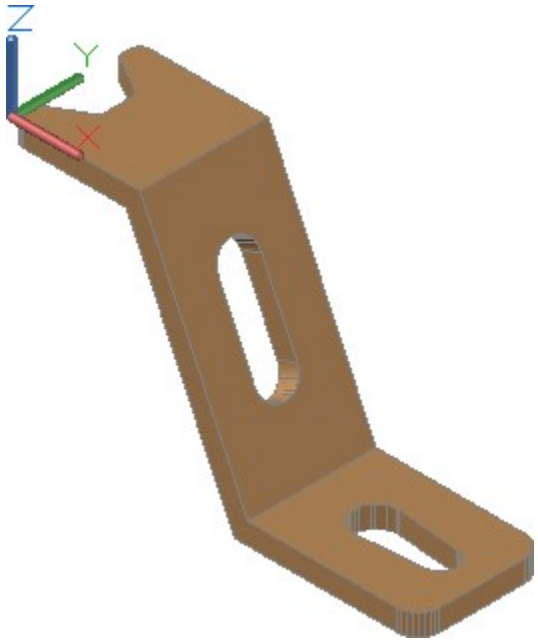


Figure Step 2C
SE Isometric View

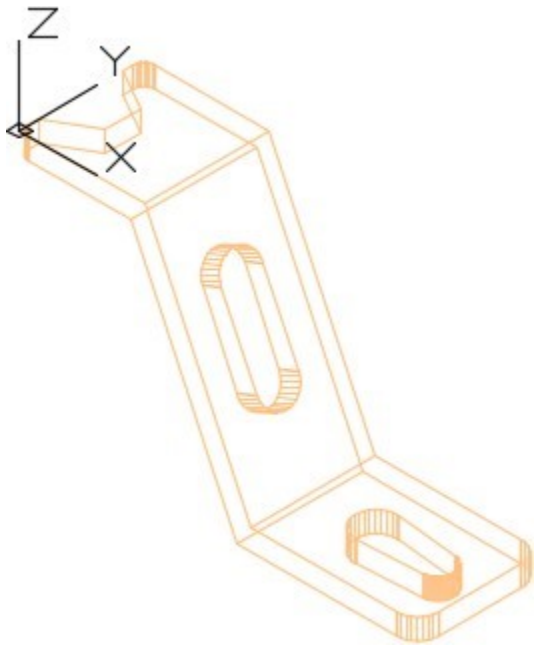


Figure Step 2D
Location of 0,0,0

Step 3

Turn layer: Pline off and set the current visual style to Realistic.

Step 4

Set the Insertion Units, change the current UCS to World, and check the model with the key.

Step 5

Save and close the drawing.

Module 20 Sectioning Solid Models

Learning Outcomes

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

1. Apply the SLICE command to cut solid models into two individual solids.
2. Apply the SECTION command to create 2D cross sectional regions through solid models.

Sectioning

The SLICE and SECTION commands are used to create cross sections from solid models. They can be very useful commands and save you drawing time.

The SLICE command, see Figure 20-1A and 20-1B, cuts a solid model into two individual solids.

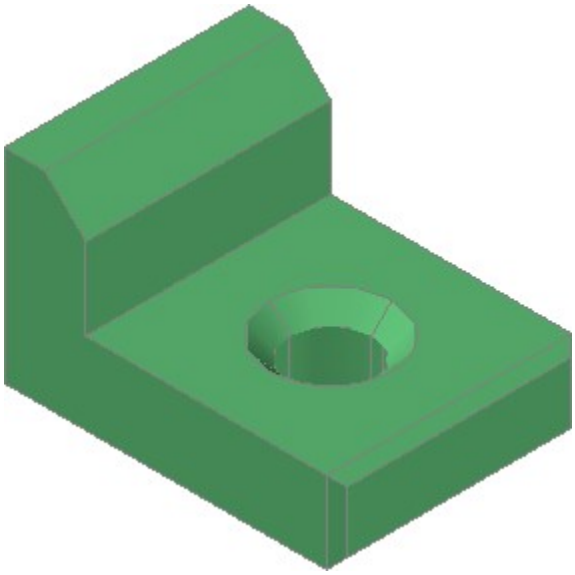


Figure 20-1A
Before Slice

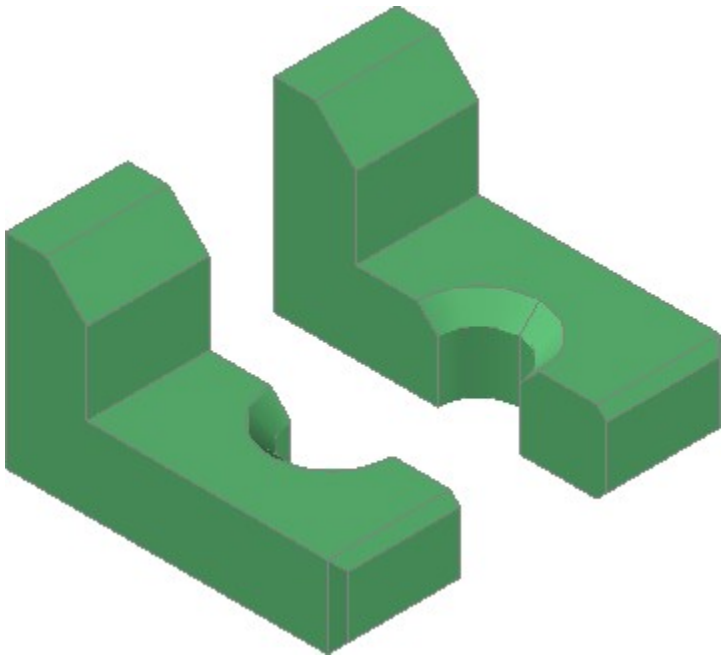


Figure 20-1B
After Slice

The SECTION command, see Figure 20-2, creates a two dimensional region along a selected cutting plane. The region can then be relocated to produce a cross section view in 2D and used as a detail drawing. See Figure 20-3 and 20-4.

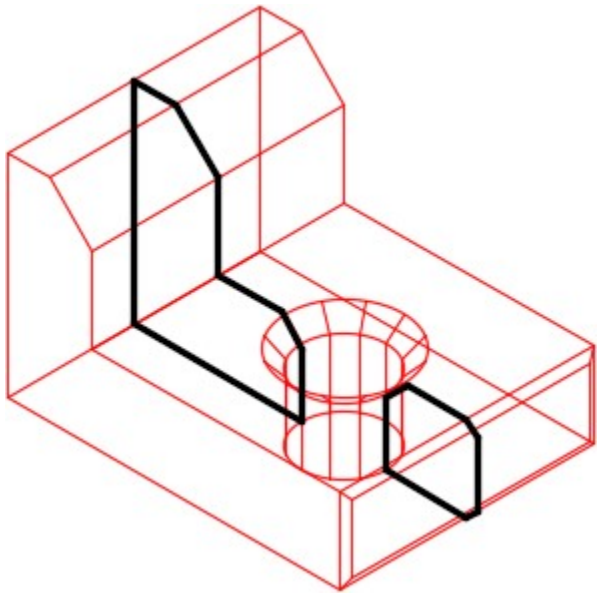


Figure 20-2
Section

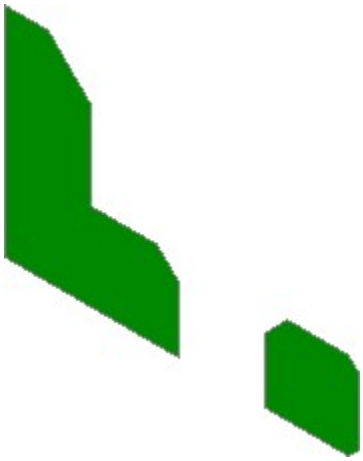


Figure 20-3 Removed Region Shaded

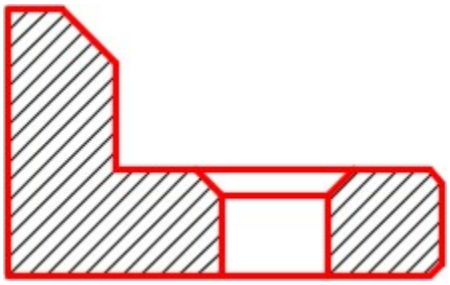
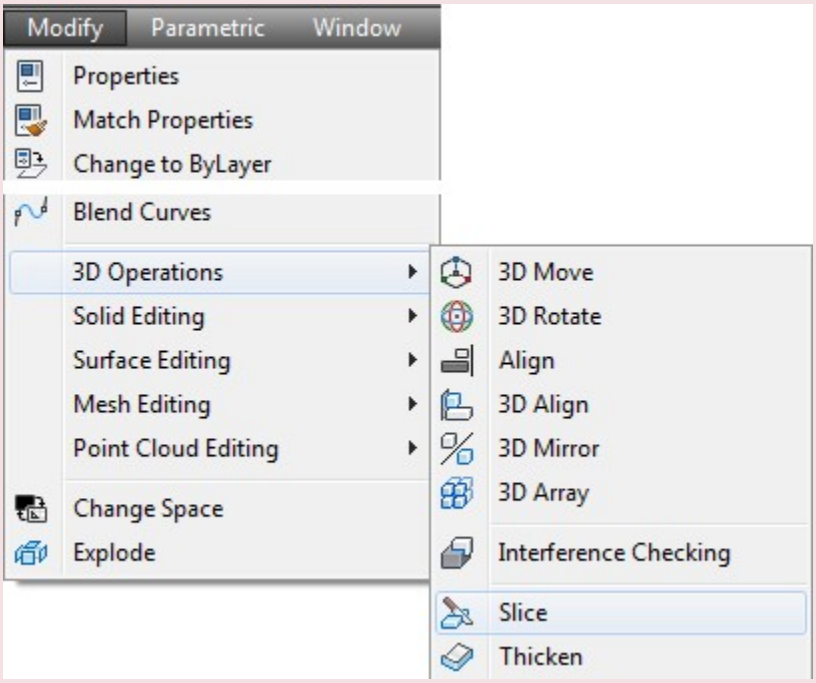


Figure Step 20-4 Completed 2D Cross Section View

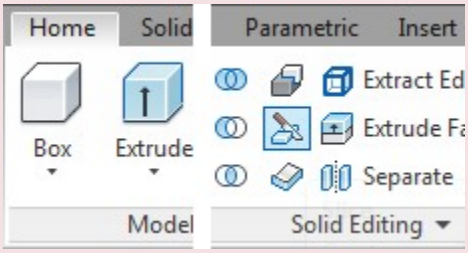
AutoCAD Command: SLICE

The SLICE command is used to cut a solid into two individual solids.

Shortcut: none



Modify Pull-down

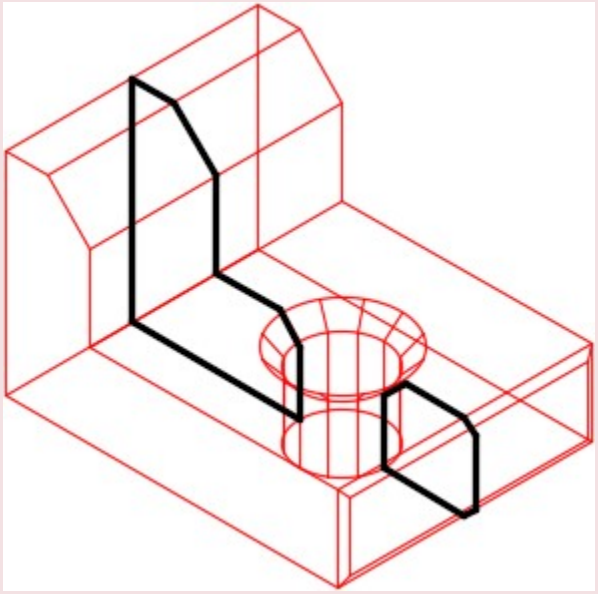


Home Ribbon

AutoCAD Command: SECTION

The SECTION command is used to create a two- dimensional region along a specified plane in a solid object.

Shortcut: **SEC**



AUTHOR'S COMMENTS: Type this command on the command line. It is not included in any standard AutoCAD menu.

WORK ALONG: Sectioning Solid Models

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 20-1.

Step 3

Using the figures as a reference, draw the closed plines on layer: Pline. (Figure Step 3)

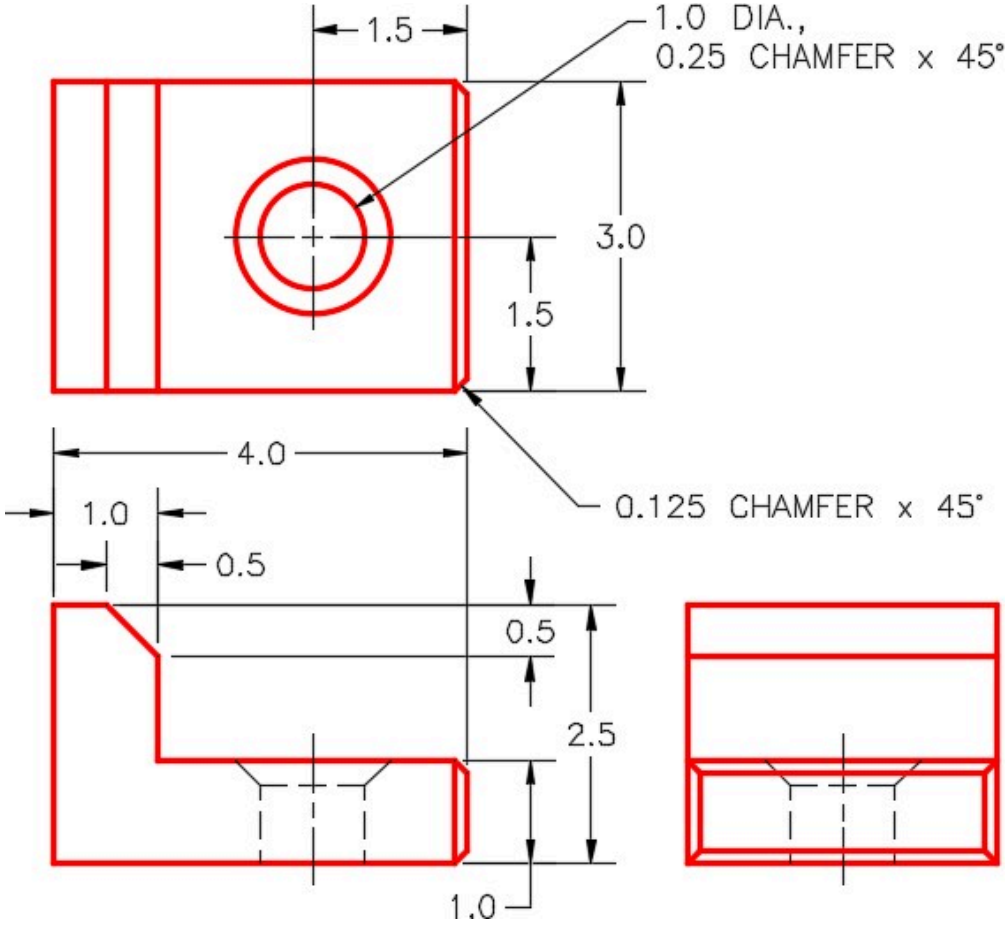


Figure Step 3

Step 4

Draw the solid model on layer: Solid 7. (Figure Step 4)

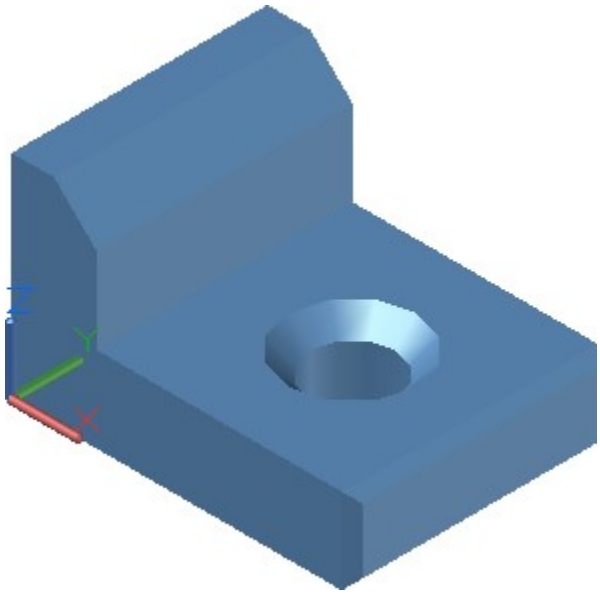


Figure Step 4

Step 5

Turn layer: Pline off. Set the current UCS set to World and make a copy of the solid model 12 inches along the X axis. (Figure Step 5)

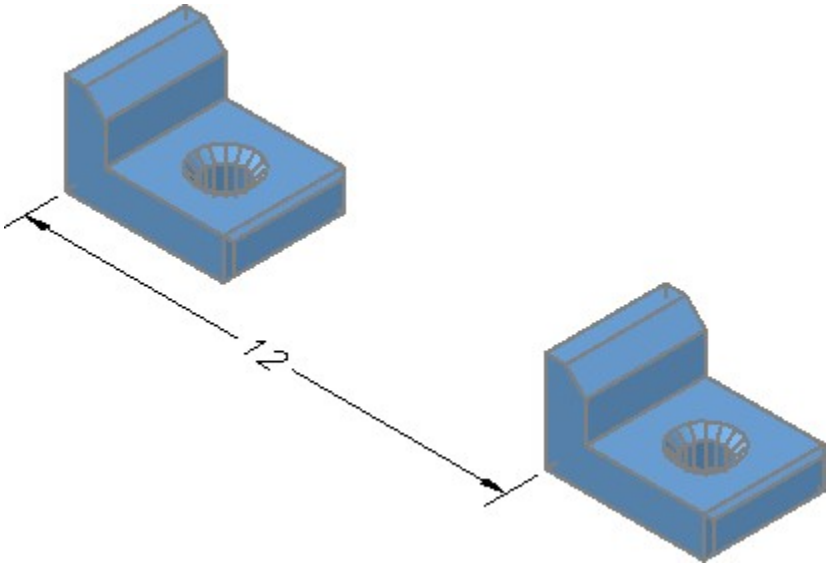


Figure Step 5

Step 6

Set the current visual style to 2D Wireframe. On the original solid model, enter the SLICE command, as shown below, to slice it into two parts. (Figure Step 6A and 6B)

Command: **SLICE**

Select objects: 1 found

(Select the solid model.)

Select objects:

(Press Enter.)

Specify first point on slicing plane by [Object/Zaxis/View/XY/YZ/ZX/3points] <3points>: *(mid)* **P1**

Specify second point on plane: *(mid)* **P2**

Specify third point on plane: *(mid)* **P3**

(Ensure you snap to the midpoint of each edge. If you drew an imaginary line from P1 to P2 to P3 and back to P1, it would form a cutting plane. You will have to orbit the model to do this.)

Specify a point on desired side of the plane or [keep Both sides]: **B**

(Enter B to retain both sides of the slice.)

Command:

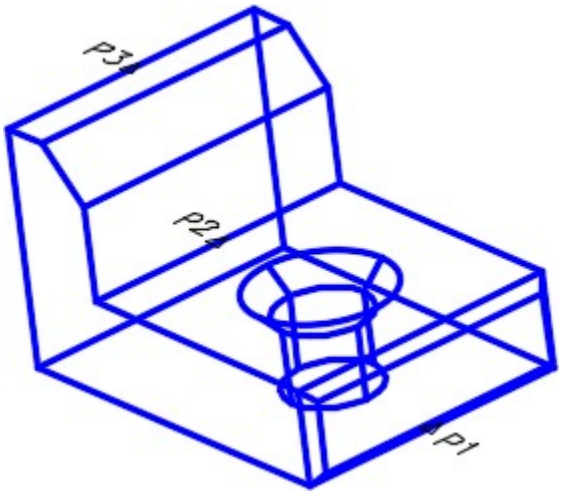


Figure Step 6A

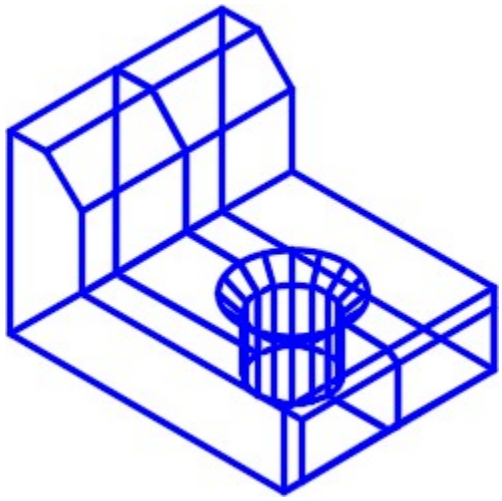


Figure Step 6B

Step 7

Change the current visual style to Realistic. (Figure Step 7)

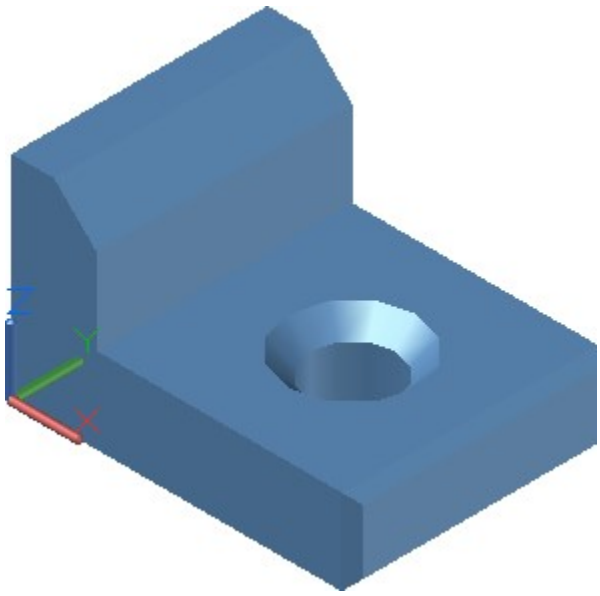


Figure Step 7

Step 8

Using the MOVE command, move the right half away from the left half. The distance you move it is not important. (Figure Step 8)

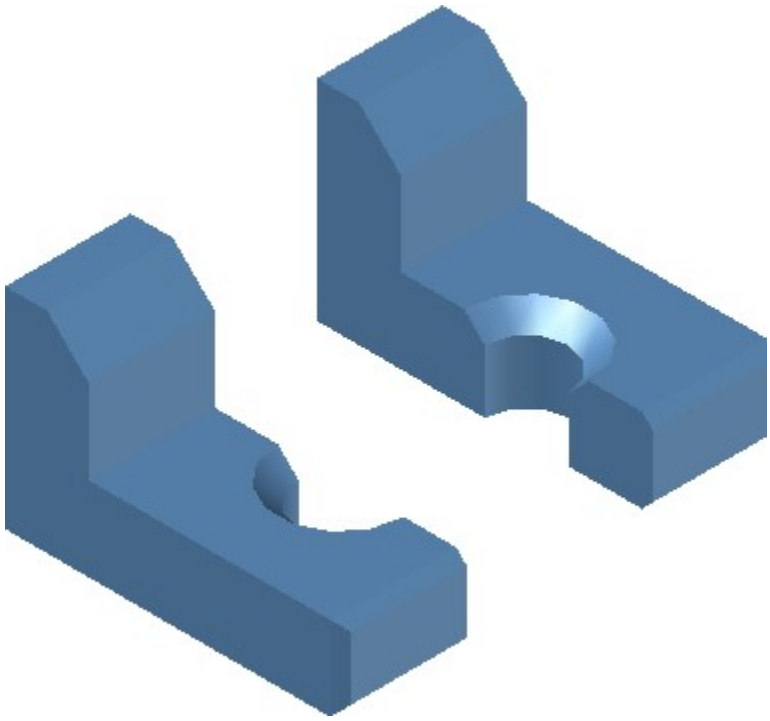


Figure Step 8

Step 9

Your model should now appear similar to the figure. (Figure Step 9)

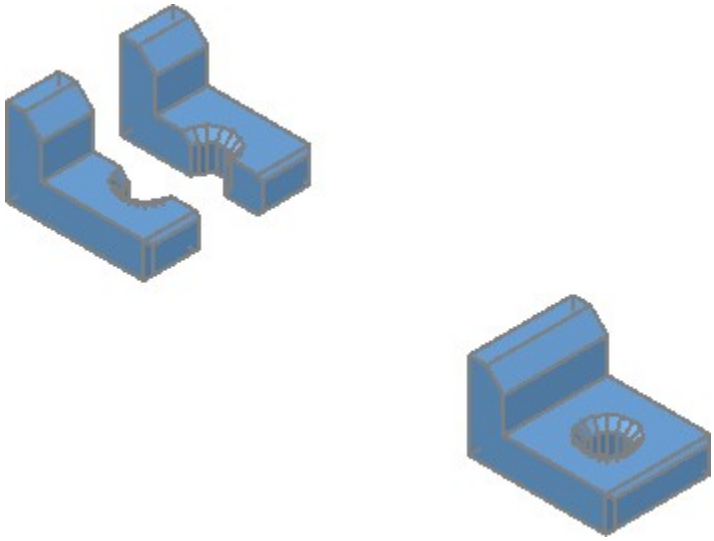


Figure Step 9

Step 10

Set the current visual style to 2D Wireframe and set layer: Hatch as the current layer. Using the copied solid, enter the SECTION command, as shown below, to create a cross section. (Figure Step 10A and 10B)

Command: **SECTION**

Select objects: 1 found

(Select the solid.)

Select objects:

Specify first point on Section plane by

[Obj/Zaxis/View/XY/YZ/ZX/3points] <3points>: *(mid)* **P4**

Specify second point on plane: *(mid)* **P5**

Specify third point on plane: *(mid)* **P6**

Command:

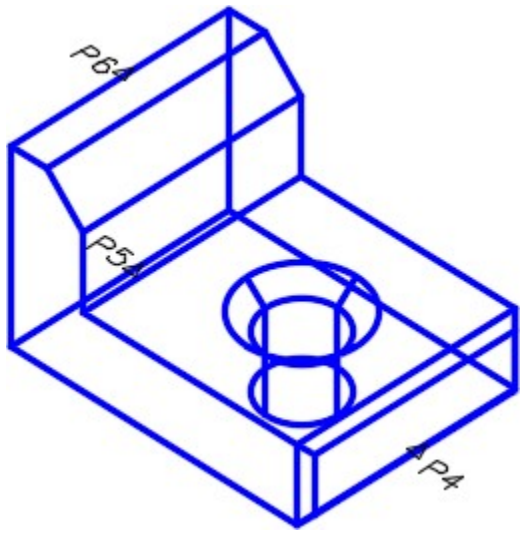


Figure Step 10A

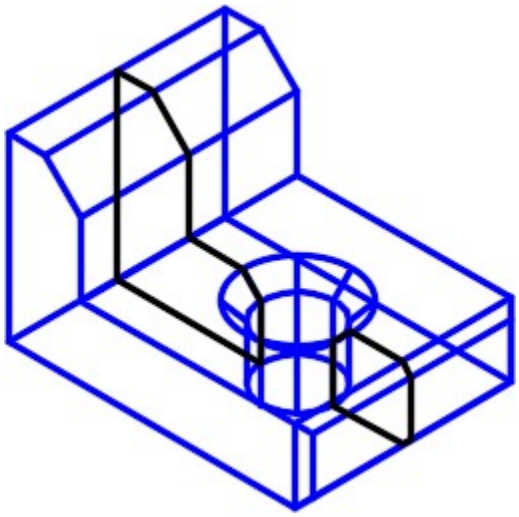


Figure Step 10B

AUTHOR'S COMMENTS: You will have to orbit the model a bit to select the correct lines.

AUTHOR'S COMMENTS: The SECTION command creates a 2D region.

Step 11

Using the MOVE command move the region, that you just created, outside of the solid. The distance you move it is not important. (Figure Step 11)

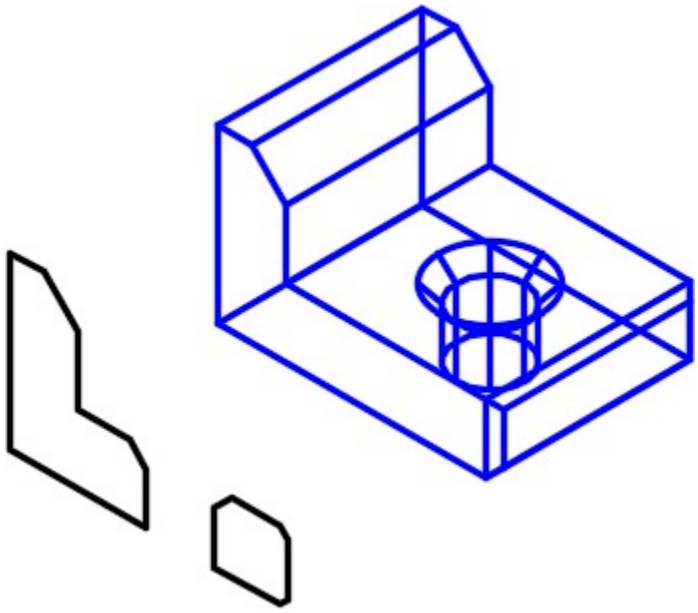


Figure Step 11

Step 12

Explode the region to convert it to lines. On layer: Hatch, add the lines to complete the cross section view as shown the figure. (Figure Step 12)

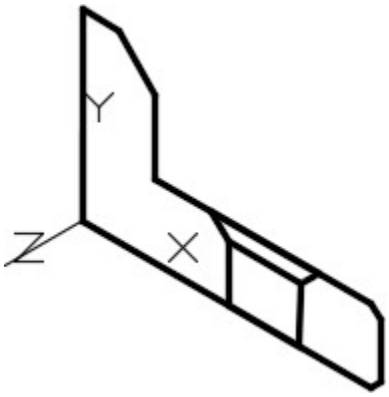


Figure Step 12

Step 13

Set the current UCS to Front, the visual style to 2D Wireframe and set layer: Hatch as the current layer. Using the HATCH command, insert the hatch pattern ANS131. (Figure Step 13)

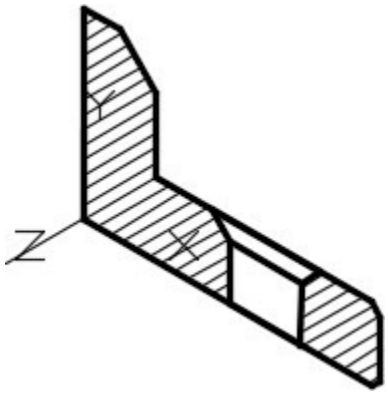


Figure Step 13

AUTHOR'S COMMENTS: Since HATCH is a 2D command, you must have the UCS located on the Front and the visual style set to 2D Wireframe.

Step 14

Your drawing should now appear similar to the figure. (Figure Step 14)

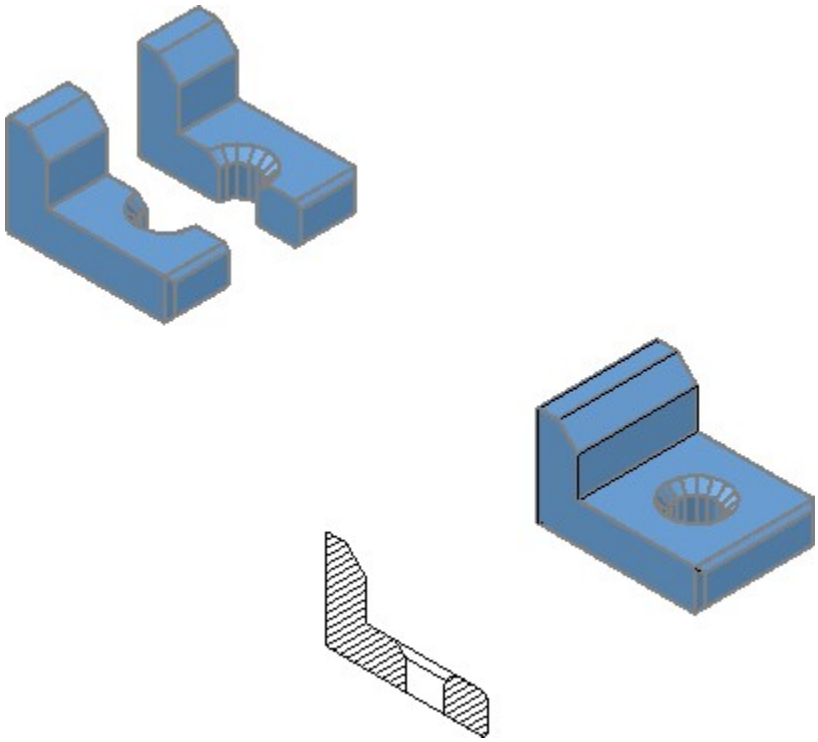


Figure Step 14

Step 15

Move the original solid back together using osnap to locate it exactly. Use the UNION command to join the two halves into one solid model. Using the Properties window, check to ensure that you only have one solid object. (Figure Step 15)

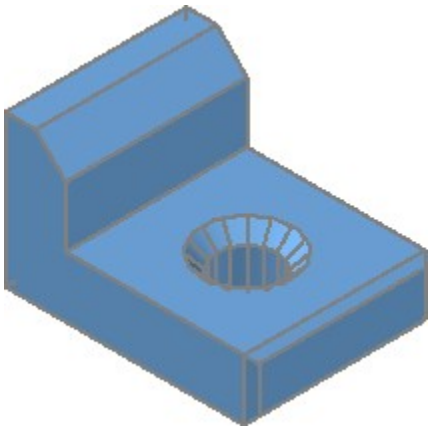


Figure Step 15

Step 16

Set the current UCS to Right and the visual style to 2D Wireframe. (Figure Step 16)

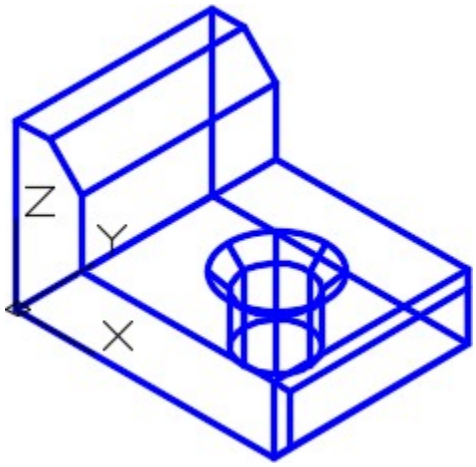


Figure Step 16

Step 17

Enter the SLICE command, as shown below, to cut the solid through the centre of the hole on the XY plane. (Figure Step 17)

Command: **SLICE**

Select objects: 1 found

(Select the solid.)

Select objects: Specify first point on slicing plane by

[Object/Zaxis/View/XY/YZ/ZX/3points] <3points>: **XY**

(The XY plane is selected to be the cutting plane. All that is required now is a point and the slice will occur. It will then slice at this point parallel to the XY plane.)

Specify a point on the XY-plane <0,0,0>: (*cen*) of **P7**

(The centre of the circle is the point selected. The solid will be sliced through centre of the circle, parallel to XY plane.)

Specify a point on desired side of the plane or [keep Both sides]: **P8**

(Only the left side is kept.)

Command:

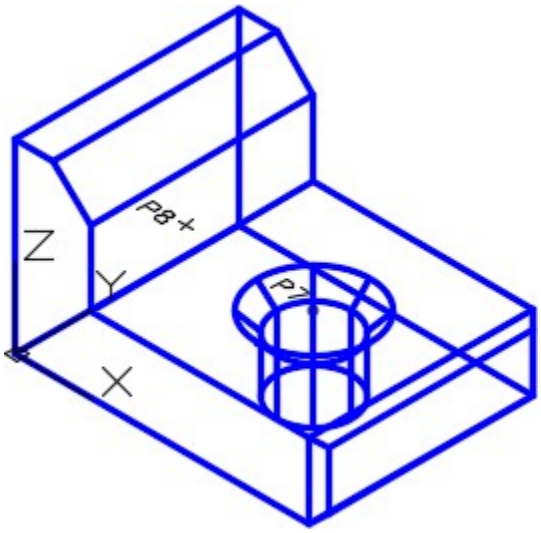


Figure Step 17

Step 18

Set the current UCS to World and the current visual style to Realistic. (Figure Step 18)

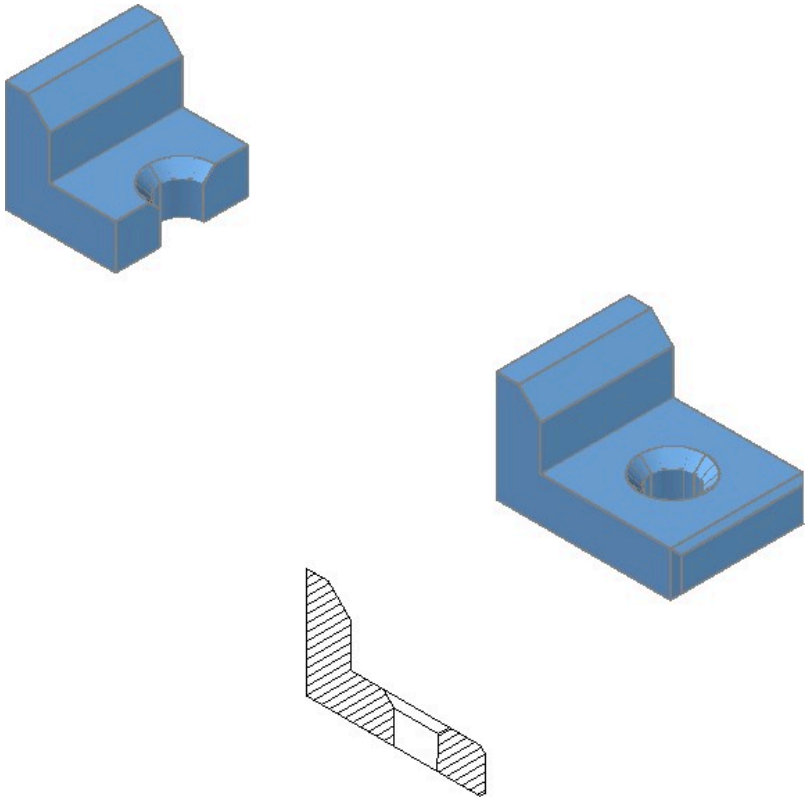


Figure Step 18

Step 19

Save and close the drawing.

Key Principles

Key Principles in Module 20

1. The SLICE command cuts a solid model into two individual solids.
2. The SECTION command creates a two dimensional region along a selected cutting plane. The region can then be relocated to produce a cross section view in 2D and used as a detail drawing.

Lab Exercise 20-1

Time allowed: 60 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 20-1	3D Layout Metric	Millimeters

Step 1

Draw the closed plines on layer: Pline.

Step 2

On layer: Solid 3, draw a solid model of the object shown in the figures. (Figure Step 2A and 2B)

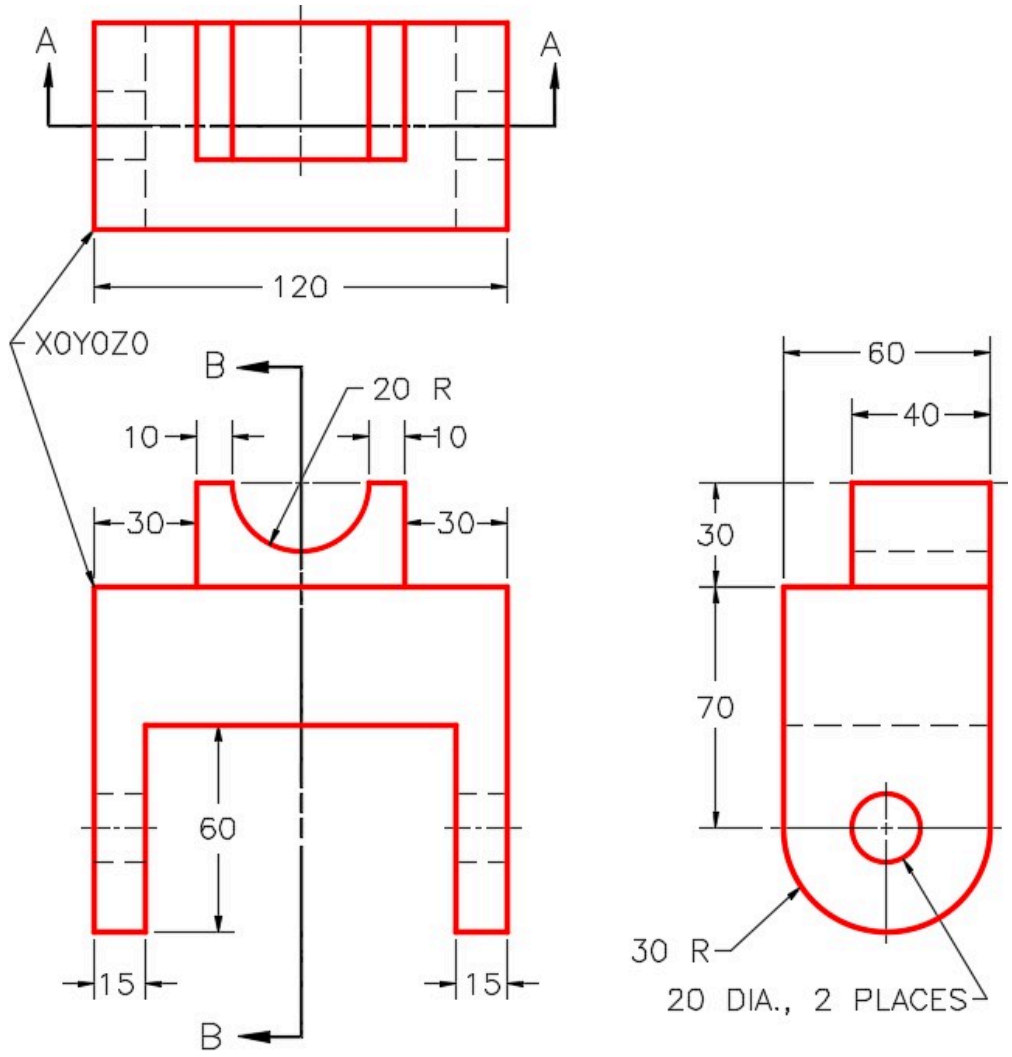


Figure Step 2A
Dimensioned Multiview Drawing

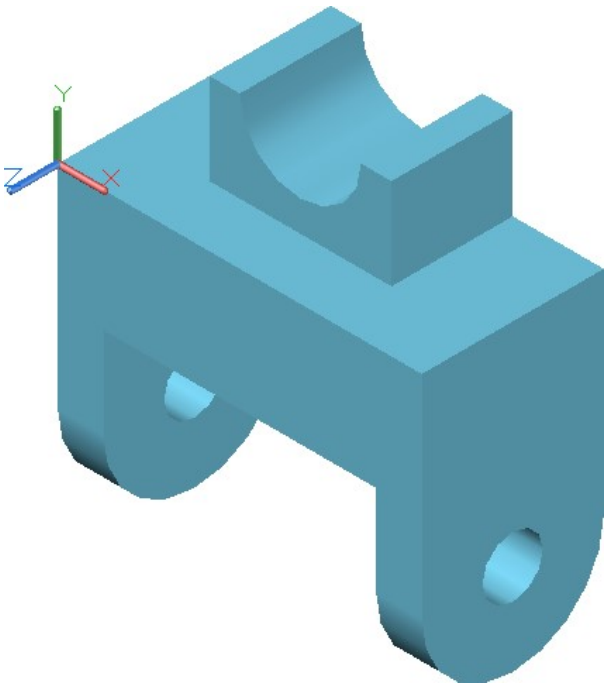


Figure Step 2B Completed Solid Model SE Isometric View

Step 3

Set the Insertion Units, change the current UCS to World and check the model with the key.

Step 4

On layer: Hatch, using the SECTION command, create a region at the cutting plane A-A. Crosshatch it with ANSI31. (Figure Step 4)

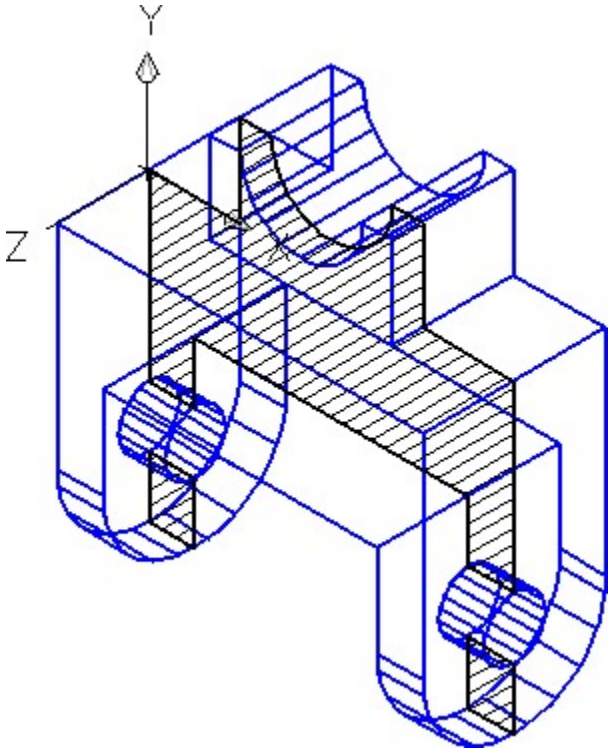


Figure Step 4

Step 5

Set the current UCS to World, make a copy of the solid model 180 millimeters apart in the X direction. Do not copy the region with the solid.

Step 6

Slice the model into two solids at cutting plane B-B. Move the two solids 30 millimeters apart. (Figure Step 6)

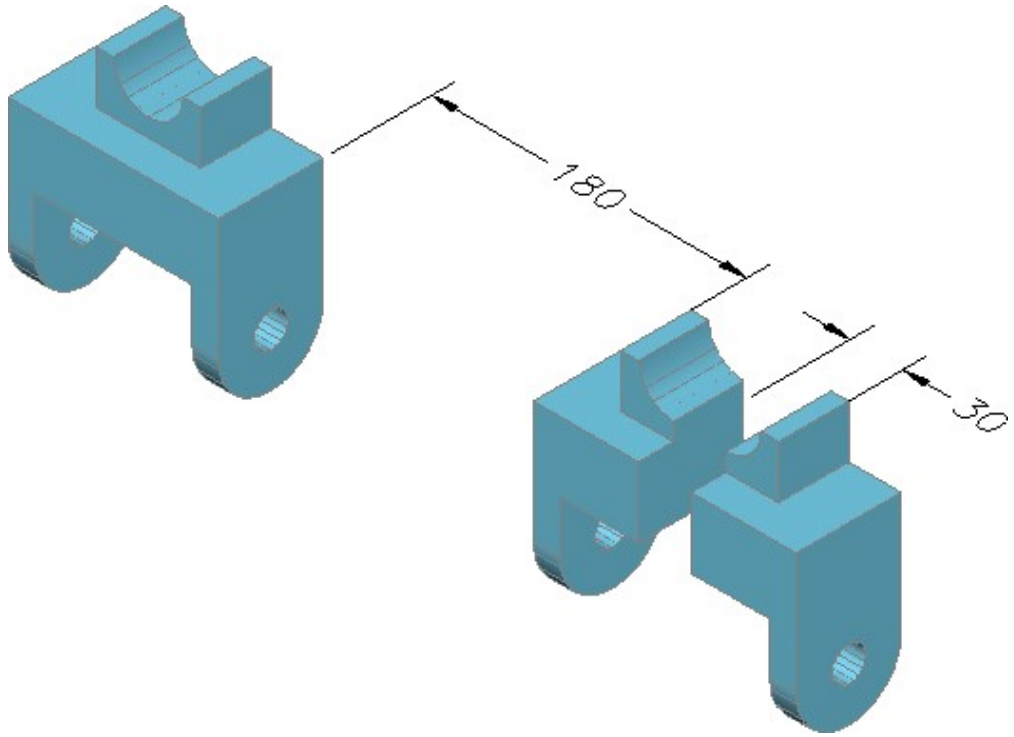


Figure Step 6

Step 7

Save and close the drawing.

Lab Exercise 20-2

Time allowed: 60 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 20-2	3D Layout Metric	Millimeters

Step 1

Draw the closed plines on layer: Pline.

Step 2

On layer: Solid 2, draw a solid model of the object shown in the figures. (Figure Step 2A and 2B)

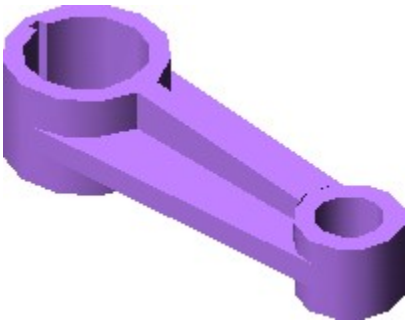


Figure Step 2A
Completed Solid Model

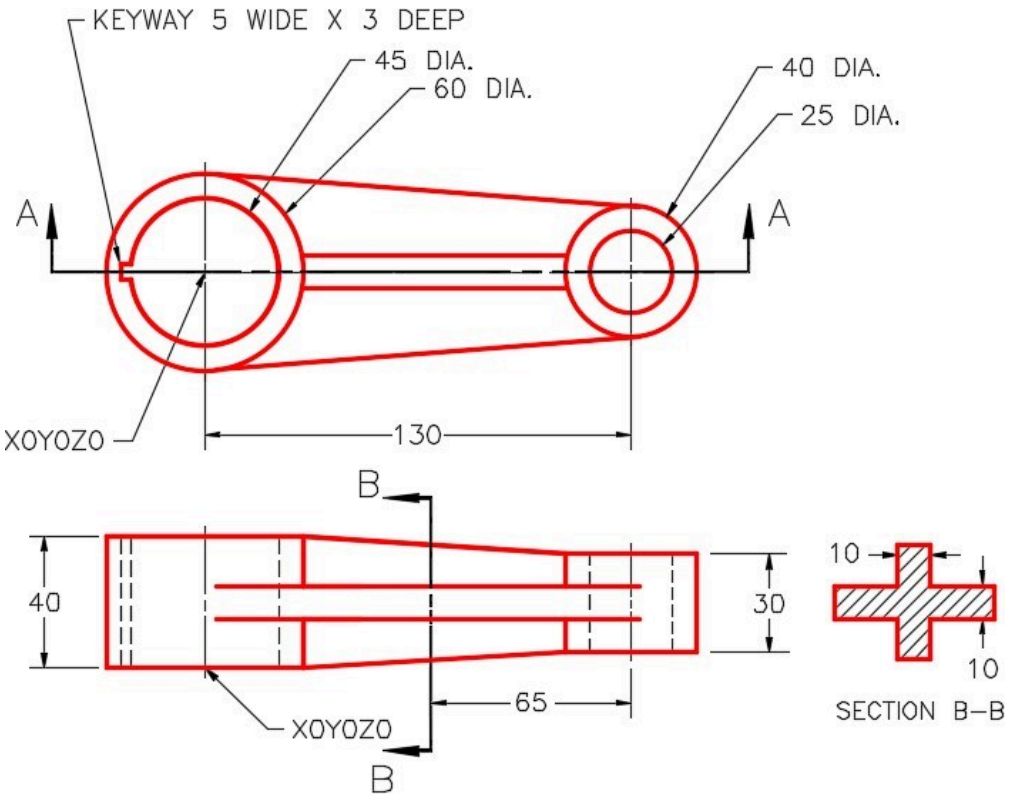


Figure Step 2B
Dimensioned Multiview Drawing

Step 3

Set the Insertion Units, change the current UCS to World and check the model with the key.

Step 4

On layer: Hatch, create a 2D region section at the cutting plane B-B. Move the region away from the model and hatch it with ANS131. (Figure Step 4)

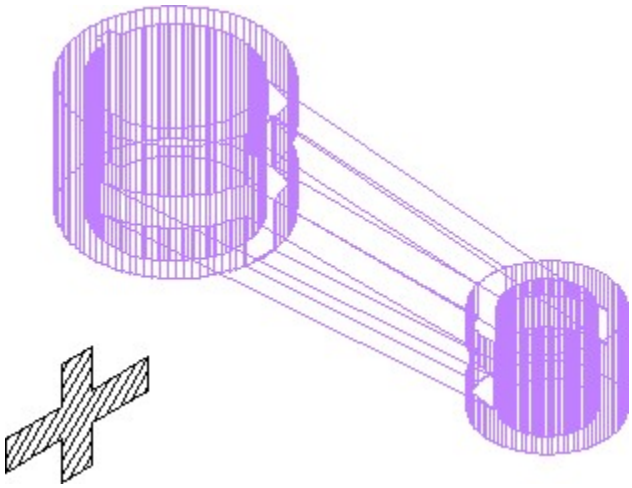


Figure Step 4

Step 5

Set the current UCS to World and make a copy of the solid model 400 millimeters from centre to centre in the X direction.

Step 6

Slice the model into two solids at cutting plane A-A. Keep only the back half of the slice. (Figure Step 6)

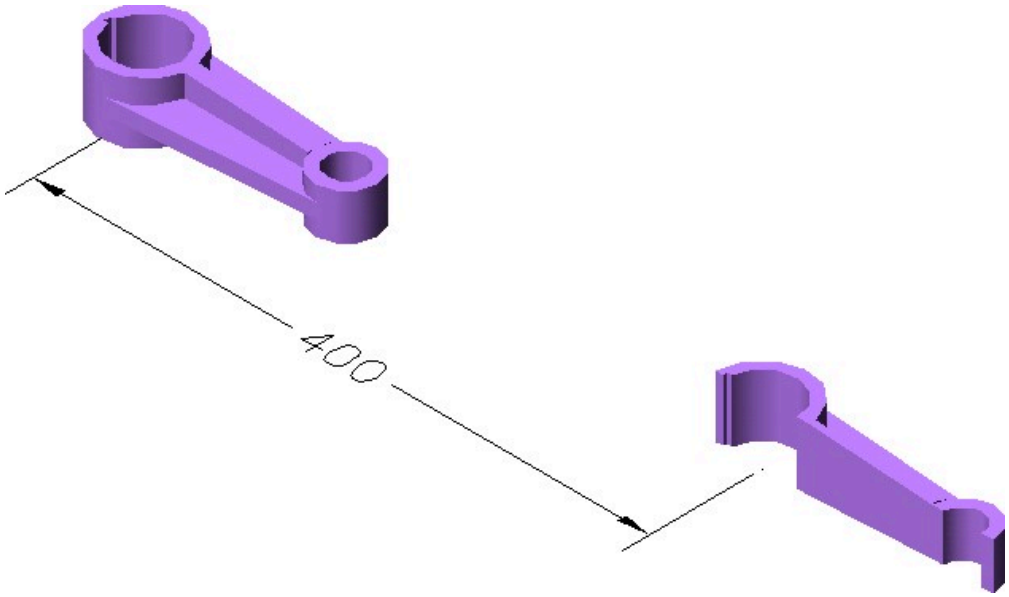


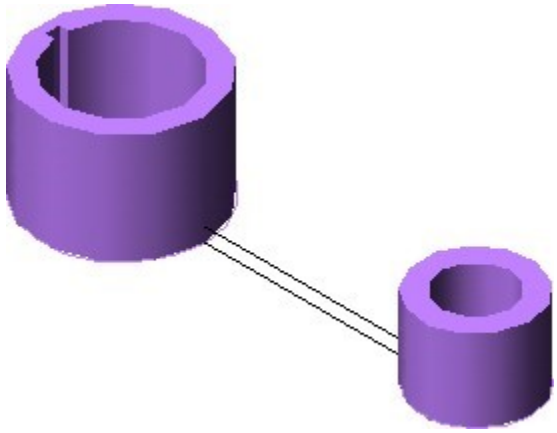
Figure Step 6

Step 7

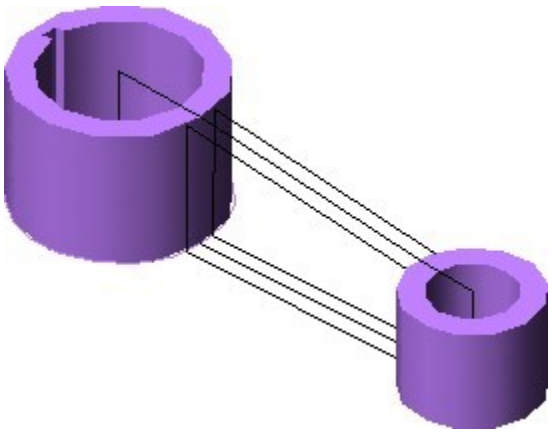
Save and close the drawing.

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

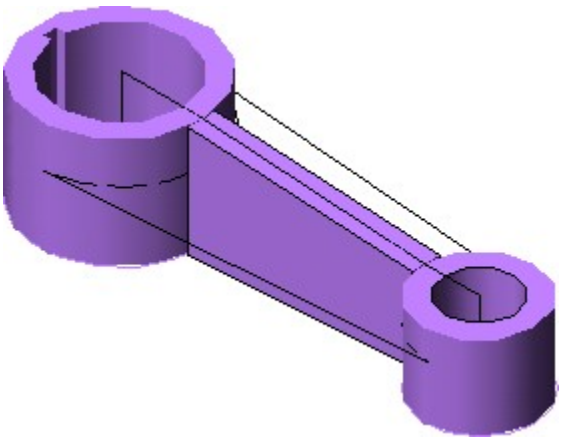
Hint 1



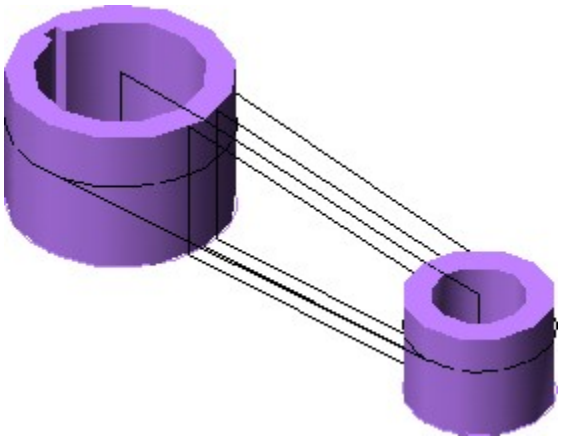
Step 1



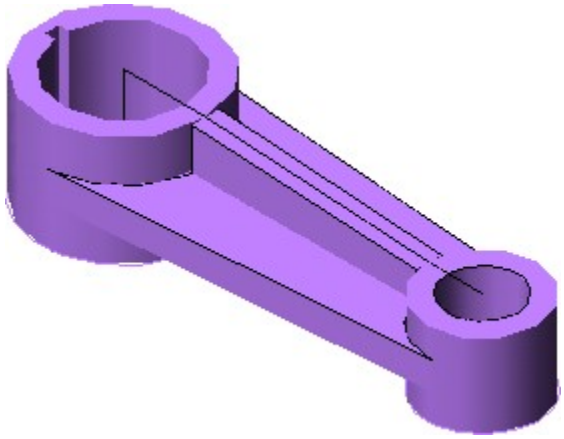
Step 2



Step 3



Step 4



Step 5

Module 21 Competency Test No.4 Open Book

Learning Outcomes

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

1. Within a four hour time limit, complete a written exam and the lab exercises without the aid of a key.

The AutoCAD 3D 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 best describes the INTERSECT command?
    A. It create a solid by finding the volume shared by two or more solids.
    B. It create a solid by subtracting one solid from another.
    C. It create a solid by joining two or more solids together.
    D. It create a solid by subtracting the largest solid from the smallest one.
    E. It create a solid by subtracting the smallest solid from the largest one.
2. What command joins two or more solids into one solid object?
    A. SUBTRACT

B. EXTRUDE

C. UNION

D. JOIN

E. INTERSECT

3. Which one of the following primitives cannot be created using the REVOLVE command?

A. Cone

B. Cylinder

C. Sphere

D. Torus

E. Box

4. What command creates a solid by revolving a 2D shape around an axis?

A. SUBTRACT

B. EXTRUDE

C. REVOLVE

D. JOIN

E. INTERSECT

5. What system variable is used to set the number of contour lines that will be used when a curved solid is constructed?

A. SOLIDLINES

B. SURTAB1

C. LINEISO

D. ISOLINES

E. SUFTAB2

6. Which direction, on the current UCS, does the EXTRUDE command project the closed object to create a solid model?

A. X

B. Y

C. Either X or Y

D. Z

E. Either X, Y or Z

7. Which one of the following statements is false?

A. A surfaced model is hollow.

B. The mass properties of a solid model can be obtained.

C. A solid model Is the best computerized representation of an object.

D. A solid model is solid.

E. The mass properties of a surfaced model can be obtained.

8. What type of object is created by the SECTION command?
    A. Solid
    B. Closed polyline
    C. Region
    D. Hatch
    E. Open polyline
9. Which one of the following is not a solid primitive?
    A. Wedge
    B. Sphere
    C. Rectangle
    D. Box
    E. Cylinder
10. What command removes one solid from another solid?
    A. SUBTRACT
    B. EXTRUDE
    C. REVOLVE
    D. JOIN
    E. INTERSECT

Lab Exercise 21-1A OPEN BOOK

Drawing Name	**Template**	**Units**
AutoCAD 3D Lab 21-1A	3D Layout English	Inches

Step 1

Draw all plines on layer: Pline.

Step 2

On layer: Solid Base, create a solid model of the Base. (Figure Step 2A and 2B)

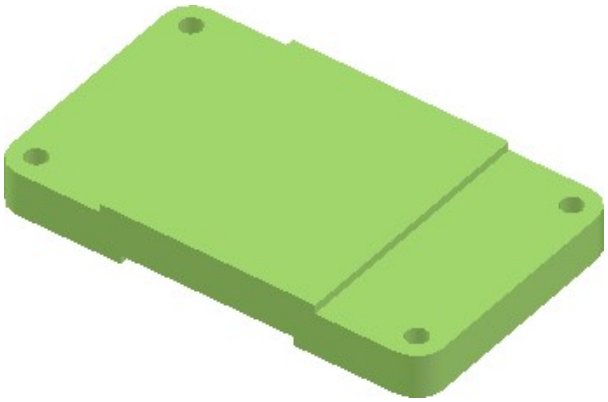


Figure Step 2A
Solid Model of the Base

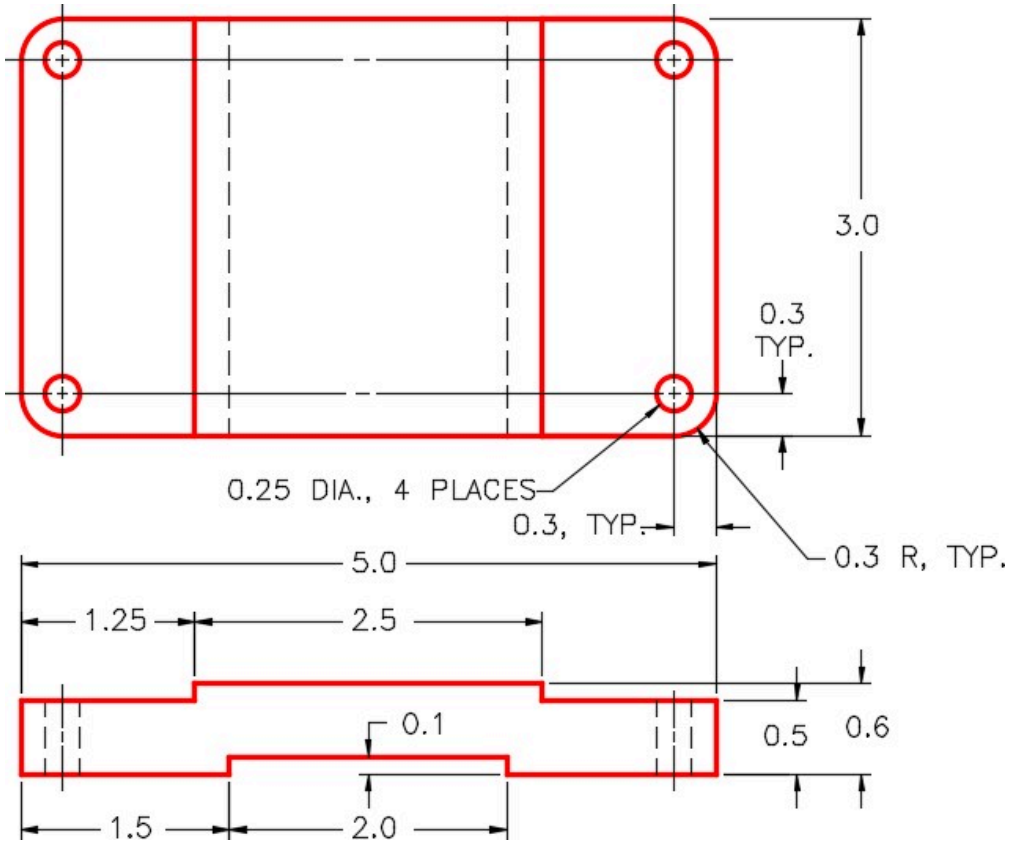


Figure Step 1
Dimensioned Multiview Drawing of the Base

Step 3

Create a block of the model. Name it:

AutoCAD 3D Lab 21-1A. Pick an appropriate location for 0,0,0. Set 0,0,0 as the insert point for the block. Make 0,0,0 as the insert point for the block. Ensure that the current UCS is located at the World when creating the block.

Lab Exercise 21-1B OPEN BOOK

Drawing Name	Template	Units
AutoCAD 3D Lab 21-1B	3D Layout English	Inches

Step 1

Draw all plines on layer: Pline.

Step 2

On layer: Construction, add any necessary construction objects that will help insert the model into the assembly.

Step 3

On layer: Solid Support, create a solid model of the Support. (Figure Step 3A and 3B)

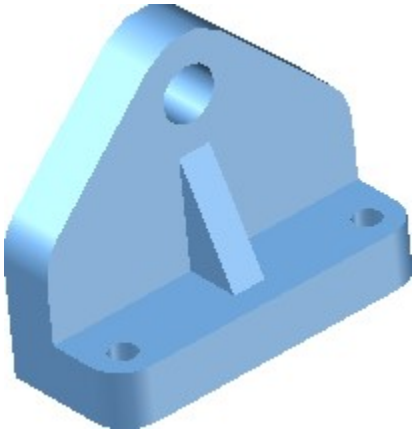


Figure Step 3A
Solid Model of the Support

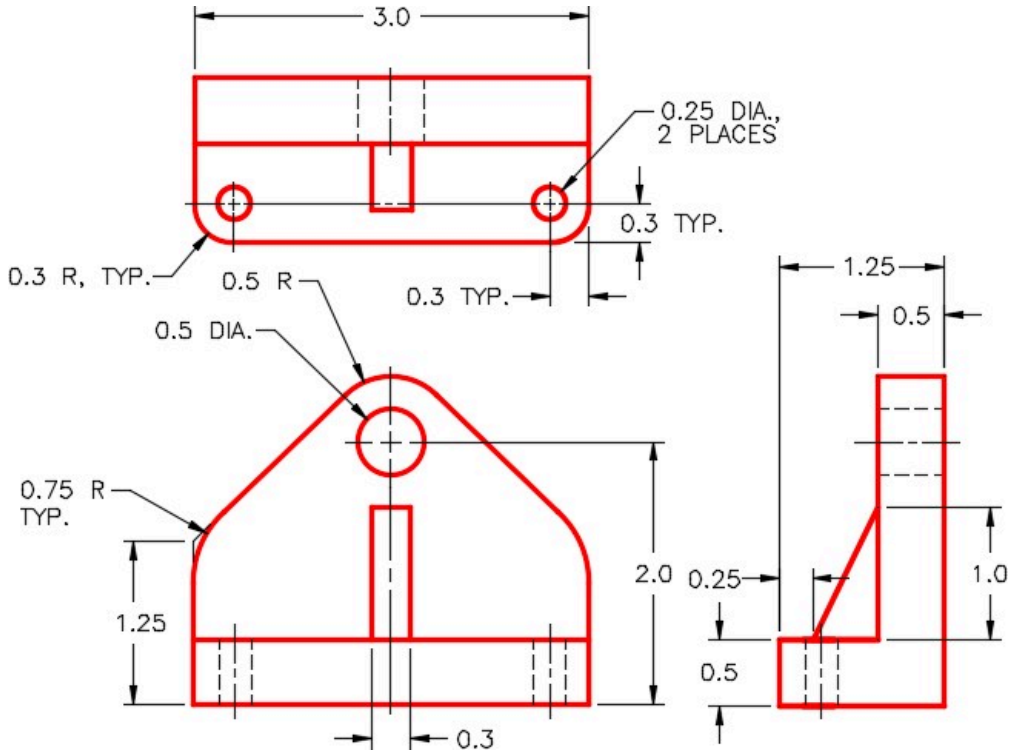


Figure Step 3B
Dimensioned Multiview Drawing of the Support

Step 4

Create a block of the model. Name it: AutoCAD 3D Lab 21-1B. Pick an appropriate location for 0,0,0. Set 0,0,0 as the insert point for the block. Ensure that the current UCS is located at the World when creating the block.

Lab Exercise 21-1C OPEN BOOK

Drawing Name	Template	Units
AutoCAD 3D Lab 21-1C	3D Layout English	Inches

Step 1

Draw all plines on layer: Pline.

Step 2

On layer: Construction, add any necessary construction objects that will help insert the model into the assembly.

Step 3

On layer: Solid Axle, create a solid model of the Axle. (Figure Step 3A and 3B)

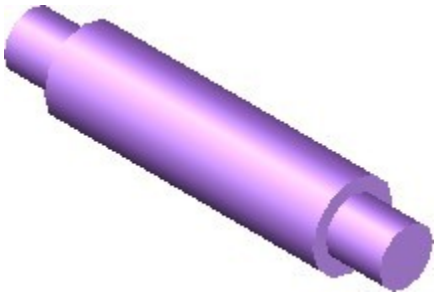


Figure Step 3A
Solid Model of the Axle

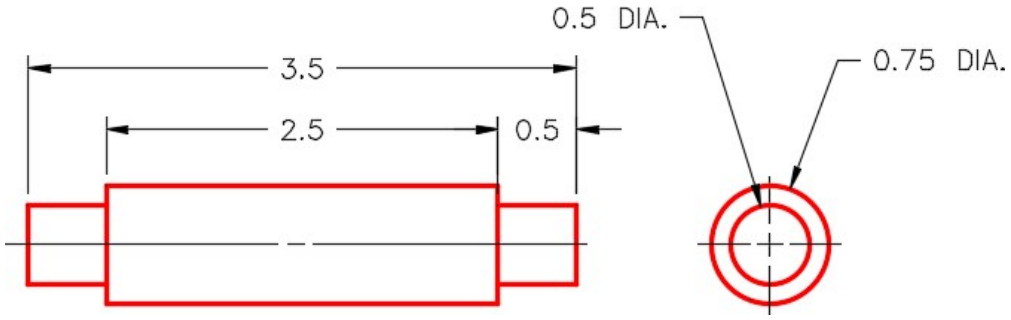


Figure Step 3B
Dimensioned Multiview Drawing of the Axle

Step 4

Create a block of the model. Name it: AutoCAD 3D Lab 21-1C. Pick an appropriate location for 0,0,0. Set 0,0,0 as the insert point for the block. Ensure that the current UCS is located at the World when creating the block.

Lab Exercise 21-1D OPEN BOOK

Drawing Name	Template	Units
AutoCAD 3D Lab 21-1D	3D Layout English	Inches

Step 1

Draw all plines on layer: Pline.

Step 2

On layer: Construction, add any necessary construction objects that will help insert the model into the assembly.

Step 3

On layer: Solid Wheel, create a solid model of the Wheel. (Figure Step 3A and 3B)

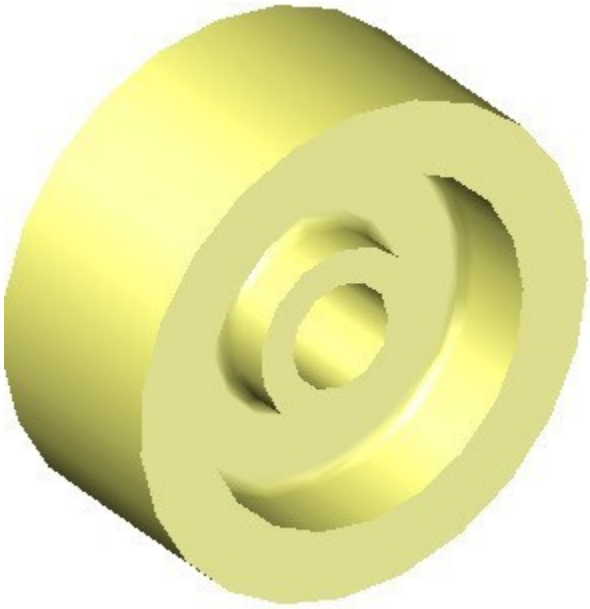


Figure Step 3A
Solid Model of the Wheel

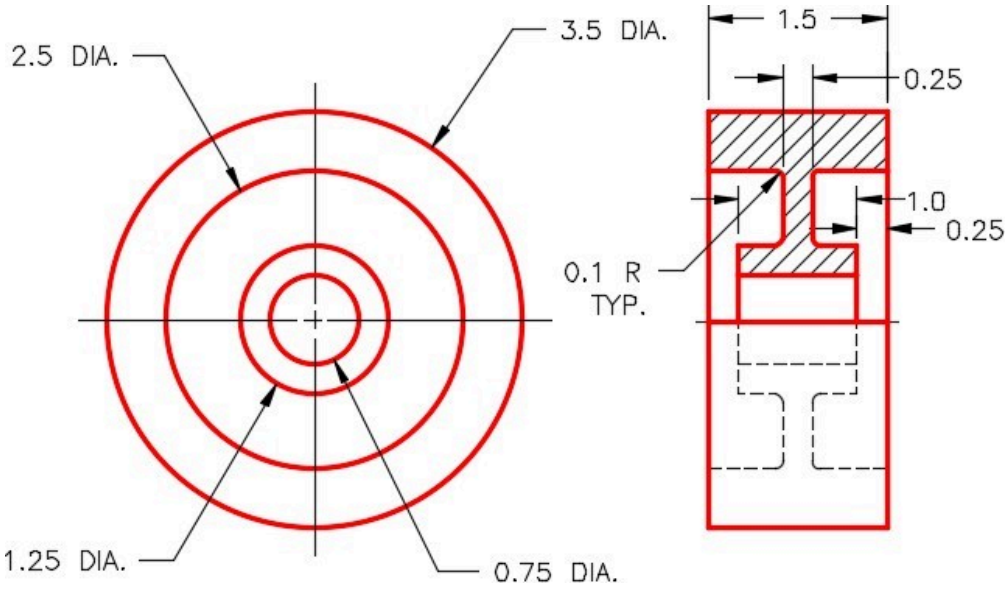


Figure Step 3B
Dimensioned Multiview Drawing of the Wheel

Step 4

Create a block of the model. Name it: AutoCAD 3D Lab 21-1D. Pick an appropriate location for 0,0,0. Make 0,0,0 as the insert point for the block. Ensure that the current UCS is located at the World when creating the block.

Lab Exercise 21-1E OPEN BOOK

Drawing Name	Template	Units
AutoCAD 3D Lab 21-1E	3D Layout English	Inches

Step 1

Draw all plines on layer: Pline.

Step 2

On layer Construction, add any necessary construction objects that will help insert the model into the assembly.

Step 3

On layer: Solid Washer, create a solid model of the Washer. (Figure Step 3A and 3B)

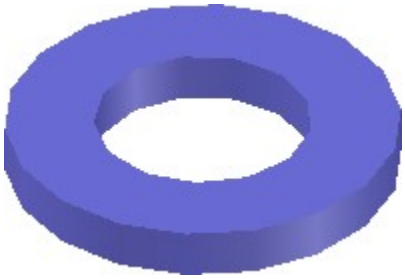


Figure Step 3A Solid Model of the Washer

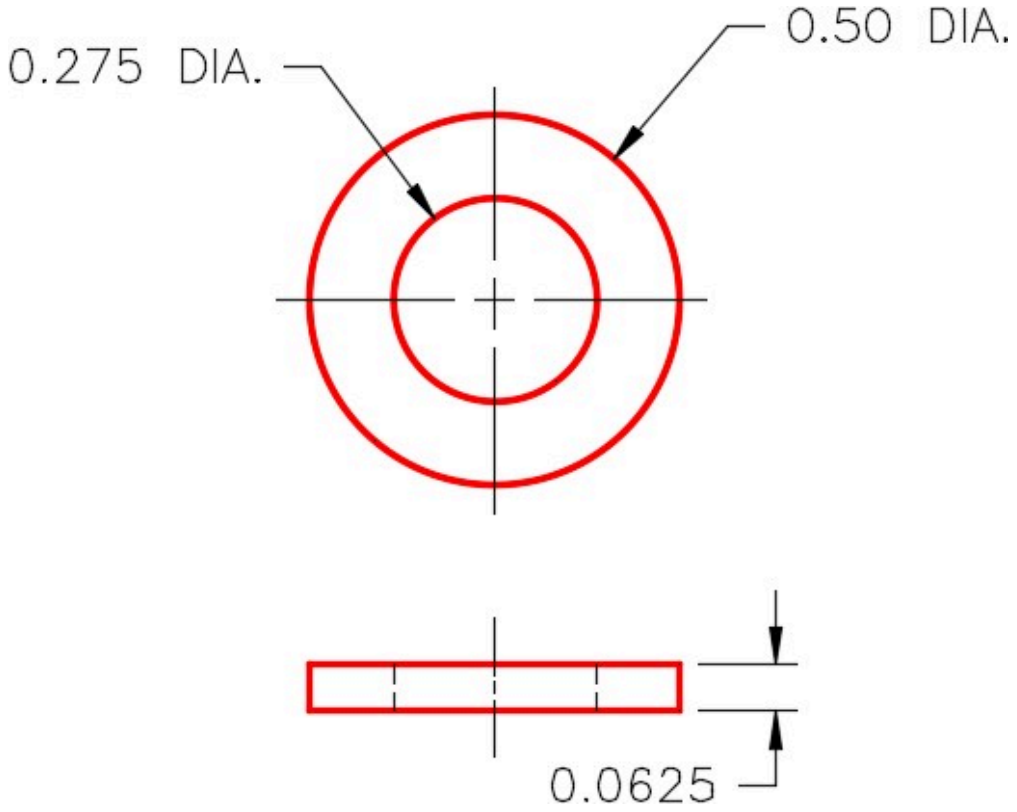


Figure Step 3B
Dimensioned Multiview Drawing of the Washer

Step 4

Create a block of the model. Name it: AutoCAD 3D Lab 21-1E. Pick an appropriate location for 0,0,0. Set 0,0,0 as the insert point for your block. Ensure that the current UCS is located at the World when creating the block.

Lab Exercise 21-1F OPEN BOOK

Drawing Name	Template	Units
AutoCAD 3D Lab 21-1F	3D Layout English	Inches

Step 1

Draw all plines on layer: Pline.

Step 2

On layer Construction, add any necessary construction objects that will help insert the model into the assembly.

Step 3

On layer: Solid Bolt, create a solid model of the Bolt. (Figure Step 3A and 3B)

Figure Step 3A Solid Model of the Bolt

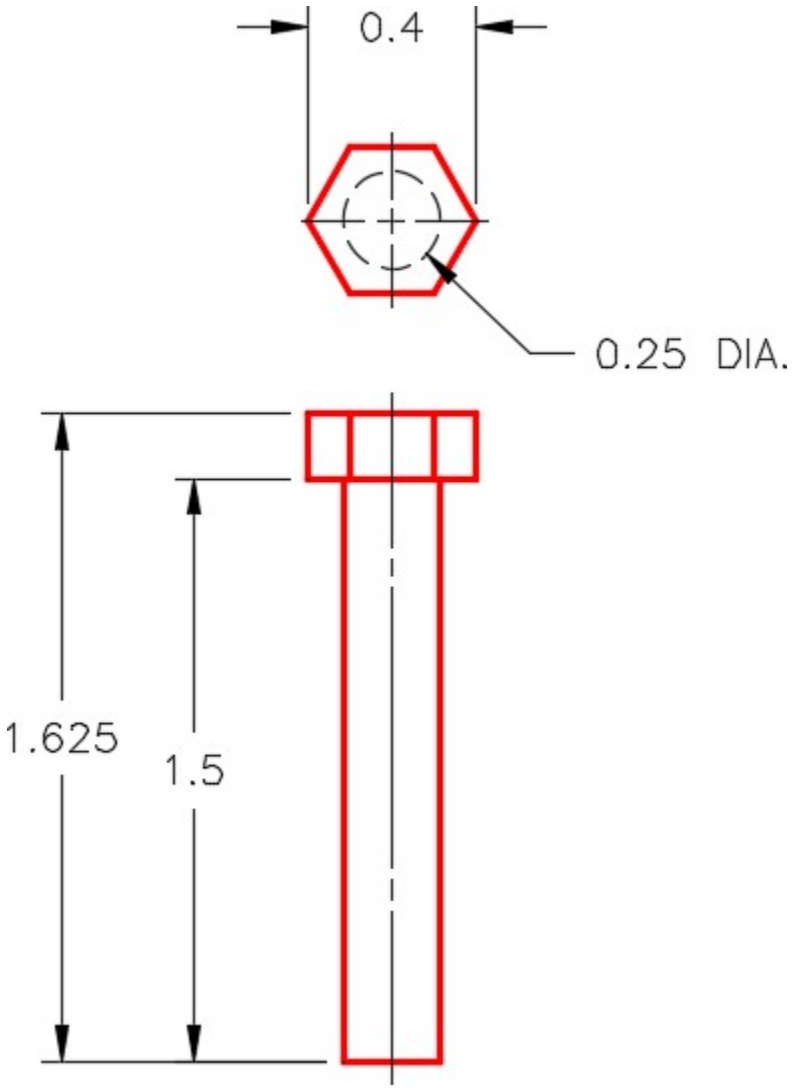


Figure Step 3B
Dimensioned Multiview Drawing of the Bolt

Step 4

Create a block of the model. Name it: AutoCAD 3D Lab 21-1F. Pick an appropriate location for 0,0,0. Set 0,0,0 as the insert point for your block. Ensure that the current UCS is located at the World when creating the block.

Lab Exercise 21-1 OPEN BOOK

Drawing Name	Template	Units
AutoCAD 3D Lab 21-1	3D Layout English	Inches

Step 1

Start a new drawing and save it with the name: AutoCAD 3D Lab 21-1.

Step 2

Open the six drawings that you created earlier in this lab exercise.

Step 3

Set the current UCS to World and using DesignCenter insert the following blocks into the drawing.

1- AutoCAD 3D Lab 21-1A

2- AutoCAD 3D Lab 21-1B

1- AutoCAD 3D Lab 21-1C

1- AutoCAD 3D Lab 21-1D

4- AutoCAD 3D Lab 21-1E

4- AutoCAD 3D Lab 21-1F

Step 4

Close all drawings except for: AutoCAD 3D Lab 21-1

Step 5

Ensure that layers: Pline and Construction are on and all solid layers are off. Move the blocks into their exact location to create the assembly using the model or construction objects locate them accurately.

Step 6

Turn on all solid layers and turn off the layers: Pline and Construction.

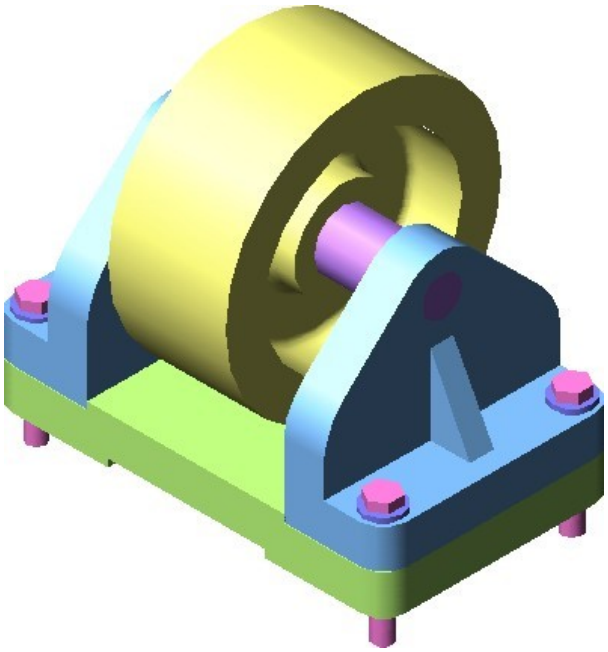


Figure Step 6

Step 7

With UCS in World, copy the complete Caster Assembly 10 inches in the X direction. Rotate the copy of the assembly 180 degrees around the X axis into the position it would be installed. (Figure Step 7)

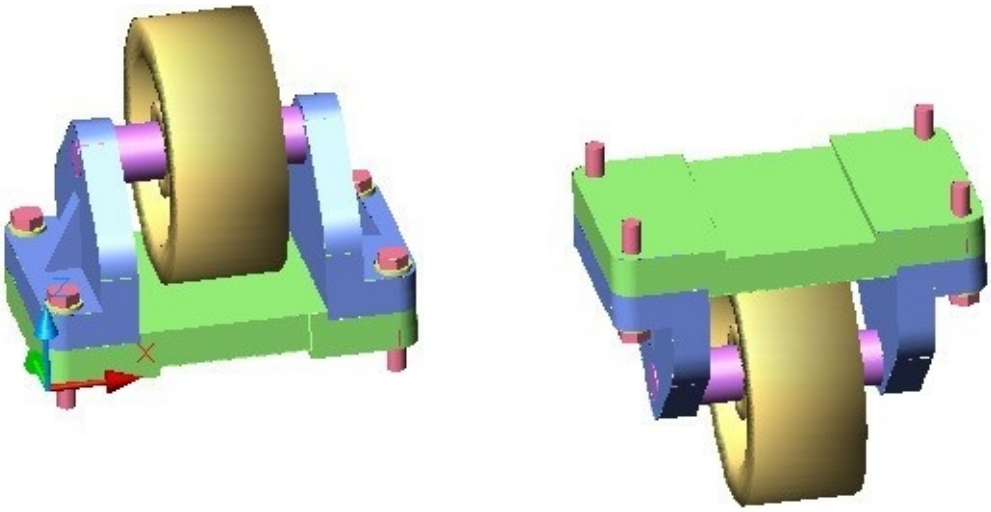


Figure Step 7

Part 6

Module 27 Editing Solid Models – Part 1

Learning Outcomes

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

1. Describe how solid models are edited including the terms, body, face and edge.
2. Apply the SOLIDEDIT command to edit faces of solid models.

Editing Solid Models

The SOLIDEDIT command is used to edit solid models. This powerful command has many options and is used to do most of the editing of existing solid models. When editing solids, you must be able to identify and select the body, faces and edges of the solid model.

Body

A *body* is the solid model itself. See Figure 27-1.

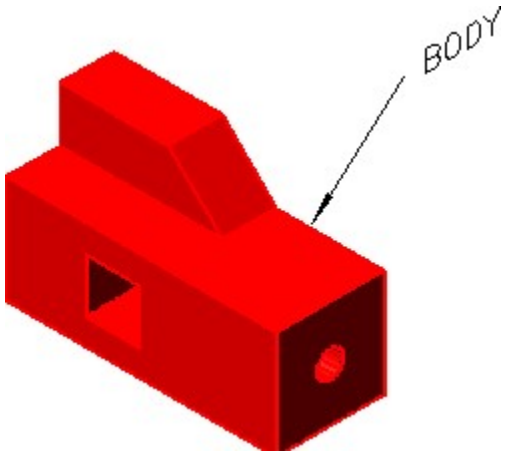


Figure 27-1
Body

Face

A *face* is the planer or curved 3D surfaces of the solid model. See Figure 27-2.

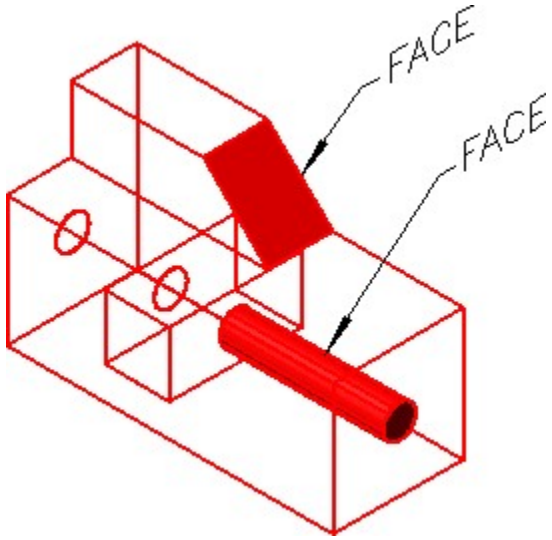


Figure 27-2
Faces

Edge

An *edge* is the shared object between two faces or surfaces of the solid. See Figure 27-3. Editing edges will be covered in Module 23.

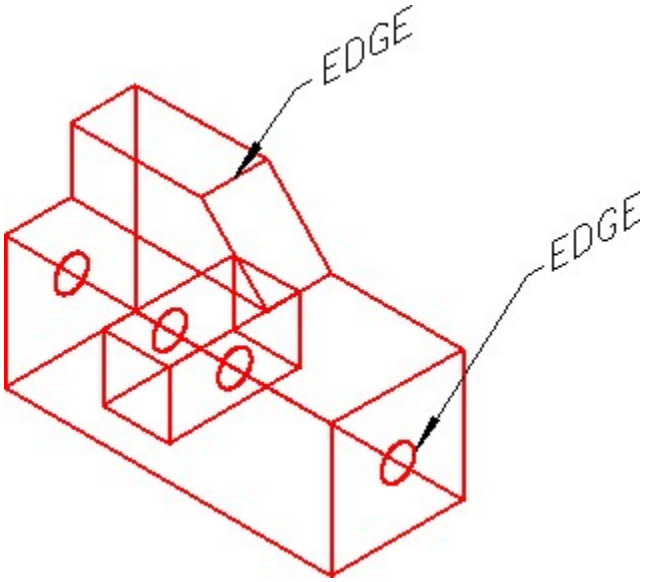


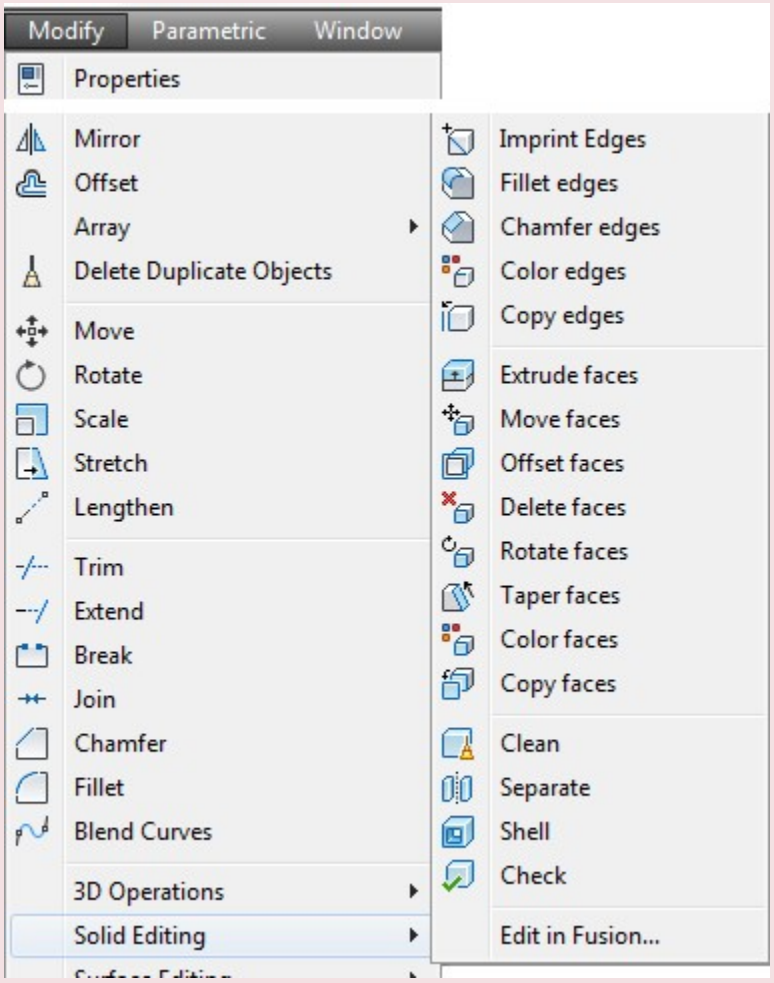
Figure 27-3
Edges

AutoCAD Command: SOLIDEDIT

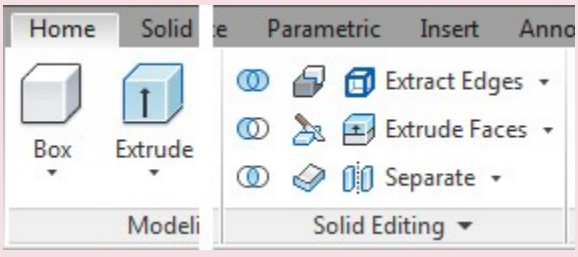
The SOLIDEDIT command is used to edit solid objects.

Shortcut: none

Solid Editing Toolbar



Modify Pull-down



Home Ribbon

AUTHOR'S COMMENTS: One of the trickiest parts of using the SOLIDEDIT command is selecting the solid faces or edges that you want to edit. This command works differently than other commands when selecting objects. In all other commands, when the selected object is picked, at a ' select object ' prompt, only one object would be selected. In the SOLIDEDIT command, when an object is selected by picking it, more than one object is usually selected with one pick. The selected faces will highlight and then you have to remove the faces you do not want to edit. i.e.

Command: **SOLIDEDIT**

Solids editing automatic checking: SOLIDCHECK=1

Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: **_F**

Enter a face editing option

[Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/eXit] <eXit>: **E**

Select faces or [Undo/Remove]: **P1** 2 faces found.

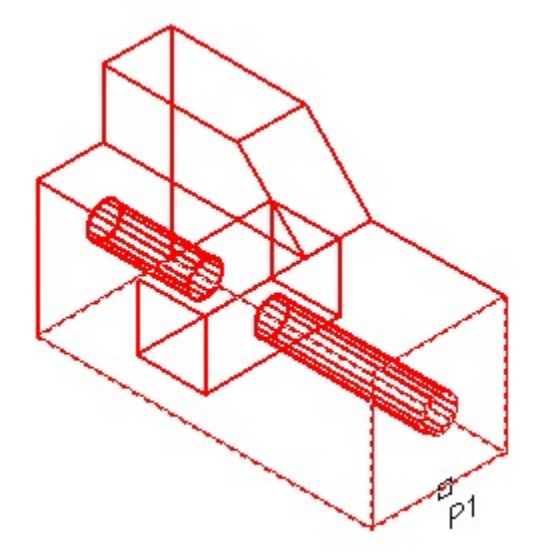
(In this example, 2 faces are selected with one pick.)

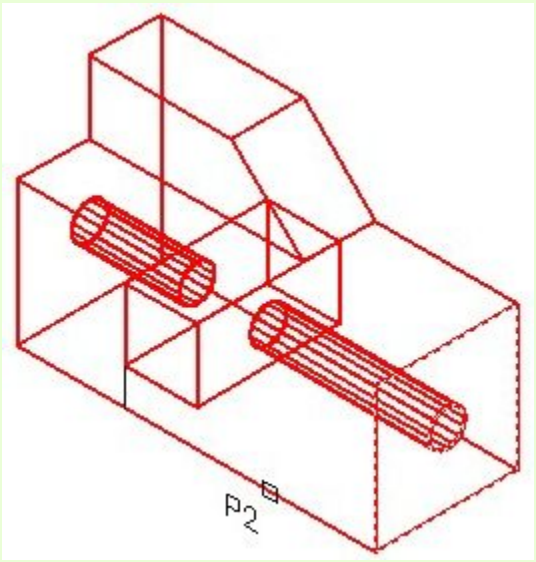
Select faces or [Undo/Remove/ALL]: **R**

Remove faces or [Undo/Add/ALL]: **P2** 2 faces found, 1 removed.

(One face is removed from the selection.)

Remove faces or [Undo/Add/ALL]:





WORK ALONG: Using the SOLIDEDIT Command

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save the drawing and name it: AutoCAD 3D Workalong 27-1.

Step 3

Draw the plines on layer: Pline and the solid model on layer: Solid 6.

Step 4

Draw a solid model of the object shown in the figures. (Figure Step 4A and 4B)

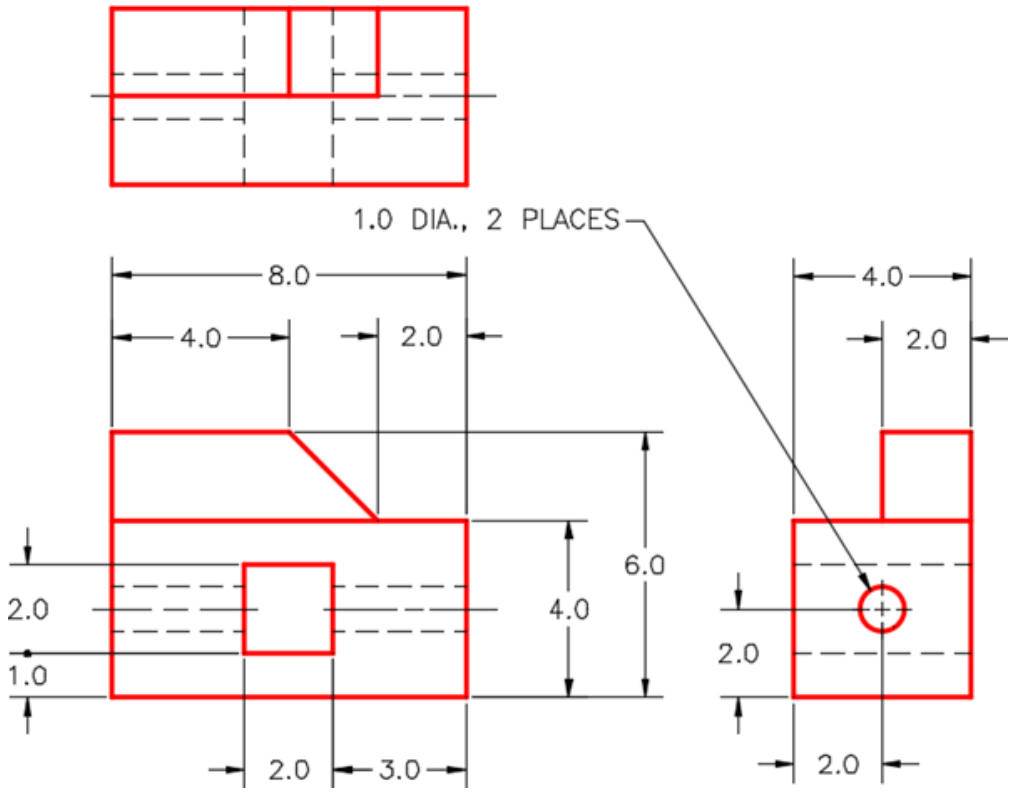


Figure Step 4A
Dimensioned Multiview Drawing

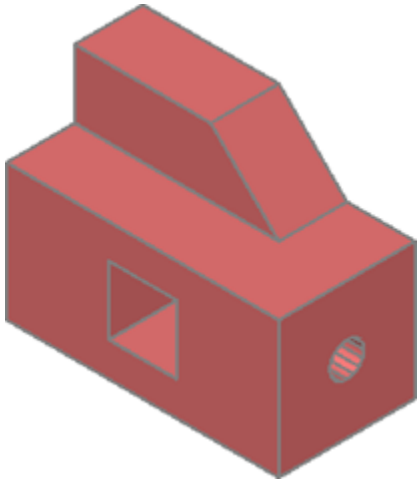
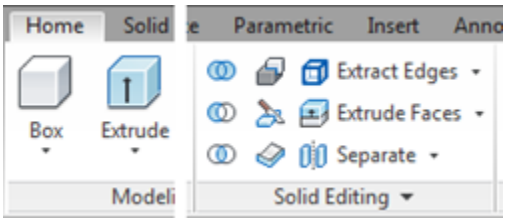


Figure Step 4B
Solid Model
SE Isometric View

Step 5

If you are using toolbars, enable the display of the Solids Editing toolbar. If you are using ribbons, use the Home tab. (Figure Step 5)

Solid Editing Toolbar



Home Ribbon Figure Step 5

AUTHOR'S COMMENTS: When editing solids, it is easier to use the Solids Editing toolbar instead of entering the commands on the keyboard.

Step 6

Set the view to SE Isometric, the current visual style to 3D Wireframe and the UCS in World. (Figure Step 6)

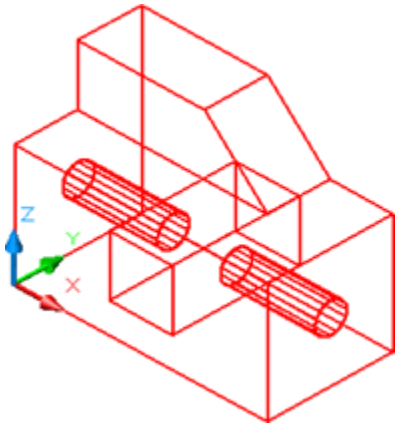


Figure Step 6

AUTHOR'S COMMENTS: The Extrude Faces option of the SOLIDEDIT command allows you to extrude any face of a solid the same way you would extrude a 2D object with the EXTRUDE command. The solid face can be extruded to modify the existing solid either making it larger or smaller.

Step 7

Click the Extrude Faces icon and, as shown below, to extrude the face on the right. (Figure Step 7A, 7B, and 7C)

Command: **SOLIDEDIT**

Solids editing automatic checking: SOLIDCHECK=1

Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: **F**

Enter a face editing option

[Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/eXit] <eXit>: **E**

Select faces or [Undo/Remove]: **P1** 2 faces found.

(Select the face. Notice it finds 2 faces.)

Select faces or [Undo/Remove/ALL]: **R**

(Remove one of the faces.)

Remove faces or [Undo/Add/ALL]: **P2** 2 faces found, 1 removed.

Remove faces or [Undo/Add/ALL]:

Specify height of extrusion or [Path]: **2**

(Extrude the face 2 units in the positive direction.)

Specify angle of taper for extrusion <0>:

Solid validation started.

Solid validation completed.

Command:

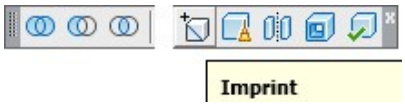


Figure Step 7A

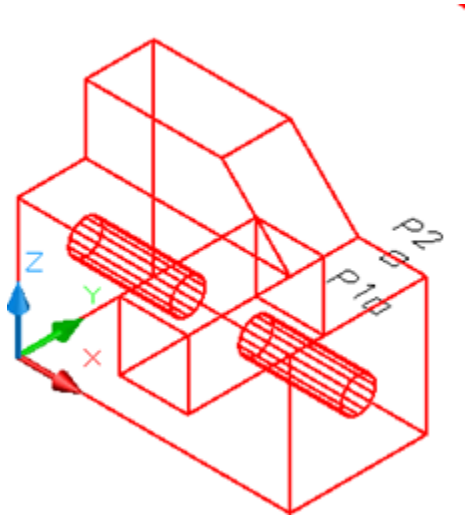


Figure Step 7B

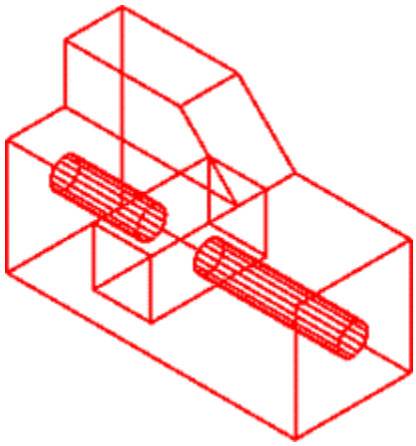


Figure Step 7C

Step 8

Click the Extrude Faces icon and complete the command, as shown below, to extrude the face to the bottom. (Figure Step 8A and 8B)

Command: **SOLIDEDIT**

Solids editing automatic checking: SOLIDCHECK=1

Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: **F**

Enter a face editing option

[Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/mAterial/Undo/eXit] <eXit>: **E**

Select faces or [Undo/Remove]: **P3** 1 face found.

Select faces or [Undo/Remove/ALL]:

Specify height of extrusion or [Path]: **-1**

Specify angle of taper for extrusion <0>:

Solid validation started.

Solid validation completed.

Enter a face editing option

[Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/mAterial/Undo/eXit] <eXit>:

Command:

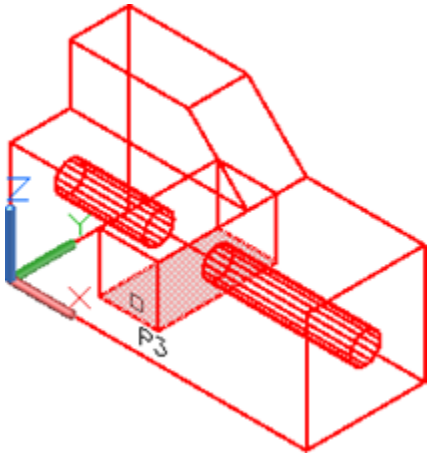


Figure Step 8A

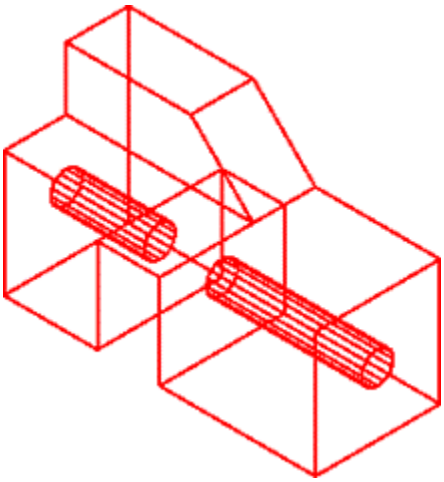


Figure Step 8B

AUTHOR'S COMMENTS: The Move Faces option of the SOLIDEDIT command allows you to move a face and have the solid take the shape of the new location of the face. The faces can be moved to make the solid larger or smaller.

Step 9

Click the Move Faces icon and complete the command shown below to move the face on the top of the model higher. (Figure Step 9A, 9B, 9C, and 9D)

Command: **SOLIDEDIT**

Solids editing automatic checking: SOLIDCHECK=1

Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: **F**

Enter a face editing option

[Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/eXit] <eXit>: **M**

Select faces or [Undo/Remove]: **P5** 2 faces found.

Select faces or [Undo/Remove/ALL]: **R**

Remove faces or [Undo/Add/ALL]: 2 faces found, 1 removed.

Remove faces or [Undo/Add/ALL]:

Specify a base point or displacement: **0,0,0**

Specify a second point of displacement: **@0,0,1**

(Move the in the positive Z or up 1 unit.)

Solid validation started. Solid validation completed.

Enter a face editing option

Command:

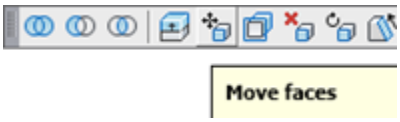


Figure Step 9A

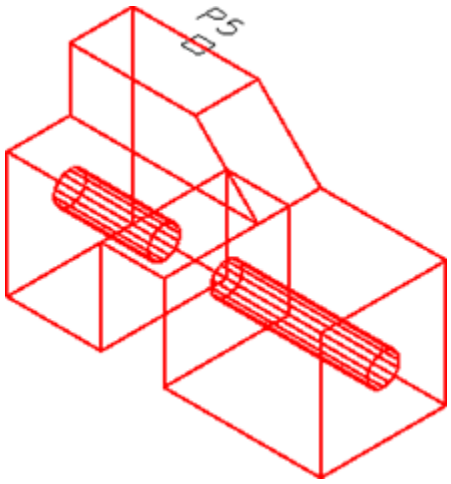


Figure Step 9B

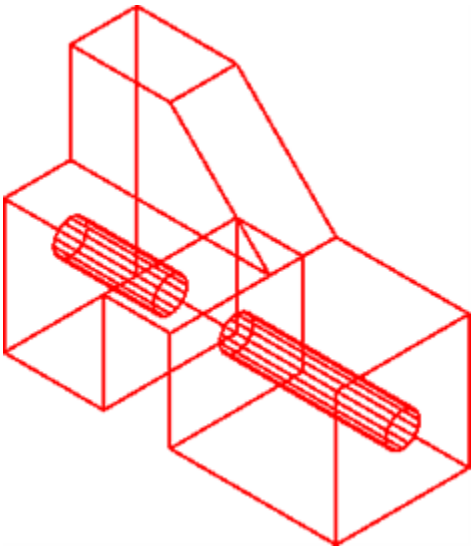


Figure Step 9C

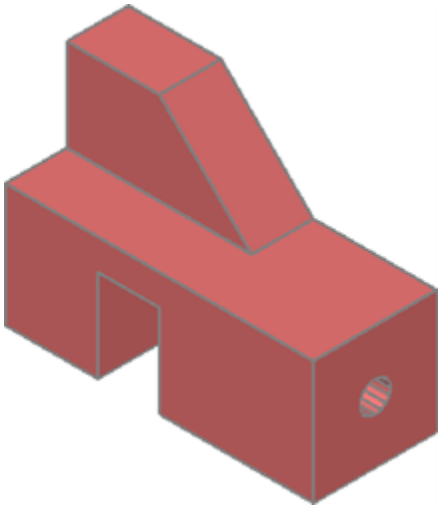


Figure Step 9D

AUTHOR'S COMMENTS: The Offset Faces option of the SOLIDEDIT command allows you to offset a face and have the solid take the shape of the new offset face. The offset can make the solid larger or smaller.

Step 10

Click the Offset Faces icon and complete the command, as shown below, to offset the face and the hole onthe right side 2 inches longer.(Figure Step 10A, 10B, 10C, and 10D)

Command: **SOLIDEDIT**

Solids editing automatic checking: SOLIDCHECK=1

Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: **F**

Enter a face editing option

[Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/eXit] <eXit>: **O**

Select faces or [Undo/Remove]: **P6** 2 faces found.

Select faces or [Undo/Remove/ALL]: **R**

Remove faces or [Undo/Add/ALL]: **P7** 2 faces found, 1 removed.

Remove faces or [Undo/Add/ALL]: **A**

(This time a face must be added in.)

Select faces or [Undo/Remove/ALL]: **P8** 2 faces found. Select faces or [Undo/Remove/ALL]: **R**

Remove faces or [Undo/Add/ALL]: **P9** 1 face found, 1 removed.

Remove faces or [Undo/Add/ALL]:

Specify the offset distance: **2**

(Offset distance is 2 units.)

Solid validation started.

Solid validation completed.

Enter a face editing option

Command:

Figure Step 10A

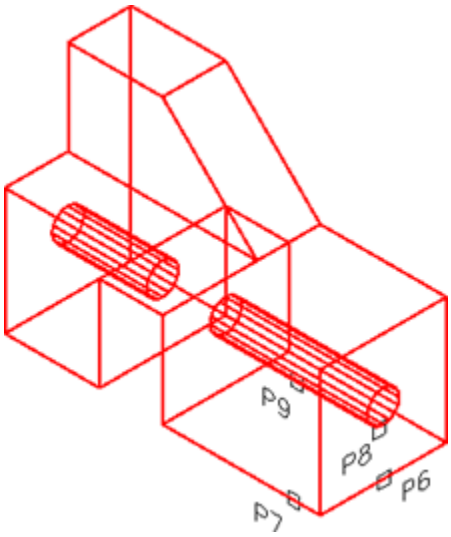


Figure Step 10B

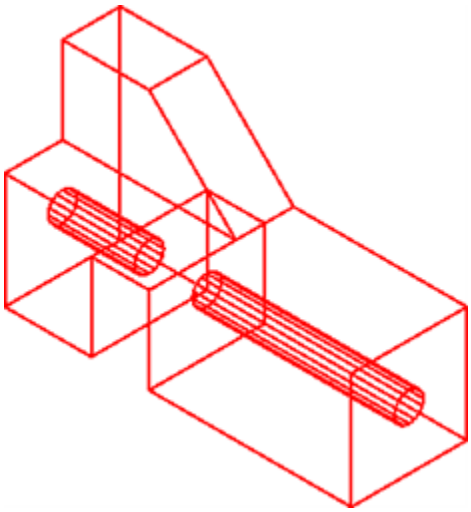


Figure Step 10C

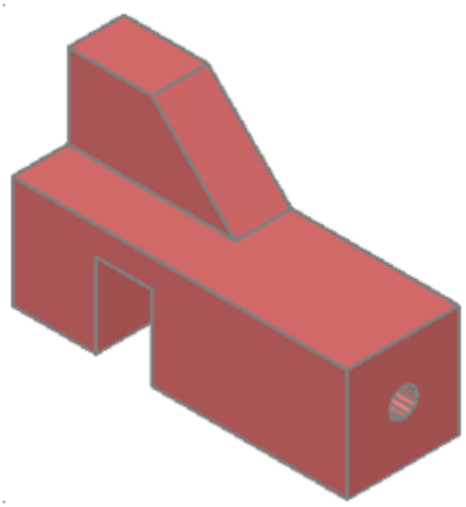


Figure Step 10D

AUTHOR'S COMMENTS: The Delete Faces option of the SOLIDEDIT command allows you to delete a solid primitive that is part of an existing solid.

Step 11

Click the Delete Faces icon and complete the command shown below to delete a the hole in the model. (Figure Step 11A, 11B, and 11C)

Command: **SOLIDEDIT**

Solids editing automatic checking: SOLIDCHECK=1

Enter a solids editing option [Face/Edge/Body/Undo/eXit]

<eXit>: **F**

Enter a face editing option

[Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/ eXit] <eXit>: **D**

Select faces or [Undo/Remove]: **P10** 2 faces found.

Select faces or [Undo/Remove/ALL]: **R**

Remove faces or [Undo/Add/ALL]: **P11** 2 faces found, 1 removed.

(Remove the left face and keep the hole face.)

Remove faces or [Undo/Add/ALL]:

Solid validation started.

Solid validation completed.

Enter a face editing option

Command:

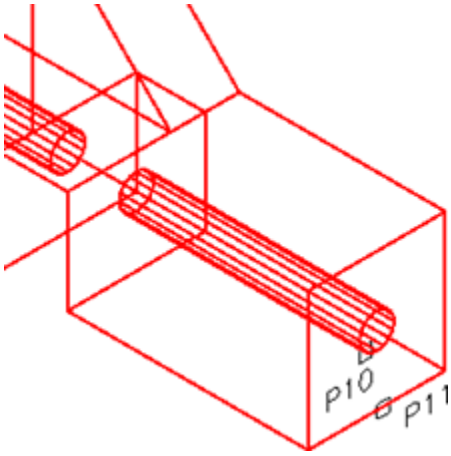


Figure Step 11A

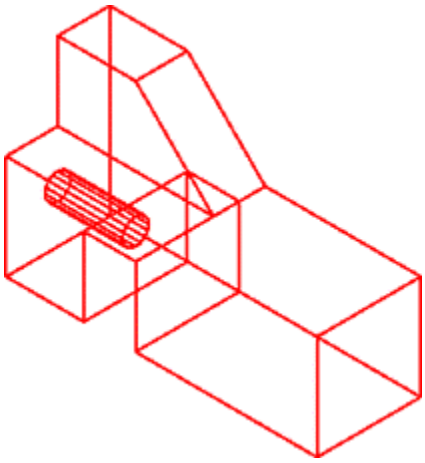


Figure Step 11B

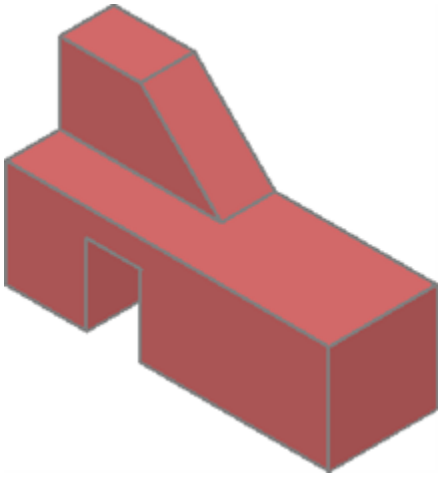


Figure Step 11C

Step 12

Save and close the drawing.

Key Principles

Key Principles in Module 27

1. When editing solids, you must be able to identify and select the body, faces and edges of the solid model.
2. When editing solids, always use the Solids Editing toolbar instead of trying to enter the commands using another method.
3. One of the trickiest parts of using the SOLIDEDIT command is selecting the solid faces or edges that you want to edit.

Lab Exercise 27-1

Time allowed: 30 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 27-1	N/A	Inches

Step 1

Open the drawing: AutoCAD 3D Lab 18-5 and save it with the name: AutoCAD 3D Lab 27-1.

Step 2

Using the SOLIDEDIT command only, edit the solid to appear as shown in the figures. (Figure Step 2A, 2B, and 2C)

AUTHOR'S COMMENTS: The rear of the solid remains in the same location. All the editing is done towards the front.

Figure Step 2A
Solid Before

Figure Step 2B
Completed Solid Model

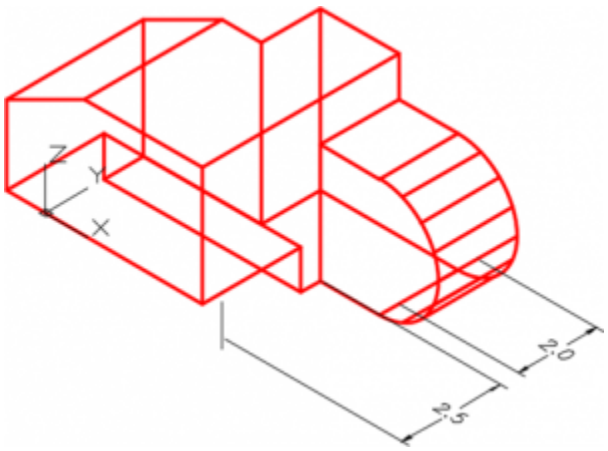


Figure Step 2C
Dimensioned Model

Lab Exercise 27-2

Time allowed: 30 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 27-2	N/A	Inches

Step 1

Open the drawing: AutoCAD 3D Lab 17-1 and save it with the name: AutoCAD 3D Lab 27-2

Step 2

Using the SOLIDEDIT command only, edit the solid to appear as in the figures. (Figure Step 2A, 2B, and 2C)

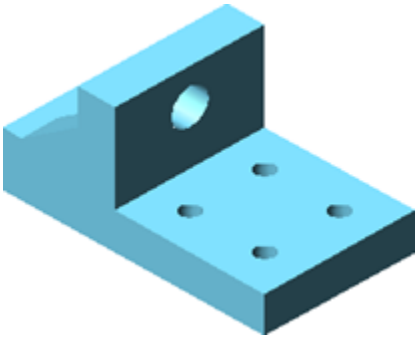


Figure Step 2A
Solid Before

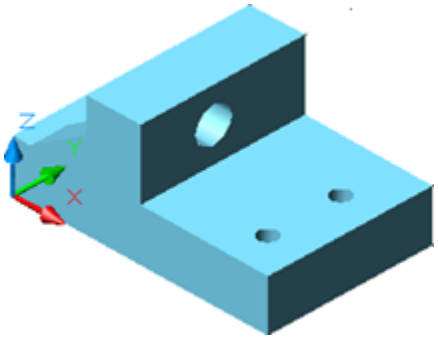


Figure Step 2B
Completed Solid Model

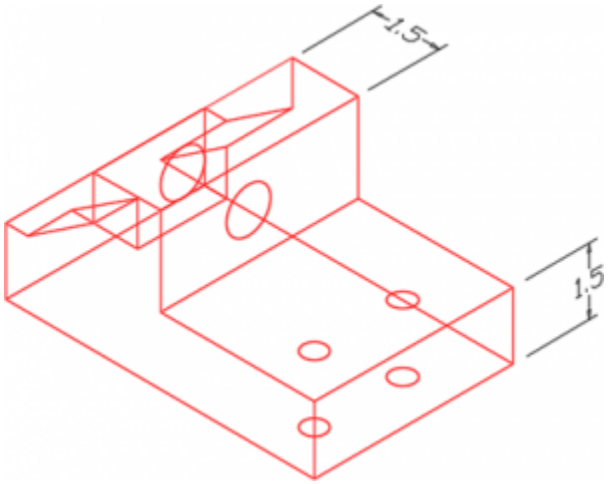


Figure Step 2C
Dimensioned Model

Module 28 Editing Solid Models – Part 2

Learning Outcomes

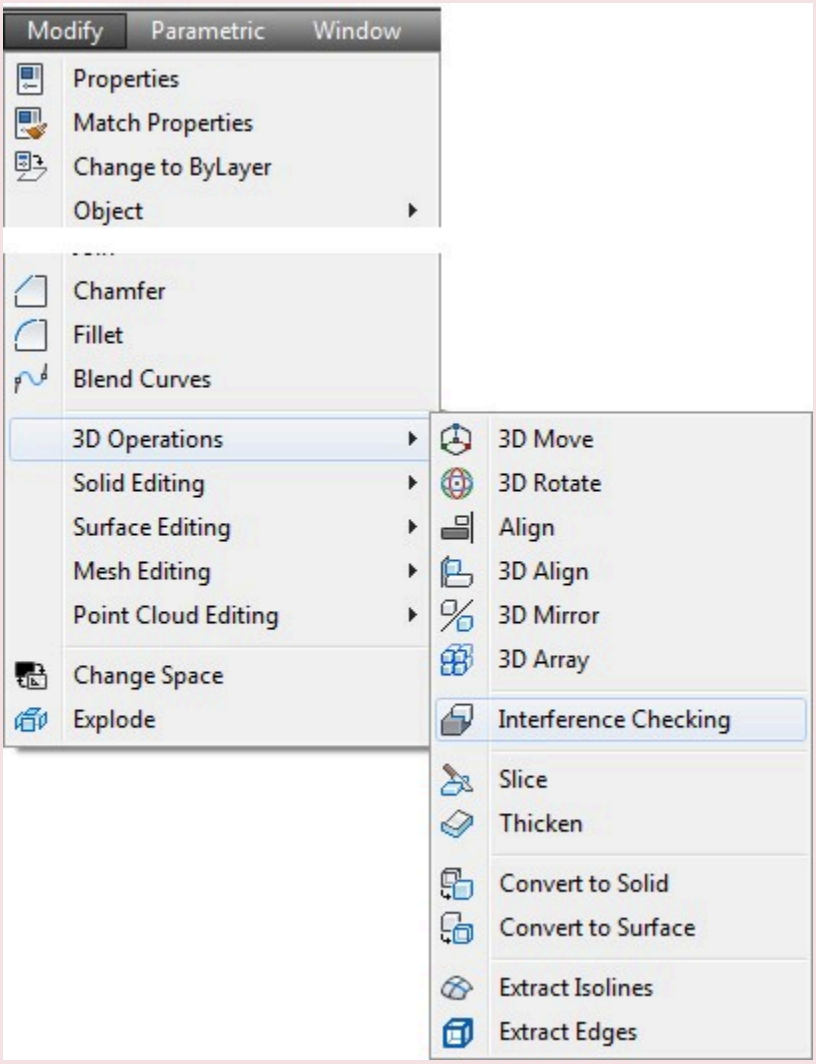
When you have completed this module, you will be able to:

1. Apply the SOLIDEDIT and SOLIDCHECK commands to modify solid models.
2. Apply the INTERFERE command to find interferences between two or more solids.

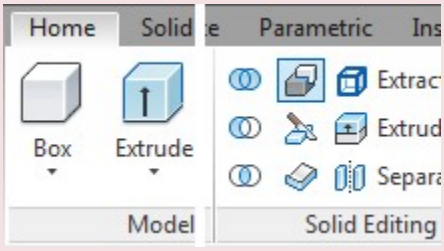
AutoCAD Command: INTERFERE

The INTERFERE command is used to check if two or more solids occupy the same 3D space. If two or more solids occupy the same 3D space, they will interfere with one another which cannot exist in real life.

Shortcut: none



Modify Pull-down



Home Ribbon

WORK ALONG: Editing Solid Models

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save the drawing and name it: AutoCAD 3D Workalong 28-1.

Step 3

Draw the plines on layer: Pline and the solid model on layer: Solid 6.

Step 4

Draw a solid model of the object shown in the figures. The location of 0,0,0 is the bottom left corner of the front view. The solid model should appear as shown in the figure. (Figure Step 4A and 4B)

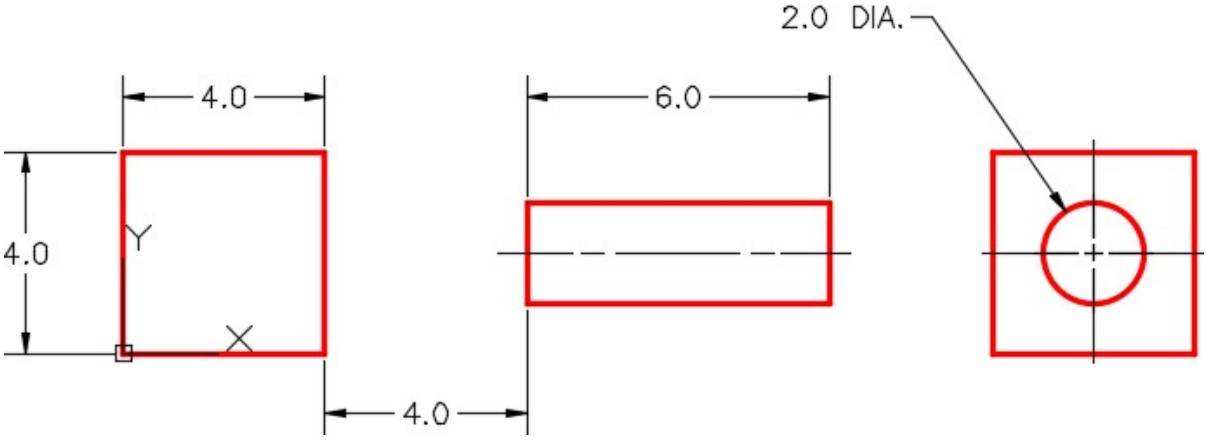


Figure Step 4A
Dimensioned Multiview Drawing

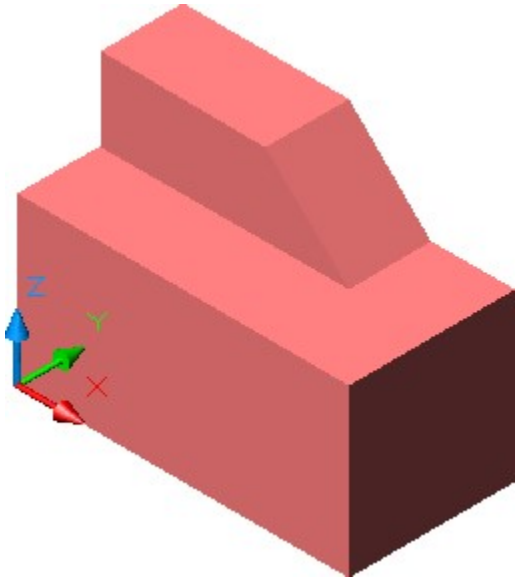
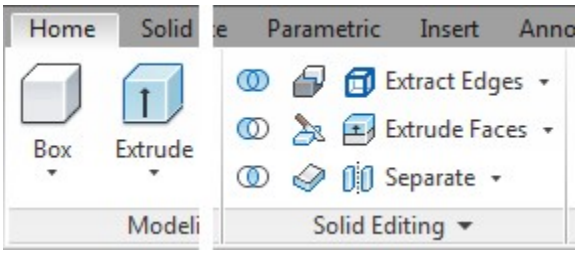


Figure Step 4B
Solid Model SE Isometric View

Step 5

If you are using toolbars, enable the display of the Solids Editing toolbar. If you are using ribbons, use the Home tab. (Figure Step 5)

Solid Editing Toolbar



Home Ribbon
Figure Step 5

Step 6

With the UCS set to Front, draw a circle by snapping to the midpoint of the bottom edge for the centre and one of the bottom corners for the radius. (Figure Step 6)

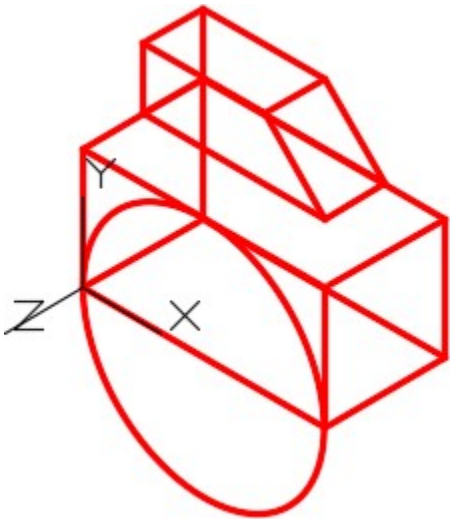


Figure Step 6

AUTHOR'S COMMENTS: The Imprint option of the SOLIDEDIT command is used to attach a 2D drawing object onto an existing solid. The attached drawing object can then be extruded to add or remove volume. The 2D drawing object must be an arc, circle, pline, ellipse, spline or a region. A 3D solid can also be attached. The attached object must touch the solid or overlap the solid model it is being attached to.

Step 7

Click the Imprint icon and complete the command, as shown below, to attach the circle to the front face of the solid. (Figure Step 7A, 7B, and 7C).

Command: **SOLIDEDIT**

Solids editing automatic checking: SOLIDCHECK=1

Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: **B**

Enter a body editing option

[Imprint/seParate solids/Shell/cLean/Check/Undo/eXit] <eXit>: **I**

Select a 3D solid: **P1**

(Select the solid model.)

Select an object to imprint: **P2**

(Select the circle.)

Delete the source object [Yes/No] <N>: **Y**

(Delete the original circle.)

Select an object to imprint:

Command:

Figure Step 7A

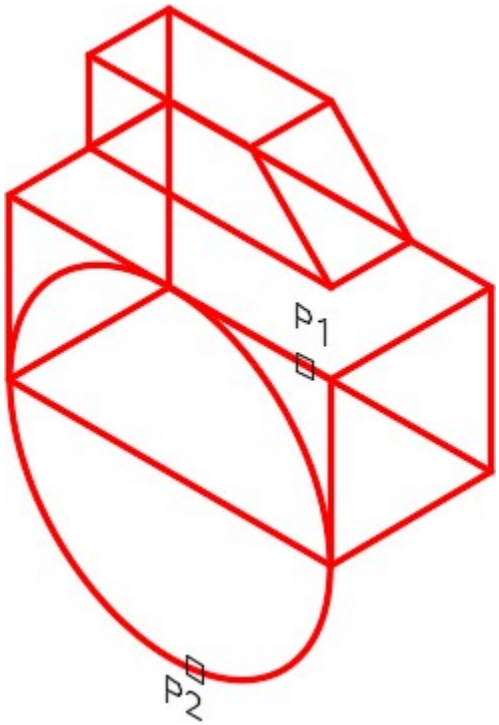


Figure Step 7B

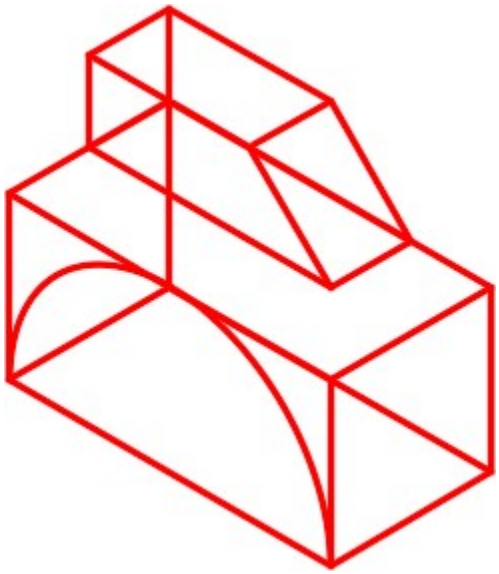


Figure Step 7C

Step 8

Using the Properties window, check to ensure that the arc and the solid are one solid model and not two separate objects.

Step 9

Using what you learned in Module 27, click the Extrude faces icon to extrude the arc 1 inch in the positive Z direction as shown in the command below. (Figure Step 9A and 9B)

Command: **SOLIDEDIT**

Solids editing automatic checking: SOLIDCHECK=1

Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>:**F**

Enter a face editing option

[Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/eXit] <eXit>: **E**

Select faces or [Undo/Remove]: **P3** 2 faces found.

Select faces or [Undo/Remove/ALL]: **R**

Remove faces or [Undo/Add/ALL]: 2 faces found, 1 removed.

Remove faces or [Undo/Add/ALL]:

Specify height of extrusion or [Path]: **1**

Specify angle of taper for extrusion <0>:

Solid validation started.

Solid validation completed.

Command:

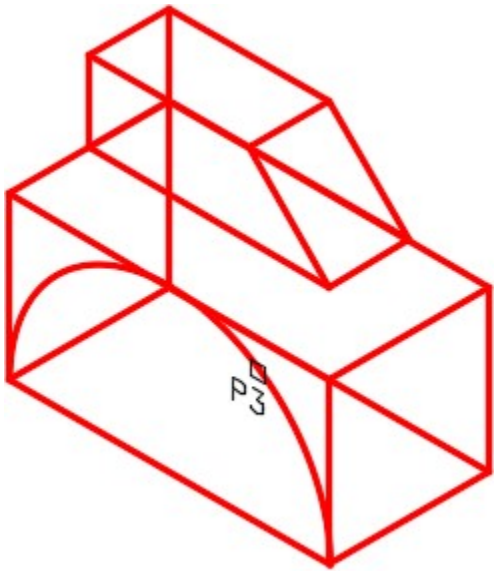


Figure Step 9A

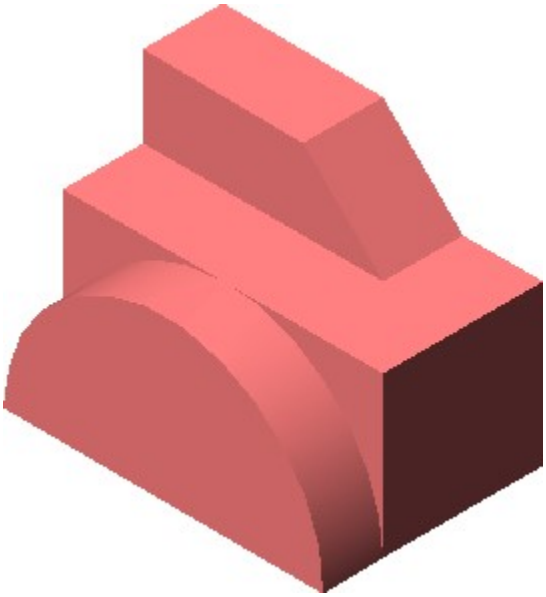


Figure Step 9B

Step 10

Using what you just learned, draw a 4 inch diameter circle locating its centre at the midpoint of the bottom edge. Imprint the circle to the solid to form the arc . Extrude the arc through the solid to create a slotted arc through the solid. (Figure Step 10A and 10B)

AUTHOR'S COMMENTS: The Check option of the SOLIDEDIT command is used to check the validity of an existing 3D solid object.

Step 11

Click the Check icon and enter the command, as shown below, to check your solid as a valid solid object. (Figure Step 11A and 11B)

Command: **SOLIDEDIT**

Solids editing automatic checking: SOLIDCHECK=1

Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: **B**

Enter a body editing option

[Imprint/seParate solids/Shell/cLean/Check/Undo/eXit] <eXit>: **C**

Select a 3D solid:

(Select the solid model.)

This object is a valid ShapeManager solid.

(This is a valid solid.)

Enter a body editing option

Command:

Figure Step 11A

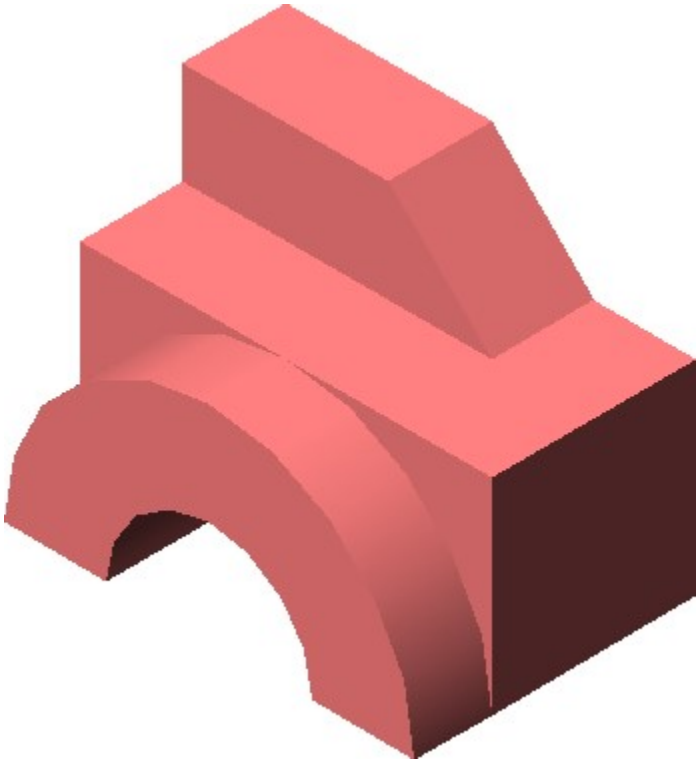


Figure Step 11B

Step 12

Save and close the drawing.

WORK ALONG: Finding the Interference Between Two Solids

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save the drawing and name it: AutoCAD 3D Workalong 28-2.

Step 3

Draw the plines on layer: Pline and the solid model on layer: Solid 7.

Step 4

Draw two solid models using the multiview drawing as a reference. Start the bottom left corner of the cube at 0,0,0. (Figure Step 4A and 4B)

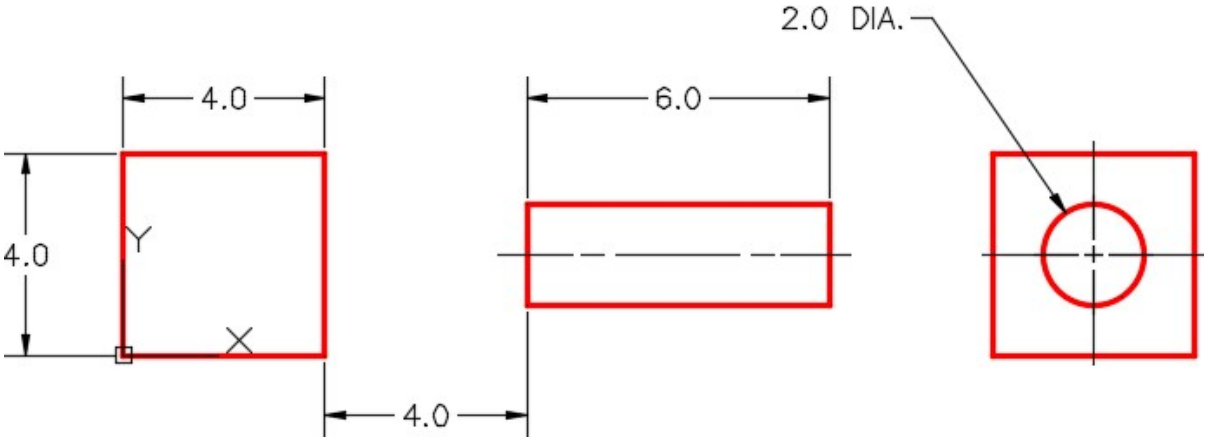


Figure Step 4A
Dimensioned Multiview Drawing

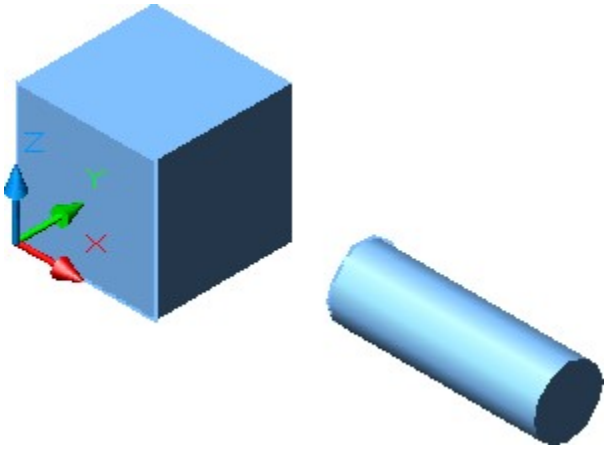
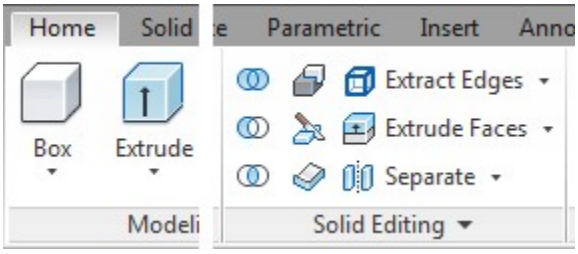


Figure Step 4B
Solid Model SE Isometric View

Step 5

If you are using toolbars, enable the display of the Solids Editing toolbar. If you are using ribbons, use the Home tab. (Figure Step 5)

Solid Editing Toolbar



Home Ribbon
Figure Step 5

Step 6

Using the UNION command, union the two solids to make them one solid model.

Step 7

Using the Properties window, check to ensure that the two 3D Solid objects are now one 3D Solid object. (Figure Step 7)

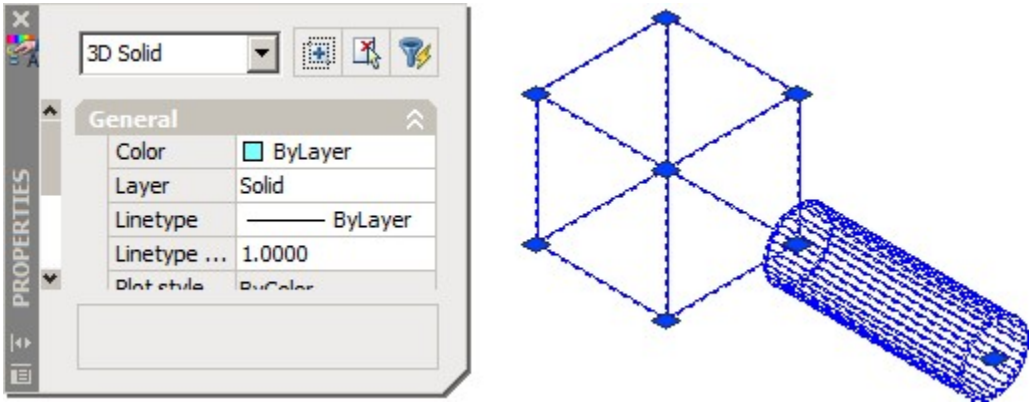


Figure Step 7

AUTHOR'S COMMENTS: The Separate option of the SOLIDEDIT command is used to separate two solids that have been joined together.

Step 8

Set layer: Solid 7 as the current layer. Click the Separate icon and enter the command shown below to separate the two solids. (Figure Step 8)

Command: **SOLIDEDIT**

Solids editing automatic checking: SOLIDCHECK=1

Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: **B**

Enter a body editing option

[Imprint/seParate solids/Shell/cLean/Check/Undo/eXit] <eXit>: **P**

Select a 3D solid:

(Select the solid.)

Enter a body editing option

Command:

Figure Step 8

Step 9

Using the Properties window, ensure that the two 3D Solid objects were separated as shown in the figure. (Figure Step 9)

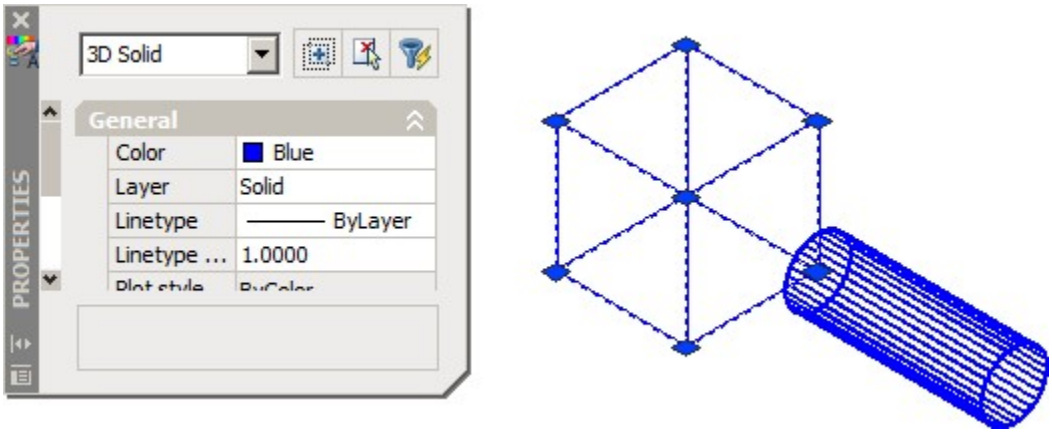


Figure Step 9

Step 10

Set the current visual style to Realistic. Move the cylinder 3 inches towards the box as shown in the figure. (Figure Step 10)

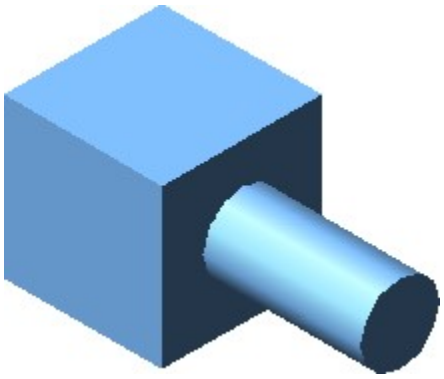


Figure Step 10

AUTHOR'S COMMENTS: The two solid now interfere with one another and for a small volume, they occupy the same 3D space. In real life, that could not happen.

Step 11

Set the current view to SE Isometric and the current visual style to 3D Wireframe. Enter the INTERFERE command, as shown below, to find the interfering solid. (Figure Step 11)

Command: **INTERFERE**

Select first set of solids:

(Select one solid.)

Select objects: 1 found

Select objects:

(Press Enter)

Select second set of solids:

Select objects: 1 found

(Select the other solid.)

Select objects:

(Press Enter)

Comparing 1 solid against 1 solid.

Interfering solids (first set): 1

(second set): 1

Interfering pairs: 1

Create interference solids? [Yes/No] <N>: **Y**

(Enter Y.)

Command:

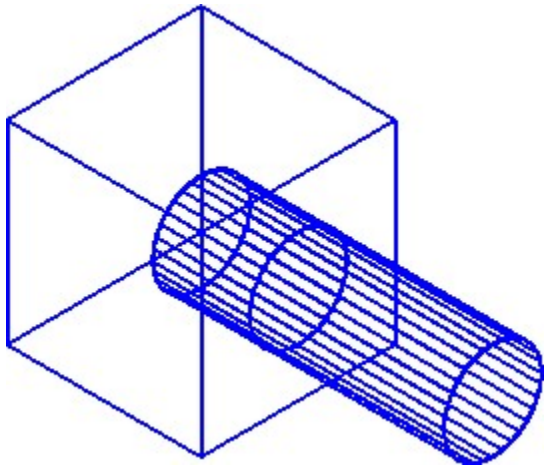


Figure Step 11

Step 12

Using the MOVE command, move the 3D Solid object created by the INTERFERE command. Where you move it is not important. (Figure Step 12)

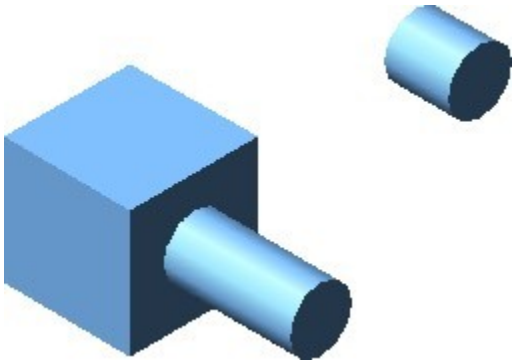


Figure Step 12

Step 13

Use the UNION command the join the two solids to one. (Figure Step 13)

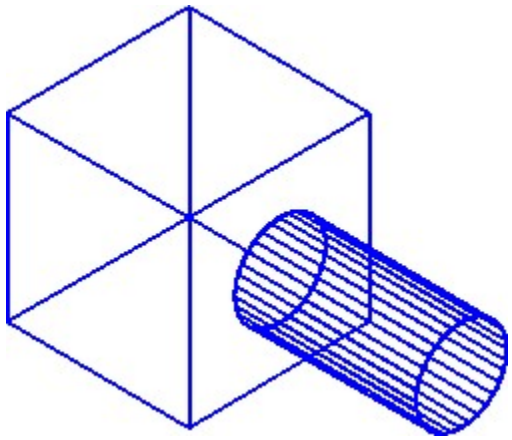


Figure Step 13

Step 14

Set the current visual style to Realistic. (Figure Step 14)

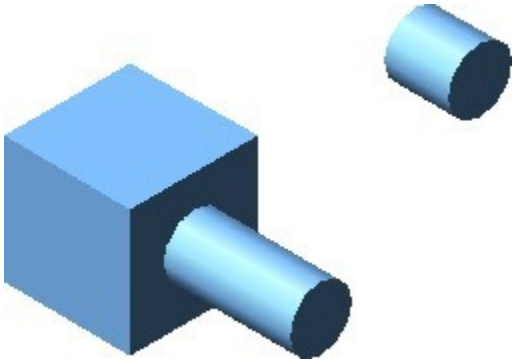


Figure Step 14

Step 15

Save and close the drawing.

System Variable: SOLIDCHECK

The SOLIDCHECK system variable toggles the solid validation on and off when a solid model is selected to be edited in the SOLIDEDIT command.

When the SOLIDEDIT command is executed, the current setting of the SOLIDCHECK system variable is displayed as shown below.

Command: **SOLIDEDIT**

Solids editing automatic checking: SOLIDCHECK=1

(SOLIDECHECK=1 solidcheck is enabled)

(SOLIDECHECK=0 solidcheck is disabled)

Key Principles

Key Principles in Module 28

1. The Separate option of the SOLIDEDIT command is used to separate two solids that have been joined together.
2. The INTERFERE command is used to check if two or more solids occupy the same If two or more solids occupy the same space, they will interfere with one another and in real life could not exist.

Module 29 Shell

Learning Outcomes

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

1. Describe and apply the SOLIDEDIT command and the Shell option to create a hollow cavity, with a specified wall thickness, in a solid model.

Shell

A *shell* is preformed when material is removed from the interior of a solid model leaving a hollow cavity with a specify wall thickness. This is done using the Shell option in the SOLIDEDIT command. See Figures 29-1, 29-2, and 29-3.

After entering the Shell option, you can specify one or more faces to be removed. That leaves the remaining faces for the shell walls. If no faces are specified for removal, the shell creates a hollow model. The thickness of the shell wall can also specified.

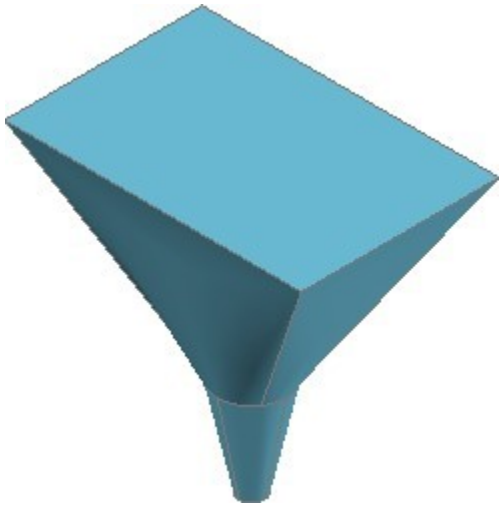


Figure 29-1 Solid Before Shell

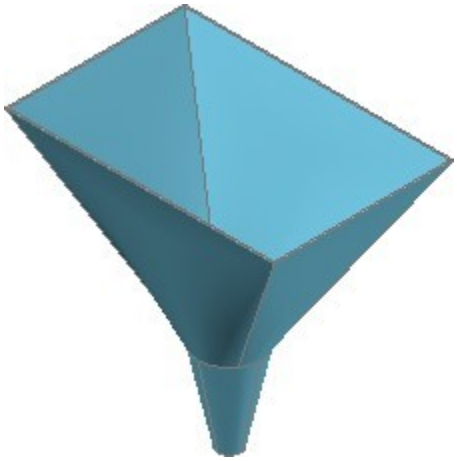


Figure 29-2
Solid After Shell

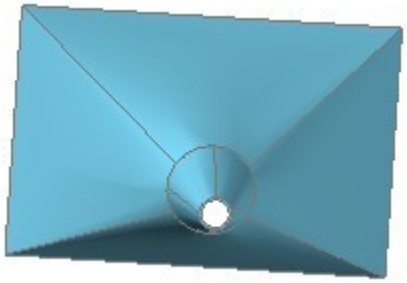


Figure 29-3 Solid After Shell Rotated View

WORK ALONG: Shelling

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save the drawing with the name: AutoCAD 3D Workalong 29-1.

Step 3

On layer: Solid 3, draw the two solid models using the multiview drawing as a reference. Start the bottom left corner of the cube at 0,0,0. (Figure Step 3A and 3B)

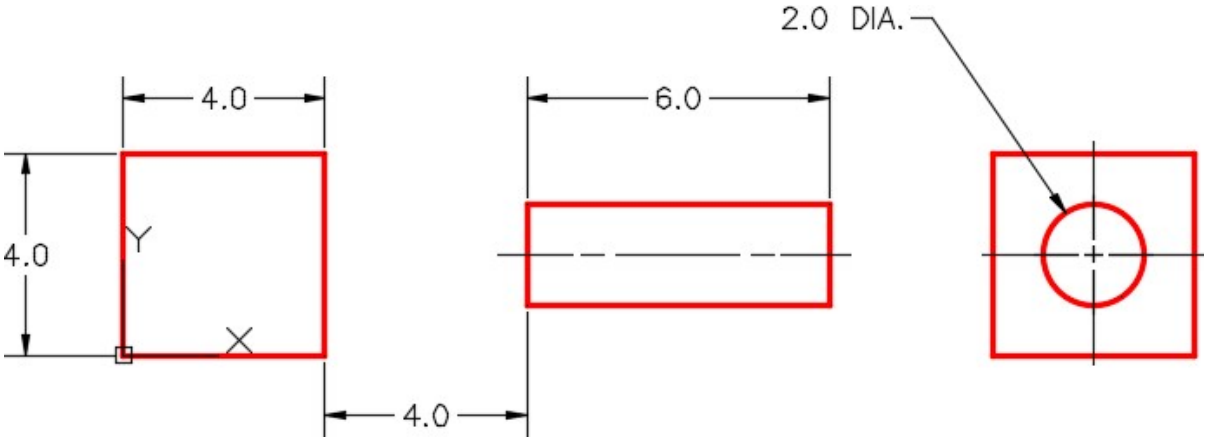


Figure Step 3A
Dimensioned Multiview Drawing

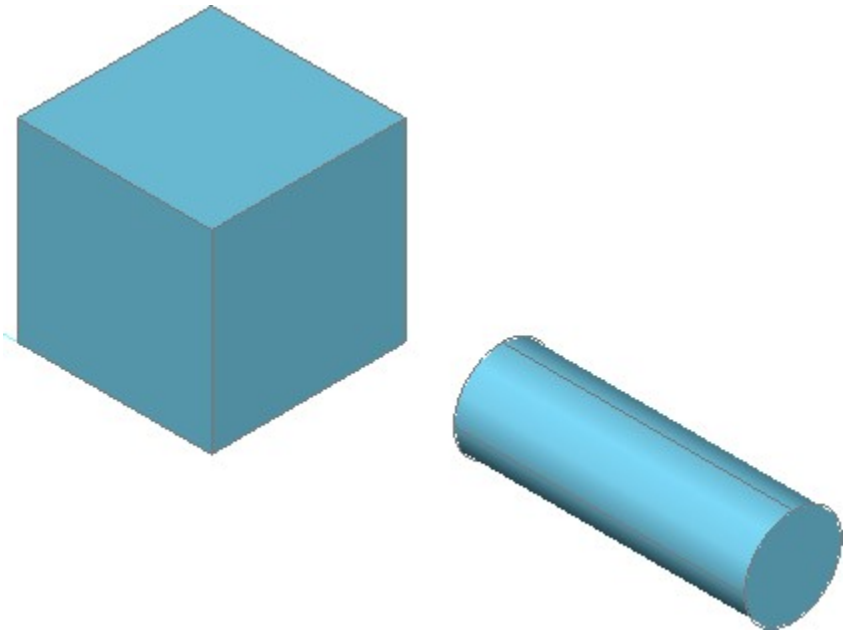


Figure Step 3B
Complete Solid Models

Step 4

If you are using toolbars, enable the display of the Solids Editing toolbar. If you are using ribbons, enable the Home Tab. (Figure Step 4A and 4B)

Figure Step 4A

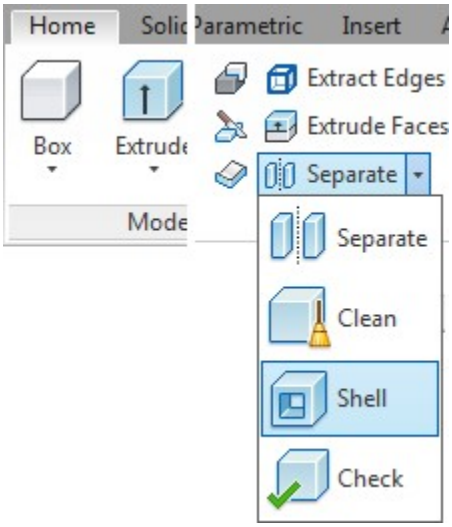


Figure Step 4B

Step 5

Click the Shell option and enter the command shown below. Enter the wall thickness 0.1 inches. Remove the top face only. (Figure Step 5A, 5B, 5C, and 5D)

Command: **SOLIDEDIT**

Solids editing automatic checking: SOLIDCHECK=1

Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>:**B**

Enter a body editing option

[Imprint/seParate solids/Shell/cLean/Check/Undo/eXit] <eXit>: **S**

Select a 3D solid:

(Select the box.)

Remove faces or [Undo/Add/ALL]: 2 faces found, 2 removed.

(Select the to of the box.)

Remove faces or [Undo/Add/ALL]: **A**

Select faces or [Undo/Remove/ALL]: 1 faces found.

(Select the side below the top.)

Select faces or [Undo/Remove/ALL]:

Enter the shell offset distance: **0.1**

Solid validation started.

Solid validation completed.

Enter a body editing option

Command:

Figure Step 5A

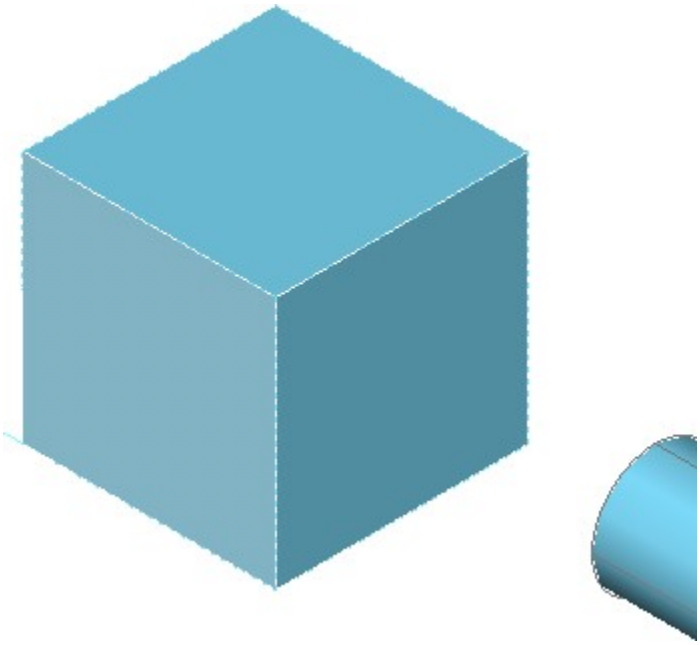


Figure Step 5B

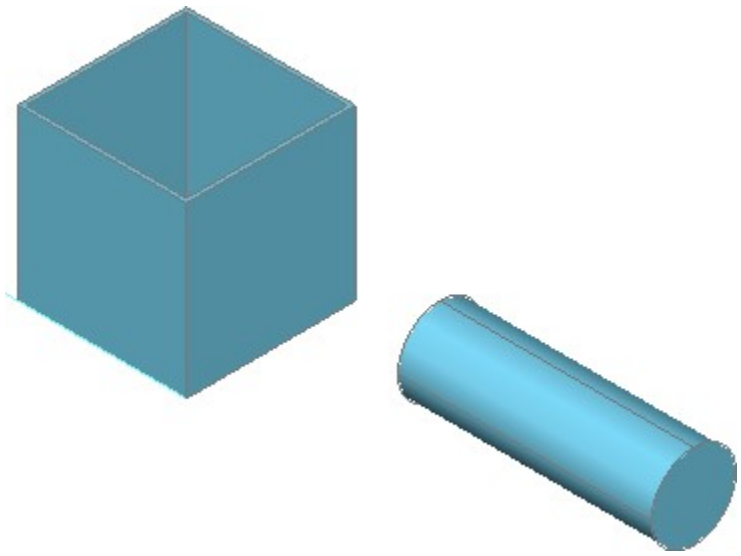


Figure Step 5C

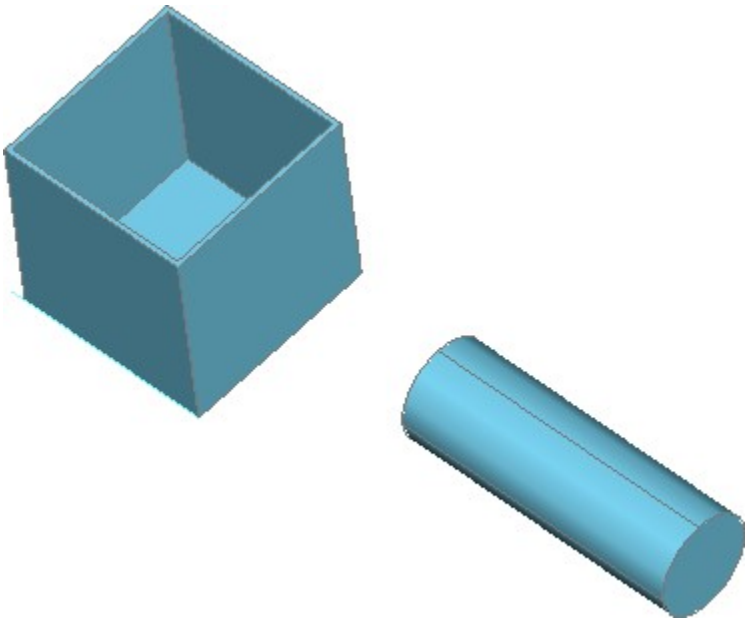


Figure Step 5D

AUTHOR'S COMMENTS: The Shell option is one of the trickiest options to use. The problem most student have is not knowing how to remove and add the faces back in. Watch the prompts as you select faces, it will tell you how many faces were removed so you know how many you have to add back in. If the commands does work the first time, undo the command try again. Keep doing this until you find it easy to do.

Step 6

Using what you just learned, click the Shell option again but this time, create a pipe with a 0.25 inch wall thickness. You will have to remove both ends of the pipe. See the command below. (Figure Step 6A, 6B, and 6C)

Command: **SOLIDEDIT**

Solids editing automatic checking: SOLIDCHECK=1

Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: **B**

Enter a body editing option

[Imprint/seParate solids/Shell/cLean/Check/Undo/eXit] <eXit>: **S**

Select a 3D solid:

Remove faces or [Undo/Add/ALL]: 2 faces found, 2 removed.

Remove faces or [Undo/Add/ALL]: **A**

Select faces or [Undo/Remove/ALL]: 1 face found.

Select faces or [Undo/Remove/ALL]: **R**

Remove faces or [Undo/Add/ALL]: 2 faces found, 2 removed.

Remove faces or [Undo/Add/ALL]: **A**

Select faces or [Undo/Remove/ALL]: 1 face found.

Select faces or [Undo/Remove/ALL]:

Enter the shell offset distance: **0.25**

Solid validation started.

Solid validation completed.

Enter a body editing option.

Command:

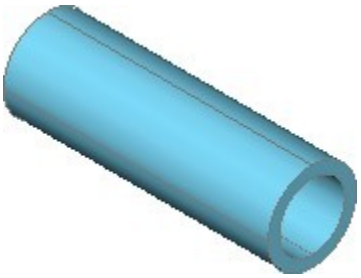


Figure Step 6A

AUTHOR'S COMMENTS: As I said before, the Shell option is one of the trickiest options to use. In this step, you will have to orbit the model in the middle of the command to remove the face on the opposite end of

the pipe.

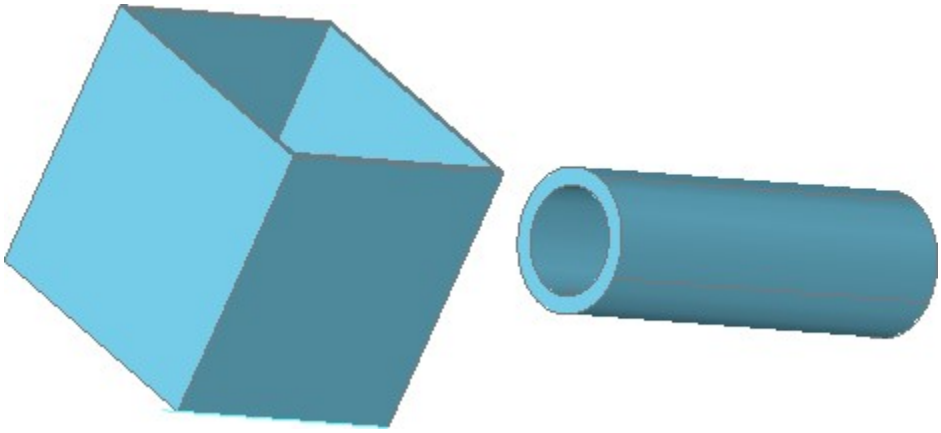


Figure Step 6B

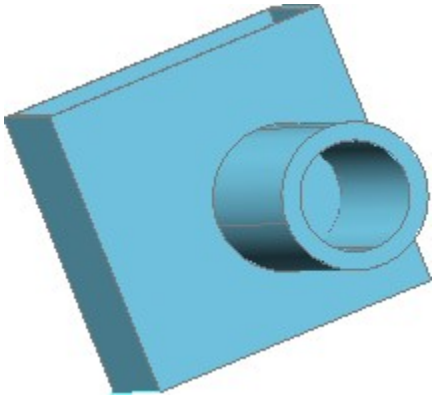


Figure Step 6C

Step 7

Orbit the model to check for completeness. Set the current view to SE Isometric. (Figure Step 7)

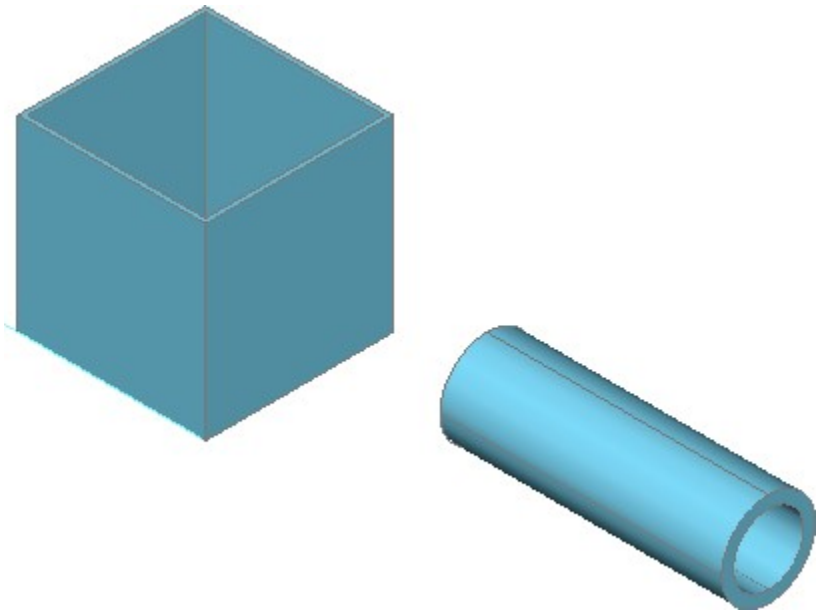


Figure Step 7

Step 8

Save and close the drawing.

Key Principles

Key Principles in Module 29

1. A shell is preformed when material is removed from the interior of a solid model leaving a hollow cavity with a specify wall This is done using the Shell option in the SOLIDEDIT command.
2. The Shell option is one of the trickiest options to The problem most student have is not knowing how to remove and add the faces back in.

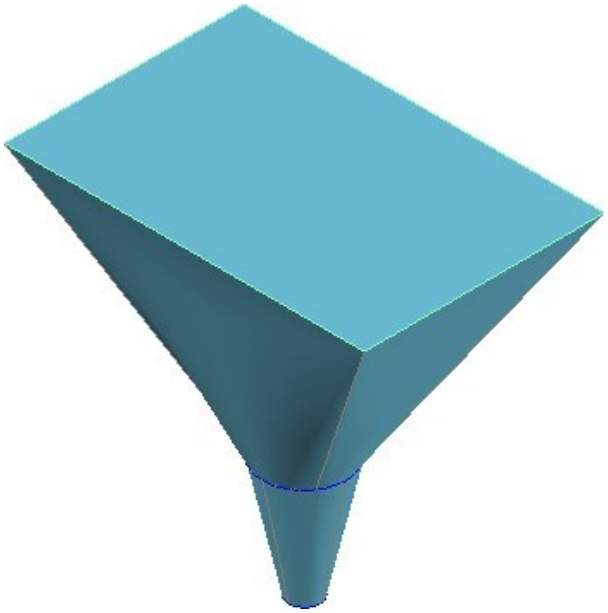
Lab Exercise 29-1

Time allowed: 30 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 29-1	N/A	Millimeters

Step 1

Open drawing: AutoCAD 3D Lab 24-1. Save and name the drawing with the name: AutoCAD 3D Lab 29-1. (Figure Step 1)



Step 1

Step 2

Shell the solid model with a wall thickness of 1.0 mm. Remove the top and bottom to create a funnel. (Figure Step 2A and 2B)

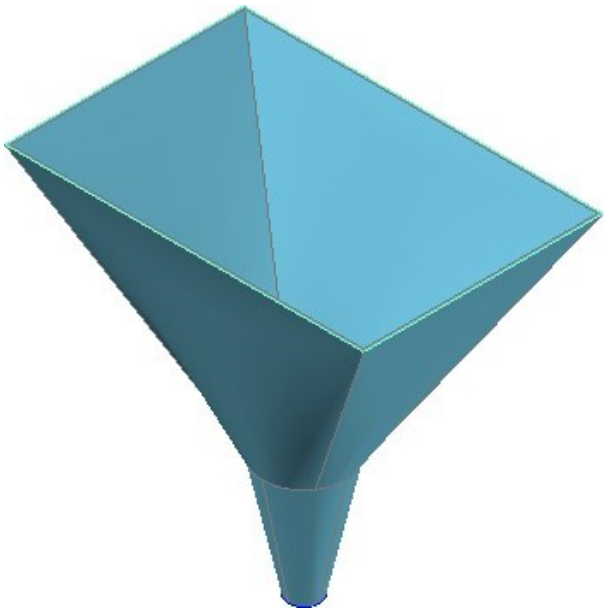


Figure Step 2A
SE Isometric View

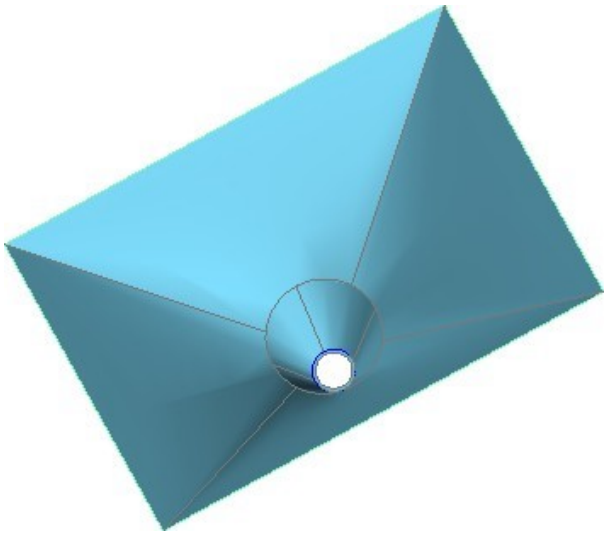


Figure Step 2B
Orbited View

Lab Exercise 29-2

Time allowed: 30 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 29-2	N/A	Inches

Step 1

Open drawing: AutoCAD 3D Lab 24-2. Save and name the drawing with the name: AutoCAD 3D Lab 29-2. (Figure Step 1)

Figure Step 1

Step 2

Shell the solid model with a wall thickness of 0.1 inches. Remove the top to create a bottle. (Figure Step 2A and 2B)

Figure Step 2A

Figure Step 2B

Lab Exercise 29-3

Time allowed: 60 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 29-3	3D Layout English	Inches

Step 1

Start a new drawing using the template shown above. Save and name it: AutoCAD 3D Lab 29-3.

Step 2

On layer: Solid Wall Plate, draw the solid model shown in the figures. (Figure Step 2A, 2B, and 2C)

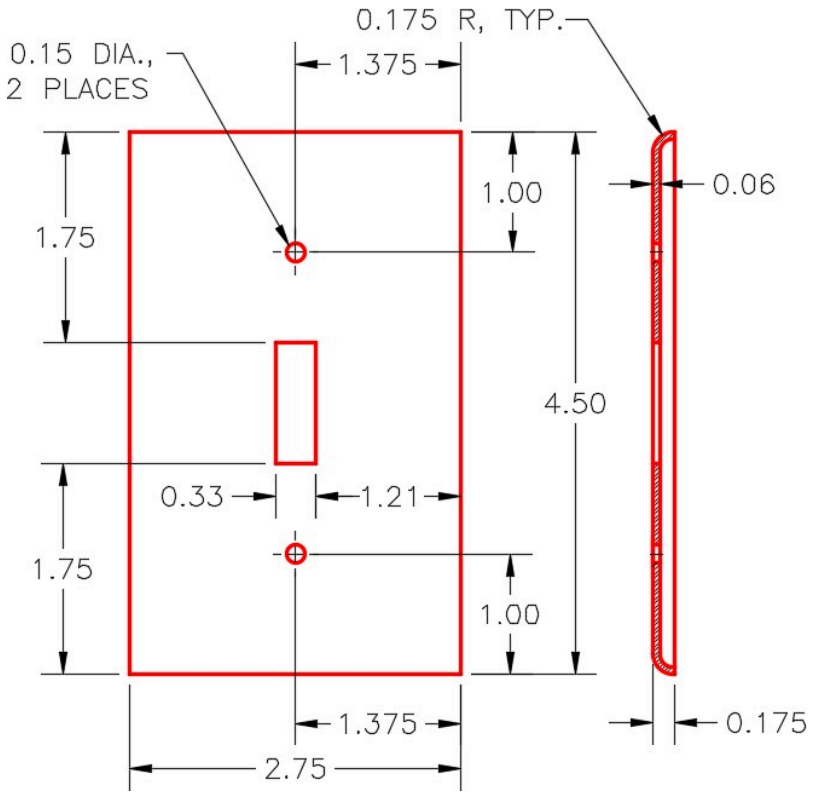


Figure Step 2A
Dimensioned Multiview Drawing

Figure Step 2B
SE Isometric View

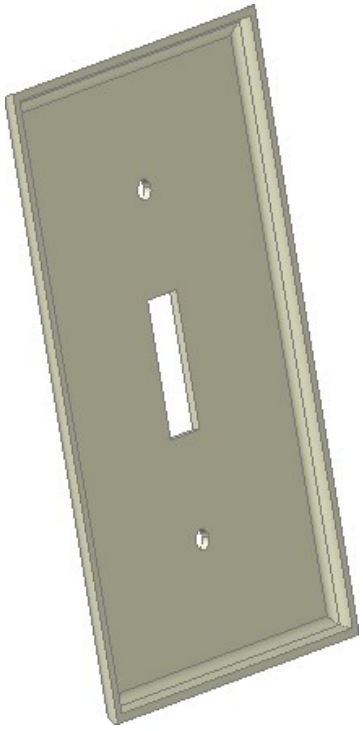


Figure Step 2C
Rotated Back View

Module 30 Creating 2D Drawings from Solid Models

Learning Outcomes

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

1. Apply the SOLVIEW, SOLDRAW, and SOLPROF commands to automatically create 2D working multiview drawings, including pictorial views, from solid models.

Creating 2D Drawings from Solid Models

After an object is constructed as a solid 3D model, there are AutoCAD commands and features available to create and dimension a 2D working multiview drawing. Before you go any farther in this module, you must know and understand the following concepts:

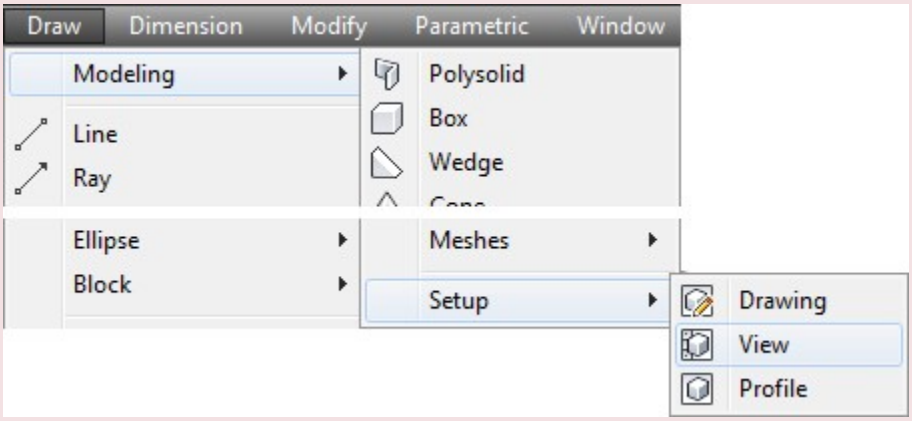
1. Creating and modifying AutoCAD layout drawings.
2. Working in paper space.
3. Creating, editing and setting the scale of viewports.

These three concepts are covered in the AutoCAD 2D book.

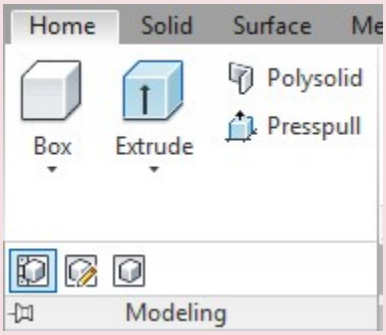
AutoCAD Command: SOLVIEW

The SOLVIEW command is used to create views and the necessary layers of a 3D solid model.

Shortcut: **none**



Draw Pull-down



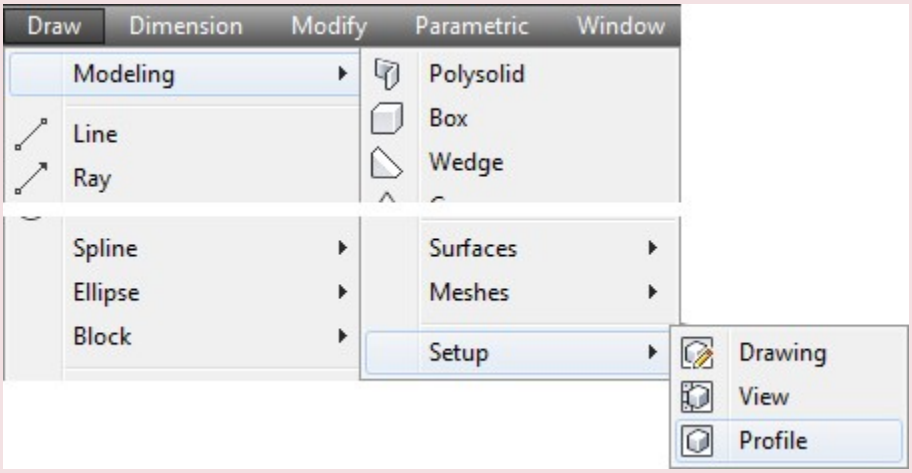
Home Ribbon

Important: Before you use the SOLVIEW command, ensure that the current visual style is 2D Wireframe and the Hidden linetype is loaded into the drawing.

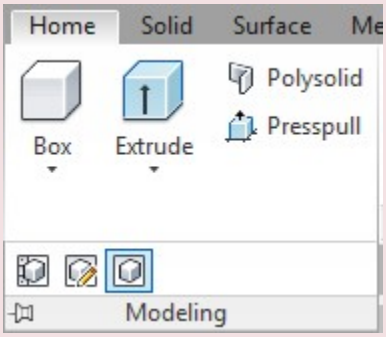
AutoCAD Command: SOLPROF

The SOLPROF command is used to create a profile of edges of straight and curved surfaces of a solid model as it is viewed from a selected viewpoint.

Shortcut: **none**



Draw Pull-down



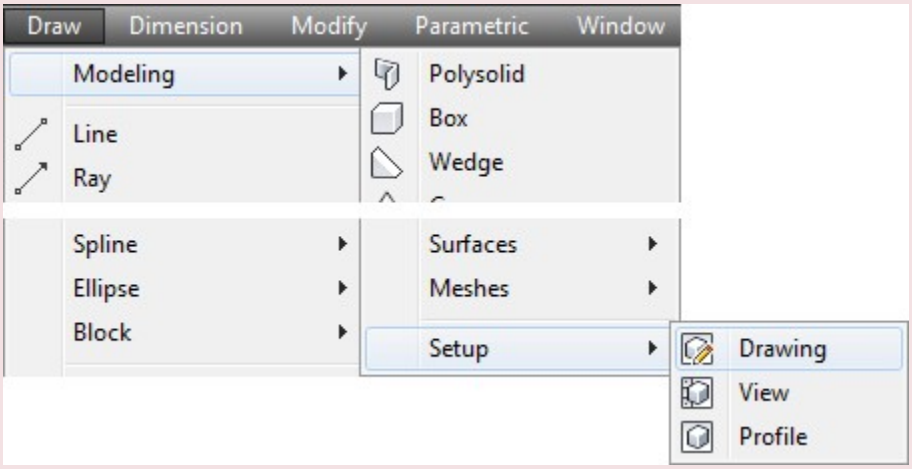
Home Ribbon

Important: Before you use the SOLPROF command, ensure that the current visual style is 2D Wireframe and the Hidden linetype is loaded into the drawing.

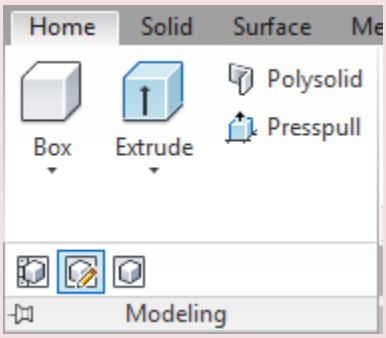
AutoCAD Command: SOLDRAW

The SOLDRAW command is used to convert views created by the SOLVIEW command by projecting the 3D objects onto a 2D plane and changing hidden objects onto the hidden layer.

Shortcut: **none**



Draw Pull-down



Home Ribbon

WORK ALONG: Creating 2D Drawings from a Solid Model

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 30-1.

Step 3

Set the current UCS as World, the current view as SE Isometric, and the current visual style to 2D Wireframe.

Step 4

Using the LINETYPE command, ensure that the Hidden linetype is loaded in the current drawing.

Step 5

Set the Insertion Units to inches and insert the block: AutoCAD 3D Workalong 30-1 at 0,0,0.

Step 6

Explode the block and change the layer that the solid model is on to layer: Solid 7. Using the Properties window, check to ensure that the object is a 3D solid. (Figure Step 6)

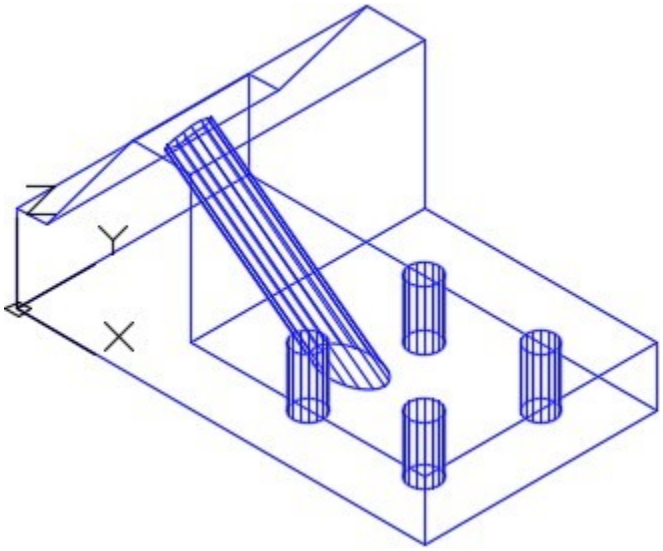


Figure Step 6

Step 7

Save the drawing but do not close it.

AUTHOR'S COMMENTS: The reason I had you save the drawing, in Step 7, is that once you start creating the layout in Steps 8 to 15 and you make an error, you will have to start the drawing over at Step 8. If you make an error, quit the drawing, without saving it, and open the drawing and start at Step 8.

Step 8

Enable layout: Module Layout D. (Figure Step 8)

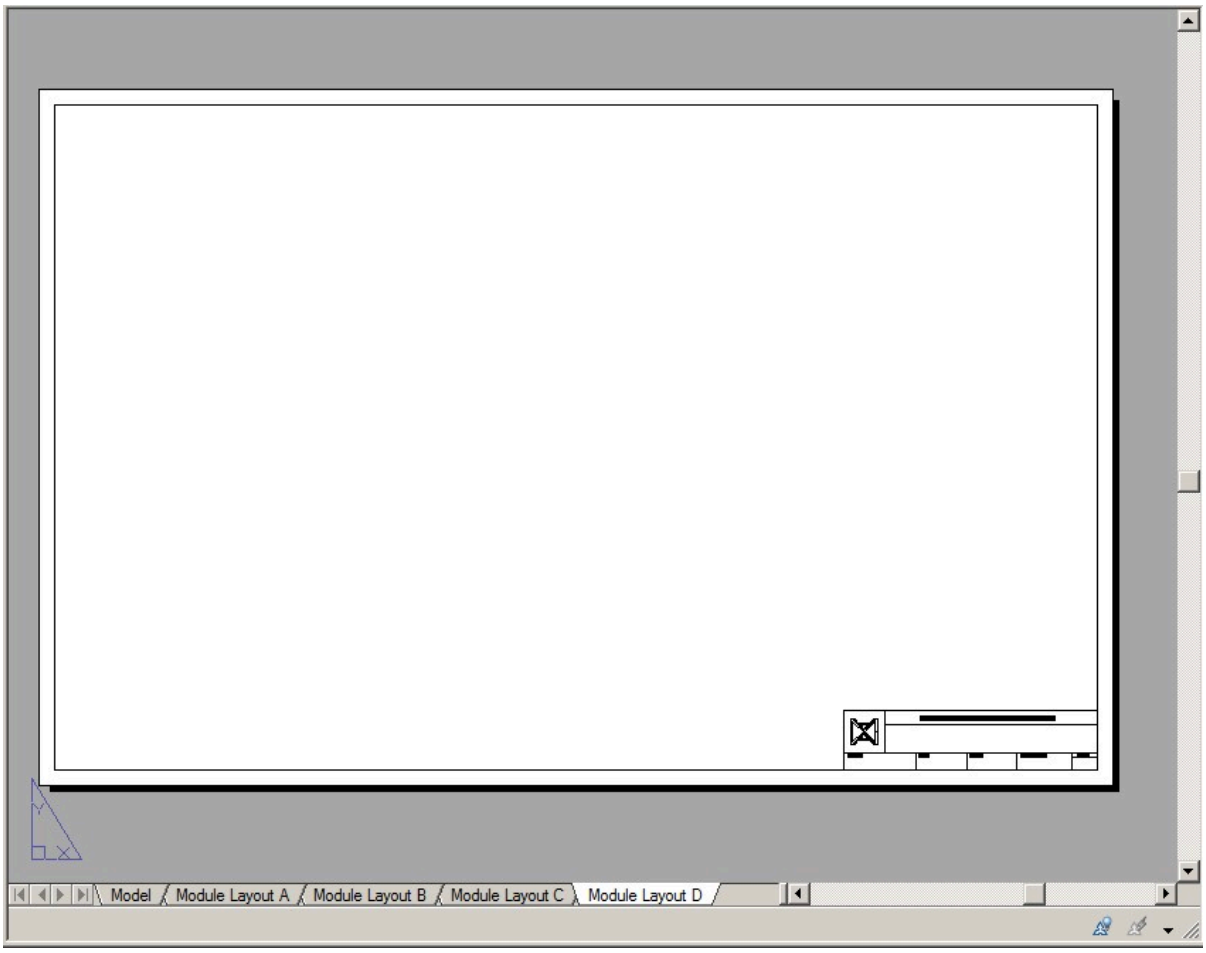


Figure Step 8

MUST KNOW: **VERY IMPORTANT** Before you use either the SOLVIEW and the SOLPROF commands, ensure that the current visual style is 2D Wireframe and the Hidden linetype is loaded into the current drawing.

Step 9

Enter the SOLVIEW command, as shown below, to create the top view of the model. Name it: Top. (Figure Step 9)

Command: **SOLVIEW**

Enter an option [Ucs/Ortho/Auxiliary/Section]: **U**

(Enter U to draw the view from the UCS.)

Enter an option [Named/World/?/Current] <Current>:

(Accept the default Current.)

Enter view scale <1.0000>:

(Accept the scale of 1.)

Specify view center: **P1**

(Select a location for the centre of the view.)

Specify view centre <specify viewport>:

(Press Enter to accept the location.)

Specify first corner of viewport: **P2**

Specify opposite corner of viewport: **P3**

Enter view name: **Top**

(The view must be named.)

Command:

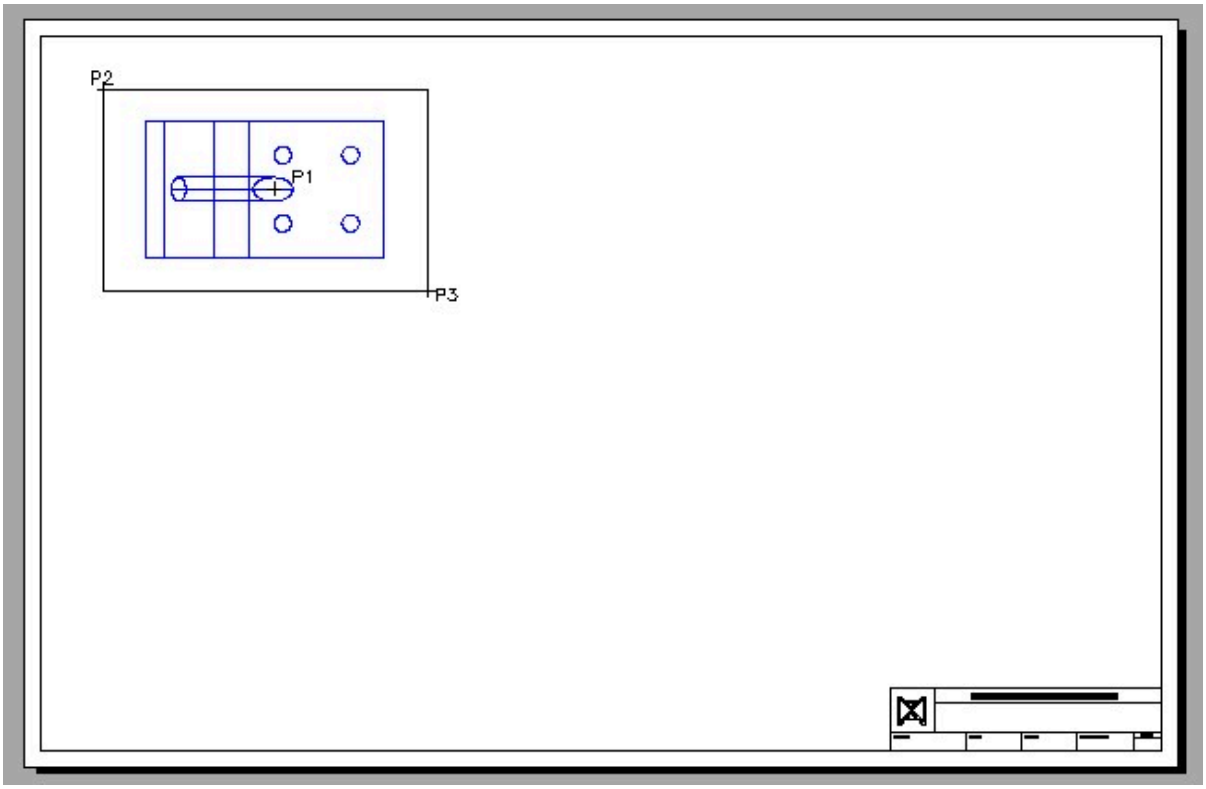


Figure Step 9

AUTHOR'S COMMENTS: You can select the locations P1, P2, and P3 by eye.

Step 10

Enter the SOLVIEW command again, but this time create the front view by projecting orthographically from the top view. Name it: Front. When done that way, the two views will be aligned. (Figure Step 10A and 10B)

Command: **SOLVIEW**

Enter an option [Ucs/Ortho/Auxiliary/Section]: **O**

Specify side of viewport to project: *(mid)*

Specify view center: **P4**

Specify view centre <specify viewport>:

Specify first corner of viewport: **P5**

Specify opposite corner of viewport: **P6**

Enter view name: **Front**

Command:

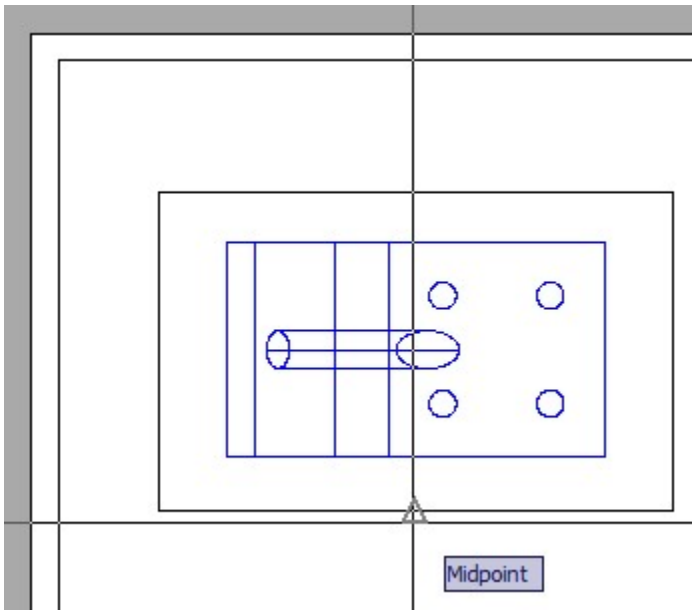


Figure Step 10A

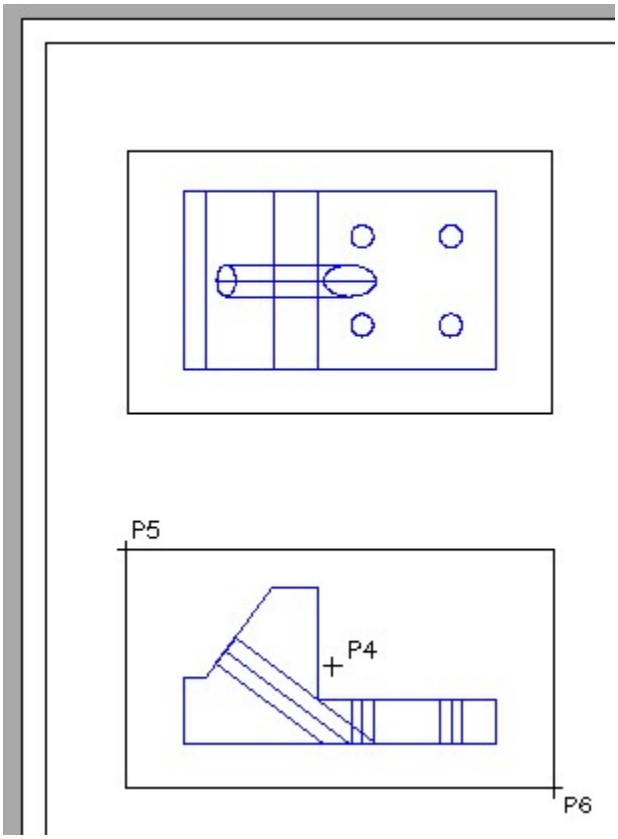


Figure Step 10B

Step 11

Using what you just learned, create the right side view and name it: Right. (Figure Step 11A and 11B)

Command: **SOLVIEW**

Enter an option [Ucs/Ortho/Auxiliary/Section]: **O**

Specify side of viewport to project:*(mid)*

Specify view center:

Specify view centre <specify viewport>:

Specify first corner of viewport:

Specify opposite corner of viewport:

Enter view name: **Right**

Enter an option [Ucs/Ortho/Auxiliary/Section]:

Command:

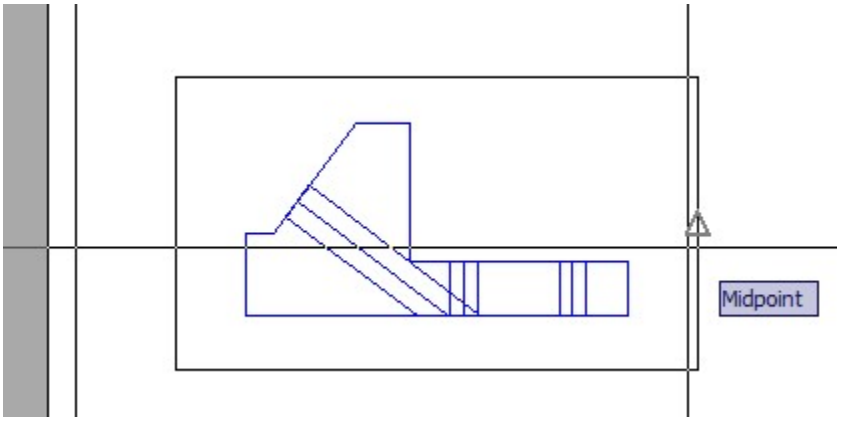


Figure Step 11A

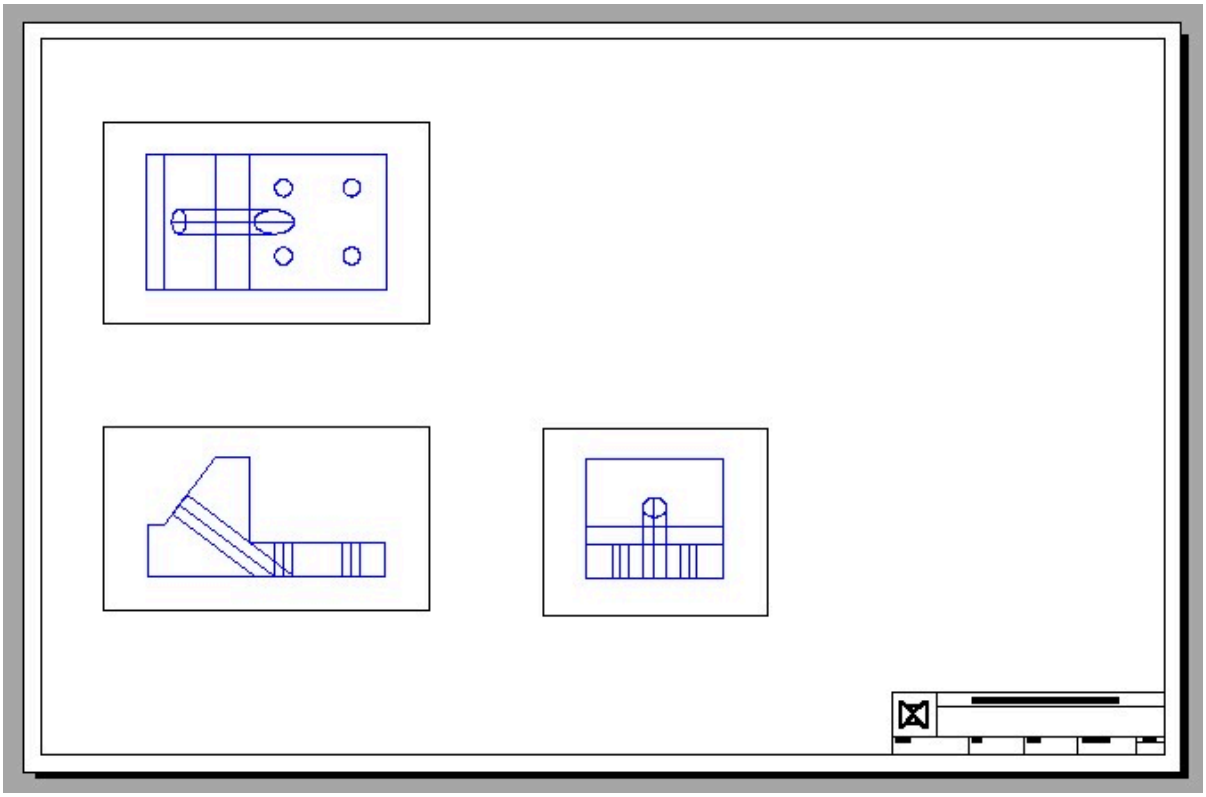


Figure Step 11B

Step 12

Use the MVIEW command to create a viewport in the top right corner of the drawing. Change the scale of the viewport to 1:1 and the current view to SE Isometric. Pan the model in the viewport to centre it. (Figure Step 12)

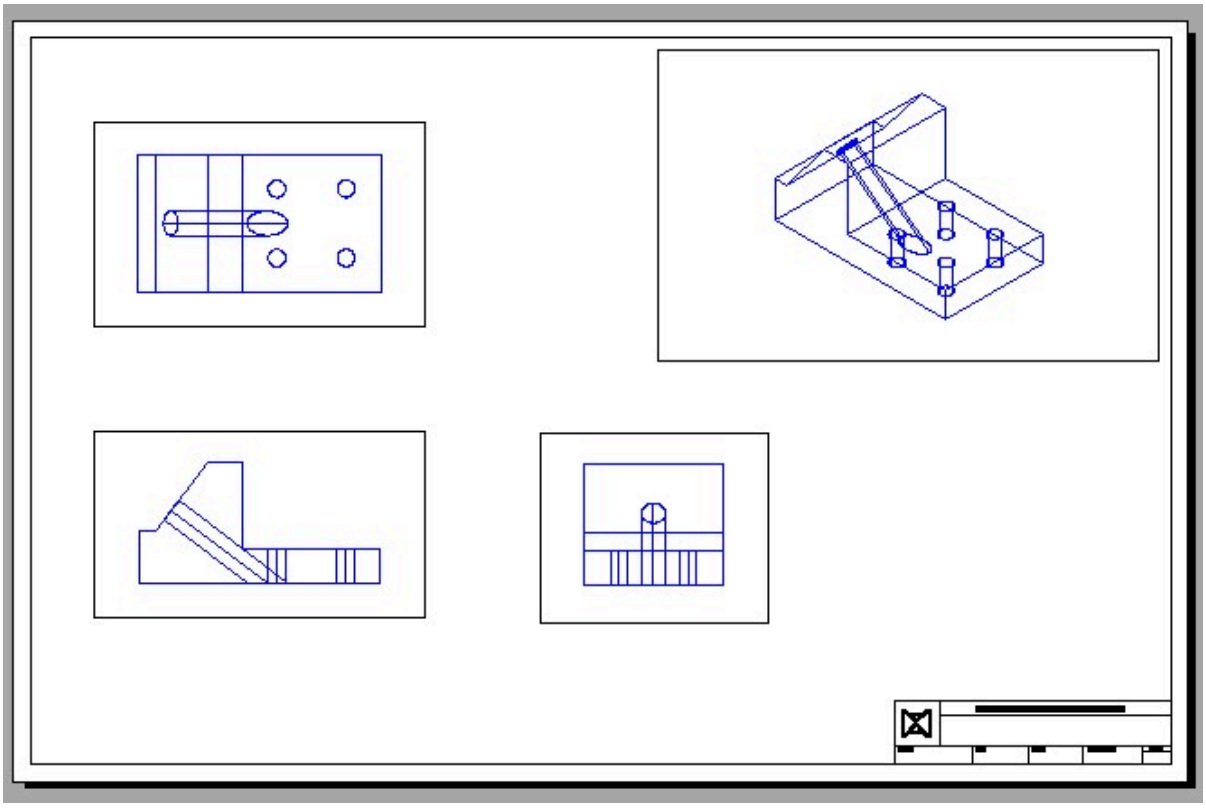


Figure Step 12

Step 13

Using the UNITS command, set the length precision to 8 decimal places. Open the Properties window and select all four viewports as shown in the figure. Check to ensure that the scale of all viewports are set to 1:1. Lock their display. (Figure Step 13)

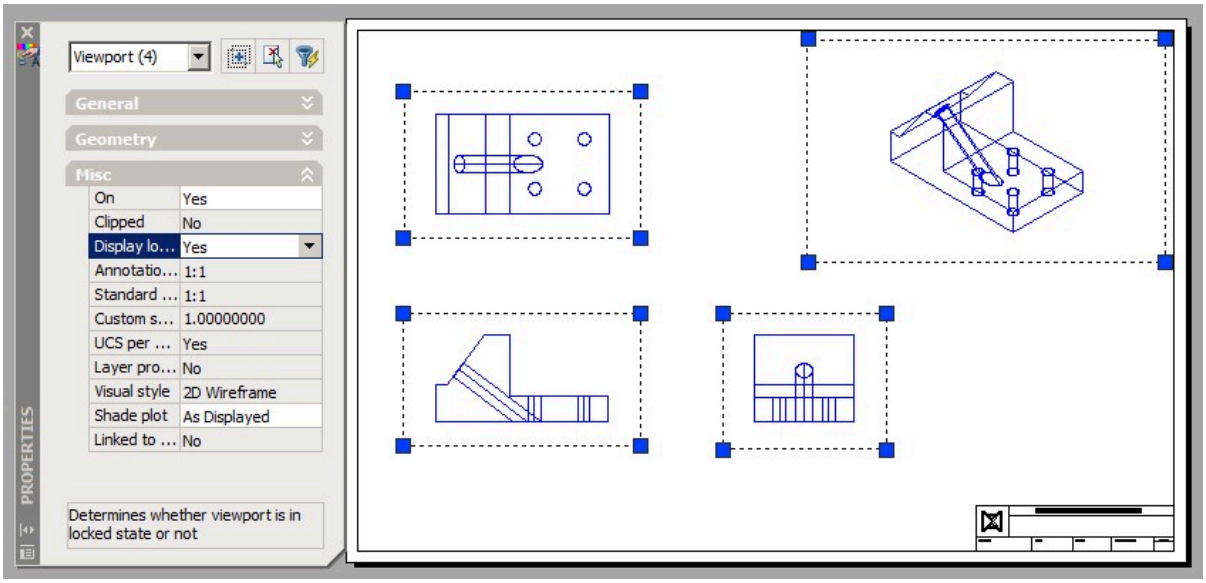


Figure Step 13

Step 14

Enter the SOLDRAW command, as shown below, to change the multiviews viewports to 2D. (Figure Step 14A and 14B)

Command: **SOLDRAW**

Select viewports to draw.

Select objects: 1 found

Select objects: 1 found, 2 total

Select objects: 1 found, 3 total

(Select the three multiview viewports.)

Select objects:

One solid selected.

One solid selected.

One solid selected.

Command:

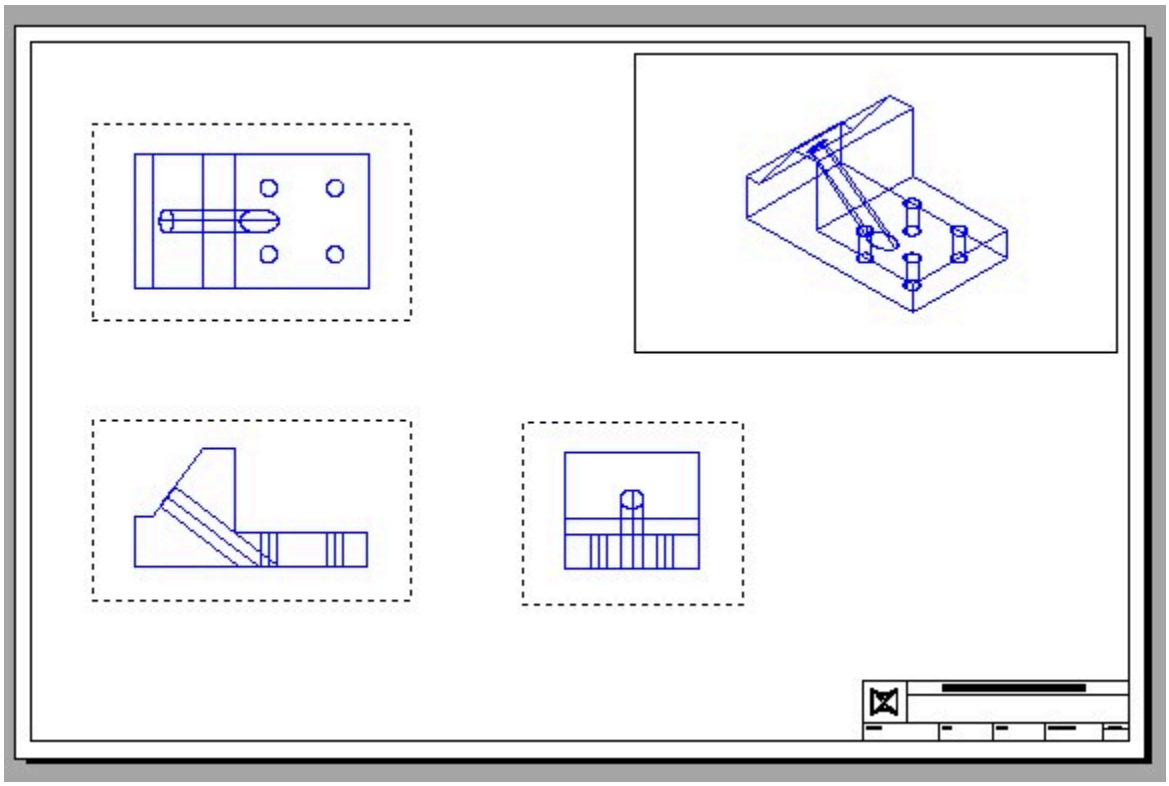


Figure Step 14A

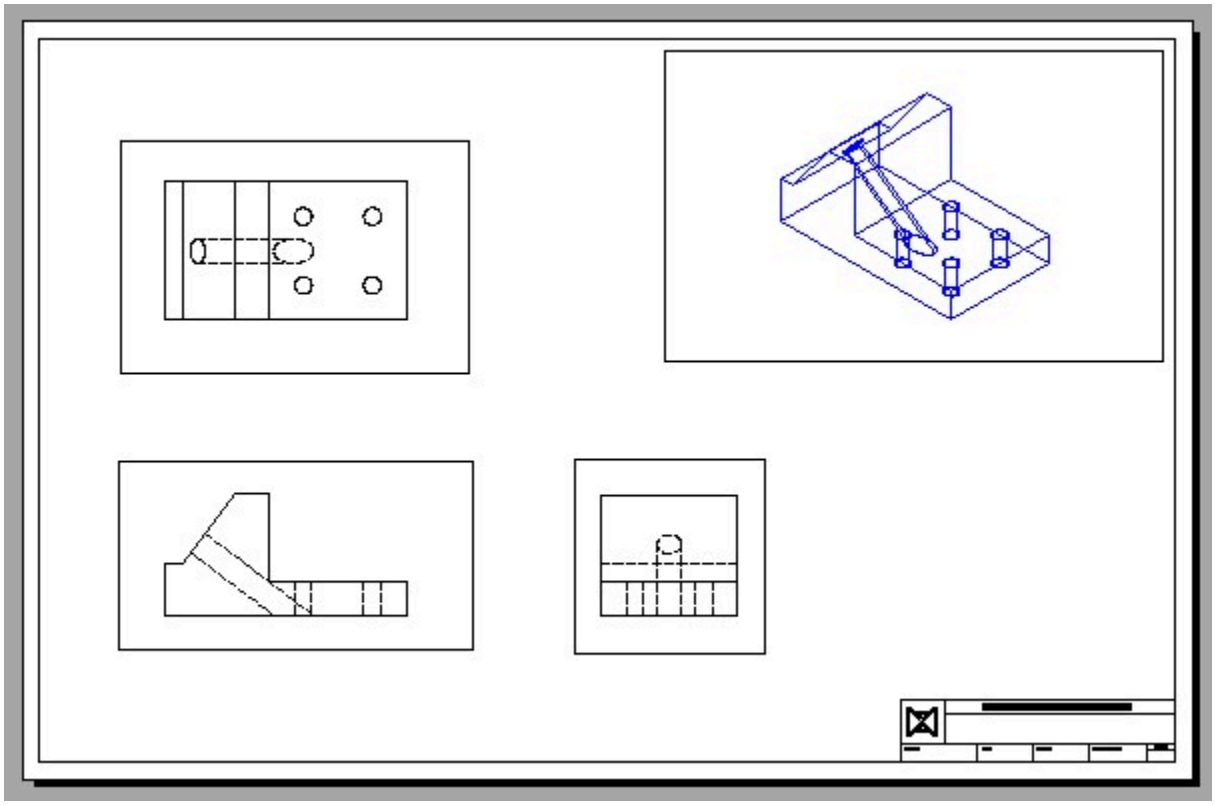


Figure Step 14B

Step 15

Change to current space to Model and select the isometric view as the current viewport. Enter the SOLPROF command as shown below. (Figure Step 15A and 15B)

Command: **SOLPROF**

Select objects: 1 found

(Select the solid model.)

Select objects:

Display hidden profile lines on separate layer? [Yes/No] <Y>:

(Select the default.)

Project profile lines onto a plane? [Yes/No] <Y>:

(Select the default.)

Delete tangential edges? [Yes/No] <Y>:

(Select the default.)

Enter an option

Command:

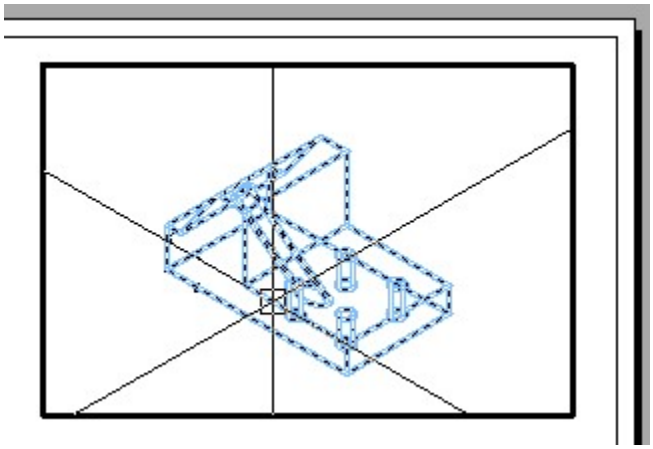


Figure Step 15A

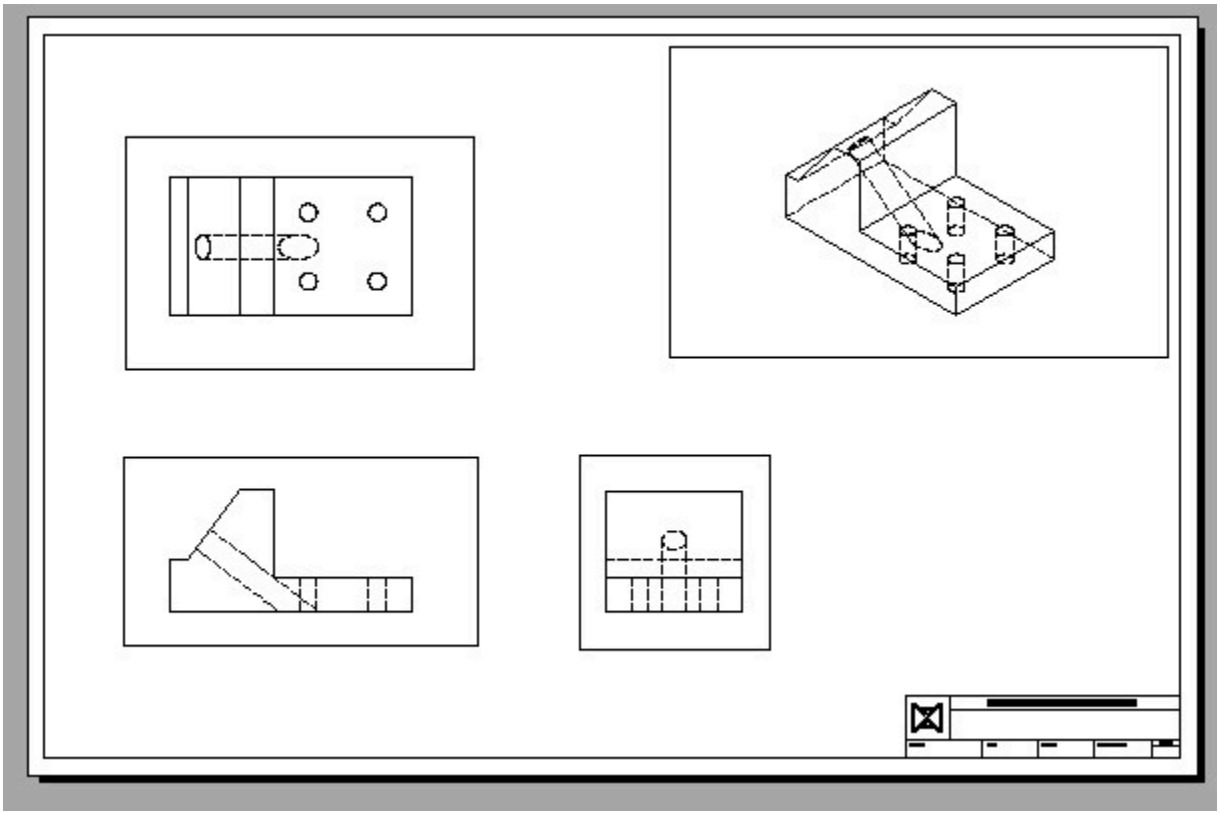


Figure Step 15B

Step 16

Change the colors of the layers as shown in the figure. (Figure Step 16)

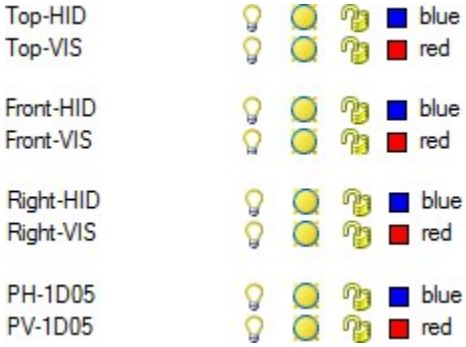


Figure Step 16

Step 17

Turn Layer: VPORT off and your completed drawing should appear similar to the figure. (Figure Step 17)

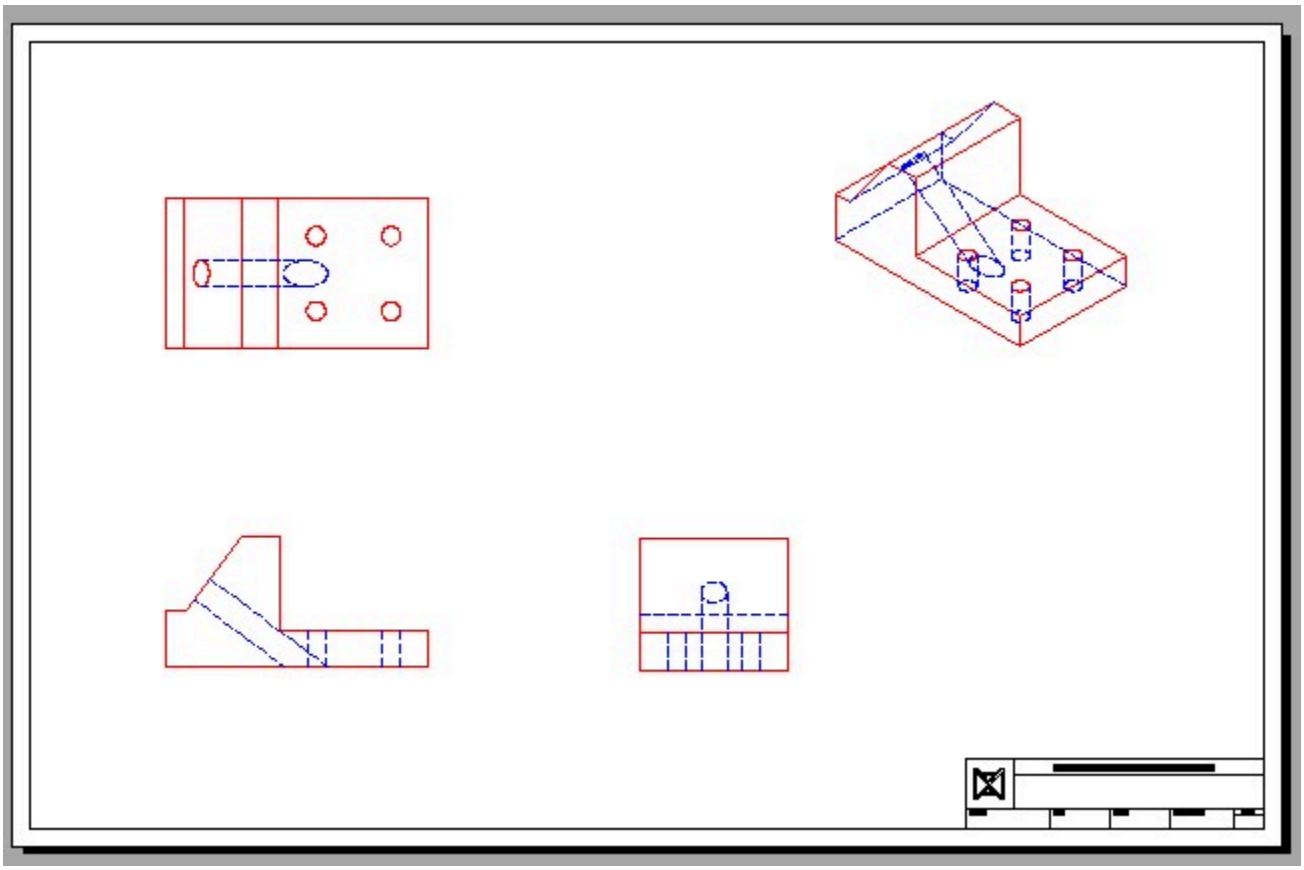


Figure Step 17

Step 18

Save and close the drawing.

Key Principles

Key Principles in Module 30

1. Before you use the SOLVIEW and the SOLPROF commands, ensure that the current visual style is 2D Wireframe and the Hidden linetype is loaded into the current drawing.

Lab Exercise 30-1

Time allowed: 45 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 30-1	3D English Layout	Inches

Step 1

Start a new drawing using the template shown above.

Step 2

Draw the plines on layer: Pline and the solid model on layer: Solid 3.

Step 3

Using the figures as a reference, draw the solid model. (Figure Step 3A, 3B, and 3C)

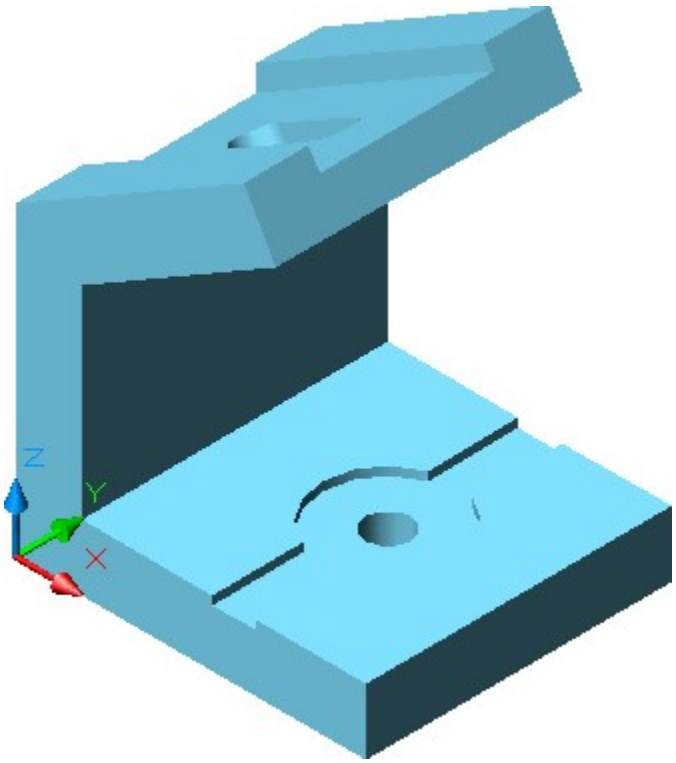


Figure Step 3A
The Solid Model – SE Isometric View

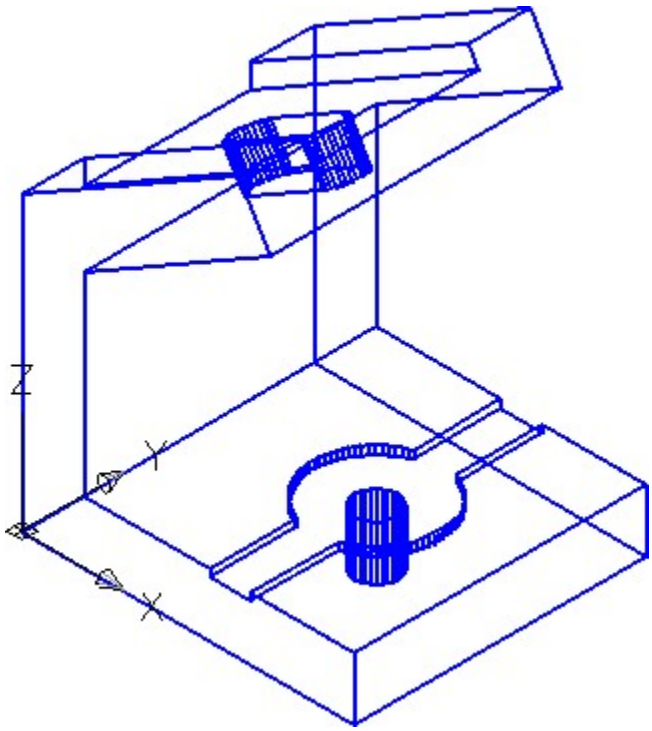


Figure Step 3B
Visual Style – 2D Wireframe

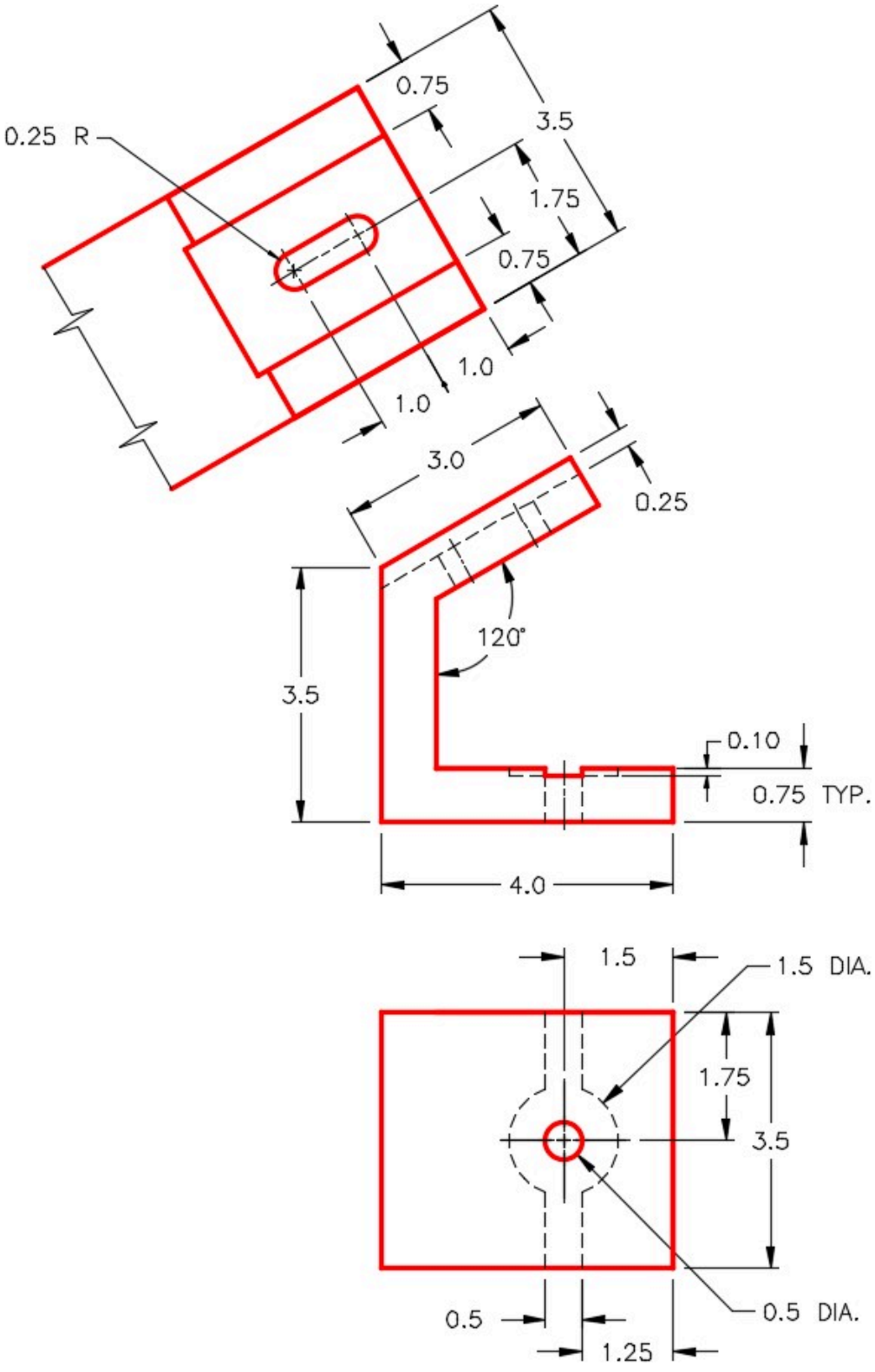


Figure Step 3C
Dimensioned Multiview Drawing

Step 4

Enable layout: Module Layout D and using the command SOLVIEW, create the three multiviews. Set the scale of the viewports to 1.5:1 and lock their display.

Step 5

Use the SOLDRAW command to change the views to 2D.

Step 6

Using the MVIEW command, create two views and set their view to SE Isometric. Scale the viewports to 1:1 and lock their display. Set one of the views to display shaded.

Step 7

Use the SOLPROF command to create the hidden lines in the isometric viewport. (Figure Step 7)

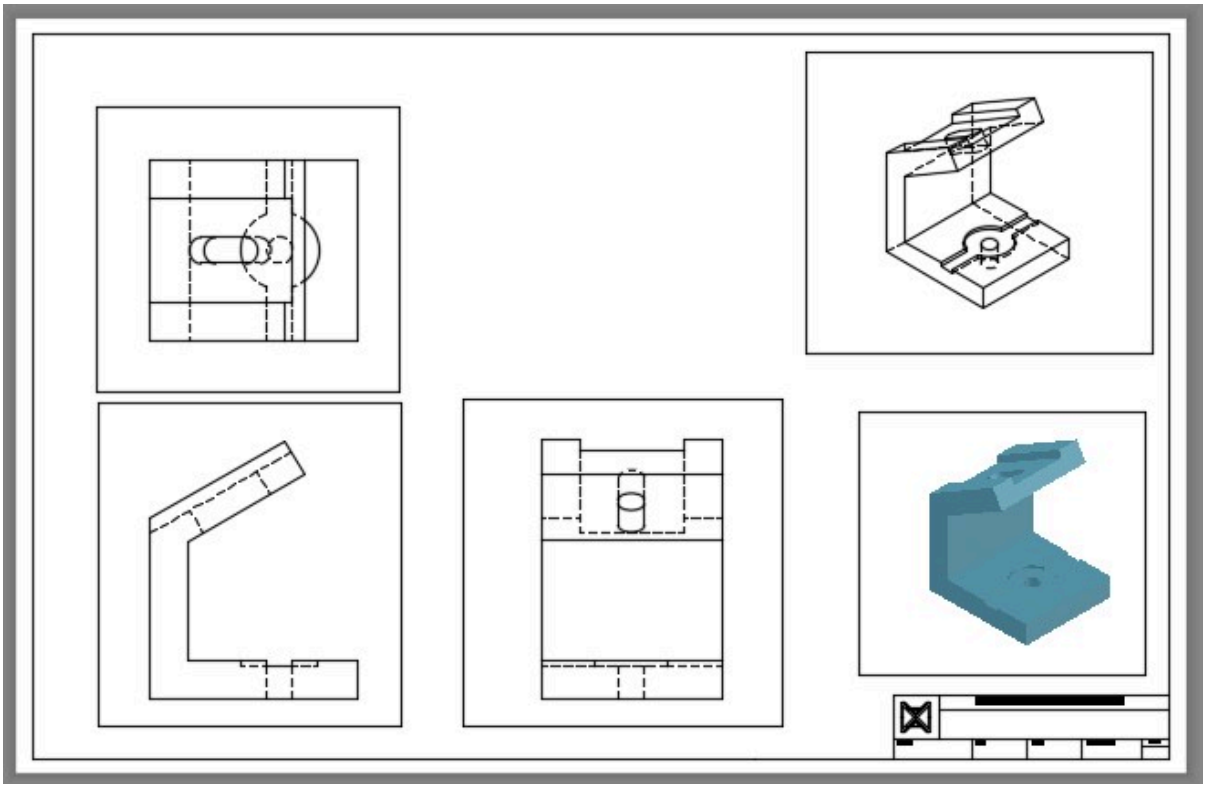


Figure Step 7

Step 8

Change the color of the layers to Red for object lines and Blue for the hidden lines. (Hint: See Step 16 of WORKALONG: Creating 2D Drawings from a Solid Model)

Step 9

Fill in the titleblock in paper space.

Step 10

Turn layer: VPORT off. (Figure Step 10)

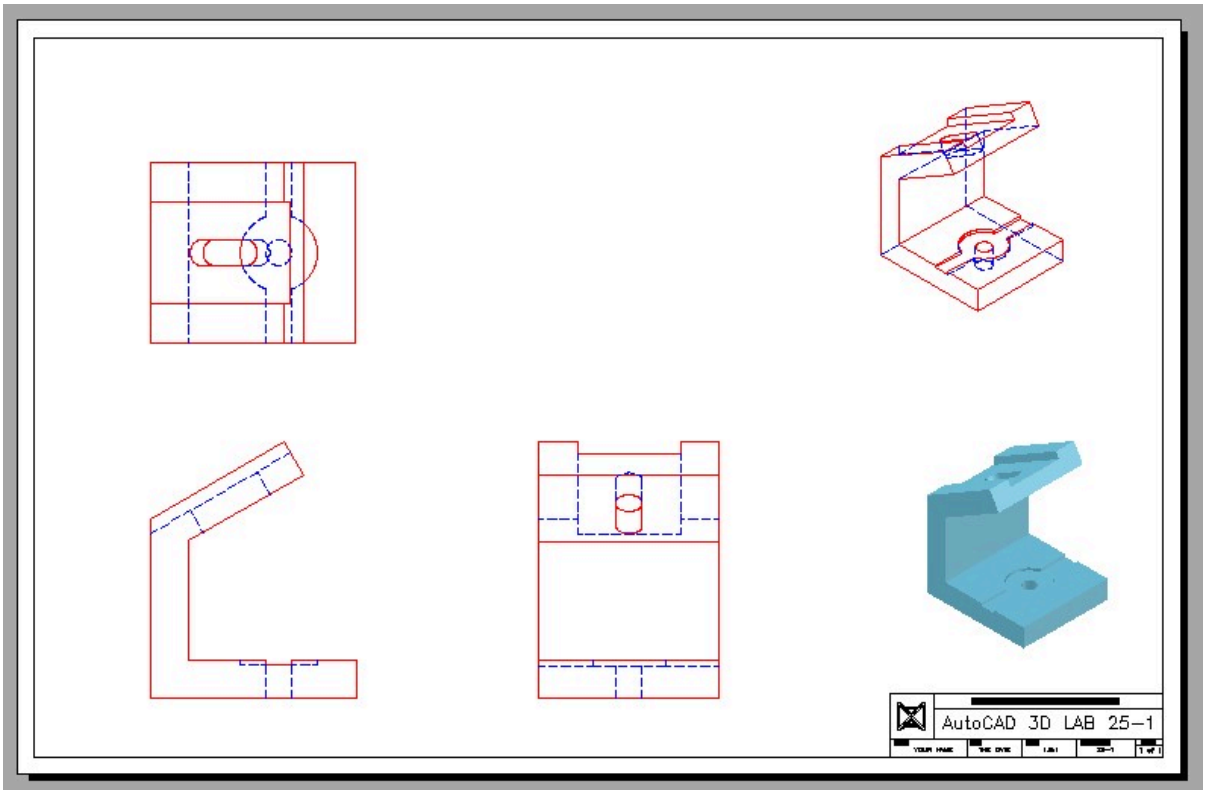


Figure Step 10

Step 11

Save and close the drawing.

Module 31 Competency Test No.6 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 exercises without the aid of a key.

The AutoCAD 3D 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 three terms, of a 3D Solid, must the user be able to identify and select when using the SOLIDEDIT command?
    A. bodies, faces and solids
    B. faces, edges and regions
    C. bodies, edges and wireframes
    D. bodies, faces and edges
    E. edges, solids and regions
2. What visual style must be current when the SOLVIEW command is used to create a 2D multiview(s) of a solid model?

A. Realistic
B. 2D Wireframe
C. Hidden
D. 3D Wireframe
E. Any visual style

3. Which one of the following item(s) must be set in the current drawing before the SOLDRAW and SOLPROF commands are used? Select the best answer.
    A. The Hidden linetype is loaded.
    B. The current visual style is 2D Wireframe.
    C. The current visual style is 3D Wireframe.
    D. A and B
    E. A and C
4. What command is used to find out if two or more solids occupy the same 3D space?
    A. SAMESPACE
    B. SUBTRACT
    C. INTERFERE
    D. UNION
    E. EXPLODE
5. Which SOLIDEDIT command option is used to divide two non-touching solid models back into individual solid models that have been joined with the UNION command?
    A. Separate
    B. Extrude
    C. Imprint
    D. Shell
    E. Check
6. Which SOLIDEDIT command option is used to attach a 2D object onto an existing solid model?
    A. Imprint
    B. Extrude
    C. Separate
    D. Shell
    E. Check
7. What two commands are used to a create 2D viewport(s), complete with hidden lines, of a solid model?

A. SOLVIEW and MSPACE

B. SOLVIEW and SOLDRAW

C. MVIEW and SOLVIEW

D. SOLPROF and MS

E. MVIEW and PSPACE

8. Which SOLIDEDIT command option is used to create a hollow cavity, with a specified wall thickness, in a solid model?

A. Imprint

B. Extrude

C. Separate

D. Shell

E. Check

9. What linetype must be loaded into the current drawing when using the SOLVIEW and SOLPROF commands?

A. hidden

B. center

C. continuous

D. dashed

E. There is no specific linetype that has to be loaded.

10. Which one of the following terms does the SOLIDEDIT command use when it prompts the user to select a solid model?

A. Model

B. Face

C. Body

D. Solid

E. Edge

Lab Exercise 31-1

Time Allowed: 3 Hours

Drawing Name	Template	Units
AutoCAD 3D Lab 31-1	3D Layout English	Inches

Step 1

Start a new drawing using the template shown above. Save the drawing with the name: AutoCAD 3D Lab 31-1.

Step 2

Draw the polylines on layer: Pline.

Step 3

On layer: Solid 1, draw a solid model of the object shown in the figures. (Figure Step 3A and 3B)

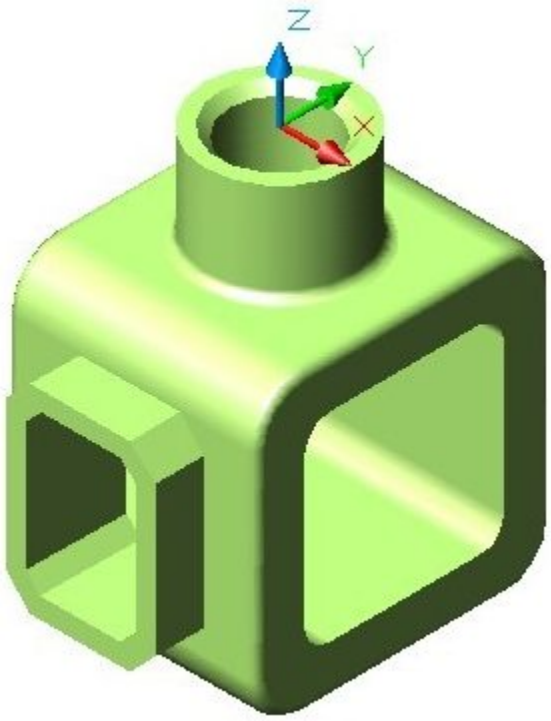


Figure Step 3A
Completed Solid Model – SE Isometric View

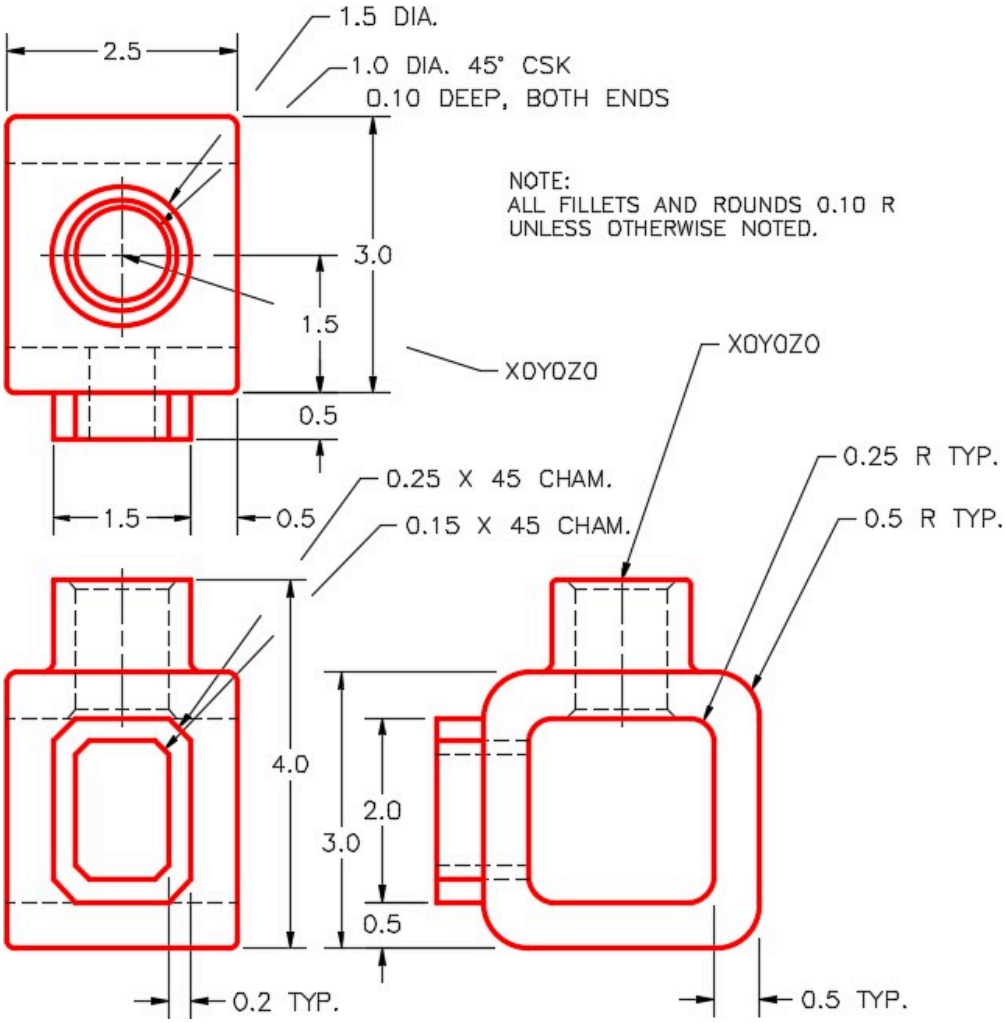


Figure Step 3B
Dimensioned Multiview Drawing

Step 4

On layout: Module Layout C, use the SOLVIEW command to create the three viewports of the model.

Step 5

On layer: VPORT, using the MVIEW command to create viewport. Set the view, in the viewport, to SE Isometric and the visual style Realistic.

Step 6

Set the scale of all four viewports to 1.25 and lock their display. (Figure Step 6)

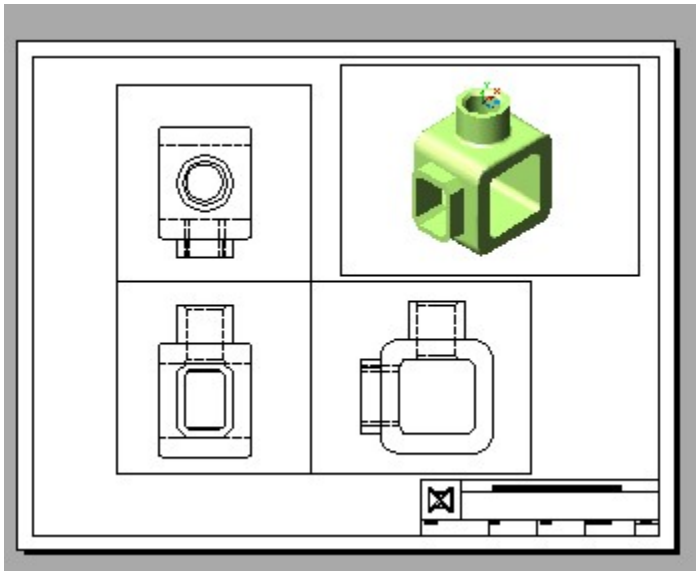


Figure Step 6

Step 7

Use the SOLDRAW command to change the three multiviews views to 2D views.

Step 8

Set the current space to Paper and fill in the titleblock. (Figure Step 8)

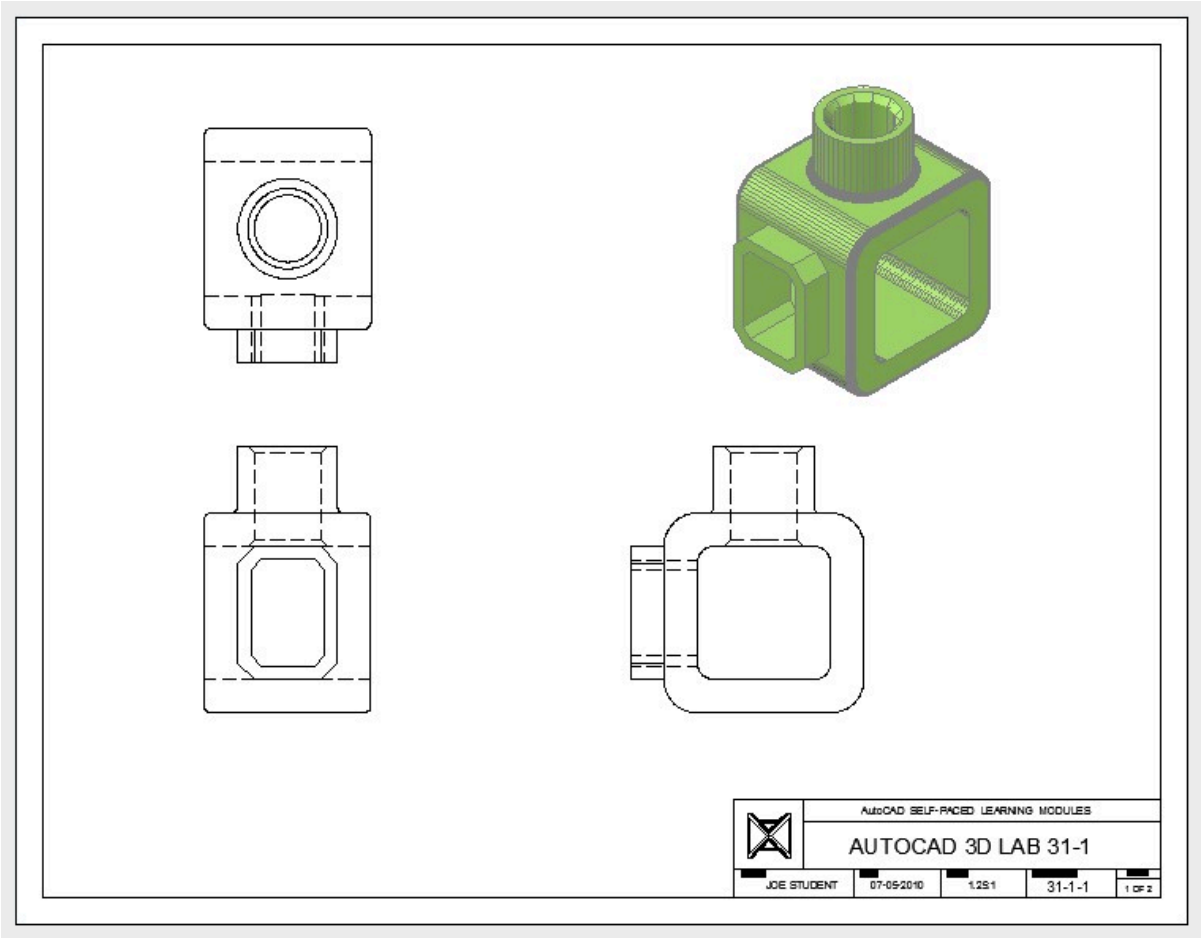


Figure Step 8

Step 9

On layout: Module Layout D, using the SOLVIEW and SOLDRAW commands, create the three multiviews of the model.

Step 10

On layer: VPORT, using the MVIEW command, create two viewports. Set the visual style to Realistic and orbit them to match the figure the best you can by eye.

Step 11

Set the scale to 2:1 for the multiviews and 1.5:1 for the isometric views. Lock their display.

Step 12

Change the color of the layers so that the object lines are red and the hidden lines are blue. (Hint: See Module 30, WORKALONG: Creating 2D Drawings from a Solid Model, Step 16)

Step 13

Turn layer VPORT off.

Step 14

Set the current space to Paper and fill in the titleblock. (Figure Step 14)

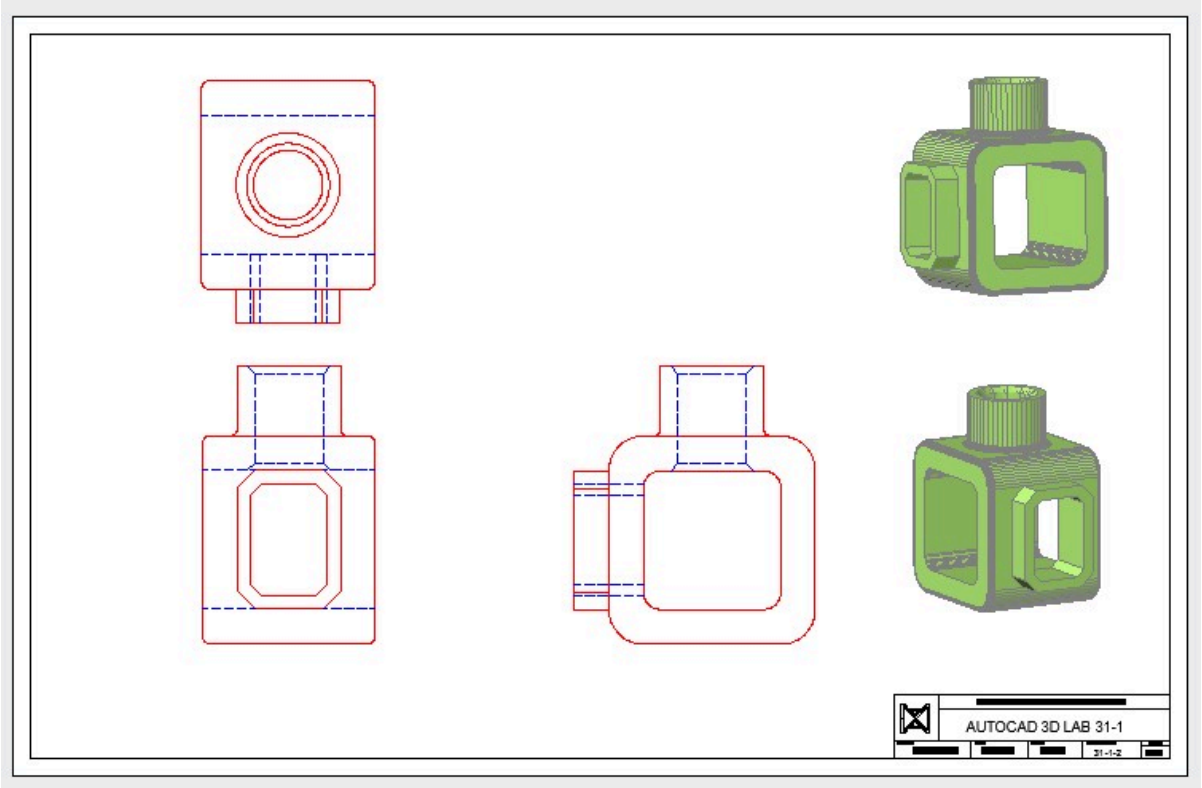


Figure Step 14

Step 15

Save and close the drawing.

Part 5

Module 22 Sweeping

Learning Outcomes

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

1. Describe and apply the SWEEP command to create solid models by sweeping a closed profile along a path.

Sweeps

Sweeps are created by sweeping (moving) one or more profiles along a selected path. See Figure 22-1 and 22-2. The *path* can be an open or closed drawing object but must be one object. If the profile is a closed object, the sweep will create a solid. If the profile is an open object, the sweep will create a surface.

Sweeps are used to create models like pipes, tubbing, drain pipes, gasket grooves, and threads.

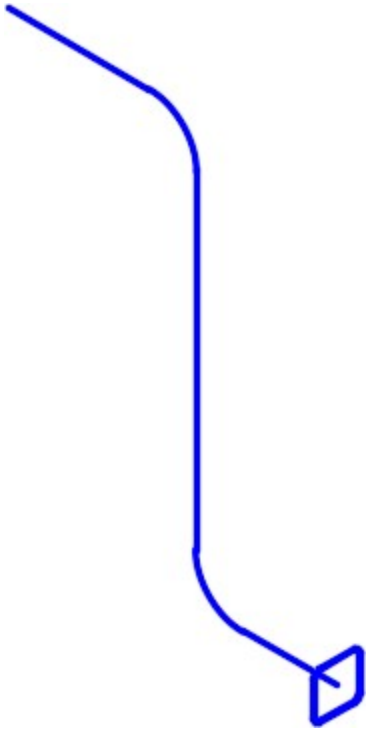


Figure 22-1
A Sweep Profile and Path

Objects that can be used for the path are 2D and 3D splines, 2D and 3D polylines, solids, surfaces and mesh edge sub-objects, helices, arcs, circles, ellipses, elliptical arcs and lines.

Objects that can be used for the profile are 2D and 3D splines, 2D polylines, 2D solids, 3D solid face subobjects, arcs, circles, ellipses, elliptical arcs, lines, regions, solids, surface mesh edge sub-objects and traces.

The system variable DELOBJ can be used to automatically delete or keep the original profile and path geometry used in the SWEEP command.

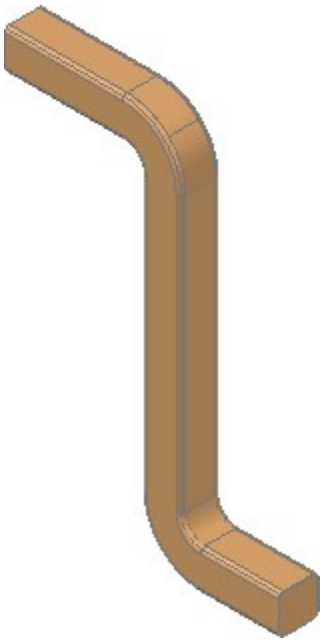
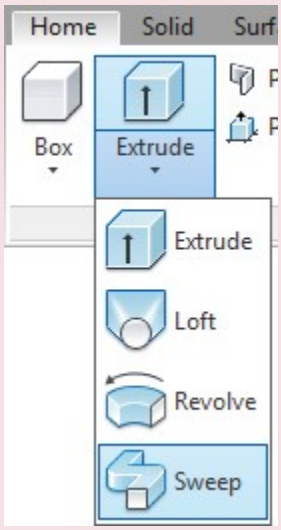


Figure 22-2
Completed Solid Model
After the Sweep is Applied

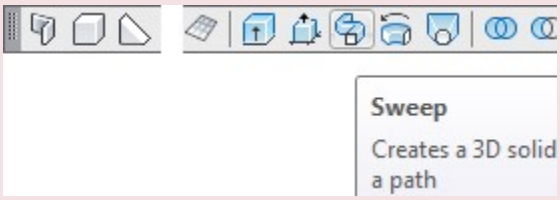
AutoCAD Command: SWEEP

The SWEEP command is used to create a 3D solid or surface model by sweeping a profile along a path.

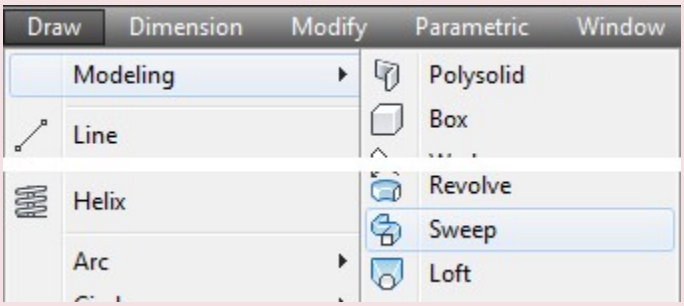
Shortcut: **none**



Home Ribbon



Modeling Toolbar



Draw Pull-down

WORK ALONG: Using the SWEEP Command – Part 1

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 22-1. (Figure Step 2A, 2B, and 2C)

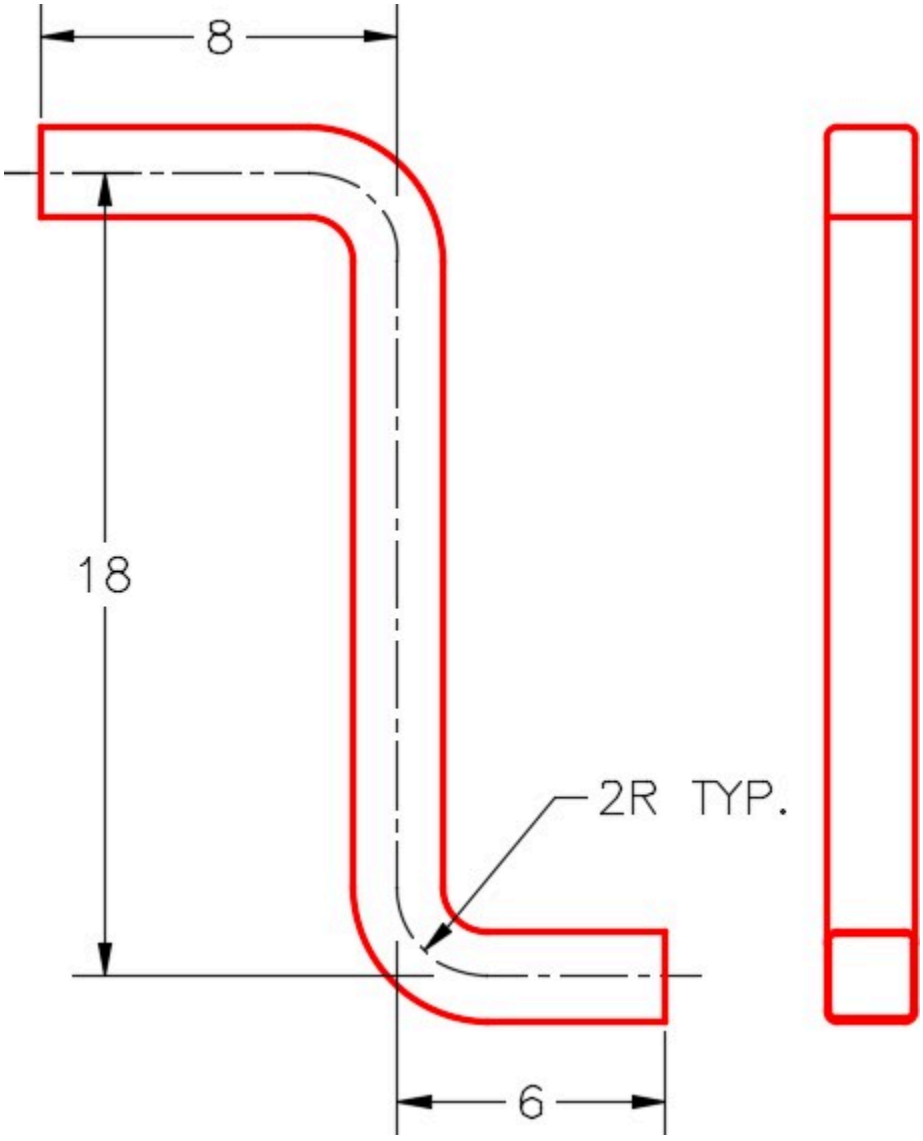


Figure Step 2A
Dimensioned Multiview Drawing

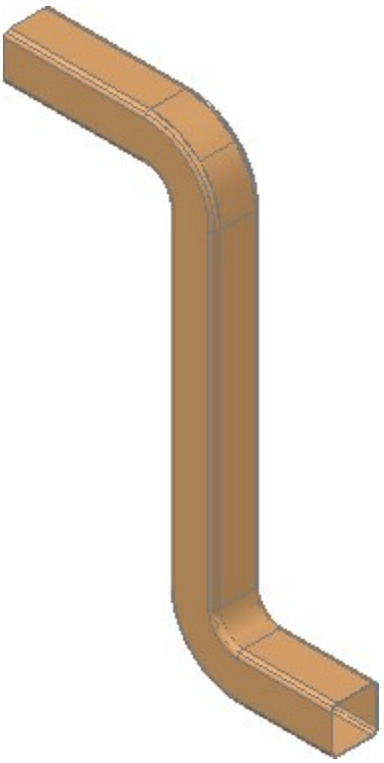


Figure Step 2B
Completed 3D Solid

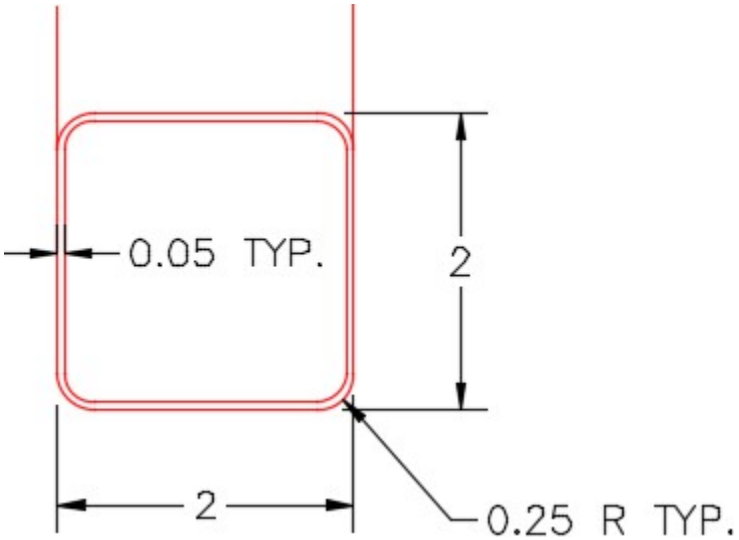


Figure Step 2C
Dimensioned Detail of Opening

Step 3

Ensure that the DELOBJ system variable is set to 0.

Step 4

Change the visual style to 3D Wireframe.

Step 5

Set layer: Path as the current layer, the current view to SE Isometric, and the current UCS to Front.

Step 6

Using Figure Step 2A as a reference, enable Ortho mode, enter the PLINE command, as shown below, to draw a pline on the front UCS. Start the pline at 0,0,0. (Figure Step 6A and 6B)

> **AUTHOR'S COMMENTS:** It is easy to draw the pline if you enable Ortho mode. Move the cursor along the applicable axis and enter the distance along that axis only.

Command: **PLINE**

Specify start point: **0,0,0**

Current line-width is 0.0000

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: <Ortho on> **6**

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **18**

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **8**

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]:

Command:

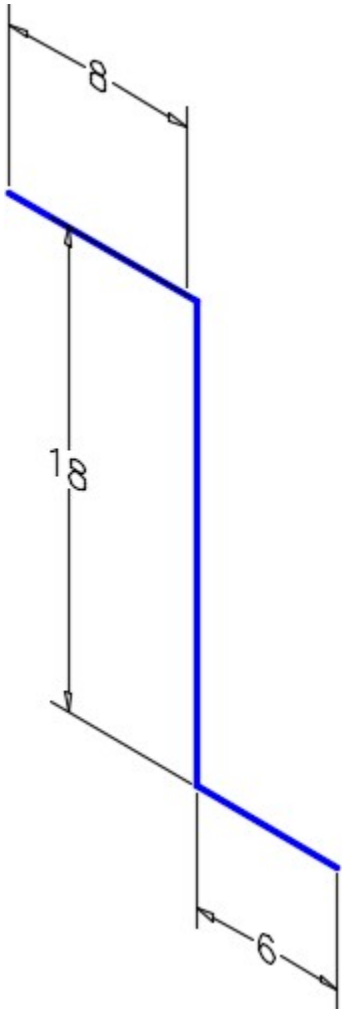


Figure Step 6A

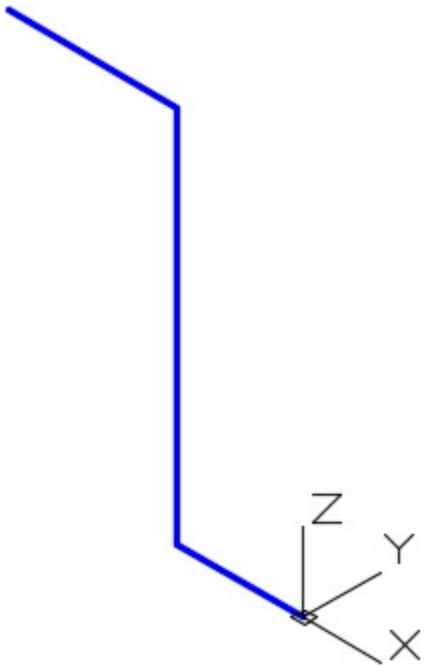


Figure Step 6B

Step 7

Using the Properties windows, check to ensure that the object is one polyline. (Figure Step 7)

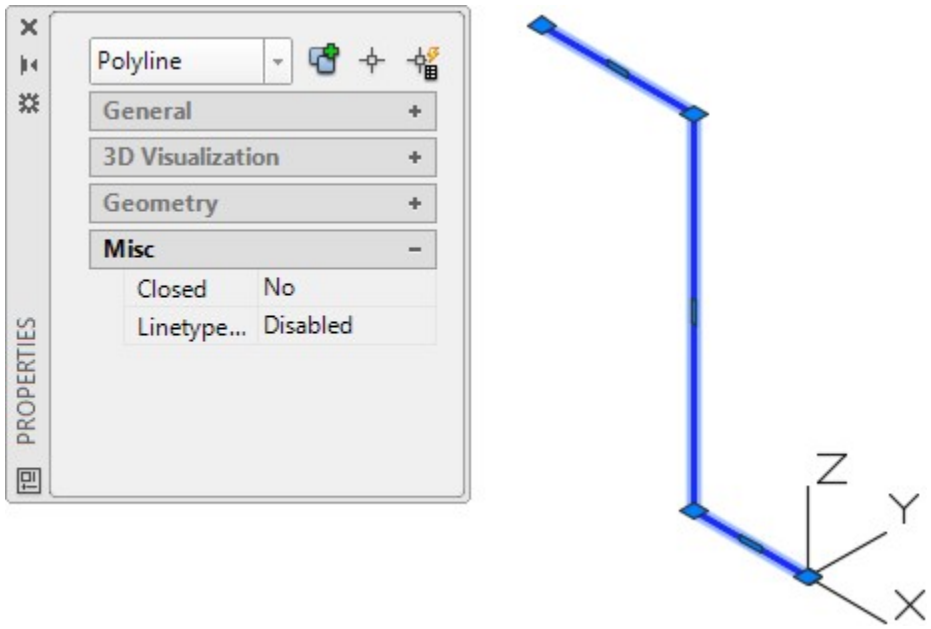


Figure Step 7

Step 8

Using the FILLET command and the Polyline option, as shown below, fillet the corners with a radius of 2. (Figure Step 8)

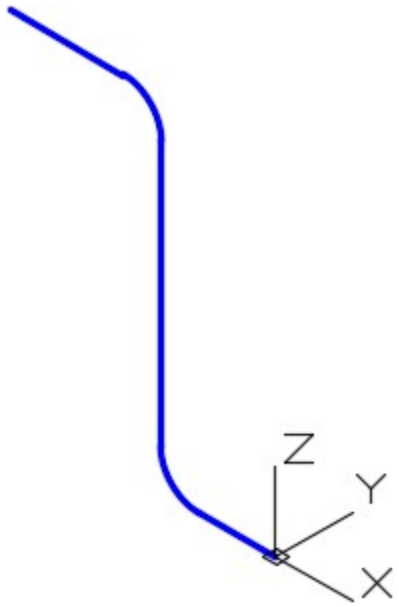


Figure Step 8

AUTHOR'S COMMENTS: All corners of a pline, with the same radius, can be filleted at one time using the Polyline option.

Command: **FILLET**

Current settings: Mode = TRIM, Radius = 0.0000

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **R**

Specify fillet radius <0.0000>: **2**

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **P**

Select 2D polyline:

(Select the polyline.)

2 lines were filleted

Command:

AUTHOR'S COMMENTS: The pline you just drew is the centerline of the object and the path for the

sweep.

Step 9

Set layer: Profile as the current layer and the current UCS to Right.

Step 10

Using the Figure Step 2C as a reference, draw a closed pline with its centre at the end of the path. The pline is 2 inches square. (Figure Step 10)

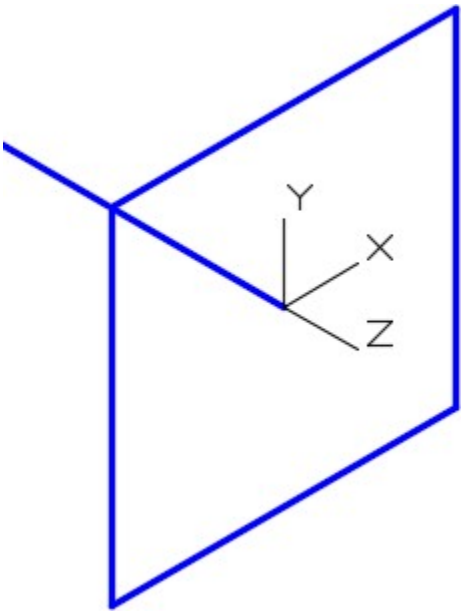


Figure Step 10

Step 11

Using what you learned earlier in the workalong, fillet the four corners of the pline with a radius of 0.25. (Figure Step 11)

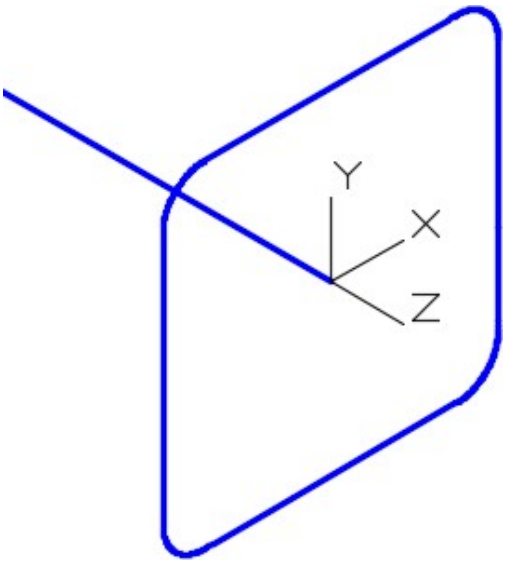


Figure Step 11

Step 12

Using what you learned earlier, use the Properties windows and insure that object is a closed polyline.

Step 13

Offset the pline, towards the inside, at a distance of 0.05 inches. (Figure Step 13A and 13B)

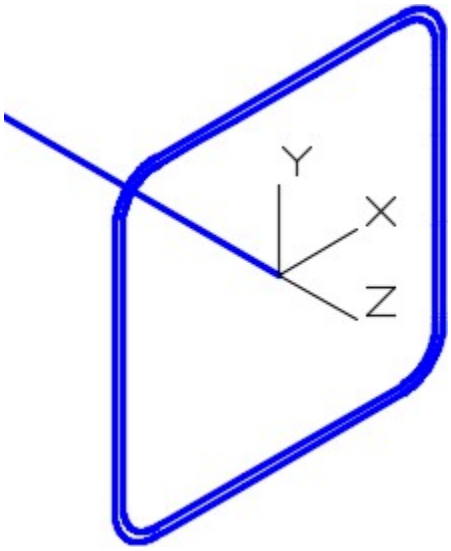


Figure Step 13A

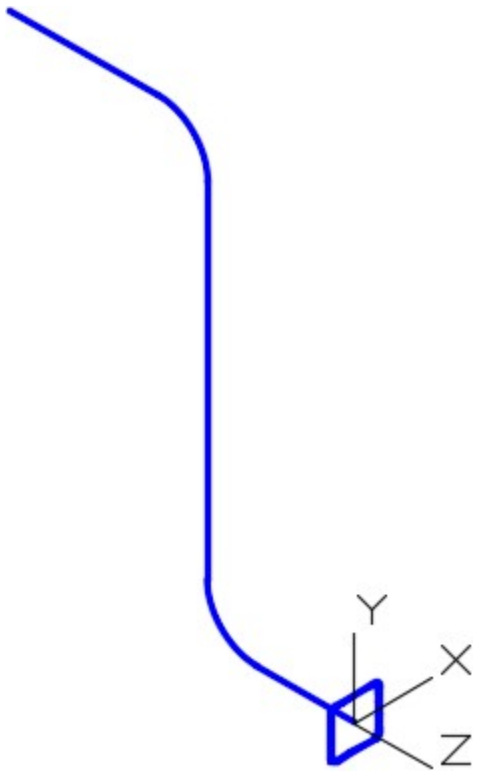


Figure Step 13B

Step 14

Set layer: Solid 4 as the current layer and the current visual style to Realistic.

Step 15

Use the SWEEP command, as shown below, to sweep both profiles along the path in one command. (Figure Step 15A and 15B)

Command: **SWEEP**

Current wire frame density: ISOLINES=4, Closed profiles creation mode = Solid

Select objects to sweep or [MOde]: 1 found

(Select one profile.)

Select objects to sweep or [MOde]: 1 found, 2 total

(Select the other profile.)

Select objects to sweep or [Mode]:

(Press Enter)

Select sweep path or [Alignment/Base point/Scale/Twist]:

(Select the path.)

Command:

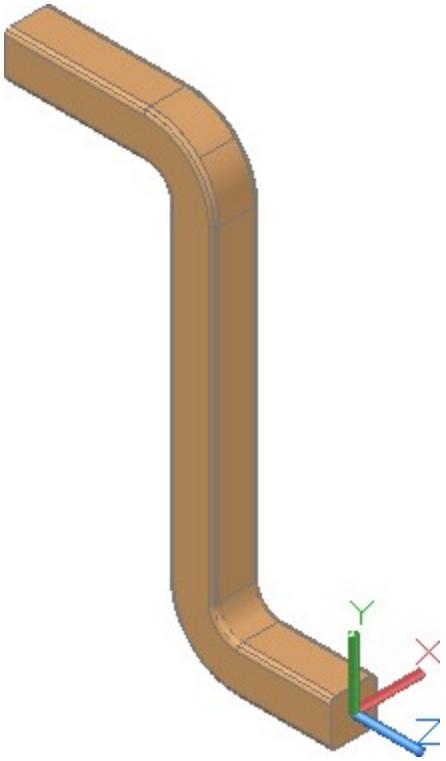


Figure Step 15A

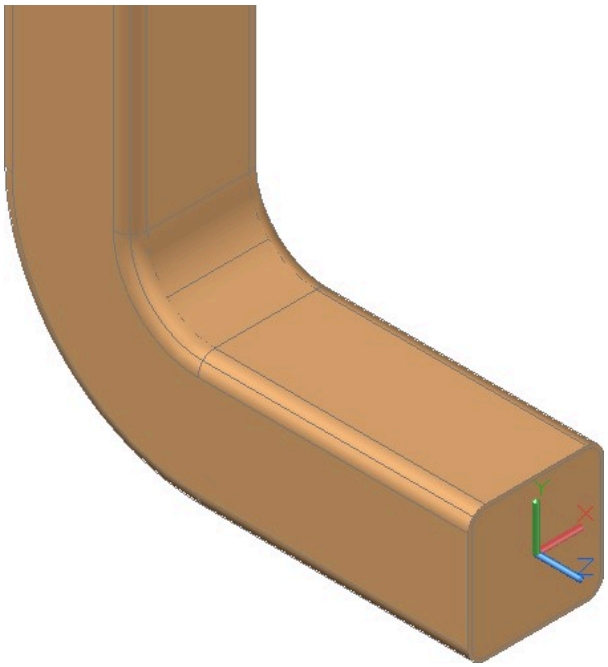


Figure Step 15B

AUTHOR'S COMMENTS: I created two solids, one inside the other, in one SWEEP command. You will see the reason for this in the following steps.

Step 15

Turn layers: Path and Profile off.

Step 16

Using what you learned earlier in the book, use the SUBTRACT command to subtract the inner solid from the outer solid. (Figure Step 16A, 16B, and 16C)

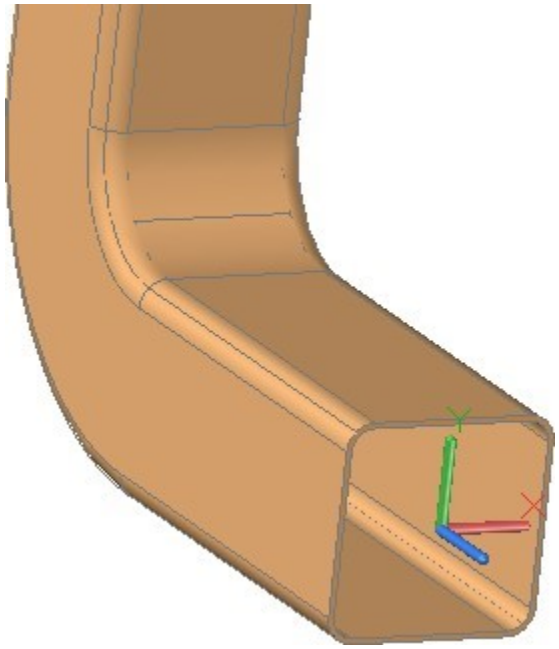


Figure Step 16A

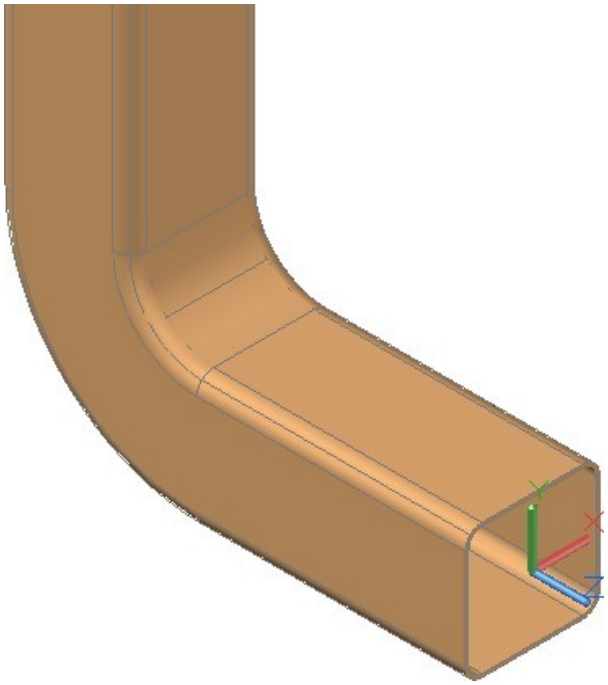


Figure Step 16B

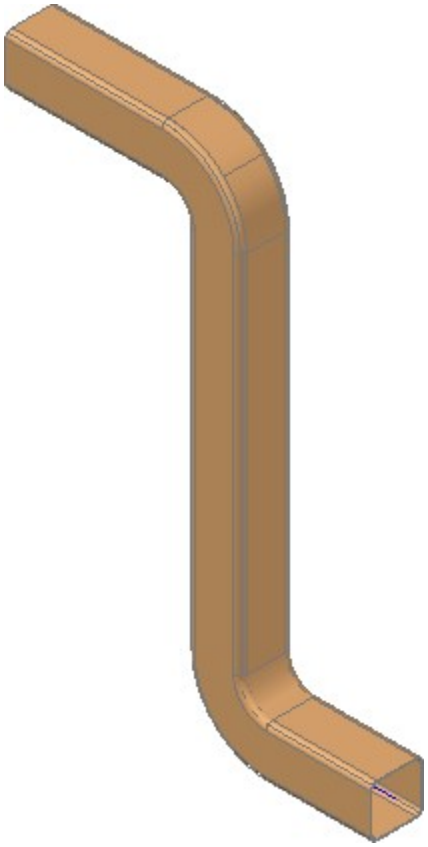


Figure Step 16C

AUTHOR'S COMMENTS: AutoCAD has an easier method, known as shelling, of creating a hollow solid without creating and subtracting an inner solid. Shelling will be taught in Module 29.

Step 17

Save and close the drawing.

WORK ALONG: Using the SWEEP Command – Part 2

Step 1

Using the NEW command, start a new drawing using template: 3D Layout Metric.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 22-2. (Figure Step 2)

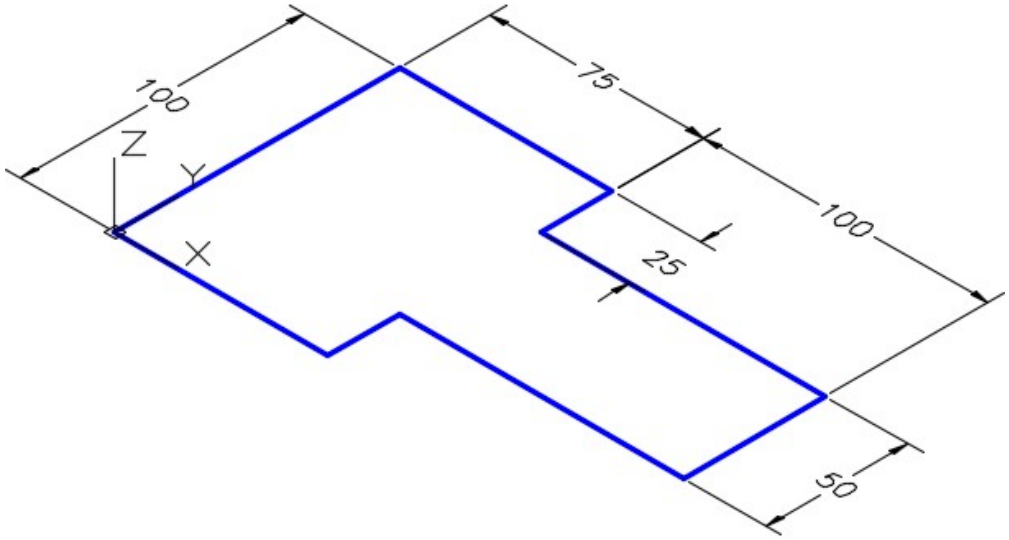


Figure Step 2
Dimensioned Drawing

Step 3

Set the current view to SE Isometric, the current UCS to World, the layer: Path as the current layer, and the current visual style to 2D Wireframe.

Step 4

Using Figure Step 2 as a reference, draw a closed pline as shown in the figure. (Figure Step 4)

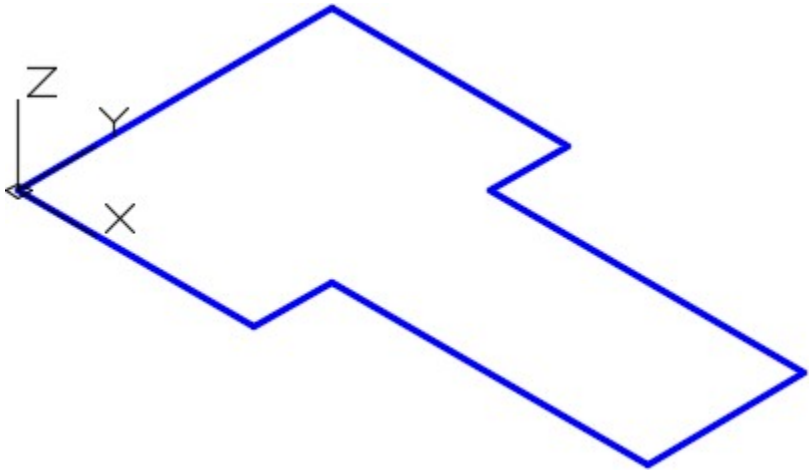


Figure Step 4

Step 6

Using what you learned in the first workalong, fillet all of the corners, in one step, with a radius of 10. (Figure Step 6)

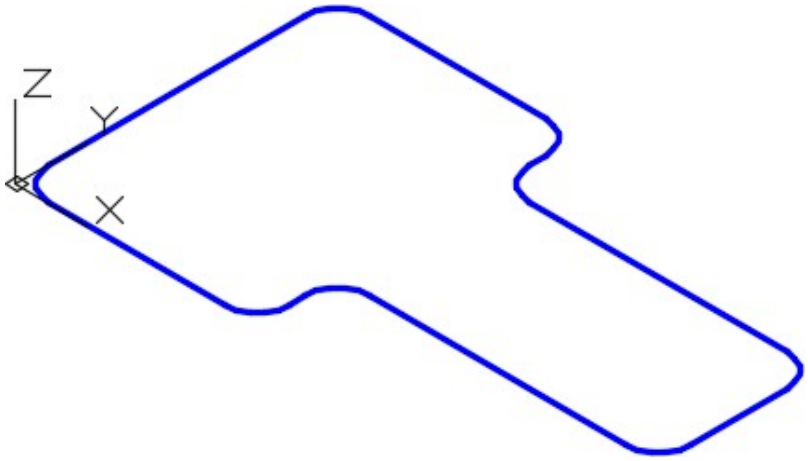


Figure Step 6

Step 7

Offset the closed pline 5 mm towards the inside. (Figure Step 7)

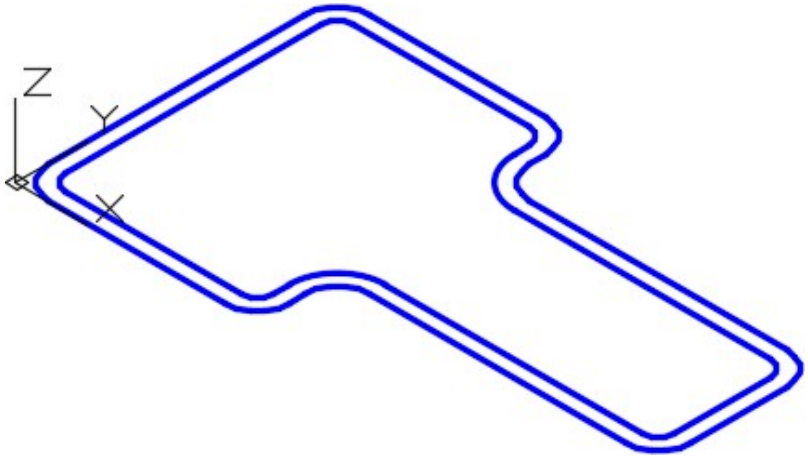


Figure Step 7

Step 8

Set the system variable ISOLINES to 24.

AUTHOR'S COMMENTS: The ISOLINES system variable controls the smoothness of the curved faces. See Module 17.

Step 9

Set layer: Solid 1 as the current layer. Extrude the outer pline 25 mm in the negative Z. (Figure Step 9)

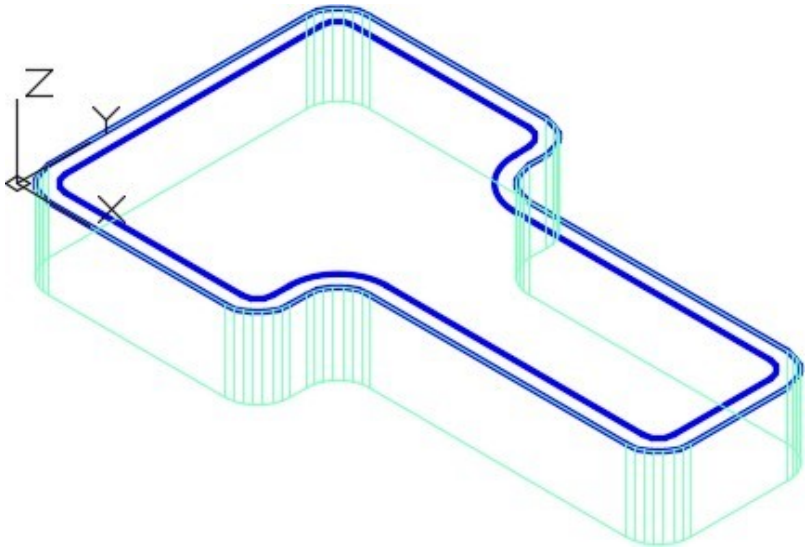


Figure Step 9

Step 10

Freeze layer: Solid Off. Change the layer of the extruded outer solid that you created in Step 9 to layer: Solid Off.

AUTHOR'S COMMENTS: I changed the solid to layer Solid Off so that the solid does not get in the way of extruding the second solid.

Step 11

Extrude the inner pline 20 mm in the negative Z. (Figure Step 11)

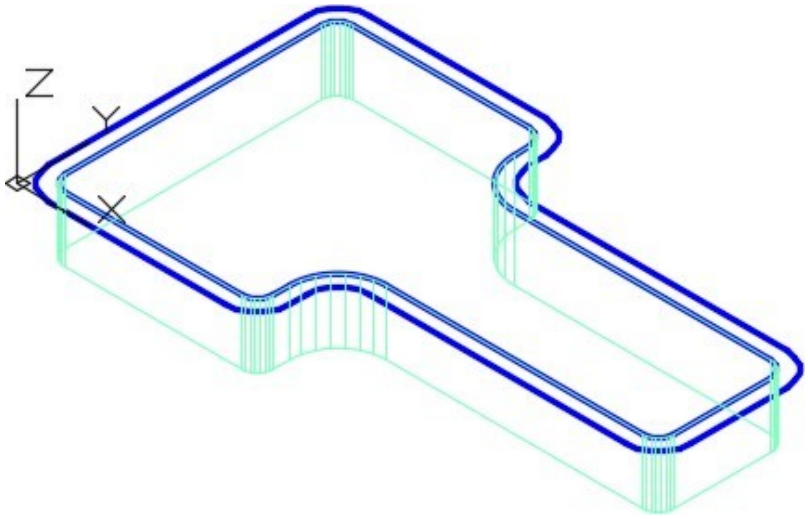


Figure Step 11

Step 12

Change the layer of the first solid from layer: Solid Off to layer: Solid 1. Turn layer: Path off. Your drawing should match the figure. (Figure Step 12)

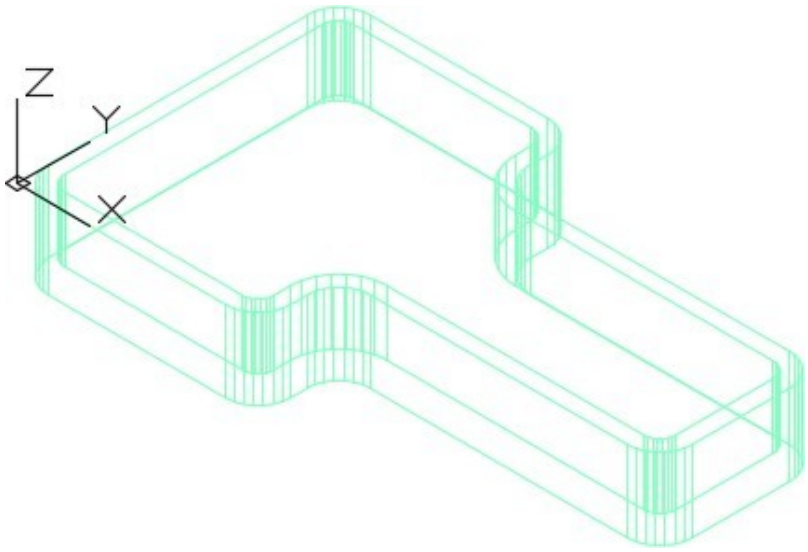


Figure Step 12

Step 13

Change the visual style to Realistic. (Figure Step 13)

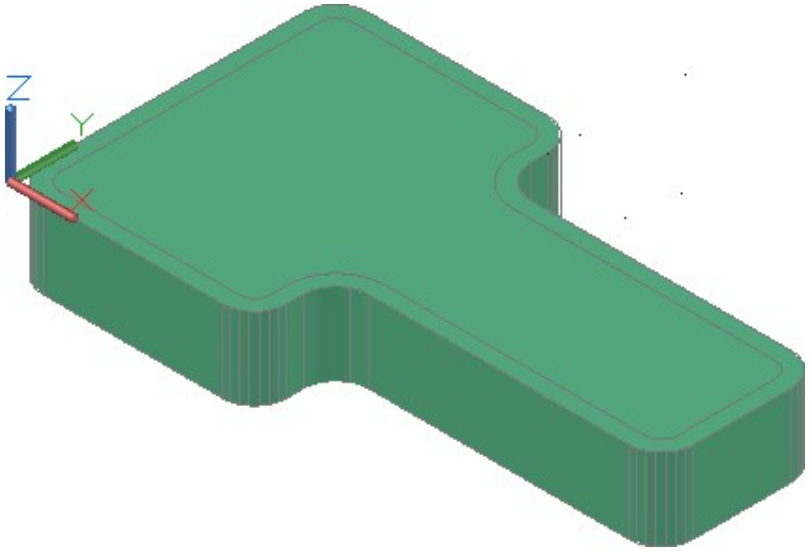


Figure Step 13

AUTHOR'S COMMENTS: When the visual style is realistic, it is easier to select the solids when you are using the SUBTRACT command.

Step 14

Using the SUBTRACT command, subtract the inner solid from the outer solid. (Figure Step 14)

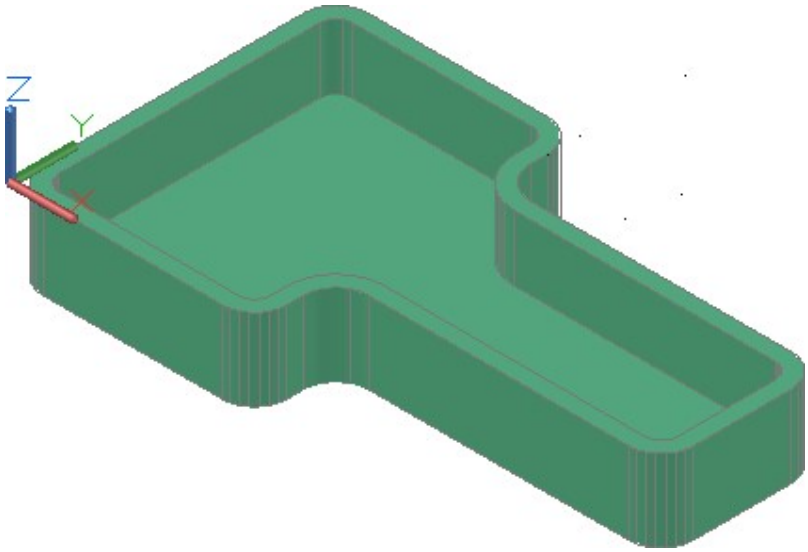


Figure Step 14

Step 15

Change the layer of the solid on layer: Solid 1 to layer: Solid Off. Turn layer: Path on and set layer: Profile as the current layer. Set the visual style to 2D wireframe. (Figure Step 15)

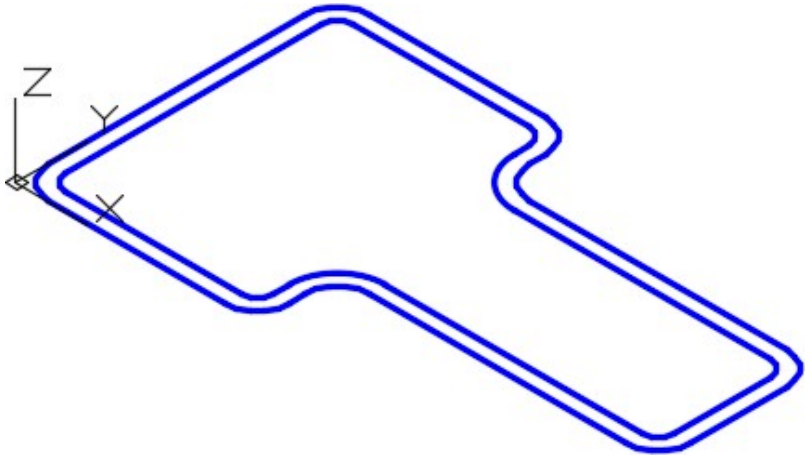


Figure Step 15

Step 16

Change the current UCS to Right.

Step 17

Draw a closed pline 2.5 x 2.5 inches. Start it by snapping to endpoint of the inner pline. (Figure Step 17A, 17B, and 17C)

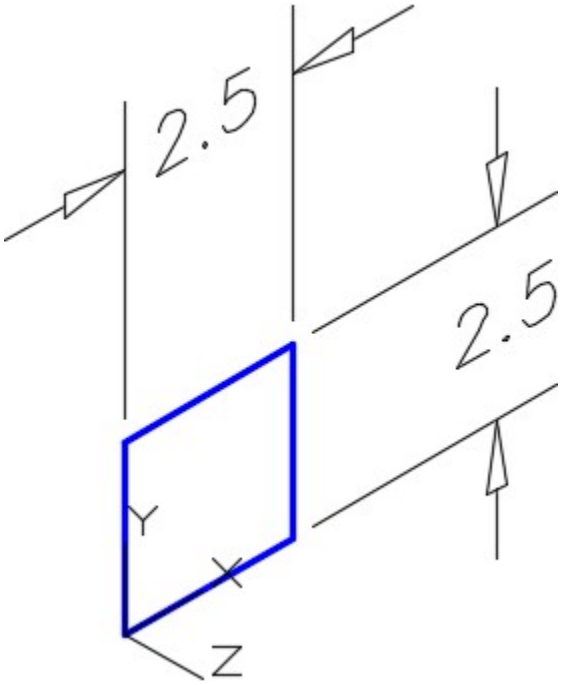


Figure Step 17A

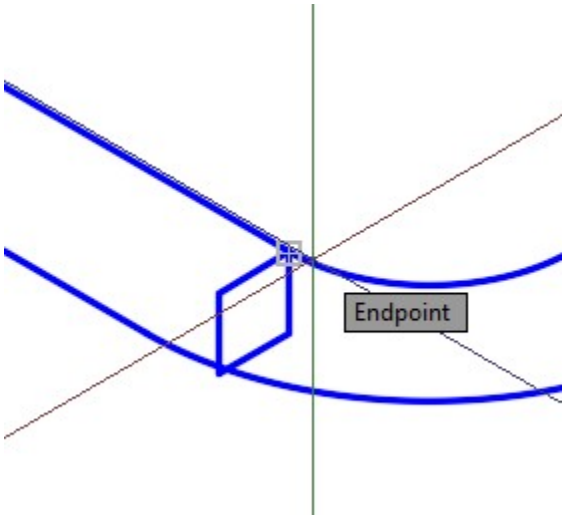


Figure Step 17B

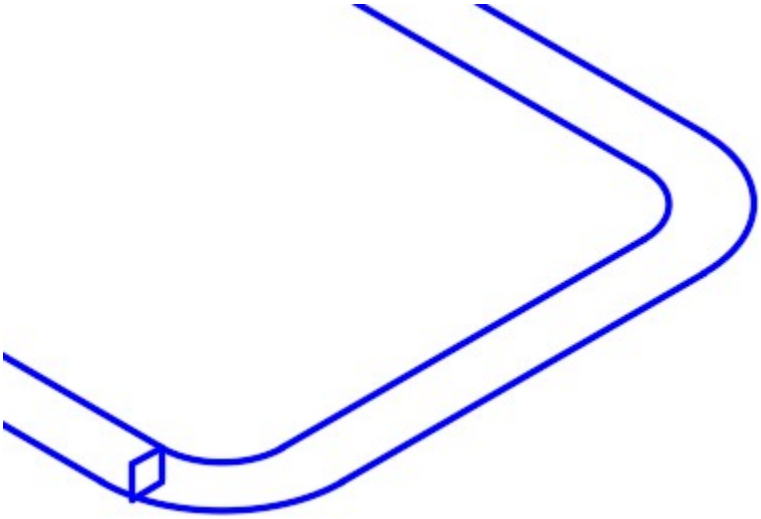


Figure Step 17C

AUTHOR'S COMMENTS: The pline you just drew is the profile for the sweep. It must connect to the path (inner pline).

Step 18

Set layer: Solid 1 as the current layer.

Step 19

Using the SWEEP command, sweep the 2.5 inch closed pline using the inner pline as the path. (Figure Step 19)

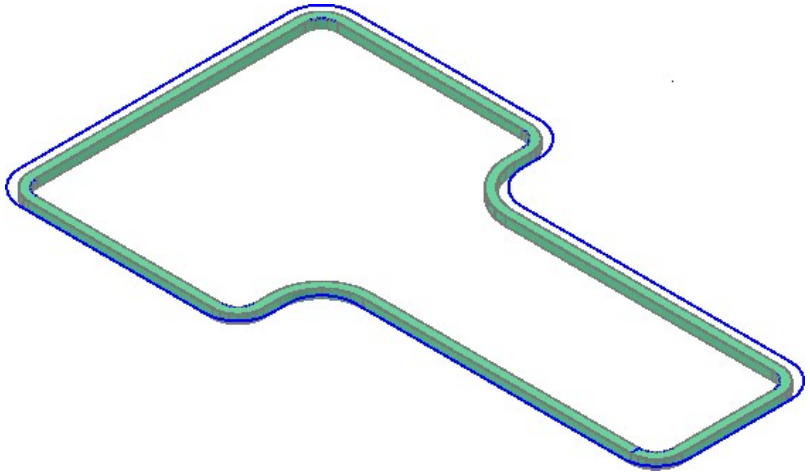


Figure Step 19

MUST KNOW: The SWEEP command requires at least one profile and a path. The path can be an open or closed object. The profile must be closed object to create a solid. The path must pierce or touch the profile.

Step 20

Change the layer of the solid on layer: Solid Off to layer: Solid 1. Turn layers: Path and Profile off and change the visual style to Realistic. (Figure Step 20)

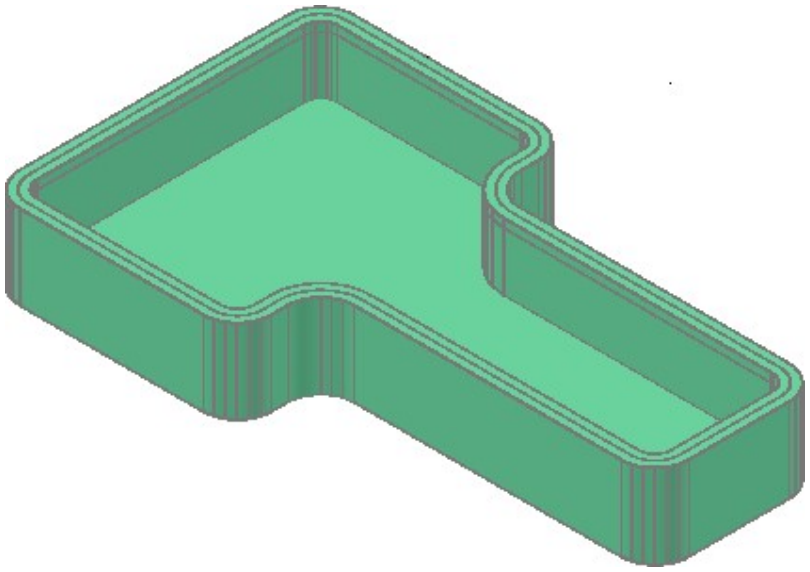


Figure Step 20

Step 21

Using the SUBTRACT command, subtract the solid that was created using the SWEEP command from the original solid. (Figure Step 21)

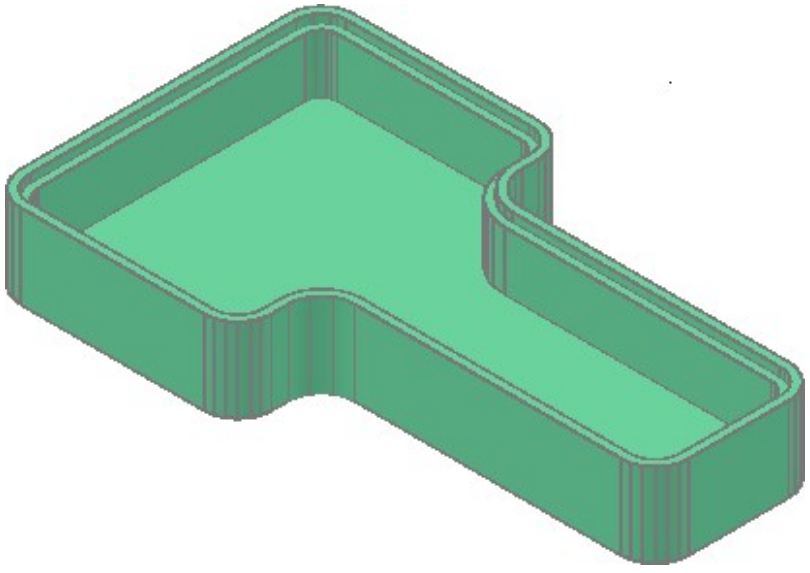


Figure Step 21

Step 22

Save and close the drawing.

Key Principles

Key Principles in Module 22

1. Sweeps are created by sweeping (moving) one or more profiles along a selected path. The path can be an open or closed object but must be one object. If the profile is a closed object, the sweep will create a solid. If the profile is an open object, the sweep will create a surface.

Lab Exercise 22-1

Time allowed: 60 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 22-1	3D Layout English	Inches

Step 1

On layer: Profile, draw the profile as a closed pline and on layer: Path, draw the path as a single pline. (Figure Step 1A and 1B)

Step 2

On layer: Solid 6, create the solid using the SWEEP command.

Step 3

Create the slots by first creating one and mirror it to create the opposite slot. Subtract them to complete the object.

Step 4

Save and close the drawing.

AUTHOR'S COMMENTS: From this module on there are no keys provided for the lab exercises. This is done so that you get used to drawing on your own without the aid of the key.

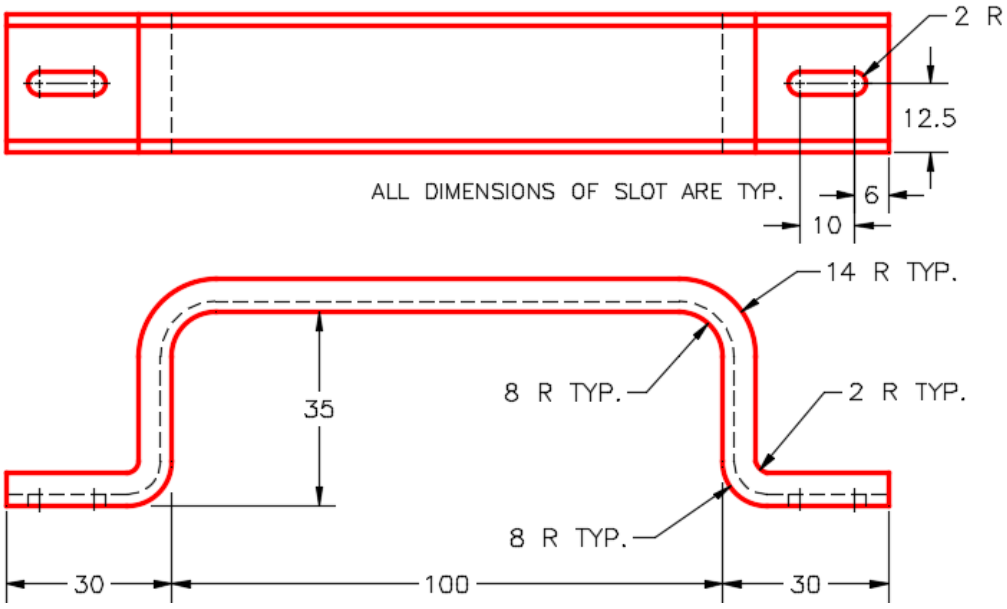


Figure Step 1A
Dimensioned Multiview Drawing

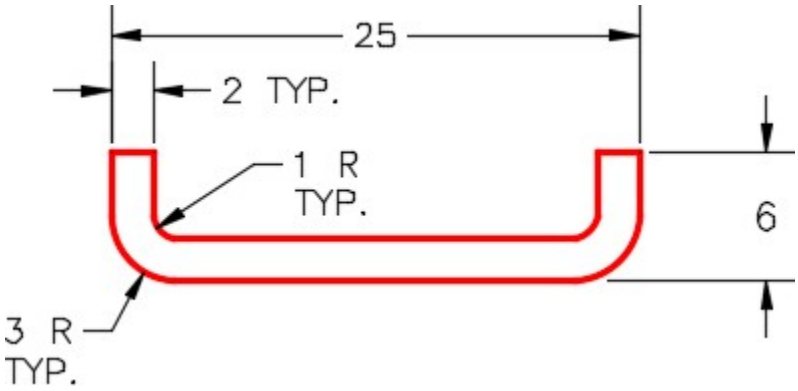
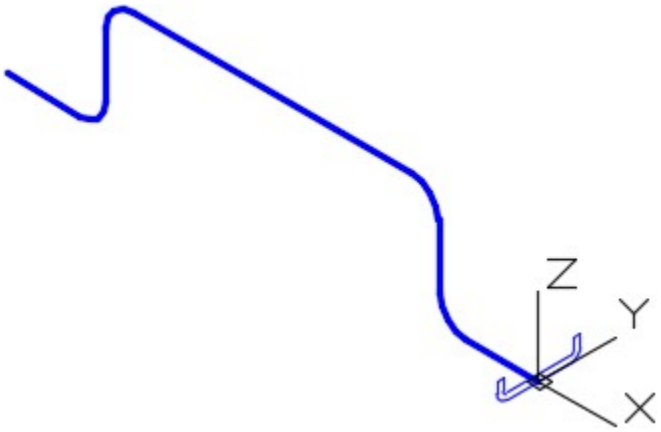
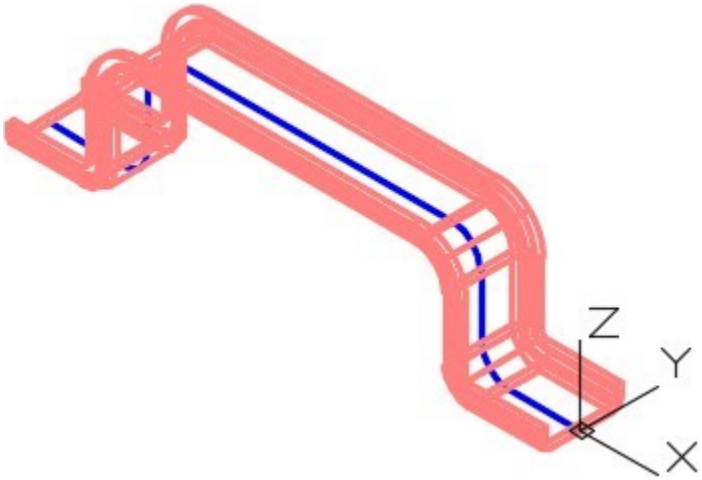


Figure Step 1B
Detail of Right Side View

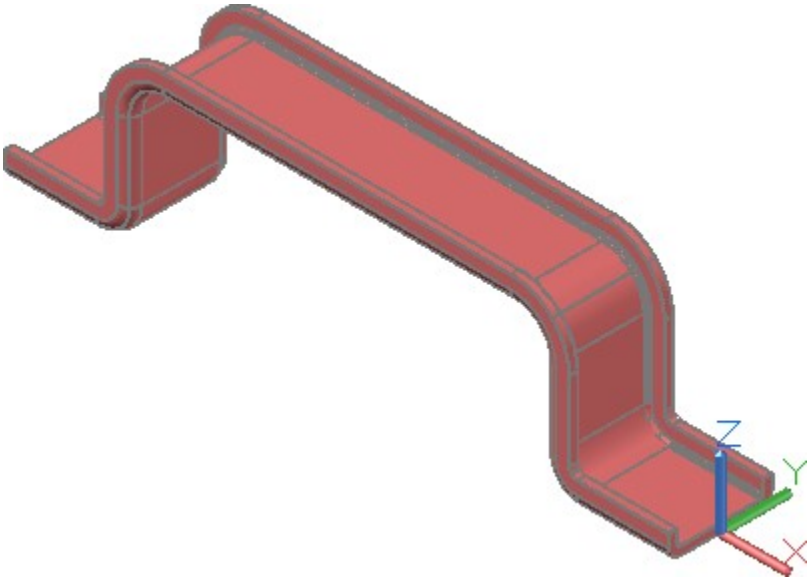
CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hint(s). If you get stuck and can't complete it on your own, use the following hint(s) to help you.



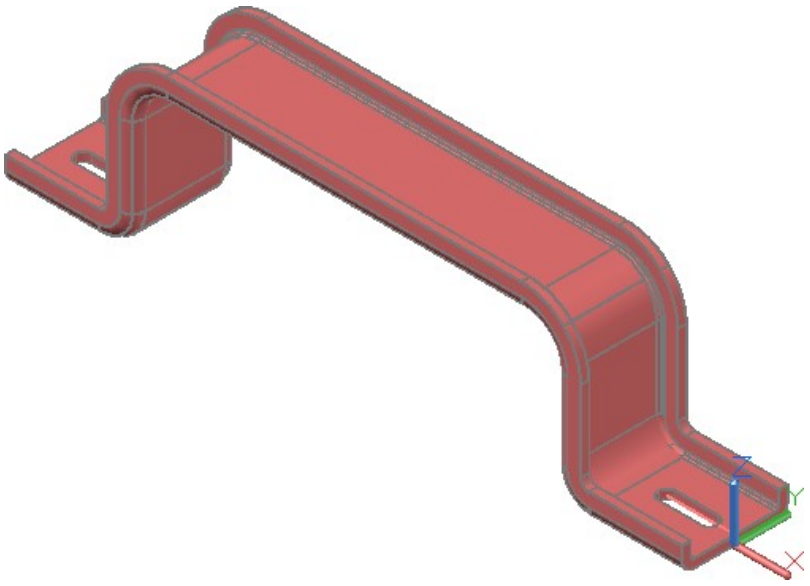
Step 1



Step 2



Step 3



Step 4

Lab Exercise 22-2

Time allowed: 60 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 22-2	3D Layout English	Inches

Step 1

On layer: Path, draw the paths as a circles, and on layer: Profile, draw the closed plines. (Figure Step 1A and 1B)

Step 2

On layer: Solid 3, create the original solid.

Step 3

On layer: Solid 3, create the solids that are to be subtracted using the SWEEP command.

Step 4

Subtract the sweep solids from the original solid.

AUTHOR'S COMMENTS: From this module on there are no keys provided for the lab exercises. This is done so that you get used to drawing on your own without the aid of the key.

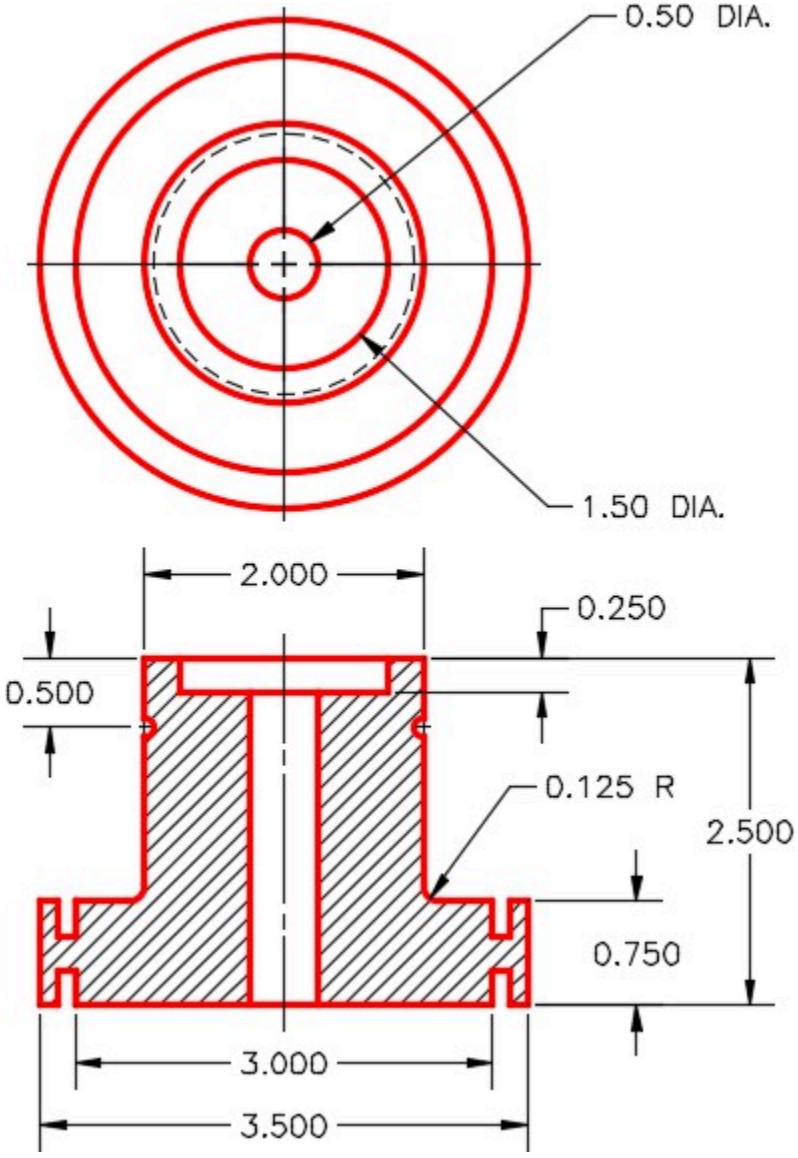


Figure Step 1A
Dimensioned Multiview Drawing

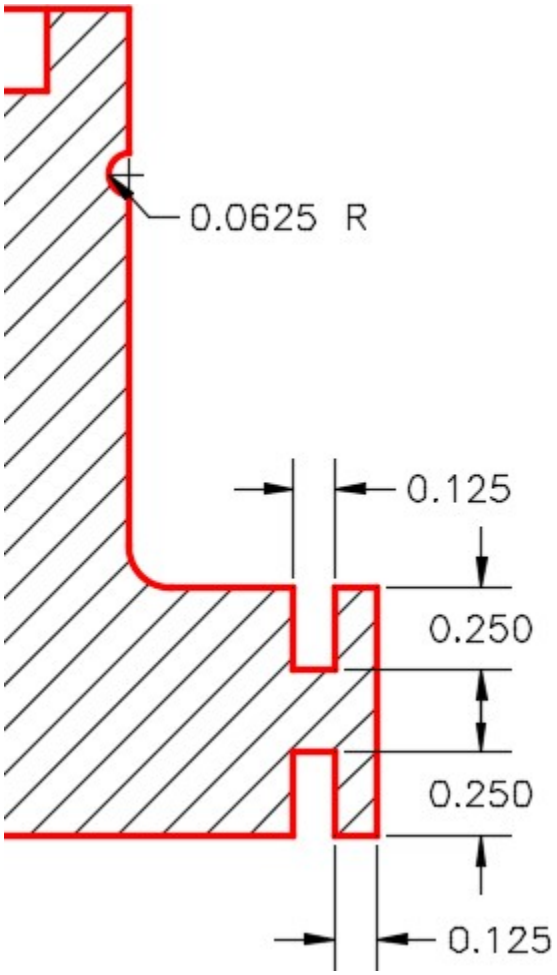
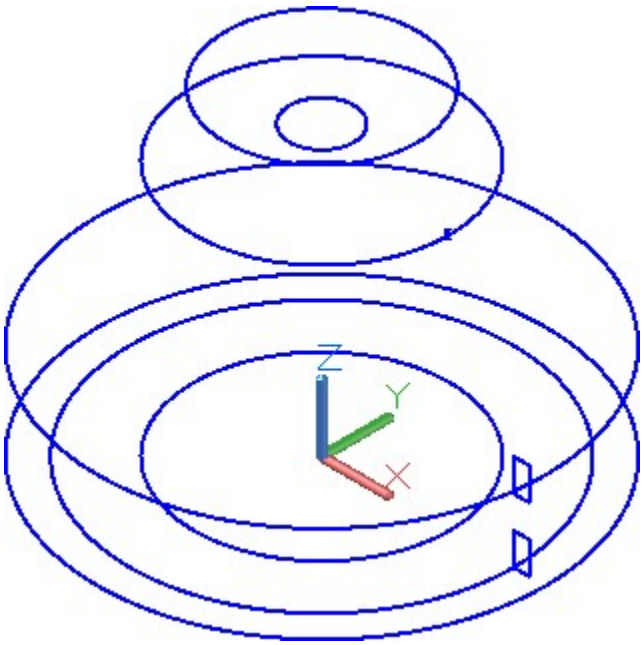
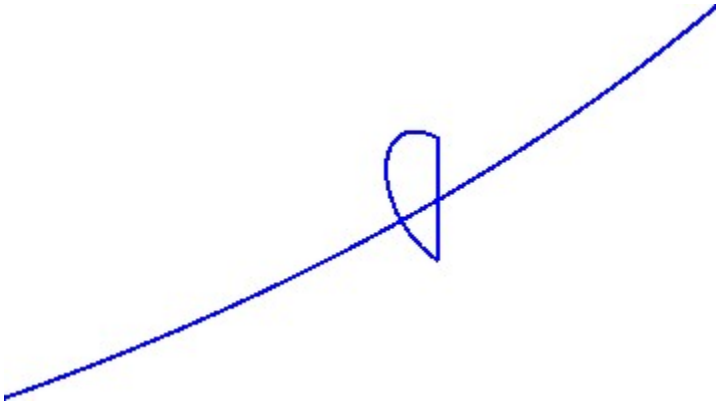


Figure Step 1B
Dimensioned Detail

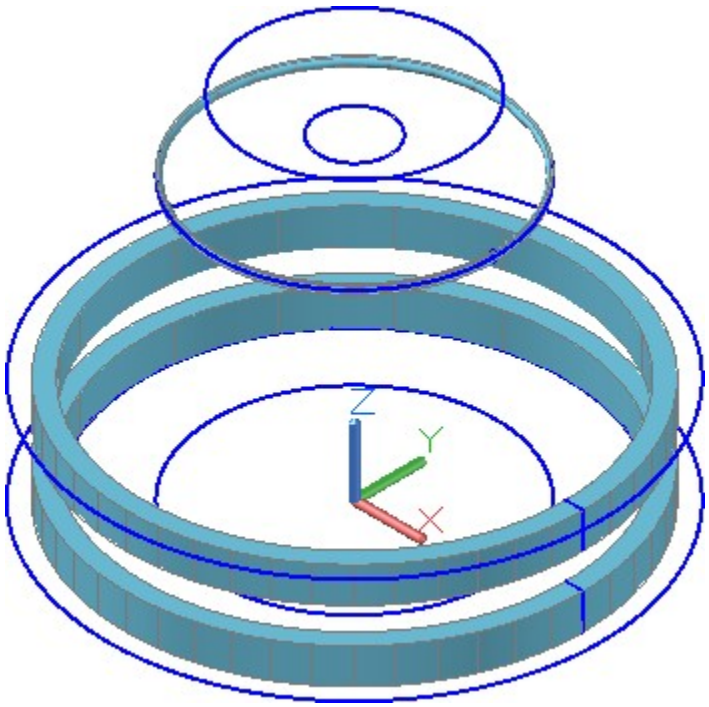
CONSTRUCTION HINTS: Do your best to complete the lab exercise drawing without using the following hint(s). If you get stuck and can't complete it on your own, use the following hint(s) to help you.



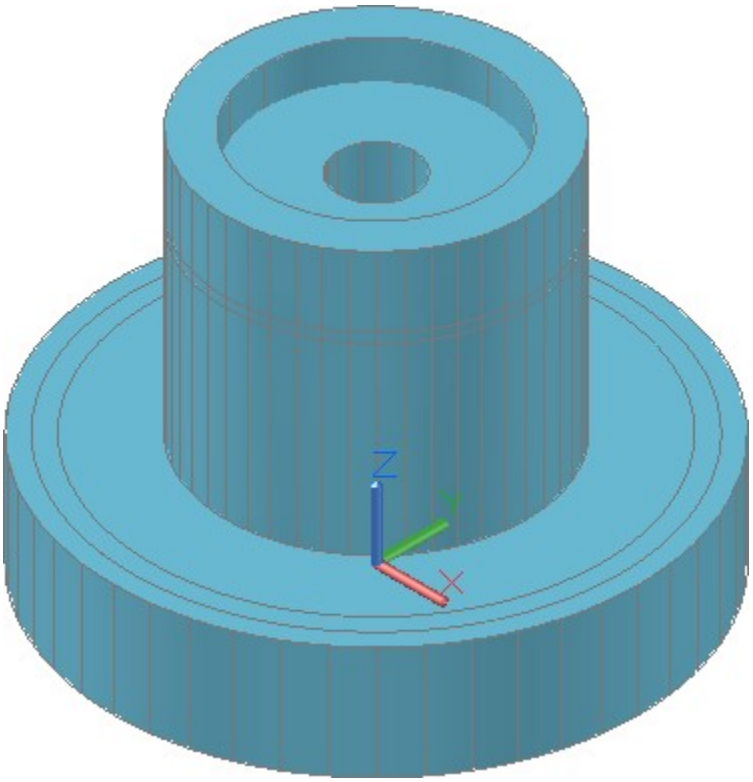
Step 1



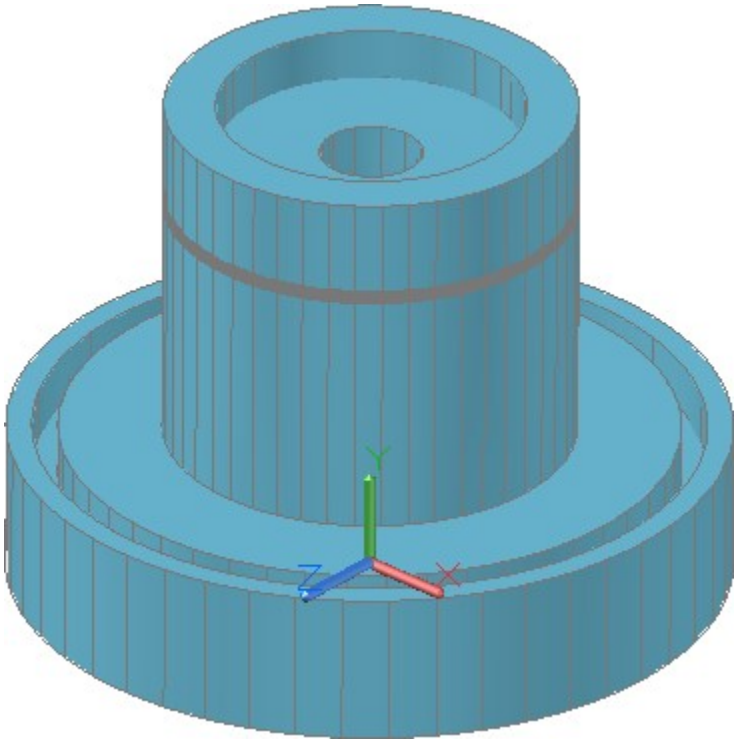
Step 1A
The arc profile must be created with an arc and line and then converted into a closed pline



Step 2



Step 3



Step 4

Step 5

Save and close the drawing.

Module 23 Helix

Learning Outcomes

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

1. Describe and apply the HELIX command to create solid models of springs, coils, and threads.

Helix

Coils, springs, and threads are constructed by drawing a profile and then using the SWEEP command, sweeping it along a path that is a helix drawn with the HELIX command. See Figure 23-1. The shape of the coil, spring or thread is controlled by the shape of the profile. See Figure 23-2, 23-3 and 23-4.

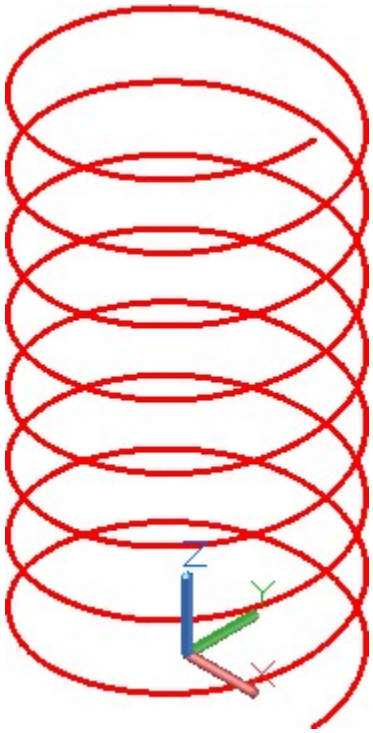


Figure 23-1
A Helix

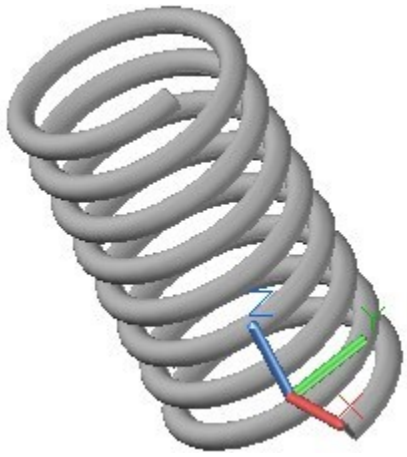


Figure 23-2
A Coil

The HELIX command is used to construct a helix. A helix is constructed by specifying one or more setting that consist of the base radius, top radius, height, number of turns, turn height or twist direction.

If the same value for both the base radius and the top radius is specified, a cylindrical helix is created. If different values are specified for the top radius and the base radius, a conical helix is created. If a height value of zero is specified, a flat 2D spiral is created.

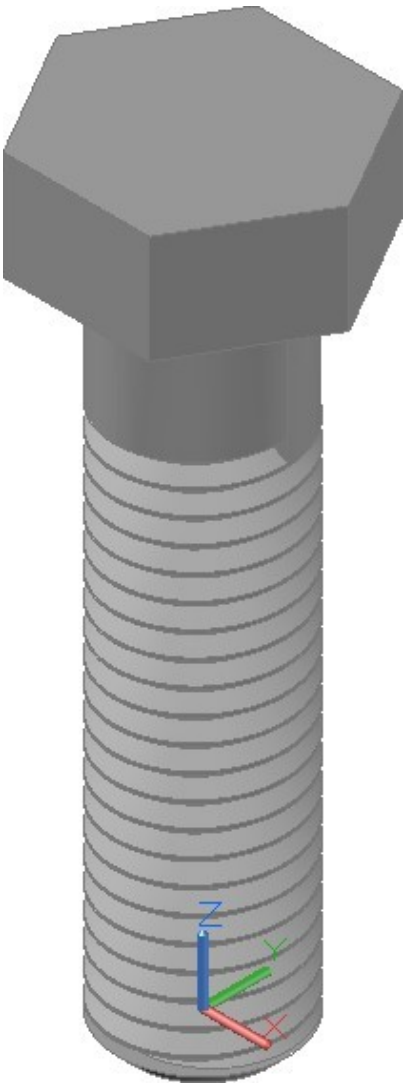


Figure 23-3
Threads

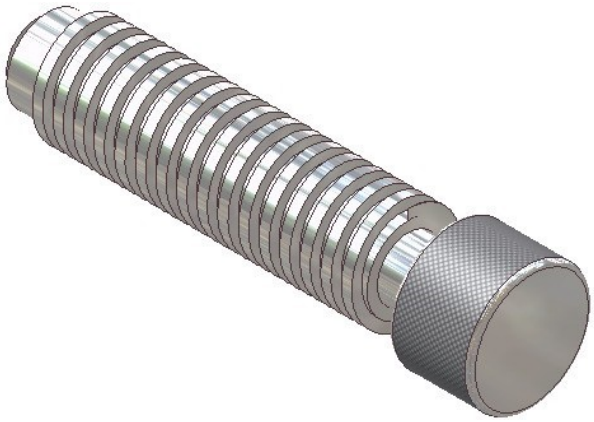
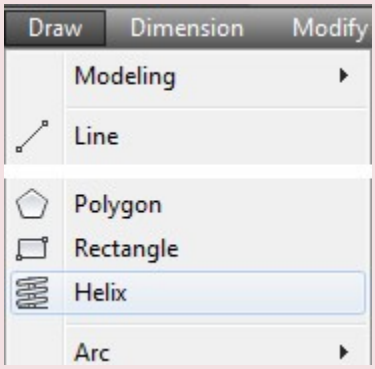


Figure 23-4
Rendered Solid Model

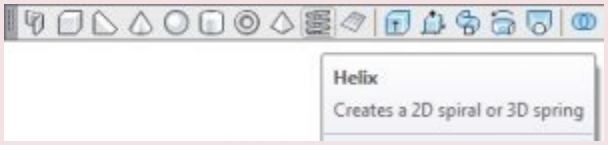
AutoCAD Command: HELIX

The HELIX command is used to create a helix.

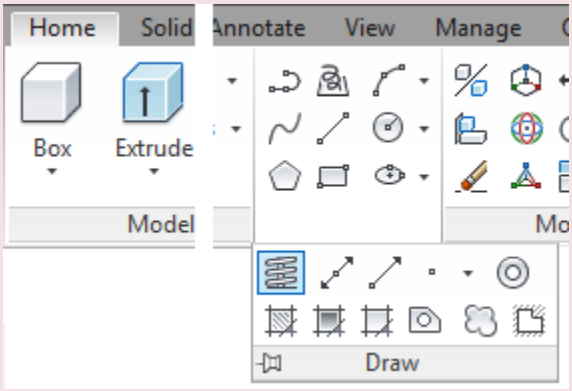
Shortcut: none



Draw Pull-down



Modeling Toolbar



Home Ribbon

Drafting Lesson: Springs and Threads

Study the Figure 23-5, shown below, to learn the terms used when drawing coil, springs and threads. *Pitch* is an important term that must be understood and is required when drawing a helix using the HELIX command. AutoCAD's HELIX command uses the term *turn height* for the pitch.

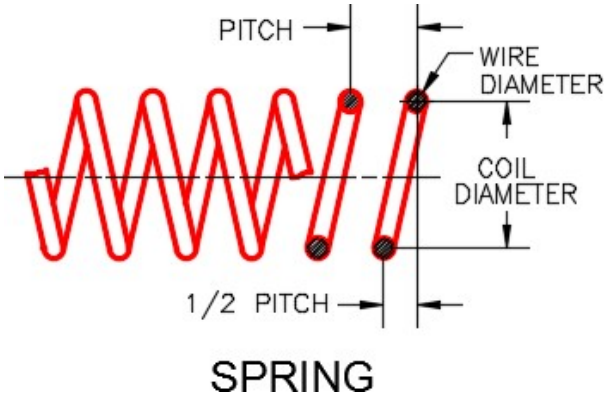


Figure 23-5
Threads and Spring Shape and Terminology

Figure 23-6 shows four common thread types and their specifications.

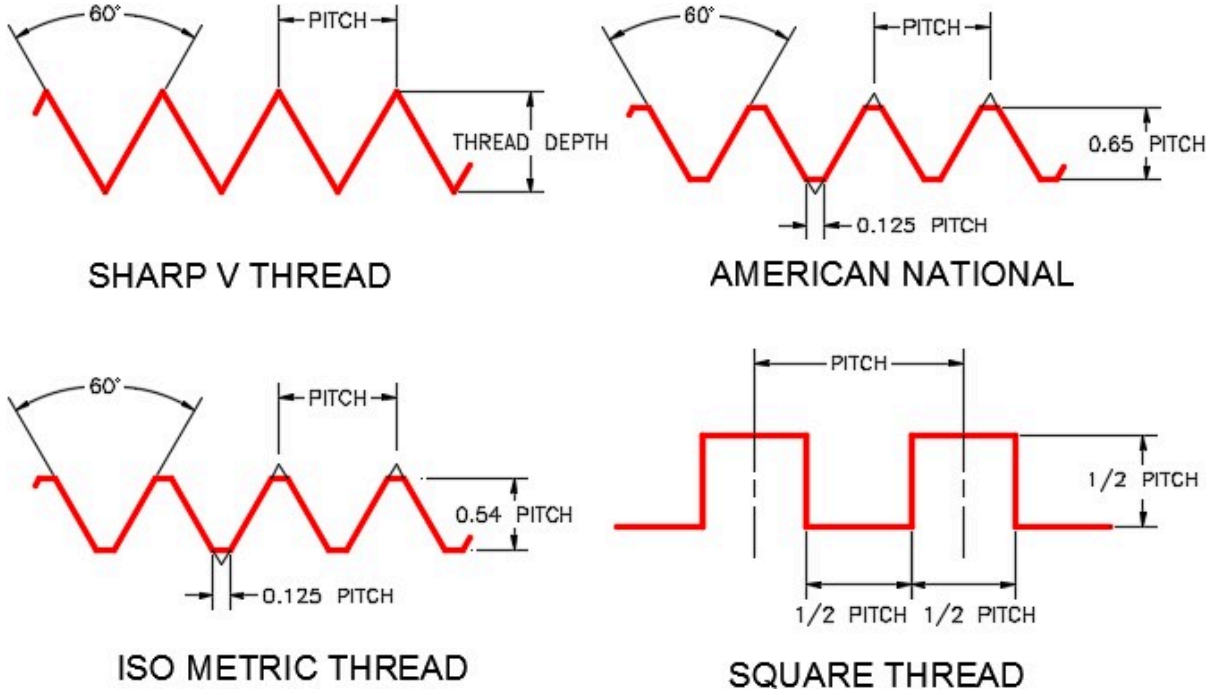


Figure 23-6
Four Common Thread Types

WORK ALONG: Creating Coils

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 23-1. (Figure Step 2A and 2B)

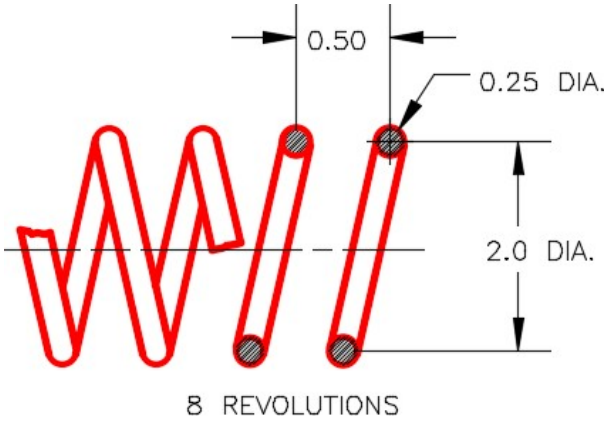


Figure Step 2A
Dimensioned Drawing

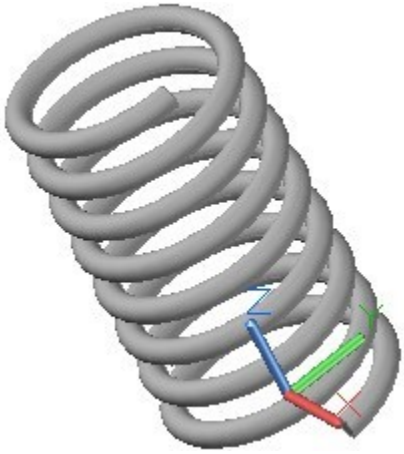


Figure Step 2B
Completed Coil

Step 3

Set layer: Helix as the current layer, the current view to SE Isometric, the current UCS to Top, and the current visual style to 3D Wireframe.

Step 4

Enter the HELIX command, as shown below, to draw the coil shown in Figure Step 2A. (Figure Step 4)

Command: **HELIX**

Number of turns = 3.0000 Twist=CCW

Specify centre point of base: **0,0,0**

Specify base radius or [Diameter] <0.5000>: **@1,0,0**

(I used coordinates to ensure that the helix start along the X axis.)

Specify top radius or [Diameter] <1.0000>: **1**

(Diameter of the coil is 2 inches.)

Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <4.0000>: **H**

Specify distance between turns <1.000>: **0.5**

(The pitch is 0.5 inches.)

Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <4.0000>: **T**

Enter number of turns <3.0000>: **8**

(8 turns means that the coil is 4 inches long.)

Command:

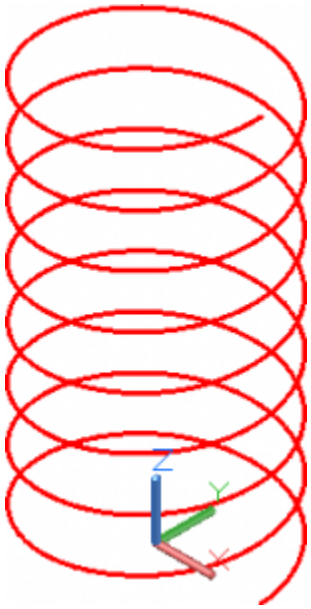


Figure Step 4

AUTHOR'S COMMENTS: Like the EXTRUDE command, the HELIX command constructs the helix along the Z direction of the current UCS.

AUTHOR'S COMMENTS: The radius is 1, the pitch (distance between turns) is 0.5, the number of turns is 8. That creates a coil that is 2 inches in diameter and 4 inches long.

Step 5

Change the current UCS to Front and set layer: Profile as the current layer. Draw a 0.25 diameter circle with its centre located at the end of the helix. (Figure Step 5)

Figure Step 5

Step 6

Set layer: Solid 8 as the current layer.

Step 7

Using what you learned in Module 22, create the coil by entering the SWEEP command. Select the circle as the profile and the helix as the path. (Figure Step 7)

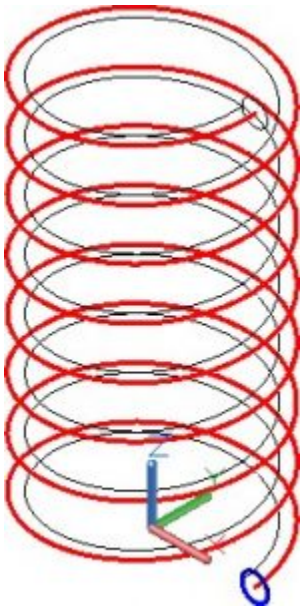


Figure Step 7

Step 8

Turn layers: Helix and Profile off. Set the current visual style to Realistic. Your completed 3D solid coil should appear as shown in the figure. (Figure Step 8)

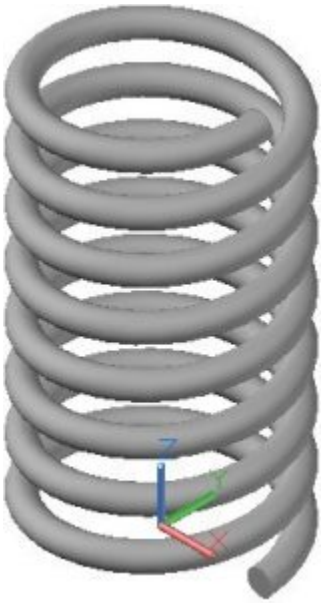


Figure Step 8

Step 9

Save and close the drawing.

WORK ALONG: Creating Square Threads

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 23-2. (Figure Step 2A and 2B)

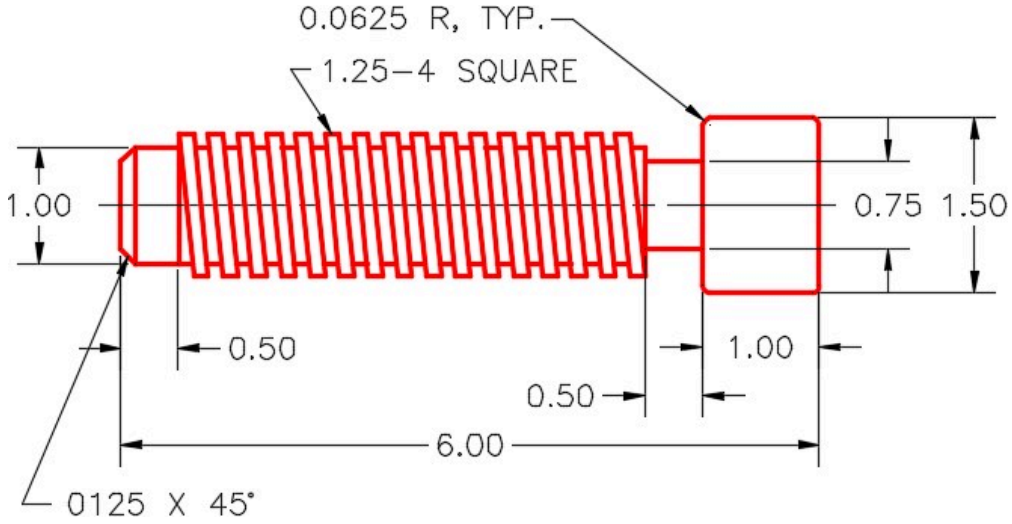


Figure Step 2A
Dimensioned Multiview Drawing

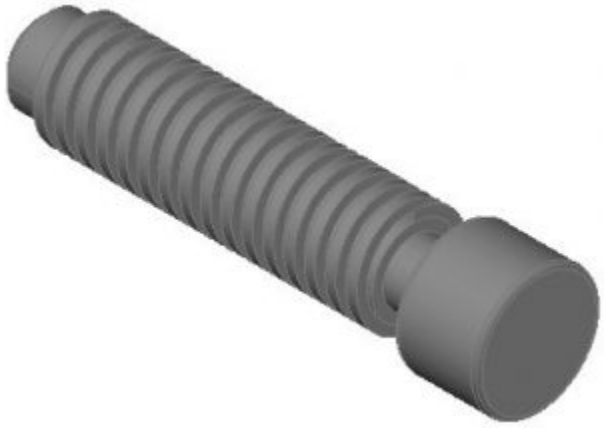


Figure Step 2B
Completed Solid Model – SE Isometric View

Step 3

Set layer: Pline as the current layer, the current UCS to Right, the current view to SE Isometric, and the current visual style to 3D Wireframe.

Step 4

Draw a circle, with a diameter of 1 inch. Locate its centre at 0,0,0. (Figure Step 4)

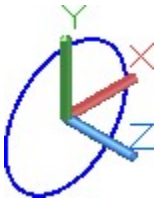


Figure Step 4

Step 5

Set layer: Solid 8 as the current layer and extrude the circle 5 inches in the negative Z direction. (Figure Step 5)

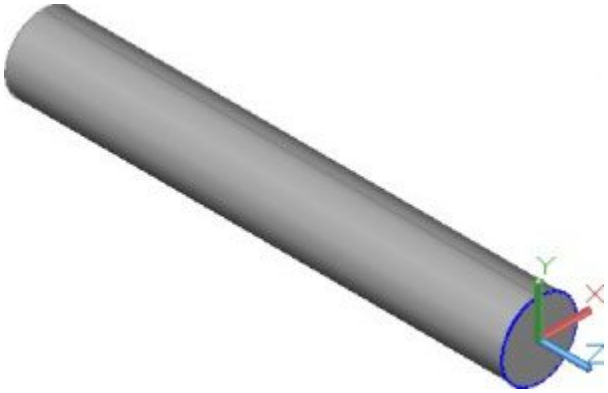


Figure Step 5

Step 6

Change the layer of the solid, that you created in Step 5, to layer: Solid Off. Turn off layers: Solid off and Pline. Set layer: Helix as the current layer.

Step 7

Enter the HELIX command, as shown below, to draw a helix for the thread. (Figure Step 7)

Command: **HELIX**

Number of turns = 3.0000 Twist=CCW

Specify centre point of base: **0,0,0**

Specify base radius or [Diameter] <1.0000>: **@0,0.5,0**

(This starts the helix at the top at the radius of 0.5 inches.)

Specify top radius or [Diameter] <0.5000>:

(Press Enter)

Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <1.0000>: **T**

Enter number of turns <3.0000>: **20**

(Since there are 4 turns per inch and the shaft is 5 inches long, there are 20 turns.)

Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <1.0000>: **A**

Specify axis end point: **@0,0,-5**

(The end of the axis is 5 inches in the negative Z direction.)

Command:

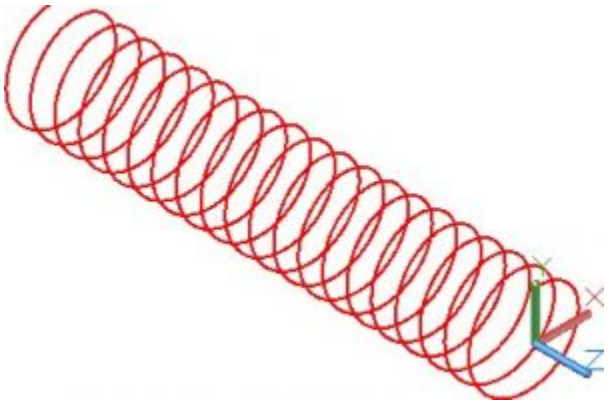


Figure Step 7

Step 8

Set layer: Profile as the current layer and the current UCS to Front.

Step 9

Using the figures as a reference, draw a closed pline starting at the end of the helix. (Figure Step 9A, 9B, and 9C)

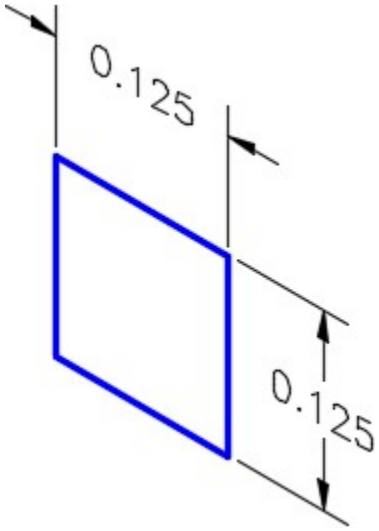


Figure Step 9A

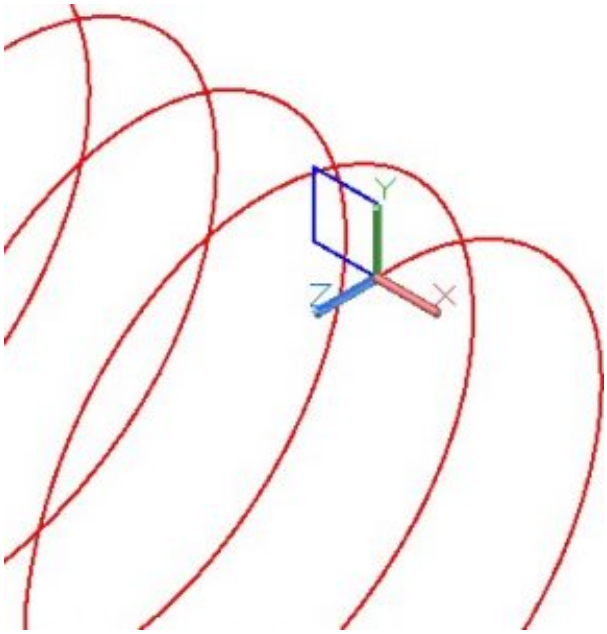


Figure Step 9B

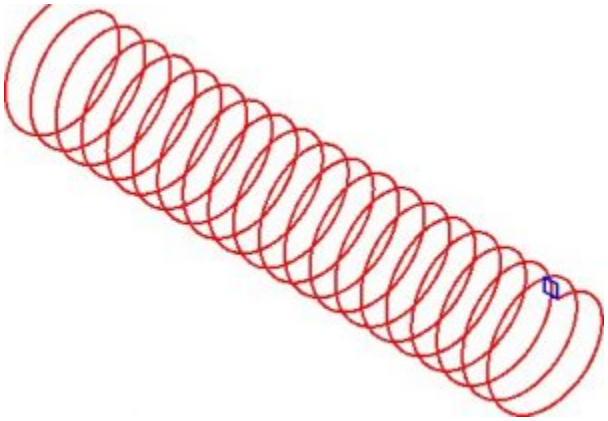


Figure Step 9C

Step 10

Set layer: Solid 8 as the current layer. Using the SWEEP command, create the square thread by sweeping the profile along the helix. (Figure Step 10)

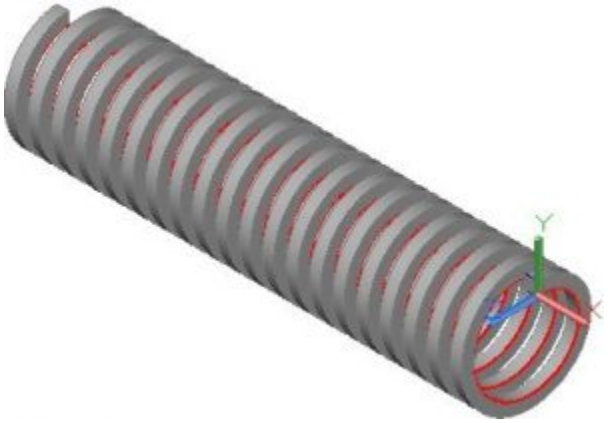


Figure Step 10

Step 11

Turn layers: Helix and Profile off. Change the solid on layer: Solid Off to layer: Solid 8.

Step 12

Using the UNION command, union the two solids to form one solid model. (Figure Step 12)

Figure Step 12

Step 13

Turn layer: Pline on and set it as the current layer.

Step 14

Change the current UCS to Right. Draw a 2 Inch diameter circle with its centre located at 0,0,0. (Figure Step 14)

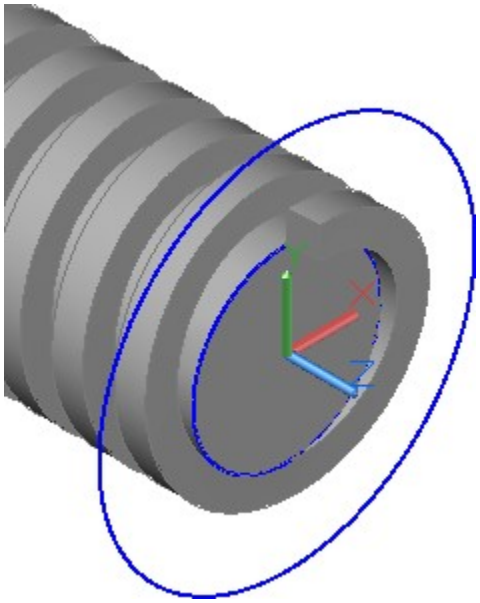


Figure Step 14

Step 15

Change the current view to Front. (Figure Step 15)

Figure Step 15

Step 16

Using the MOVE command, move the 2 inch diameter circle 0.125 inches in the negative Z direction. Using the SLICE command, as shown below, slice the solid model using the 2 inch diameter circle as the Object to create a plane at the slice location. (Figure Step 16)

Command: **SLICE**

Select objects to slice: 1 found

(Select the solid model.)

Select objects to slice:

(Press Enter.)

Specify start point of slicing plane or [planar Object/Surface/Zaxis/View/XY/YZ/ZX/3points]

<3points>: **O**

Select a circle, ellipse, arc, 2D-spline, 2D-polyline to define the slicing plane:

(Select the circle.)

Specify a point on desired side or [keep Both sides] <Both>:

(Press Enter.)

Command:

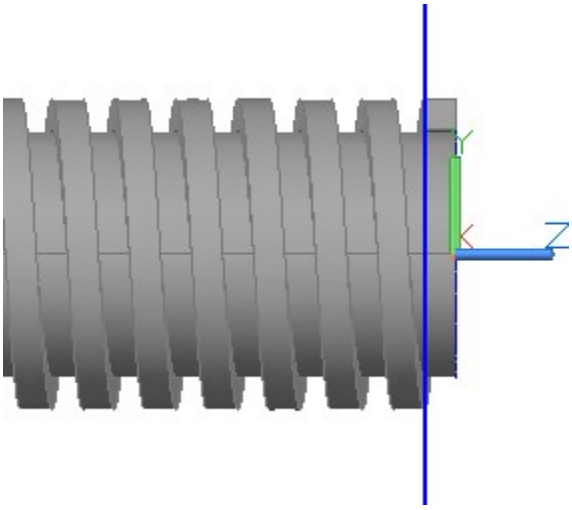


Figure Step 16

AUTHOR'S COMMENTS: I find it is easiest to keep both sides when I use the SLICE command. After I slice the model, I delete the side I don't want to keep.

Step 17

Delete the end of the solid that you sliced off. (Figure Step 17)

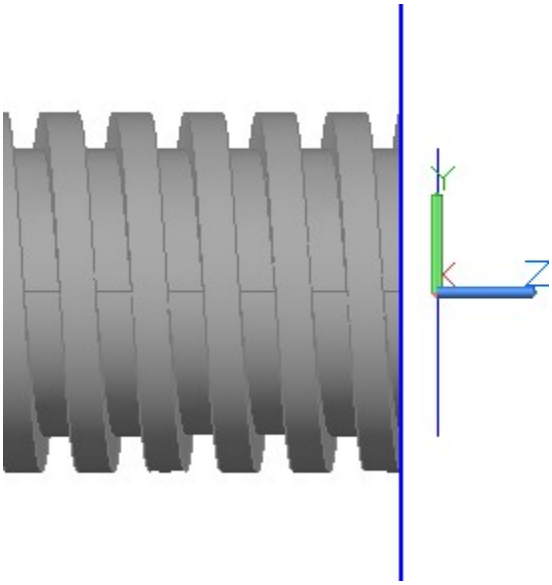


Figure Step 17

Step 18

Move the circle 4 inches in the negative Z direction. Using what you learned in the last two steps, slice the solid and delete the portion you sliced off. Set the current view to SE Isometric. (Figure Step 18A, 18B, and 18C)

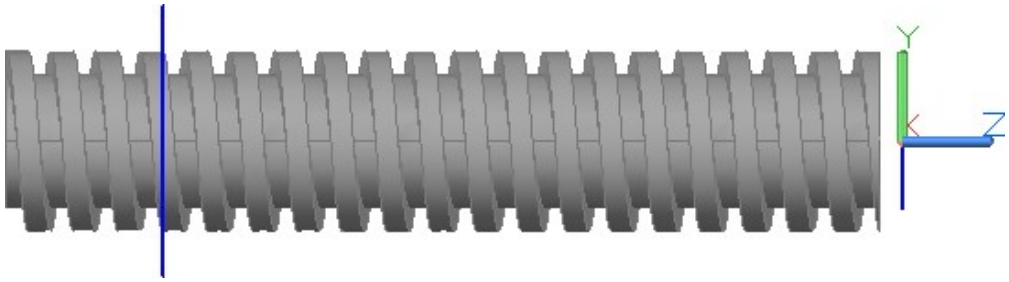


Figure Step 18A

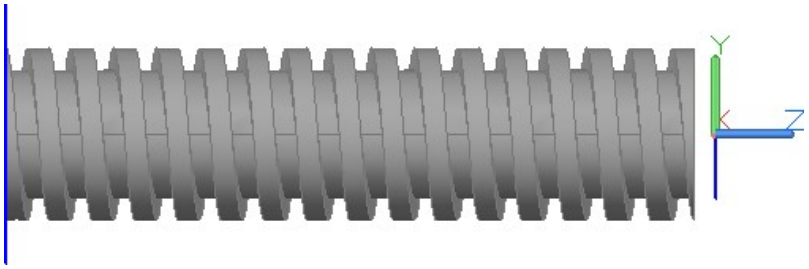


Figure Step 18B

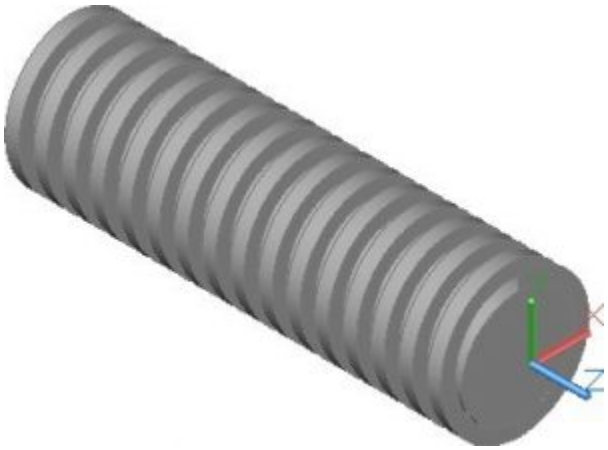


Figure Step 18C

Step 19

Locate the UCS at the centre of the end of the solid thread. Draw a 0.75 inch diameter circle locating the centre at 0,0,0. On layer: Solid 8, extrude the circle 0.5 inches in the positive Z direction. (Figure Step19)

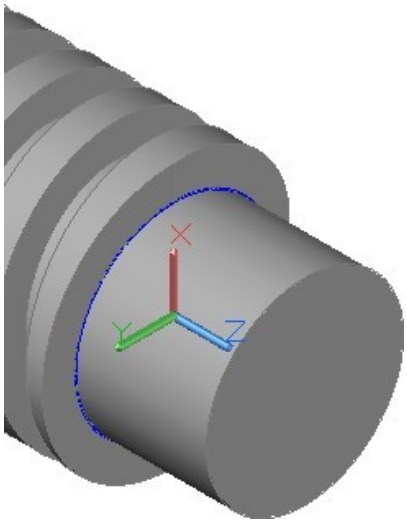


Figure Step 19

Step 20

On layer: Pline, draw a 1.5 inch diameter circle locating its centre at the end of the extrusion that you created in Step 19. On layer: Solid 8, extrude the circle 1 inch in the positive Z direction. (Figure Step 20A and 20B)

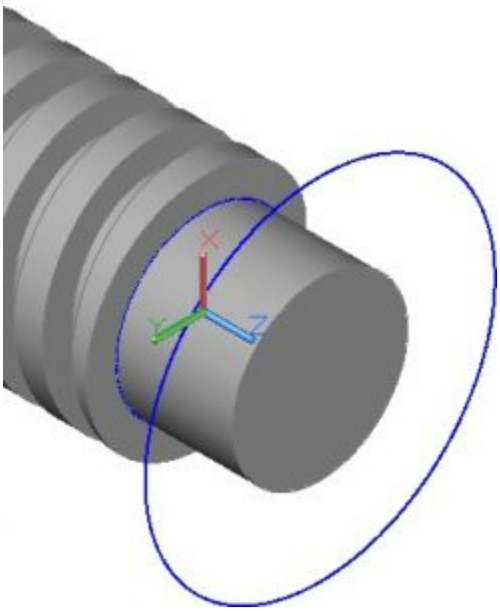


Figure Step 20A

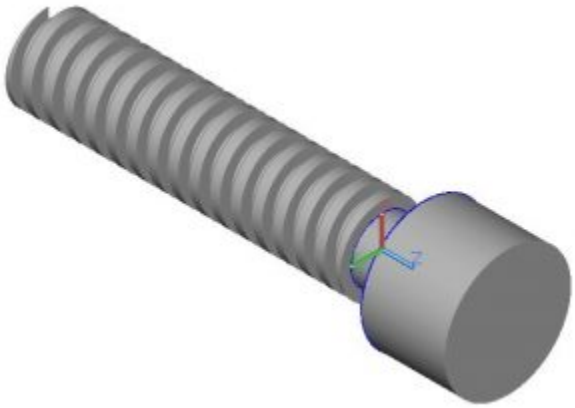


Figure Step 20B

Step 21

Using what you just learned, orbit the model, relocate the UCS and extrude a 1 inch diameter circle 0.5 inches in the negative Z direction. (Figure Step 21A, 21B, and 21C)

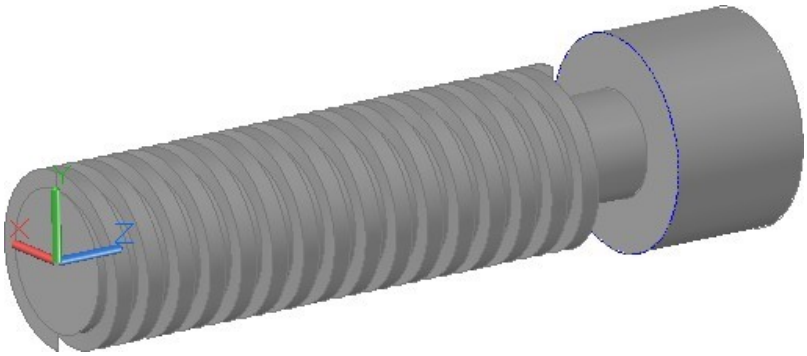


Figure Step 21A

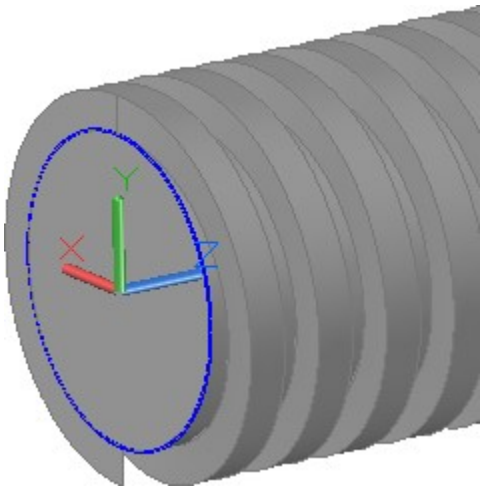


Figure Step 21B

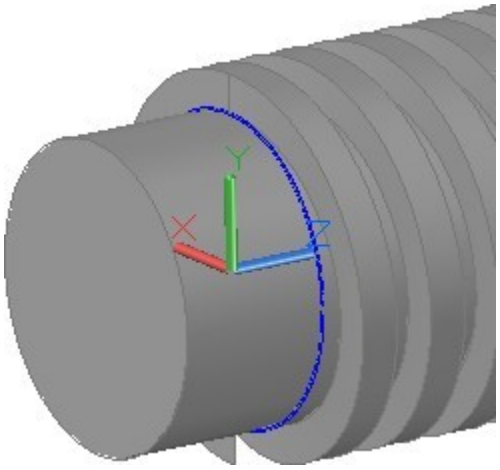


Figure Step 21C

Step 22

Chamfer the end of the shaft 0.125 inches at 45 degrees. (Figure Step 22)

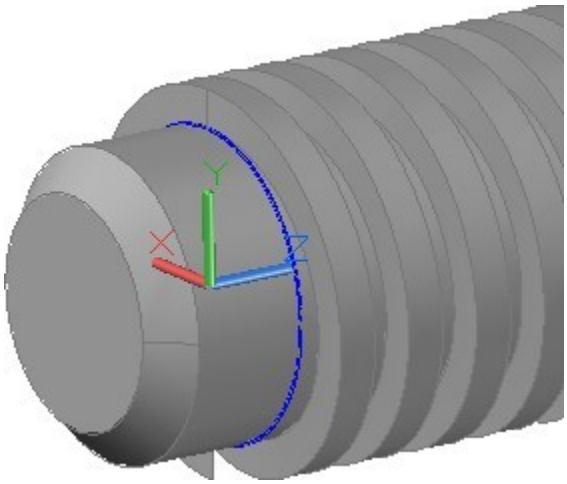


Figure Step 22

Step 23

Fillet the two edges of the head of the bolt using the radius 0.0625.

Step 24

Change the current view to SE Isometric. Using the UNION command, union the four solids to create on solid model. Your completed model should appear as shown in the figure. (Figure Step 24)

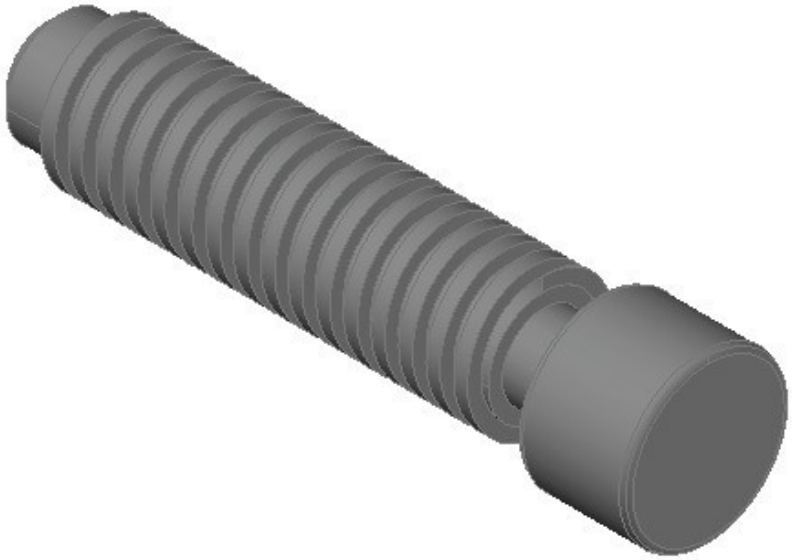


Figure Step 24

Step 25

Save and close the drawing.

WORK ALONG: Using the HELIX Command – Part 3

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 23-3. (Figure Step 2A and 2B)

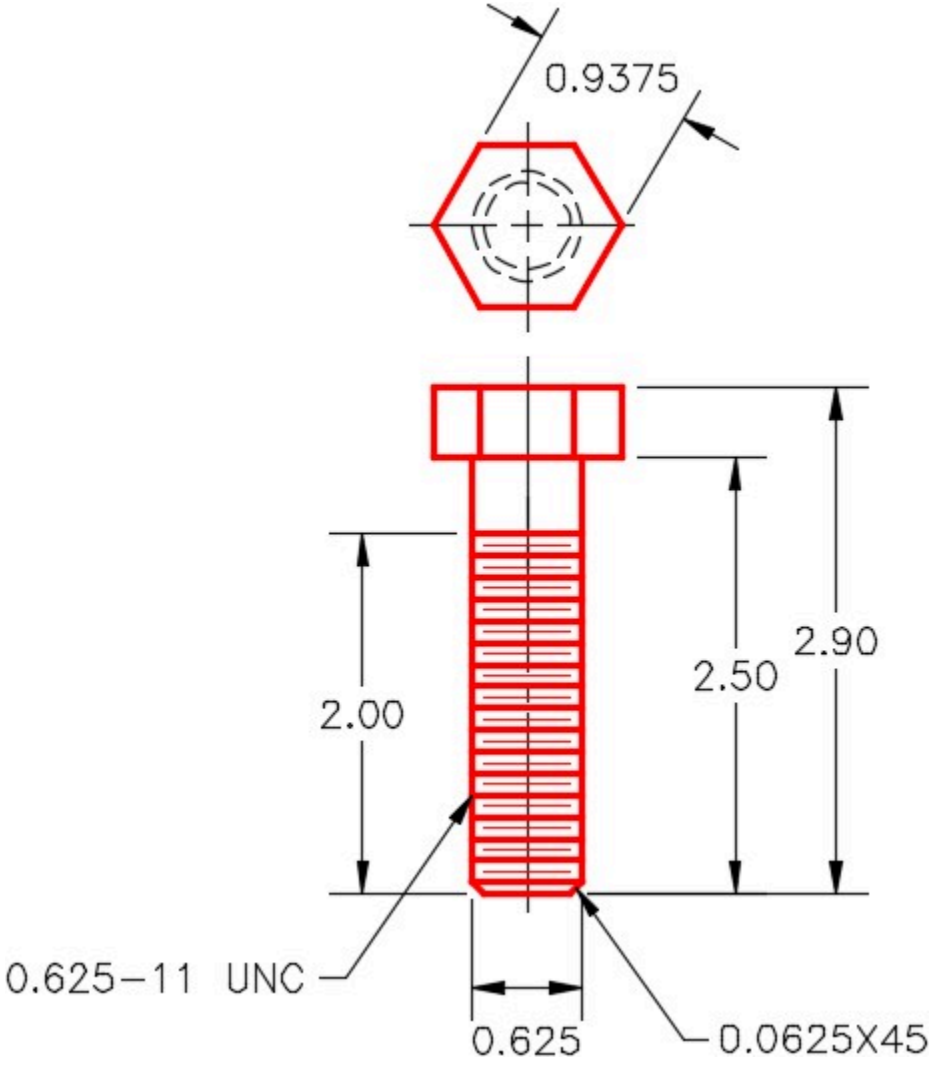


Figure Step 2A
Dimensioned Multiview Drawing

Figure Step 2B Completed Solid Model

Step 3

Using what you learned in the last workalong and using Figure Step 2A as a reference, draw the solid model on layer: Solid 8 to match the figure. Ensure that you union the hexagon and cylinder to create one solid model. (Figure Step 3)

Figure Step 3

Step 4

Using what you learned in the last workalong, on layer: Helix, draw the helix using the figure as a reference. (Figure Step 4A and 4B)

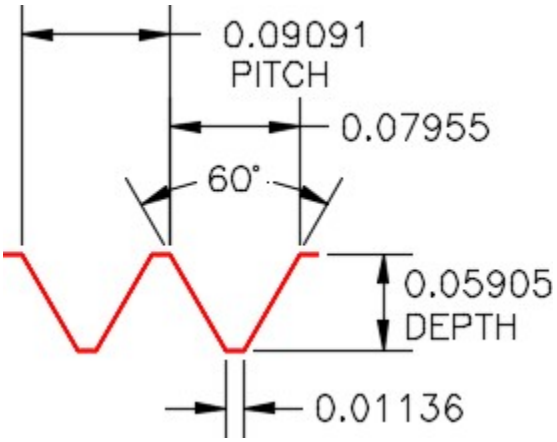


Figure Step 4A
Thread Details

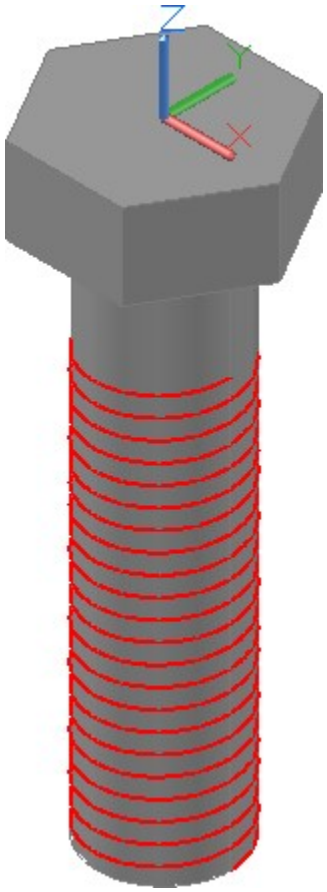


Figure Step 4B

Step 5

Change the layer of the solid that you drew in Step 3 to layer: Solid Off. Turn off layers: Solid off and Pline.

Step 6

Set layer: Construction as the current layer and the current UCS to Front. (Figure Step 6)

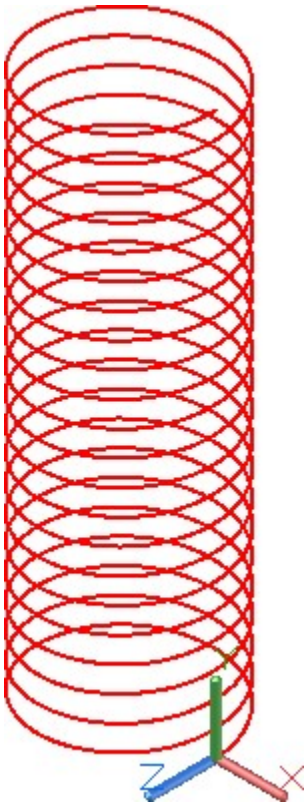


Figure Step 6

Step 7

On layer: Construction, draw the construction lines for the thread profile. (Figure Step 7A and 7B)

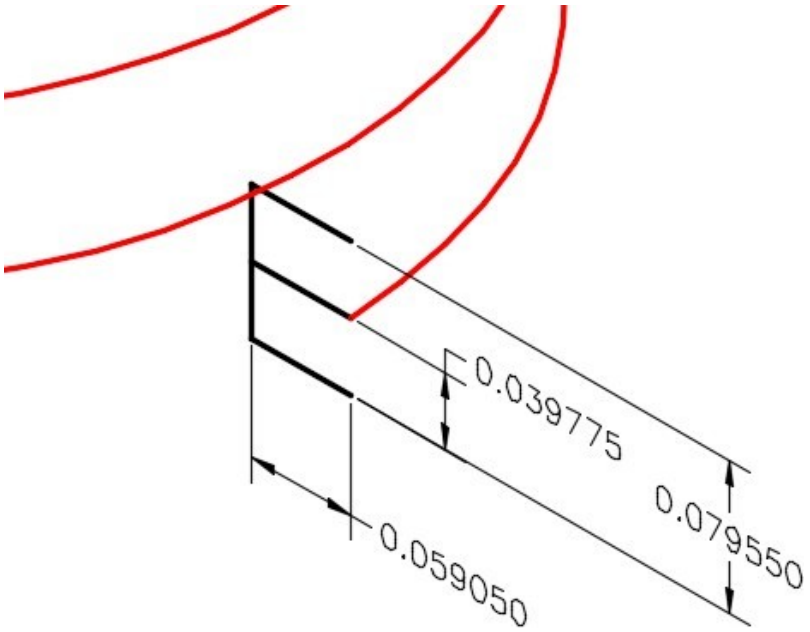


Figure Step 7A

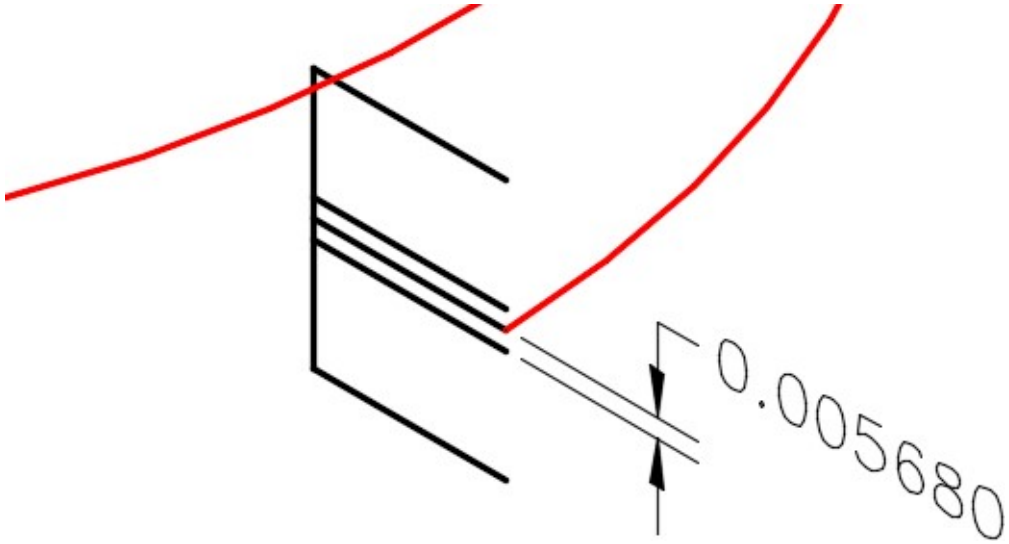


Figure Step 7B

AUTHOR'S COMMENTS: The end of the centre construction line must be snapped to the end of the helix.

Step 8

On layer: Profile, snapping to the ends of the construction lines, draw the thread profile as a closed pline. (Figure Step 8)

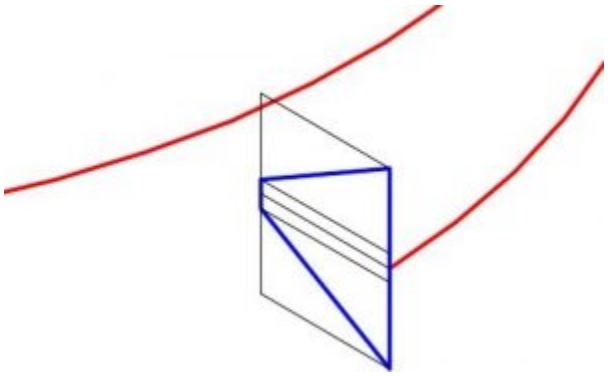


Figure Step 8

Step 9

Turn layer: Construction off. (Figure Step 9A and 9B)

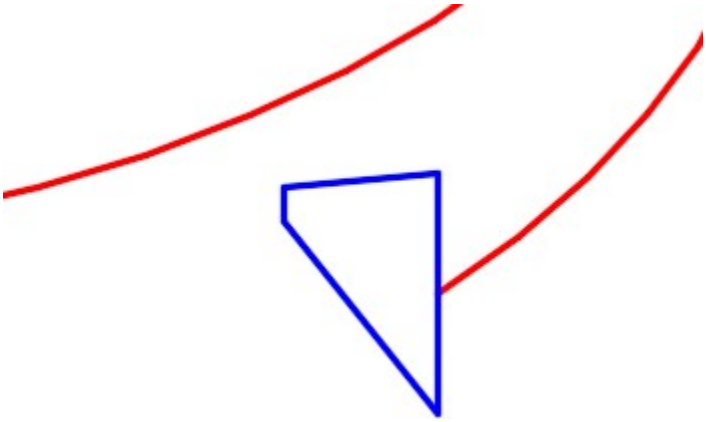


Figure Step 9A

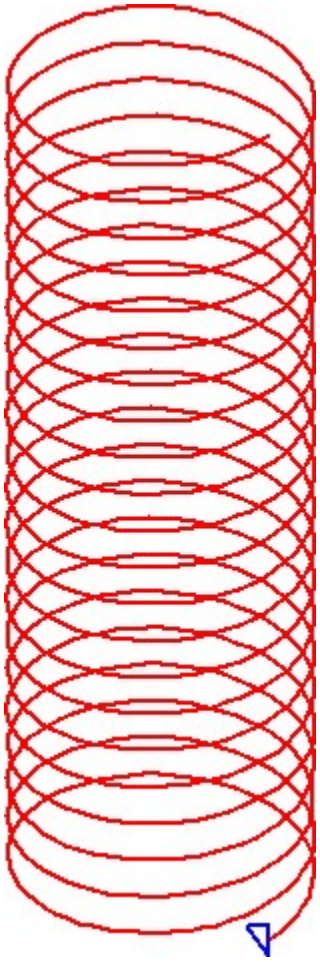


Figure Step 9B

Step 10

Set layer: Solid 8 as the current layer and the current UCS to Top.

Step 11

Using the SWEEP command, sweep the profile using the helix as a path. (Figure Step 11)

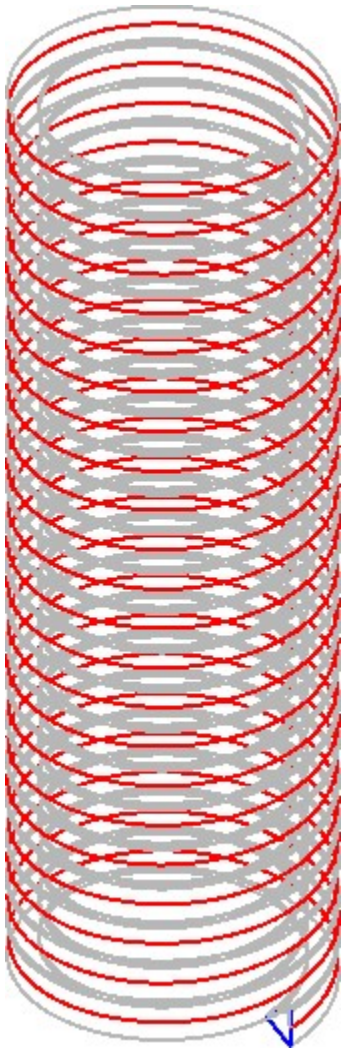


Figure Step 11

Step 12

Turn layers: Helix and Profile off. Change the solid from layer: Solid Off to layer: Solid 8.

Step 13

Using the SUBTRACT command, subtract the solid thread, that you created in the SWEEP command, from the solid model to create the bolt. (Figure Step 13)

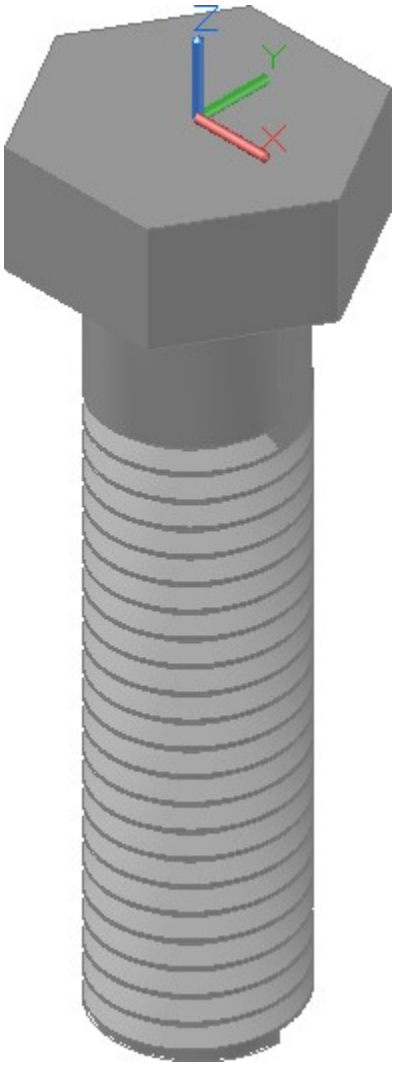


Figure Step 13

Step 14

Using what you learned in the last workalong, on layer: Pline, draw 1.5 diameter circle locating its centre at the centre of the bottom of the bolt. (Figure Step 14)

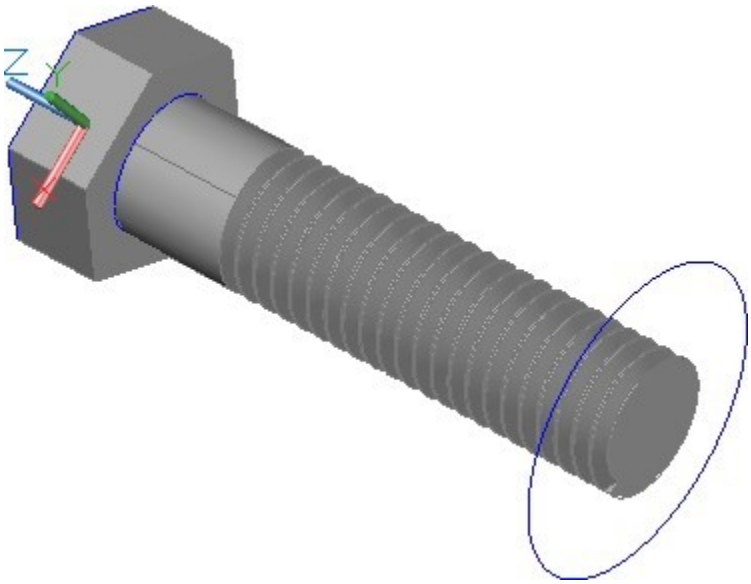


Figure Step 14

Step 15

Set the current view to Front. Move the circle 0.0625 inches in the positive Z direction. (Figure Step 15)

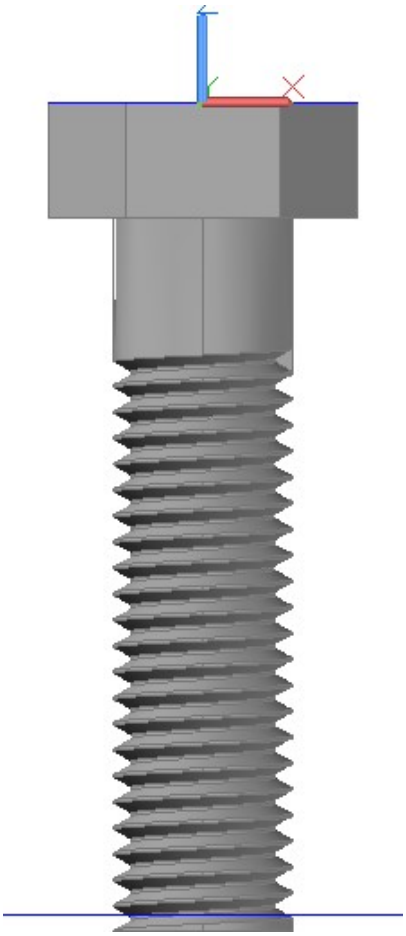


Figure Step 15

Step 16

Slice the bolt and delete the end. (Figure Step 16)

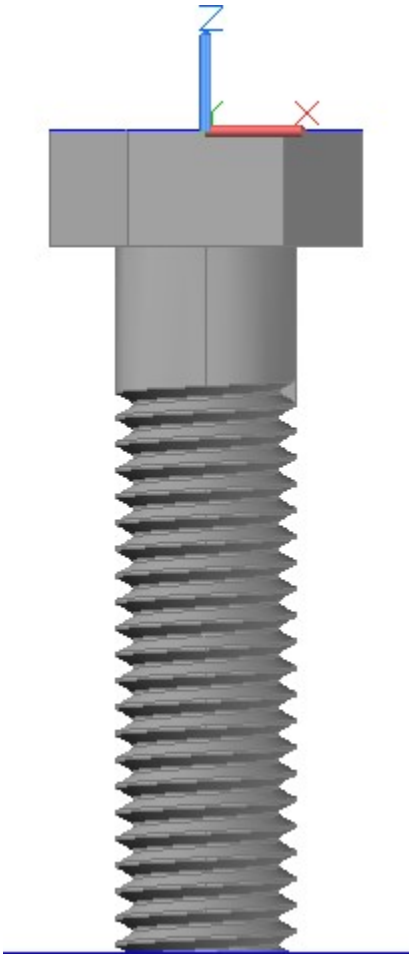


Figure Step 16

Step 17

Draw a circle at the end of the threaded shaft and extrude it 0.0625 inches. (Figure Step 17)

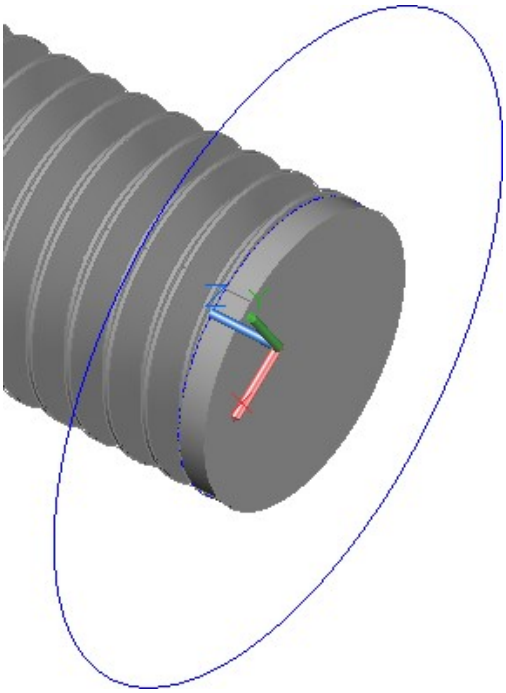


Figure Step 17

Step 18

Union the two solids together and chamfer the end of the bolt using the dimensioned drawing as a reference. (Figure Step 18A and 18B)

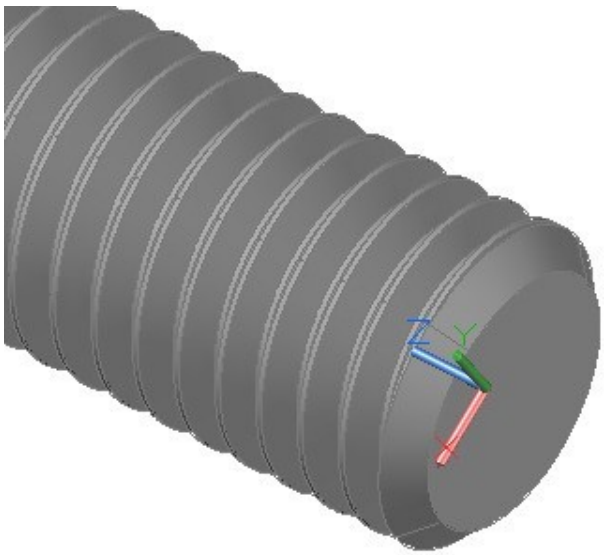


Figure Step 18A

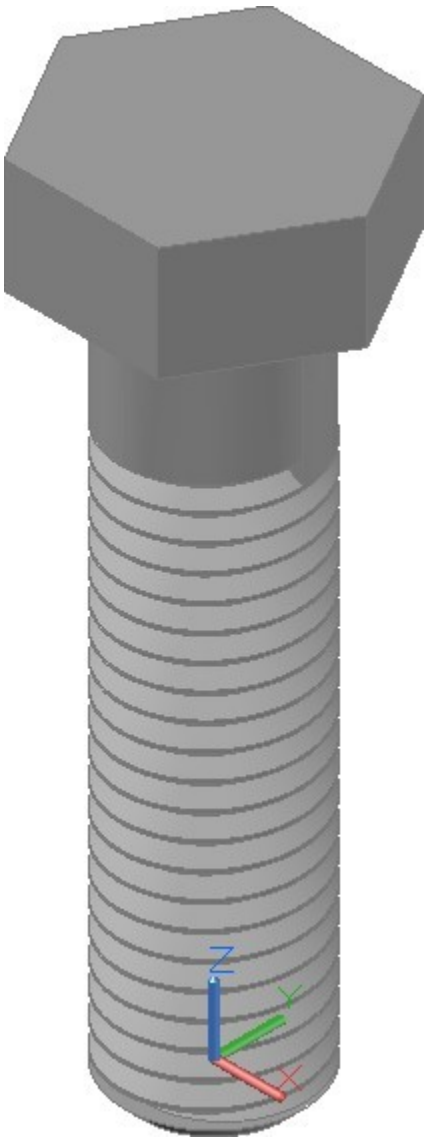


Figure Step 18B

Step 19

On layer: Solid 8, anywhere in model space, draw the solid model of the nut to match the figure. Do not draw the hole in the centre. (Figure Step 19A and 19B)

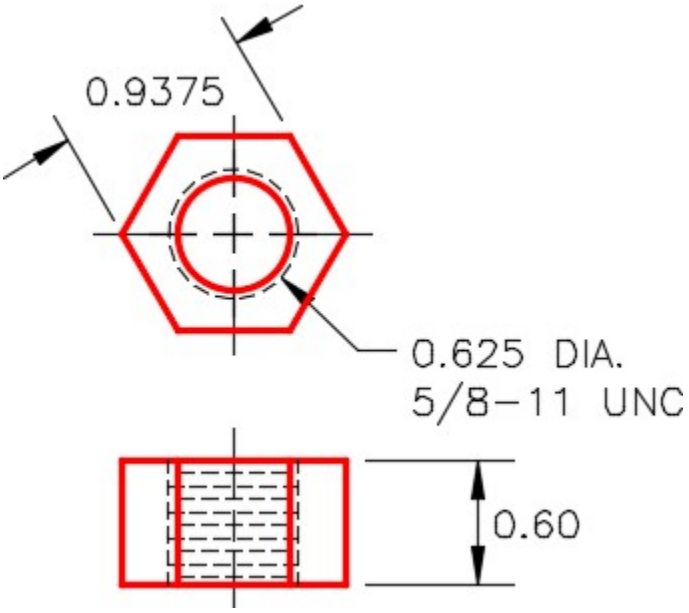


Figure Step 19A
Dimensioned Multiview Drawing

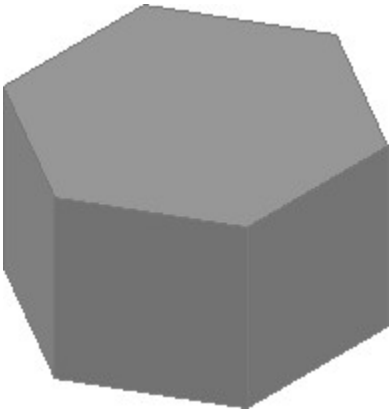


Figure Step 19B

Step 20

On layer: Center Line, draw a line on the centerline of the bolt and the nut. The length of the lines is not important. (Figure Step 20)

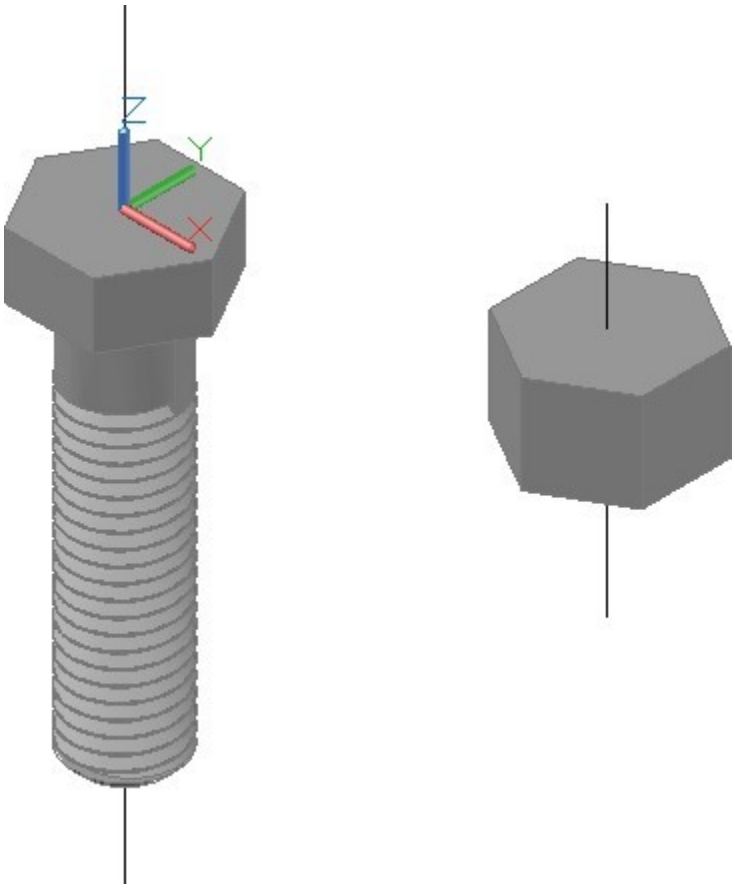


Figure Step 20

Step 21

Using the centerlines to snap to, copy the bolt exactly in the centre of the nut. The vertical location is not important as long as the nut is totally in the threaded area. (Figure Step 21)

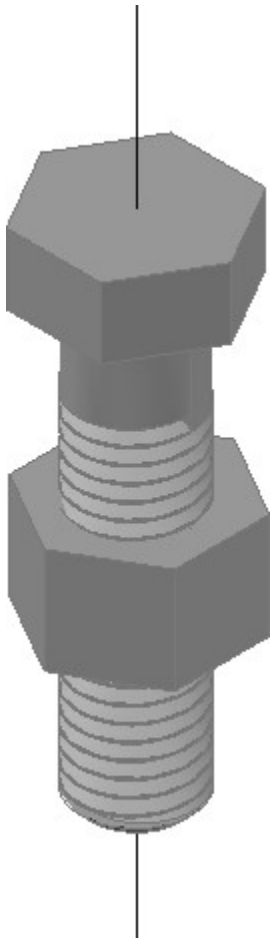


Figure Step 21

Step 22

Using the SUBTRACT command, Subtract the bolt from the nut to leave the nut threaded. (Figure Step 22)

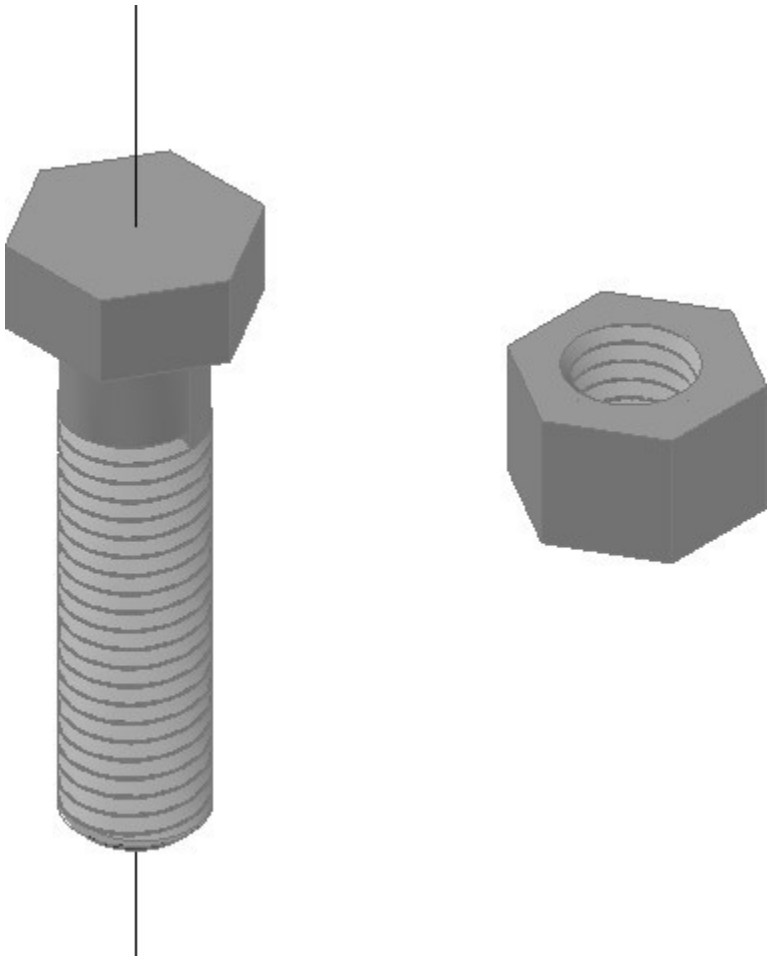


Figure Step 22

AUTHOR'S COMMENTS: This is a little trick I use to easily draw the threads on the inside of the nut.

Step 23

Using the centerlines, copy the solid models to create an assembly solid model. After you assemble the two parts by snapping to the centerlines, with Ortho enabled, move the nut, by eye, to locate it vertically. Do not union the bolt and nut together. (Figure Step 23A and 23B)

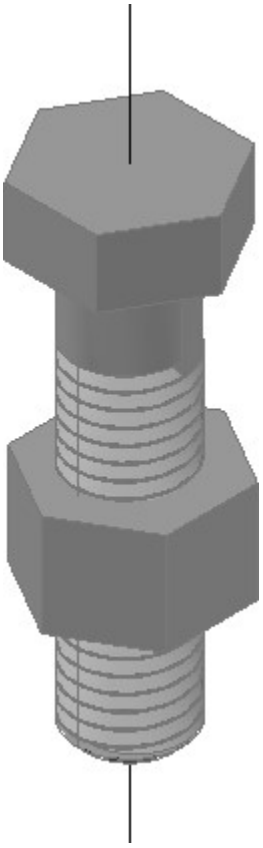


Figure Step 23A

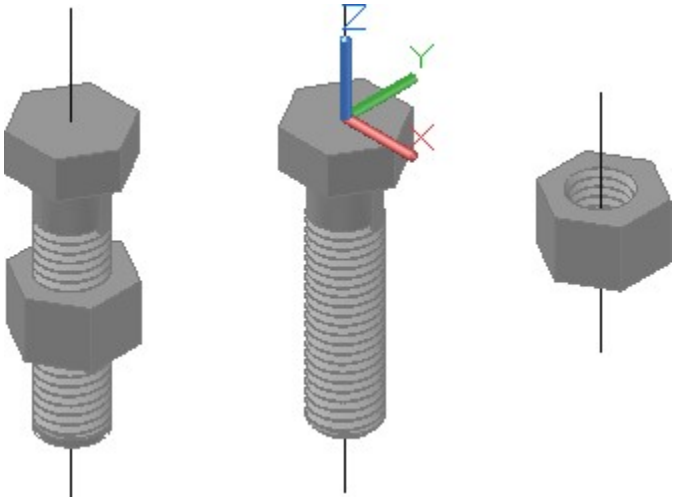


Figure Step 23B

Step 24

Save and close the drawing.

Key Principles

Key Principles in Module 23

1. Coils, springs and threads are constructed by drawing a profile and then using the SWEEP command, sweeping it along a path that is a helix drawn with the HELIX command.
2. A helix is constructed by specifying one or more setting that consist of the base radius, top radius, height, number of turns, turn height or twist direction.

Module 24 Loft

Learning Outcomes

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

1. Describe and apply the LOFT command to create solid or surface models by blending them between two or more cross sections.

Loft

A *loft* is a blend or transition solid or surface that connects two or more cross sections. The cross sections, also called *profiles*, can be the same or different shapes. See Figure 24-1. The cross sections can also be projected along a path.

The most common drawing objects that can be used for the cross sections are 2D and 3D splines, 2D polylines, 2D solids, arcs, circles, ellipses, elliptical arcs, and lines. For a complete list, see the AutoCAD Help files.

The path must be a single open or closed object. The most common objects that can be used for the path are 2D and 3D splines, 2D and 3D polylines, helices, arcs, circles, ellipses, elliptical arcs, and lines. See AutoCAD's Help.

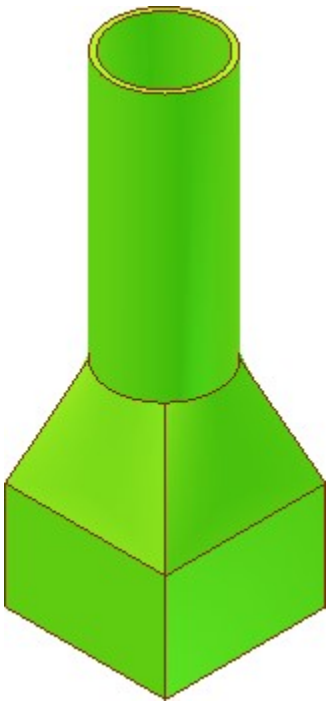
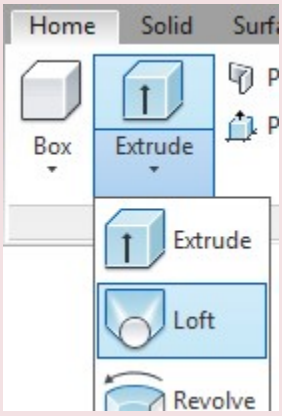


Figure 24-1
A Loft

AutoCAD Command: LOFT

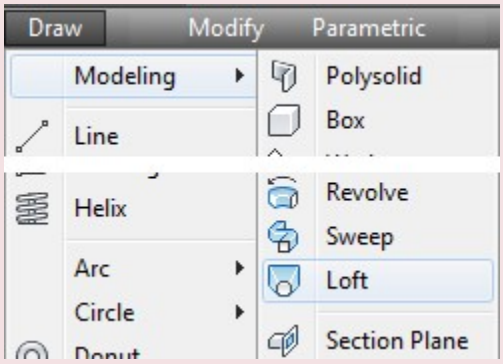
The LOFT command is used to create a solid or surface by blending two or more cross sections with the same or different shapes.

Shortcut: none



Home Ribbon

Modeling Toolbar



Draw Pull-down

WORK ALONG: Using the LOFT Command

Step 1

Using the NEW command, start a new drawing using template: 3D Layout English.

Step 2

Save and name the drawing: AutoCAD 3D Workalong 24-1.

Step 3

Set the current visual style to 3D Wireframe, the current layer to Profile, the current view to SE Isometric, and the current UCS to Right.

Step 4

Using Figure Step 4A as a reference, draw a 12 × 12 inch closed pline. Start the bottom left corner at 0,0,0 as shown in the figure. (Figure Step 4A and 4B)

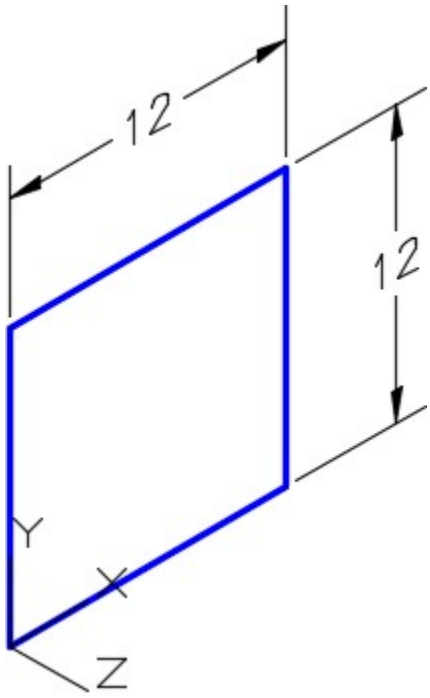


Figure Step 4A

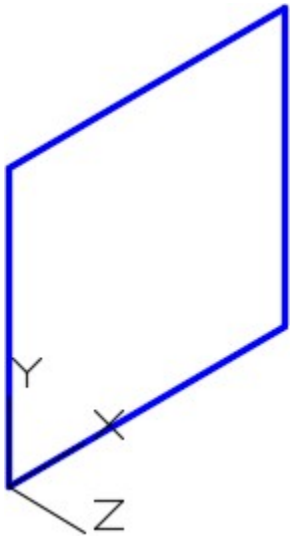


Figure Step 4B

Step 5

Using the Properties window, ensure that the square is a closed pline.

AUTHOR'S COMMENTS: If the square is not a closed object, the LOFT command will create a surface and not a solid.

Step 6

Copy the closed pline 6 inches in the positive Z direction. (Figure Step 6)

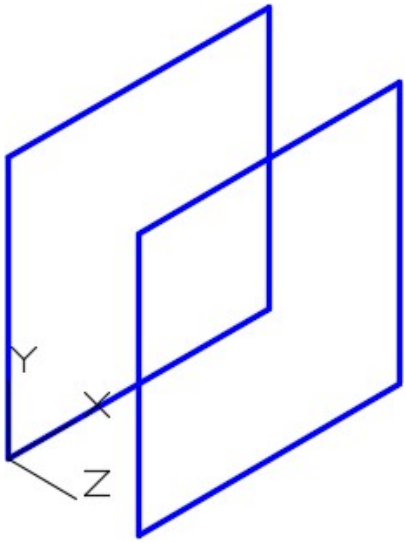


Figure Step 6

Step 7

Draw a 6 inch diameter circle exactly in the centre of the copied square. (Figure Step 7)

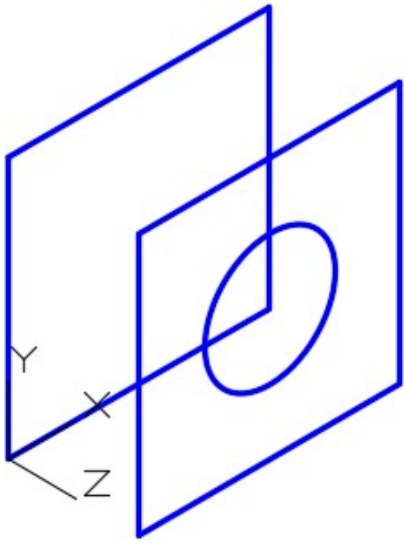


Figure Step 7

AUTHOR'S COMMENTS: You will have to draw a construction line to do this.

Step 8

Move the circle 8 inches in the positive Z direction. (Figure Step 8)

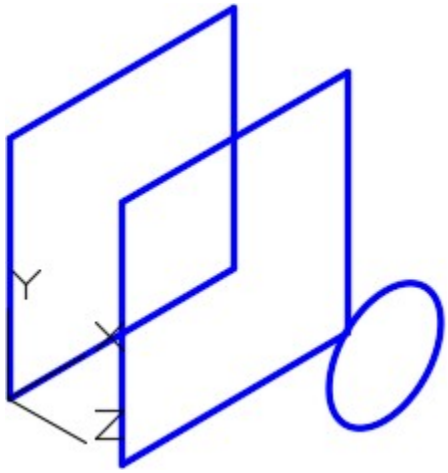


Figure Step 8

Step 9

Change the current UCS to Front and the set layer: Path as the current layer.

Step 10

Draw a pline 36 inches in the positive X direction and then 24 inches in the negative Y direction. Start it at the centre of the circle. (Figure Step 10)

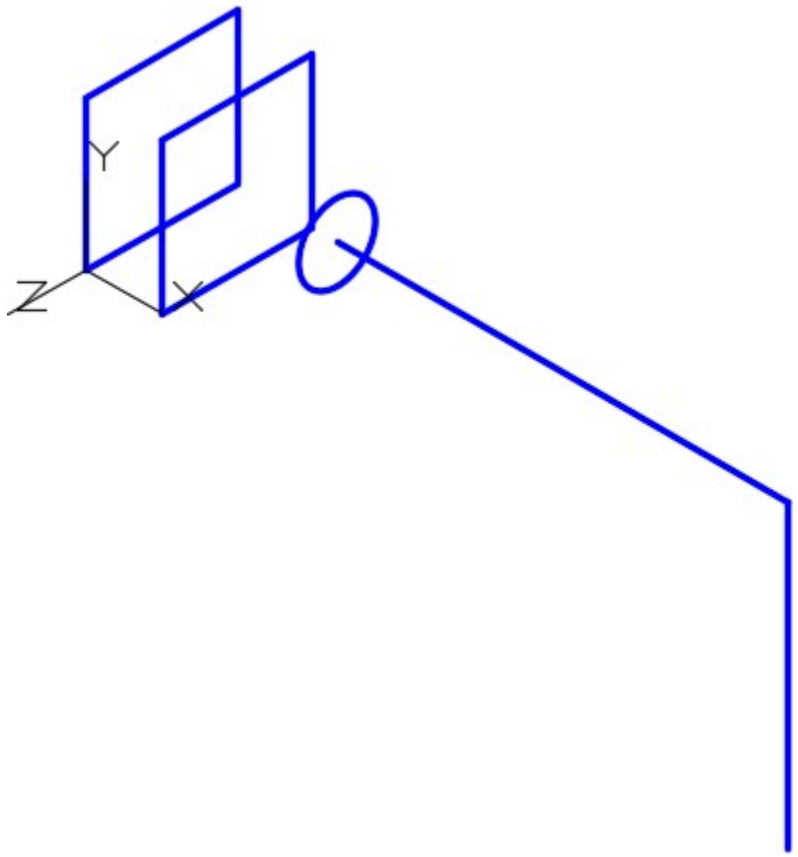


Figure Step 10

AUTHOR'S COMMENTS: The system variable DELOBJ is used to control if the cross sections and paths are kept or deleted when the solid or surface is created in a loft.

Step 11

Fillet the pline with a radius of 12 inches. (Figure Step 11)

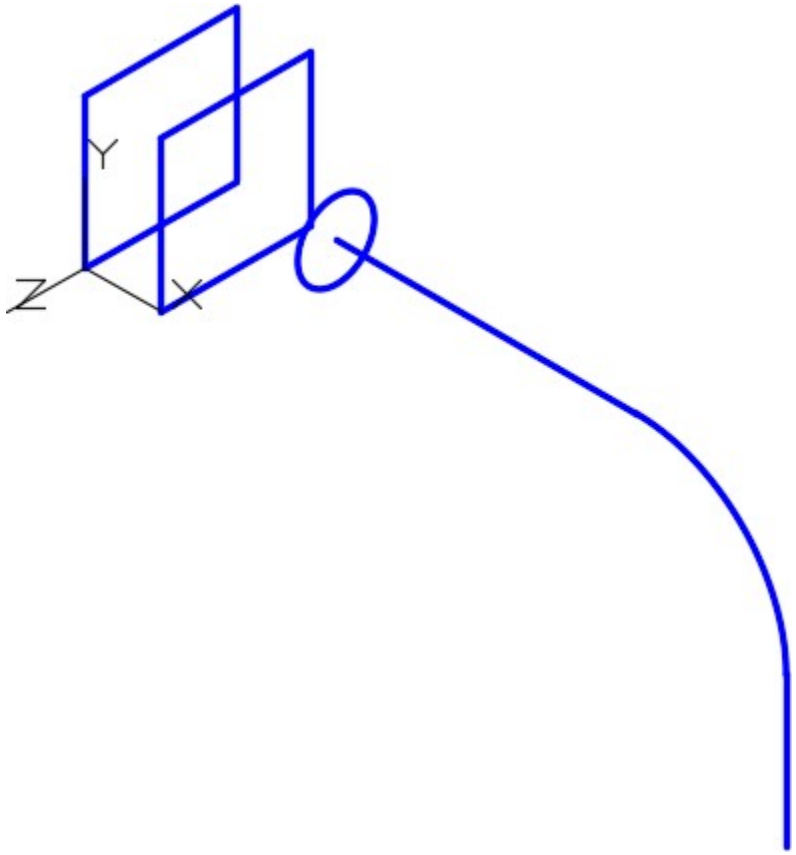


Figure Step 11

Step 12

Change the current UCS to Top and the set layer: Profile as current layer.

Step 13

Draw a 6 diameter circle locating its centre at the bottom end of the pline that you just drew. (Figure Step 13)

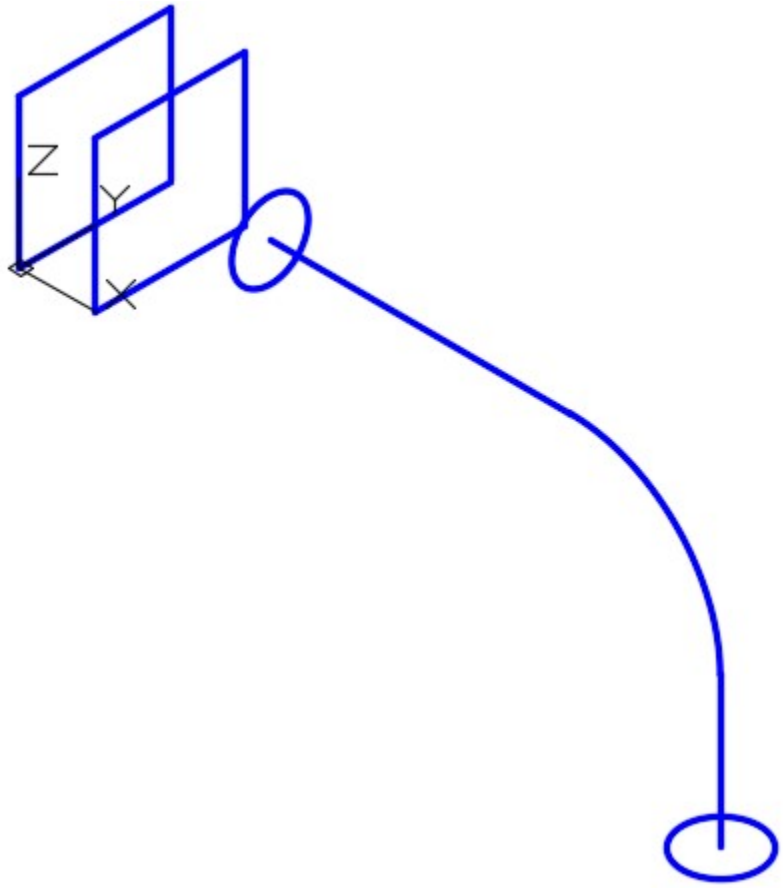


Figure Step 13

Step 14

After drawing the necessary construction lines, draw a 3 x 8 closed rectangular pline with its centre located at the end of the pline. (Figure Step 14A and 14B)

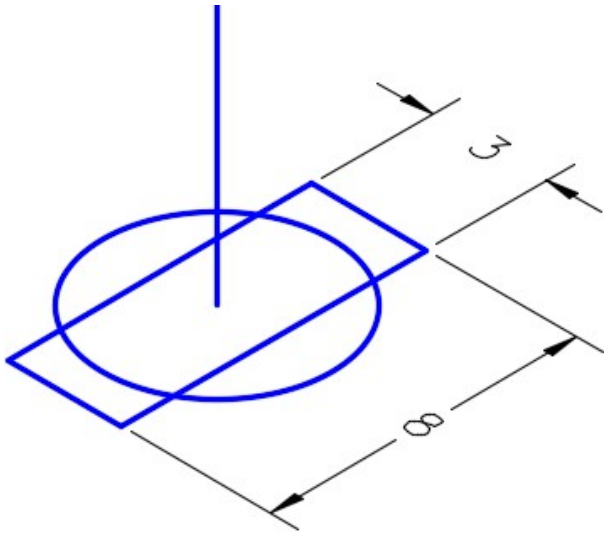


Figure Step 14A

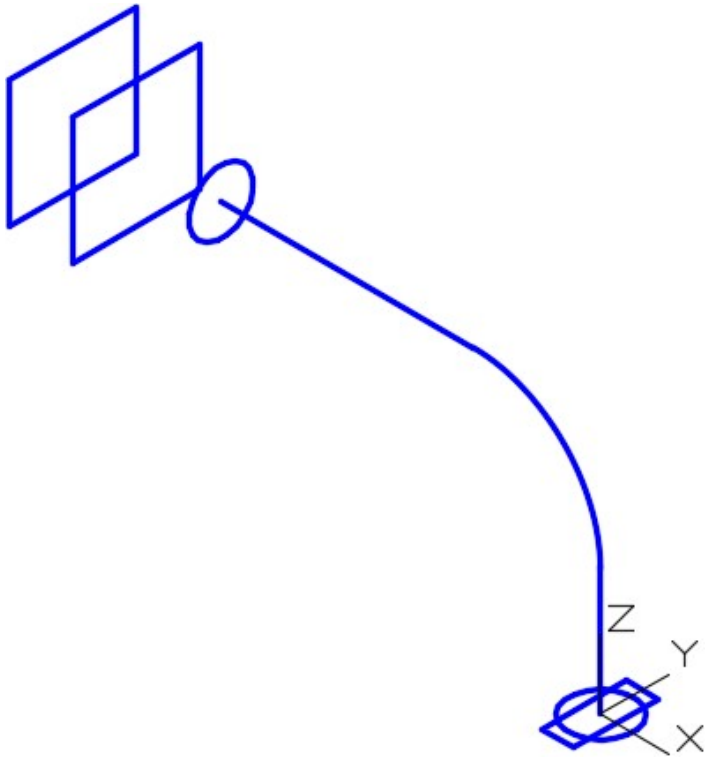


Figure Step 14B

Step 15

Move the rectangular pline 10 inches in the negative Z direction. (Figure Step 15)

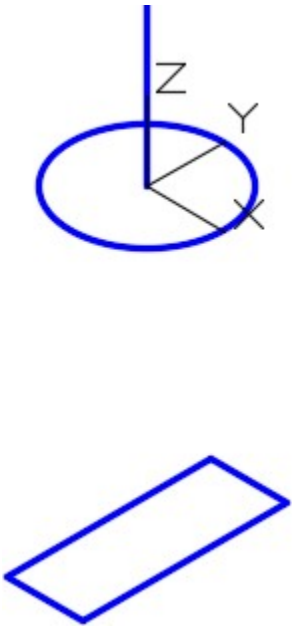


Figure Step 15

Step 16

Set the current view to SE Isometric. (Figure Step 16)

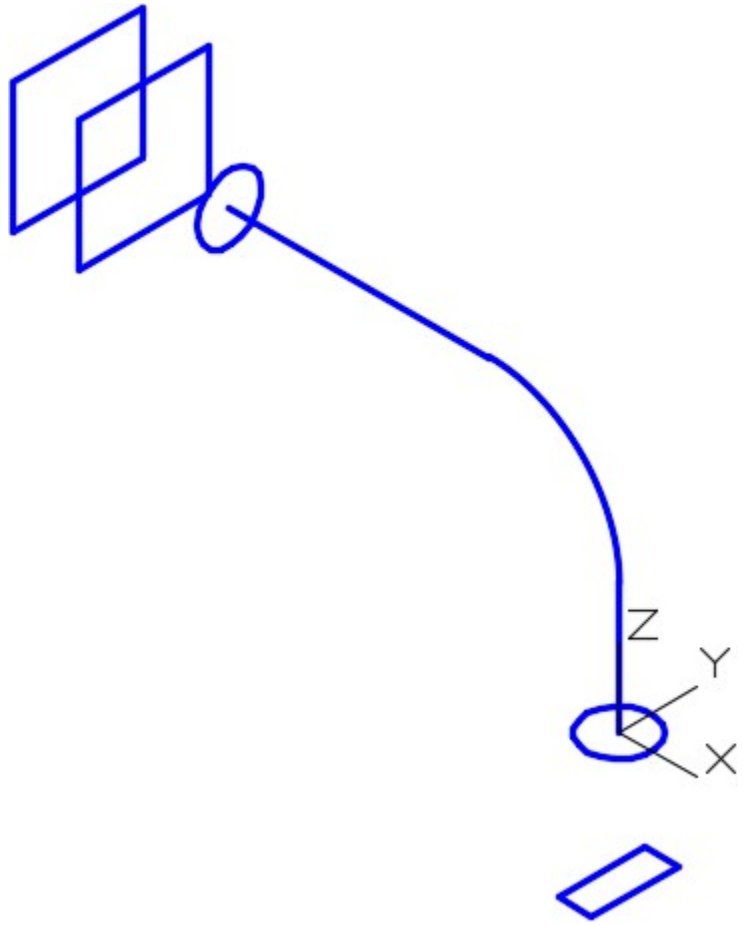


Figure Step 16

Step 17

Ensure that the system variable DELOBJ is set to 0.

Step 18

Set the current layer to Solid 4 and the system variable ISOLINES to 4.

Step 19

Enter the LOFT command, as shown below, and select the two squares for the cross sections. (Figure Step 19)

Command: **LOFT**

Current wire frame density: ISOLINES=4, Closed profiles creation mode = Solid

Select cross sections in lofting order or [POint/Join multiple edges/MOde]: 1 found

Select cross sections in lofting order or [POint/Join multiple edges/MOde]: 1 found, 2 total

Select cross sections in lofting order or [POint/Join multiple edges/MOde]: 2 cross sections selected

Enter an option [Guides/Path/Cross sections only/Settings] <Cross sections only>:

(Press Enter)

Command:

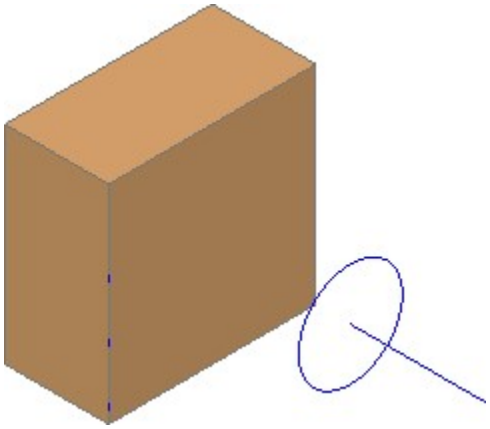


Figure Step 19

Step 20

Turn off layer: Solid Off. Change the layer of the solid that you just created in Step 18 to layer: Solid Off. (Figure Step 20)

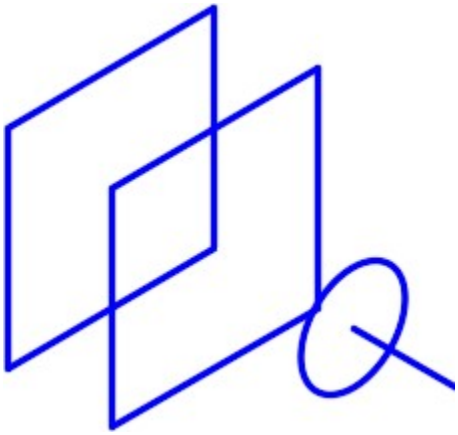


Figure Step 20

Step 21

Using the LOFT command, create a loft selecting the square and circle as the cross sections. (Figure Step 21)

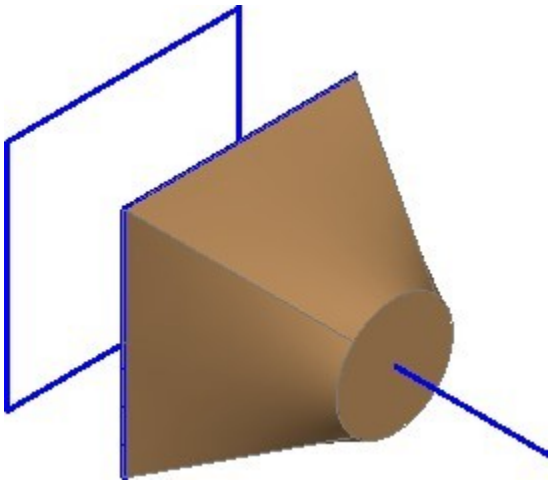


Figure Step 21

Step 22

Change the layer of the solid that you just created in the loft to layer: Solid Off.

Step 23

Enter the LOFT command, as shown below, to draw a loft between two cross sections that follow a path. Select the two circles for the cross sections, and the pline as the path. (Figure Step 23A and 23B)

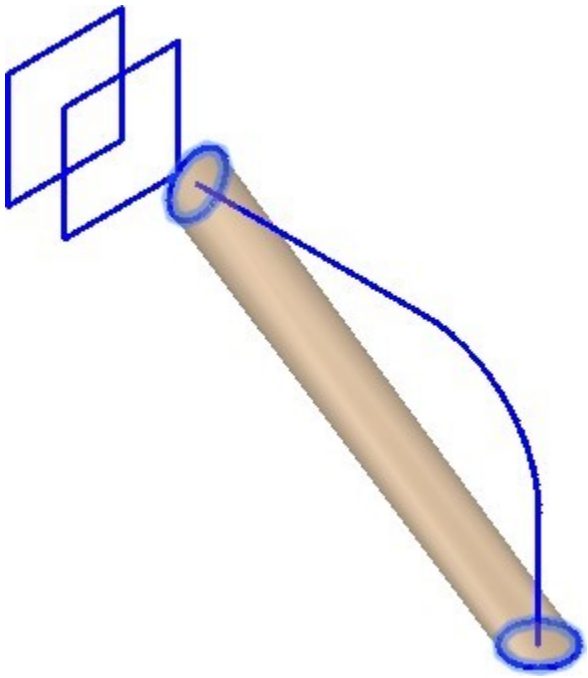


Figure Step 23A

Command: **LOFT**

Current wire frame density: ISOLINES=4, Closed profiles creation mode = Solid

Select cross sections in lofting order or [Point/Join multiple edges/MOde]: 1 found

Select cross sections in lofting order or [POint/Join multiple edges/MOde]: 1 found,2 total

Select cross sections in lofting order or [POint/Join multiple edges/MOde]: 2 cross sections selected

Enter an option [Guides/Path/Cross sections only/Settings] <Cross sections only>: **P**

Select path profile: 1 found

(Select the pline.)

Command:

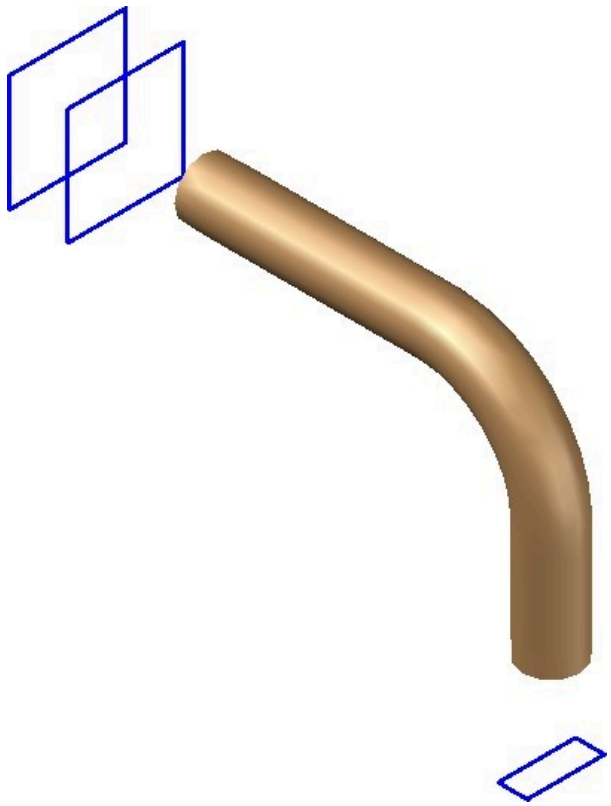


Figure Step 23B

Step 24

Using what you already learned in the workalong, change the layer of the solid created in Step 23 to layer: Solid off. Create a solid using the loft command between the circle and the rectangle. (Figure Step 24)

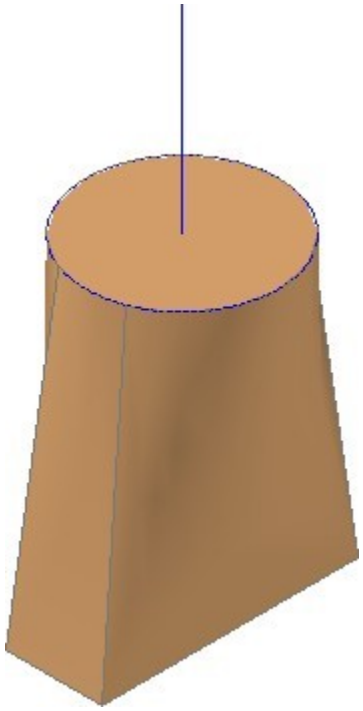


Figure Step 24

Step 25

Turn off layers: Profile and Path.

Step 26

Change the layer of all solids, that are on layer: Solid Off, to layer: Solid 4.

Step 27

Using the UNION command, union all solids to form a single solid. (Figure Step 27)

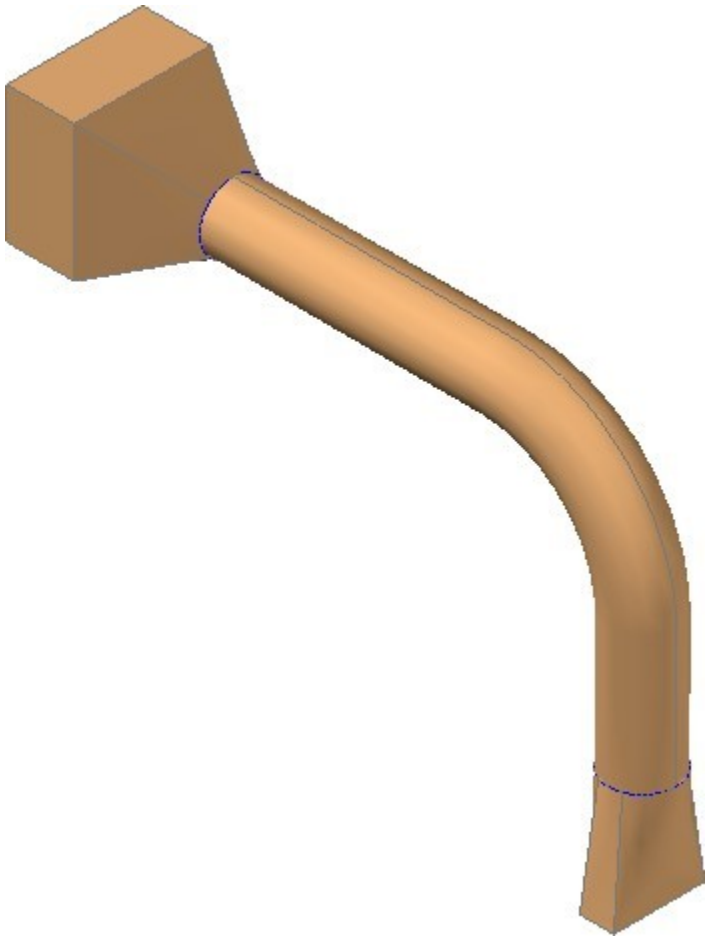


Figure Step 27

Step 28

Save and close the drawing.

Key Principles

Key Principles in Module 24

1. A loft, created with the LOFT command, is a blend or transition solid or surface that connects two or more cross sections. The cross sections, sometimes called profiles, can be the same or different shapes. The cross sections can also be projected along a path.

Lab Exercise 24-1

Time allowed: 60 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 24-1	3D Layout Metric	Millimeters

Step 1

On layer: Solid 3, draw a solid model of the object shown in the figure by drawing cross sections and then using the LOFT command. (Figure Step 1)

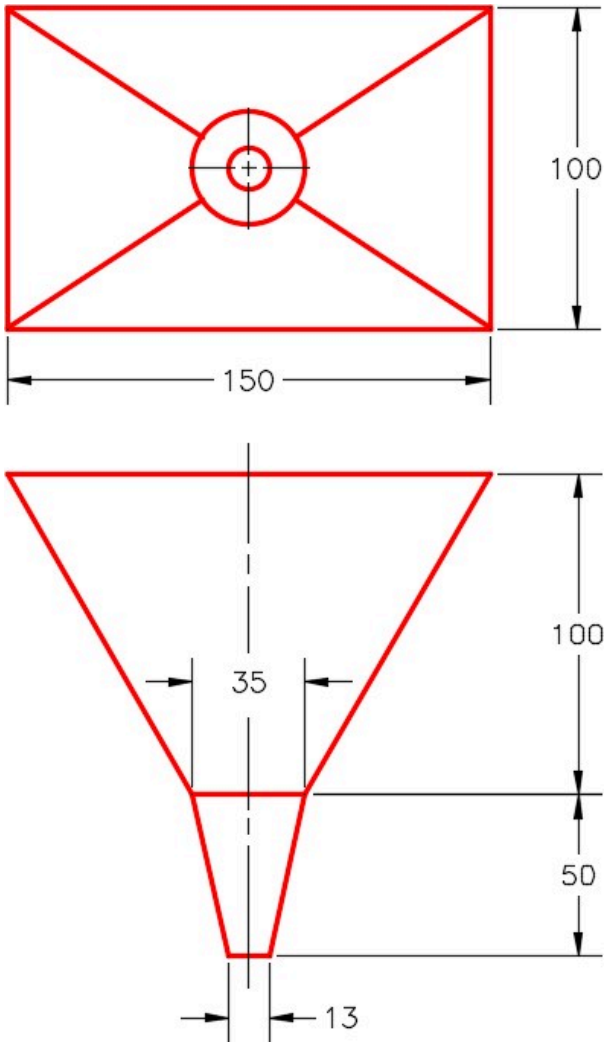


Figure Step 1
Dimensioned Multiview Drawing

Step 2

Draw the cross sections on layer: Profile.

Step 3

Union the solids to create one single solid model. (Figure Step 3)

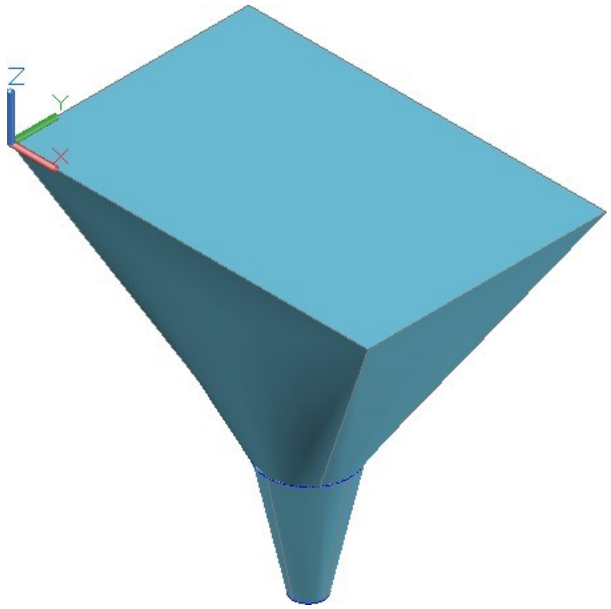


Figure Step 3

AUTHOR'S COMMENTS: This solid model is not complete. In Module 29, you will be shelling the model to remove the inside to create a large funnel.

Step 4

Save and close the drawing.

Lab Exercise 24-2

Time allowed: 60 minutes.

Drawing Name	Template	Units
AutoCAD 3D Lab 24-2	3D Layout English	Inches

Step 1

On layer: Solid 2, draw a solid model of the object shown in the figure by drawing cross sections and then using the LOFT command. (Figure Step 1)

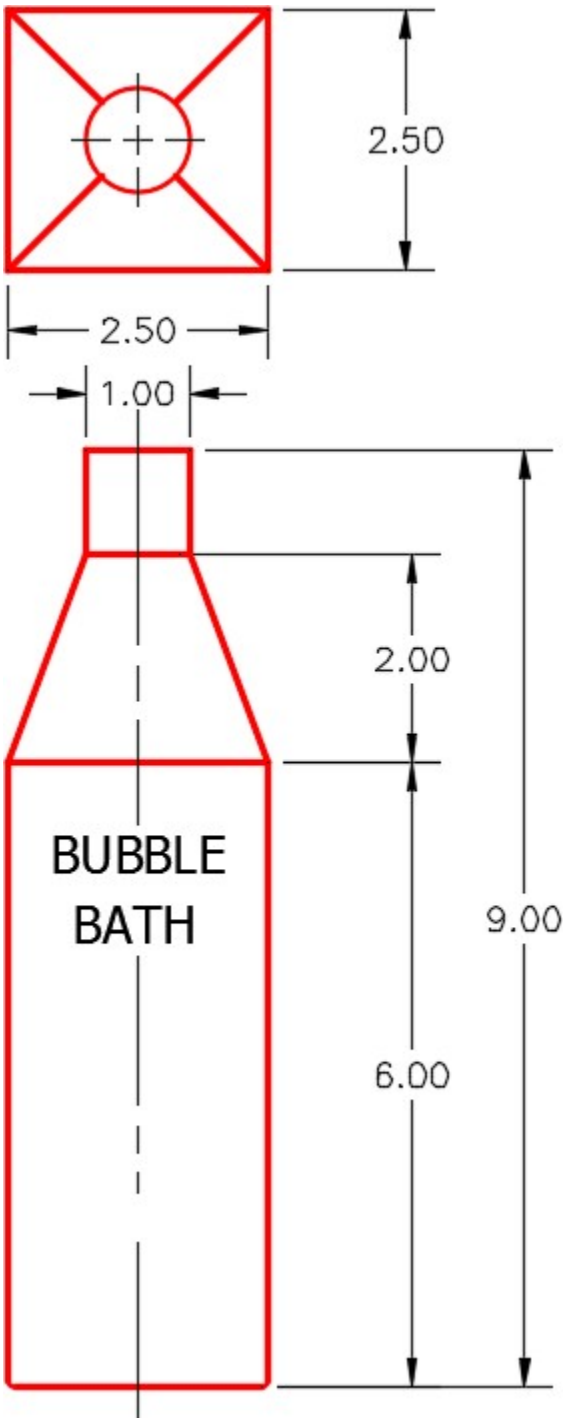


Figure Step 1
Dimensioned Multiview Drawing

Step 2

Draw the cross sections on layer: Profile.

Step 3

Draw the text on layer: Text after you create the solid. Use the font: Tohoma. Select the height of your choice.

Step 4

Union the solids together to create one single solid model. (Figure Step 4)

Figure Step 4

AUTHOR'S COMMENTS: This solid model is not complete. In Module 29, you will be shelling the model to remove the inside to create a bottle.

Step 5

Save and close the drawing.

Module 25 Mass Properties

Learning Outcomes

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

1. Describe and apply the MASSPROP commands to find the mass properties of solid models.

Geometry Lesson: Mass Properties

The *mass properties* of a solid object are its mass, centre of gravity, centroid, volume, moments of inertia, products of inertia and radii of gyration. Since the mass refers to a solid object, you can only find the mass properties of a solid model. AutoCAD expresses the mass properties in inches. The mass properties include the following:

Mass	the quantity of the matter contained in the solid object. This is determined by the density of the material and the volume of the solid. Mass is not dependent on gravity which makes it different but proportional to weight. Mass is used when considering a measure of a solid's resistance to inertia.
Volume	The amount of space occupied by the solid object.
Bounding box	The smallest possible box that the object will fit inside. The box is constructed on the XYZ axis.
Centroid	Geometrical centre of the object. If the density of the object is uniform, the centroid is the centre of the mass or the centre of gravity.
Moments of Inertia	Is the measure of how the mass is distributed around XYZ axis of the current UCS. The values in the moments of inertia are a measure of the objects resistance to angular acceleration and are used when calculating the stress on the object.
Products of Inertia	Similar to moments of inertia, the products of inertia measures the objects resistance to the angular acceleration, except it measures it around each 2D Axis (XY YZ XZ).
Radii of Gyration	If the object was a uniform solid, with no holes, the radii of gyration would be the radius the moments of inertia would use to perform its calculations around each axis.
Principal Moments of XYZ Direction	This is a measure of the maximum moments of inertia around an axis. When they are maximum, the principle moments are zero.

MUST KNOW: Since AutoCAD cannot assign a material to a solid model, the volume and mass of a solid will be identical. This is because AutoCAD assigns the density of 1 to all solids. To calculate the weight of the solid model, do the following:

The formula to calculate mass or weight of the solid is:

mass= volume X density

Following is a list of the density for three common materials. You can find the density of all materials by searching for lists on the internet.

Steel – 0.2836 lb/in3

Copper – 0.3237 lb/in3

Aluminum – 0.0975 lb/in3

For example, the weight of a cubic foot of steel compared to a cubic foot of aluminum:

Steel – 1728 in3 x 0.2836 = 490.06 lbs

Aluminum – 1728 in3 x 0.0975 = 168.48 lbs

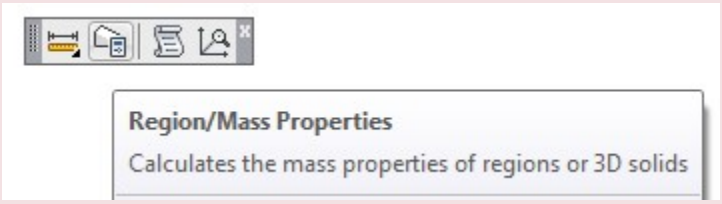
Hint: (12″X12″X12″=1728 in3)

Keep in mind that AutoCAD always works in inches.

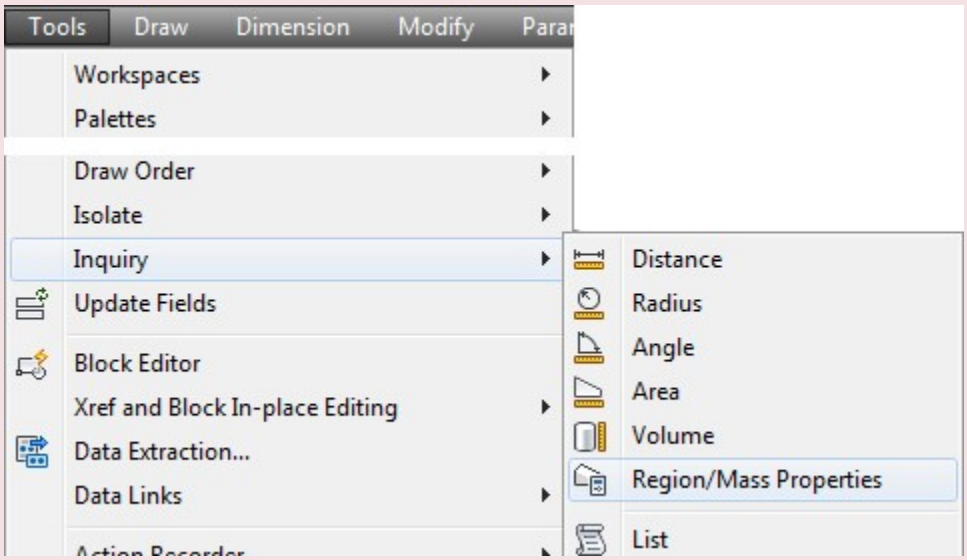
AutoCAD Command: MASSPROP

The MASSPROP command is used to compute and display the mass properties of a solid model or a region.

Shortcut: none



Inquiry Toolbar



Tools Pull-down

WORK ALONG: Finding the Mass Properties of Solids and Regions

Step 1

Start a new drawing using the template: 3D Layout English. Save the drawing with the name: AutoCAD 3D Workalong 25-1

Step 2

Using the INSERT command, insert the block: AutoCAD 3D Workalong 25-1 at the coordinates 0,0,0. Explode the block. Do not explode the solids. Change the solid models to layer: Solid 2.

AUTHOR'S COMMENTS: Use the Properties window to ensure that the objects are solid models.

Step 3

Set the current view to SE Isometric, the current UCS to World and the current visual style to Realistic. Your drawing should appear as shown in the figure. (Figure Step 3)

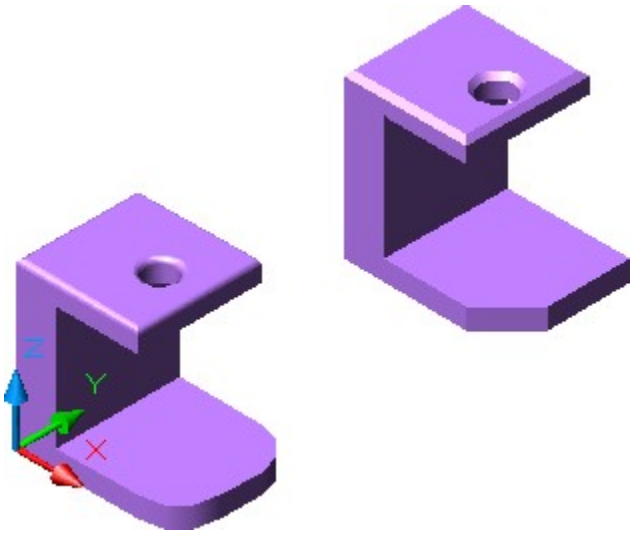


Figure Step 3

Step 4

Enter the MASSPROP command, as shown below, selecting the solid model on the left side.

Command: **MASSPROP**

Select objects: 1 found

(Select the solid model on the left.)

Select objects:

SOLIDS

Mass:	19.0166
Volume:	19.0166Creating Solid Models Using Boxes, Wedges and Cylinder
Bounding box:	X: 0.0000—— 4.0000 Y: 0.0000—— 3.0000 Z: 0.0000—— 4.0000
Centroid:X:	1.1528 Y: 1.5000 Z: 1.8564
Moments of inertia:	X: 159.7292 Y: 147.0995 Z: 101.3495
Products of inertia:	XY: 32.8824 YZ: 52.9538 ZX: 35.1506
Radii of gyration:	X: 2.8982 Y: 2.7812 Z: 2.3086
Principal moments and X-Y-Z directions about centroid:	I: 52.9685 along [0.9625 0.0000 0.2712] J: 56.2930 along [0.0000 1.0000 0.0000] K: 31.7293 along [-0.2712 0.0000 0.9625]

Write analysis to a file? [Yes/No] <N>:

(Accept No as the default.)

Command:

Step 5

Enter the MASSPROP command again and select the solid model on the right side.

Command: **MASSPROP**

Select objects: 1 found

(Select the solid on the right.)

Select objects:

SOLIDS

Mass:	18.8792
Volume:	18.8792
Bounding box:	X: 0.0000—— 4.0000 Y: 8.0000—— 11.0000 Z: 0.0000—— 4.0000
Centroid:X:	X: 1.1424 Y: 9.5154 Z: 1.8485
Moments of inertia:	X: 1824.7216 Y: 144.5649 Z: 1766.8808
Products of inertia:	XY: 205.9529 YZ: 331.6084 ZX: 34.1723
Radii of gyration:	X: 9.8312 Y: 2.7672 Z: 9.6741
Principal moments and X-Y-Z directions about centroid:	I: 52.3639 along [0.9426 0.1825 0.2795] J: 55.5396 along [-0.1827 0.9828 -0.0257] K: 31.1772 along [-0.2793 -0.0269 0.9598]

Write analysis to a file? [Yes/No] <N>: **Y**

(This time enter Y to write the data to a file.)

Command:

Step 6

Entering Y for yes will open the Create Mass and Area Properties File dialogue box. Save the file in the Lab Exercises folder assign it the same name as the drawing. The extension .mpr is automatically added.

AUTHOR'S COMMENTS: The extension for the Mass and Area Properties File is .mpr. Once you save the mass properties data in the text file you can use the data in other programs or calculations. This saves you retyping the numbers or writing them down and eliminates the possibility of making errors.

AUTHOR'S COMMENTS: The Notepad program that comes with Windows is a very useful program to use when working with text files in AutoCAD. It is just a simple text editor but what makes it unique is that it works with pure text characters only. It does not add any special characters like fonts and bolding. This is important when the AutoCAD program has to use the text file.

Step 7

Start Notepad. (Figure Step 7)

Figure Step 7

Step 8

In Notepad, open the file: AutoCAD 3D Workalong 25-1.mpr in the folder you saved it in. (Figure Step 8A, 8B, and 8C)

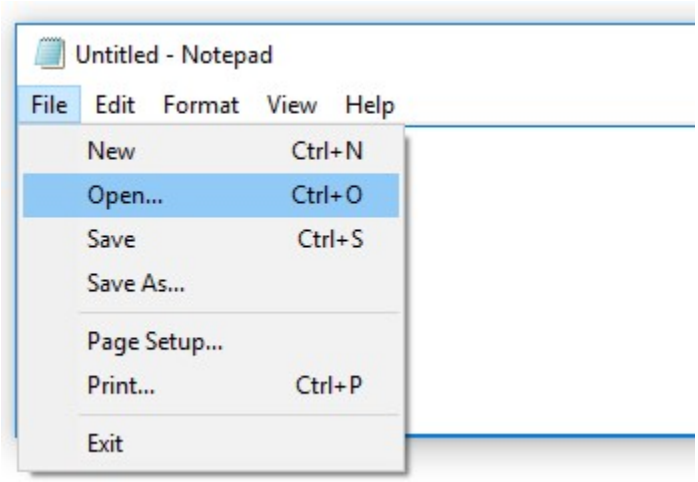


Figure Step 8A

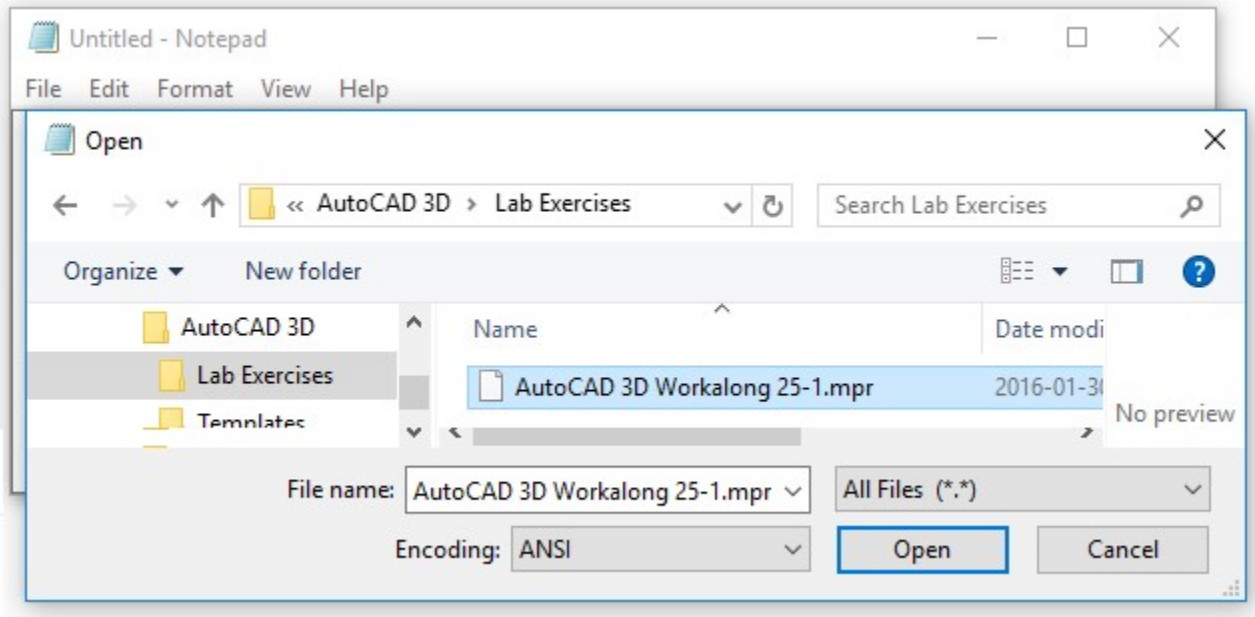


Figure Step 8B

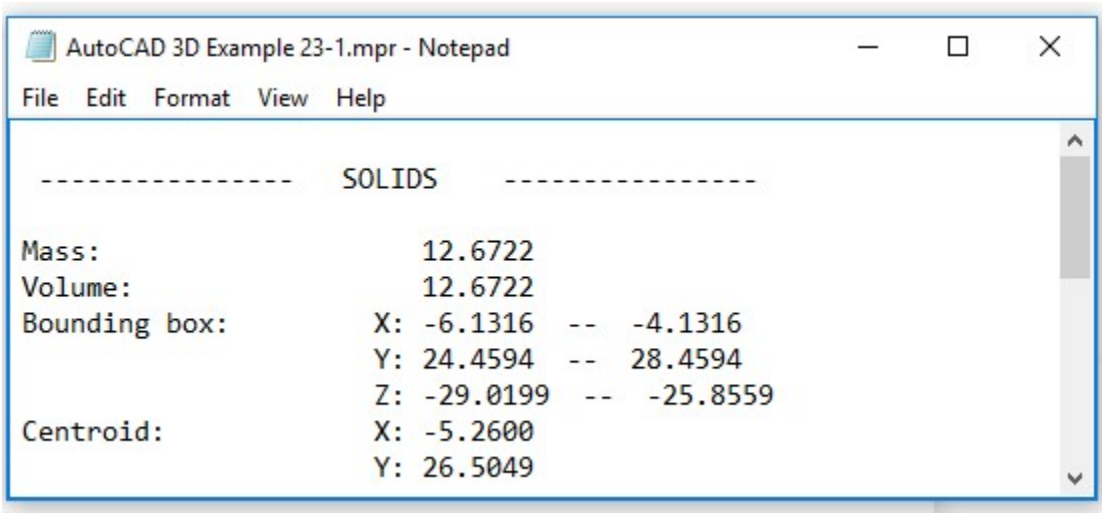


Figure Step 8C

Step 9

In the Notepad window, you can find the centroid as shown below. This is the centre of gravity of the model.

Centroid: X: 1.1424

Y: 9.5154

Z: 1.8485

Step 10

On layer: Construction, insert a point at the centroid of the model using the POINT command as shown below. Change the point style and your model should appear similar to the figure. The shaded model rotated also is displaying a point at the centroid. (Figure Step 10A and 10B)

Command: **POINT**

Current point modes: PDMODE=34

PDSIZE=0.0000

Specify a point: **1.1424,9.5154,1.8485**

Command:

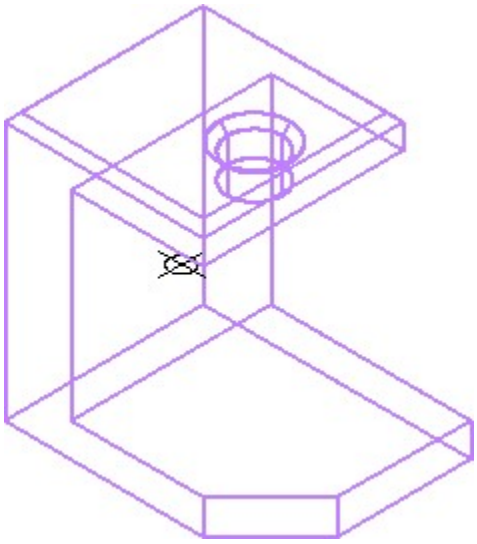


Figure Step 10A

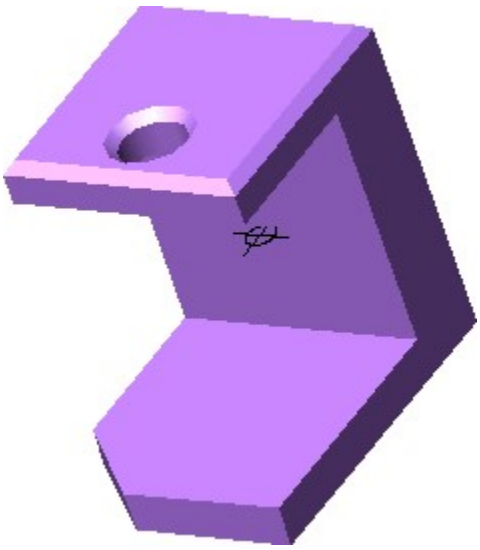


Figure Step 10B

Step 11

Calculate and record the weight of the solid model made out of aluminum and steel. See MUST KNOW in Geometry Lesson: Mass Properties. Check your answers after Step 18.

Steel -_____ lbs

Aluminum -_____ lbs

AUTHOR'S COMMENTS: See the MUST KNOW under Geometry Lesson: Mass Properties.

Step 12

Using what you learned previously, offset a construction line 0.5 units from the far chamfered edge of the model on its right side. (Figure Step 12)

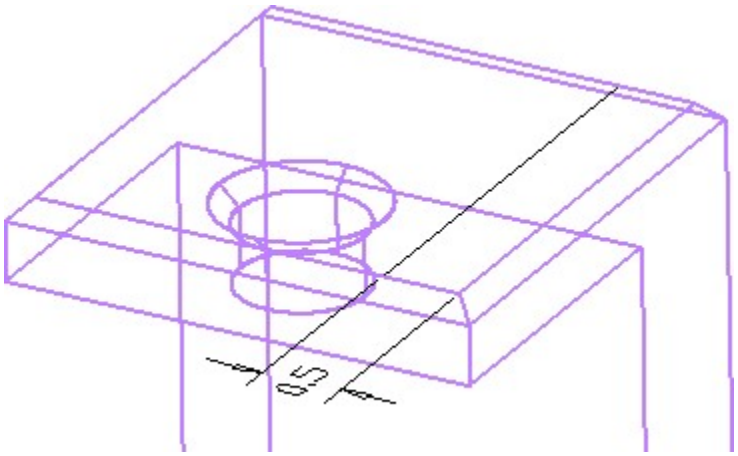


Figure Step 12

Step 13

Locate the Top UCS at the end of the line (Figure Step 13)

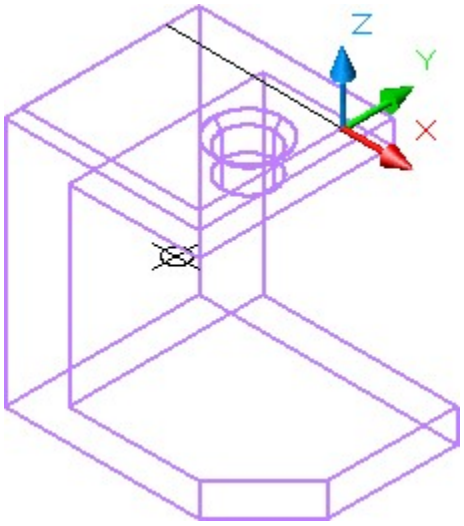


Figure Step 13

Step 14

Using what you learned in Module 20, use the SECTION command, as shown below, to create a section at the location of the construction line you just drew. (Figure Step 14)

Command: **SECTION**

Select objects: 1 found

Select objects: Specify first point on Section plane by

[Object/Zaxis/View/XY/YZ/ZX/3points] <3points>: **ZX**

Specify a point on the ZX-plane <0,0,0>:

Command:

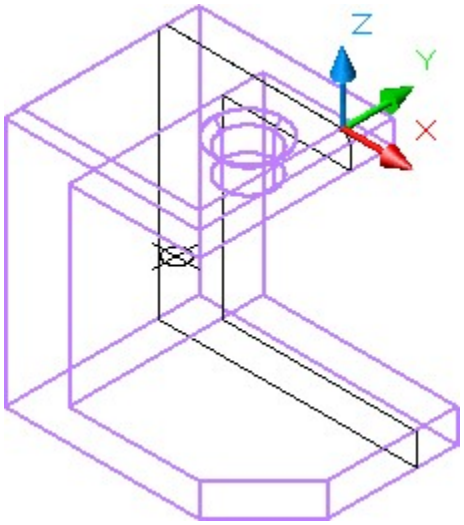


Figure Step 14

Step 15

Move the region you just created outside of the model. It is not important where you locate it. Shaded and rotated, your model should appear similar to the figure. (Figure Step 15).

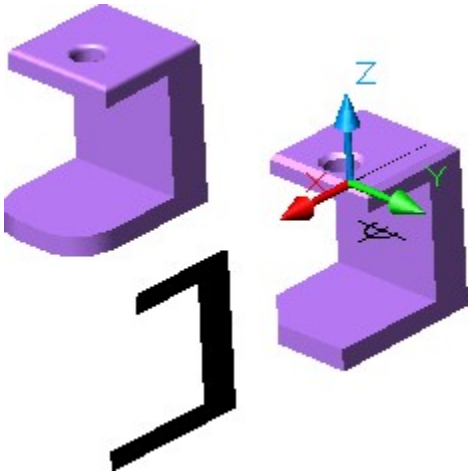


Figure Step 15

Step 16

Using the MASSPROP command, as shown below, find the mass properties of the region.

Command: **MASSPROP**

Select objects: 1 found

(Select the region.)

Select objects:

REGIONS

Area:	6.4844
Perimeter:	19.8536
Bounding box	X: 3.8918 — 7.8918 Y: 0.0000 — 0.0000 Z: -4.0000 — 0.0000
Centroid:	X: 5.0834 Y: 0.0000 Z: -2.1397

Write analysis to a file? [Yes/No] <N>: **N**

Command:

Step 17

Using what you learned earlier in this module, insert a point at the centroid of the region as shown in the figure. (Figure Step 17)

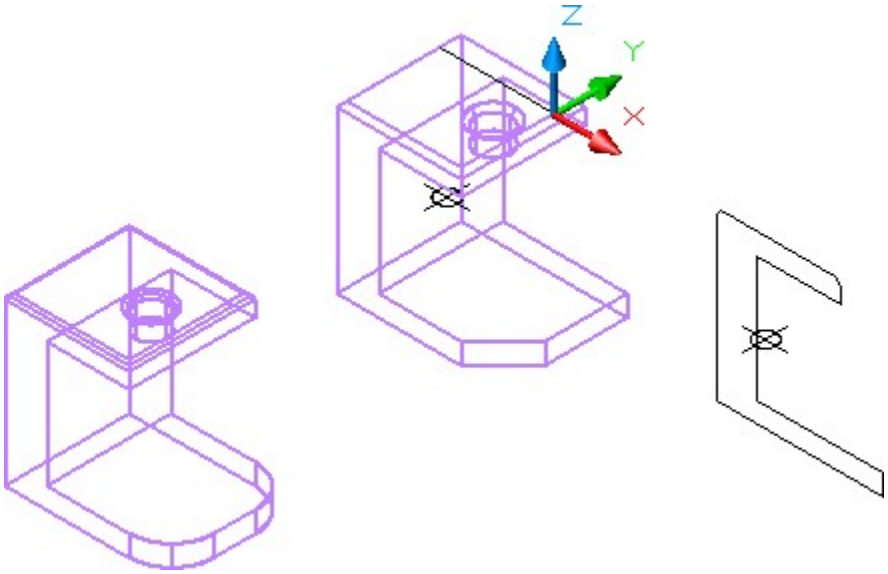


Figure Step 17

Step 18

Save and close the drawing.

Answers from Step 11

Steel – 5.3541 lbs

Aluminum – 1.8407 lbs

Key Principles

Key Principles in Module 25

1. Mass properties can only be found for solid models and regions.
2. The formula to calculate mass or weight of a solid is: mass= volume X density
3. AutoCAD always works in inches and reports the mass properties in inches regardless of the what units the drawing was constructed in.

Module 26 Competency Test No.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 the lab exercise without the aid of a key.

The AutoCAD 3D 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. For which AutoCAD objects can Mass Properties be found?
    A. Wireframes and Solids
    B. Regions and Faces
    C. Solids and Regions
    D. Regions and Wireframes
    E. Edges and Faces
2. What command is used to create a solid model by moving a profile along a selected path?
    A. LOFT

B. SHELL

C. HELIX

D. SWEEP

E. MOVEP

3. What formula is used to find the weight of a solid object?

A. inertia x mass + density

B. density x mass

C. volume + density

D. mass x volume

E. volume x density

4. By default, what extension file name will be assigned to the mass properties report generated from the MASSPROP command?

A. .MRP

B. .MAP

C. .TGA

D. .MPR

E. .MPP

5. What system variable setting controls if the original profile and path geometry are kept or deleted after being selecting in the SWEEP command?

A. DBMODE

B. ISOLINES

C. DELOBJ

D. SURFTAB

E. OBJDEL

6. What command is used to create a coil?

A. HELIX

B. SHELL

C. LOFT

D. SWEEP

E. MOVEP

7. What command is used to create a solid by blending two or more cross sections, that are closed objects, of different shapes?

A. SHELL

B. LOFT

C. HELIX

D. SWEEP

E. MOVEP

8. Which one of the following objects cannot be used as a path for the SWEEP command?

A. 2D Polyline

B. Ellipse

C. Circle

D. Trace

E. 3D Polyline

9. Which two of the following commands are most important commands used to create a thread?

A. LOFT and SHELL

B. SHELL and HELIX

C. HELIX and SWEEP

D. SWEEP and LOFT

E. MOVEP and HELIX

10. When creating a thread, which one of the following objects is used as the path in the SWEEP command?

A. Arc

B. Circle

C. Helix

D. 3D Spline

E. 3D Polyline

Lab Exercise 26-1 OPEN BOOK

Drawing Name	**Template**	**Units**
AutoCAD 3D Lab 26-1	3D Layout English	Inches

Step 1

Complete the following steps to draw the individual solid models and assemble them. They can be drawn anywhere in model space. (Figure Step 1A and 1B)

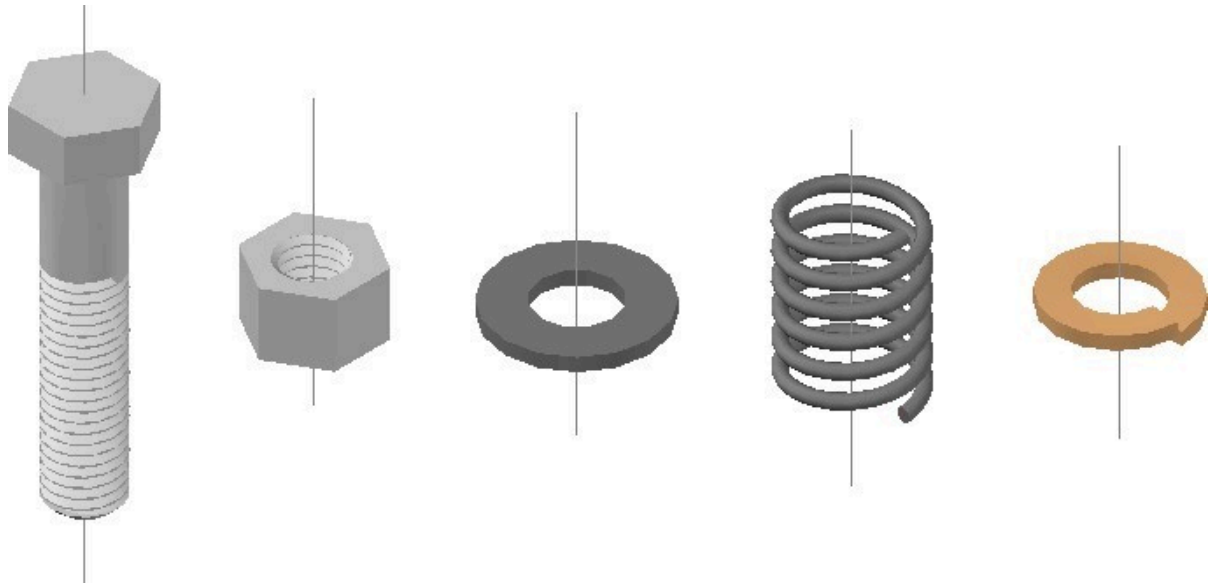


Figure Step 1A
Completed Solid Models – SE Isometric View

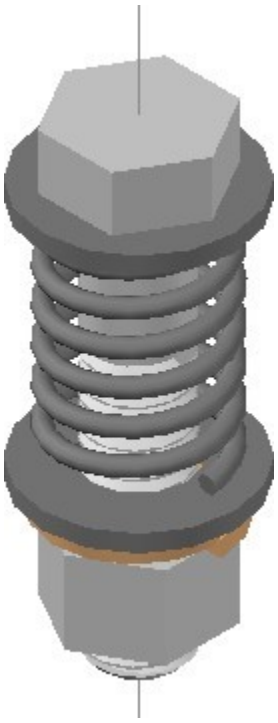


Figure Step 1B
Completed Assembly –
SE Isometric View

Step 2

Set the system variable DELOBJ to 0.

Step 3

For all solid models, draw the profiles on layer: Profile, the paths on layer: Path, the helixes on layer: Helix, the plines on layer: Pline, and the centre lines on layer: Center Line.

Step 4

Draw a solid model of the bolt on layer: Solid 9 using the dimensioned drawings as a reference. Draw a centre line in the model. (Figure Step 4A, 4B, and 4C)

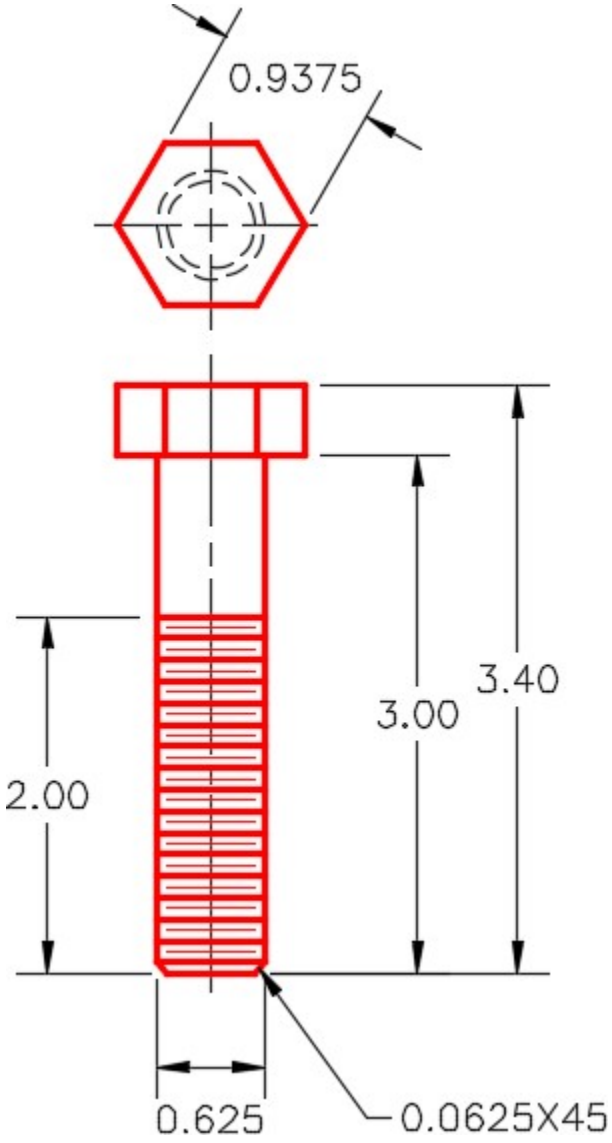


Figure Step 4A
Dimensioned Multiview Drawing of Bolt

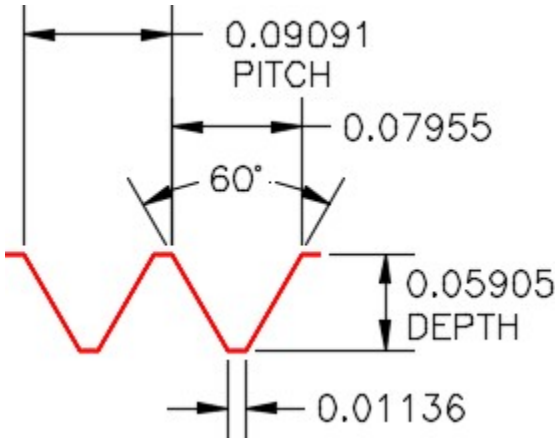


Figure Step 4B
Dimensioned Thread Details

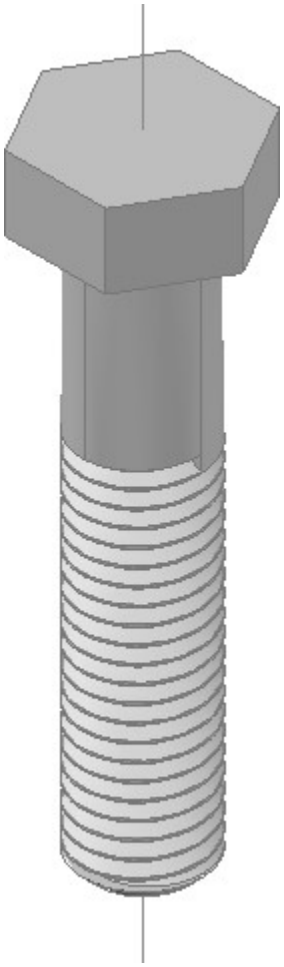


Figure Step 4C
Bolt –
SE Isometric View

Step 5

Draw a solid model of the nut on layer: Solid 9 using the dimensioned drawing as a reference. Draw a centre line in the model. (Figure Step 5A and 5B)

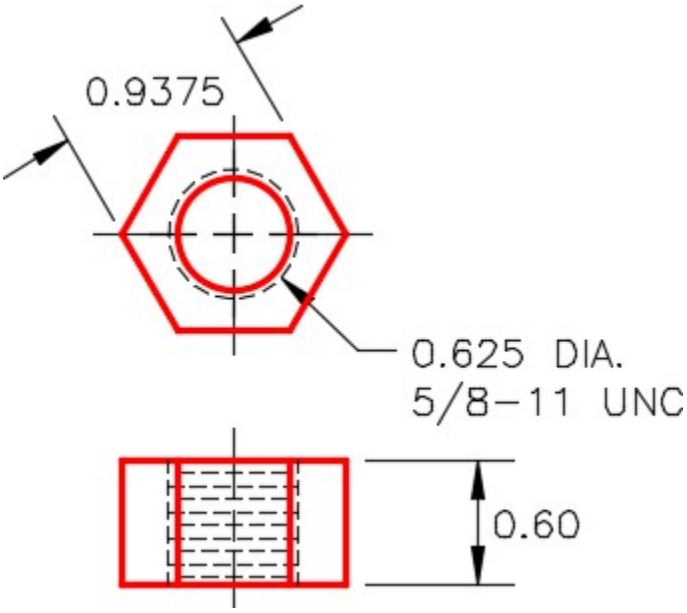


Figure Step 5A
Dimensioned Multiview Drawing of Nut

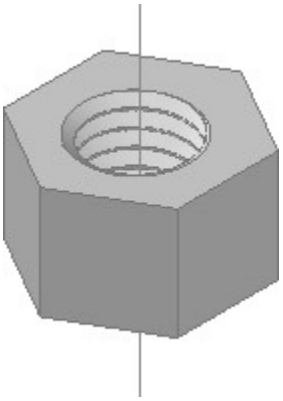


Figure Step 5B
Nut –
SE Isometric View

Step 6

Draw a solid model of the washer on layer: Solid 8 using the dimensioned drawing as a reference. Draw a centre line in the model. (Figure Step 6A and 6B)

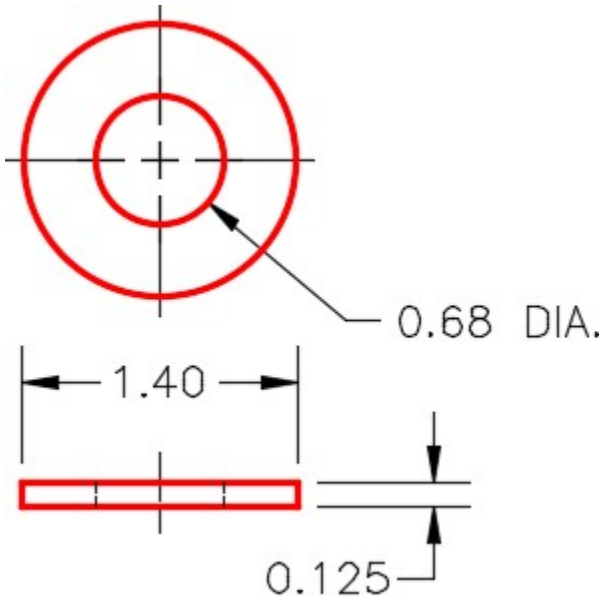


Figure Step 6A
Dimensioned Drawing of Washer

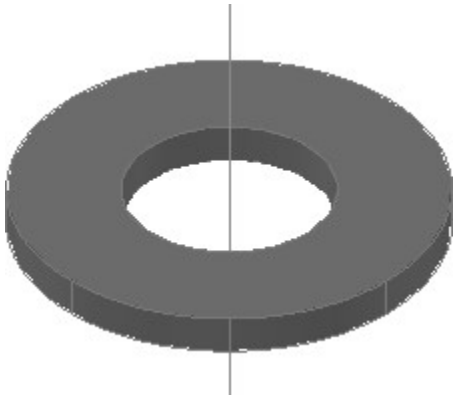


Figure Step 6B
Washer –
SE Isometric View

Step 7

Draw a solid model of the spring on layer: Solid 8 using the dimensioned drawing as a reference. Draw a centre line in the model. (Figure Step 7A and 7B)

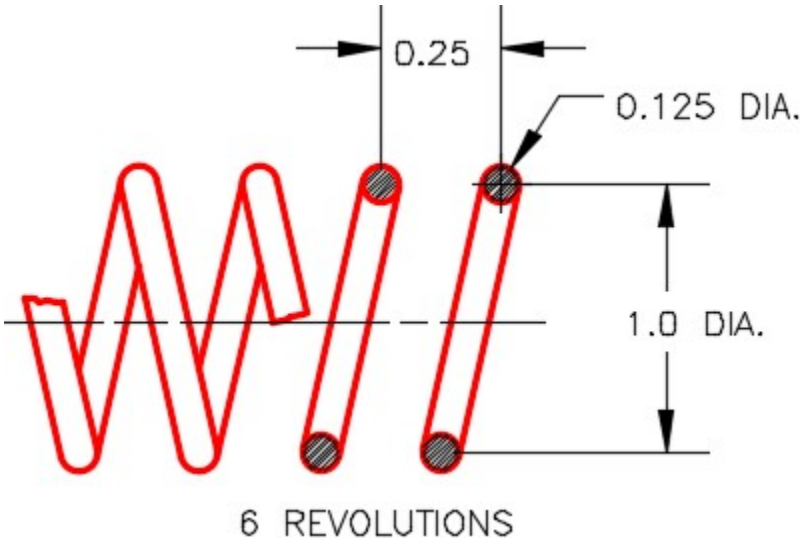


Figure Step 7A
Dimensioned Drawing of Spring

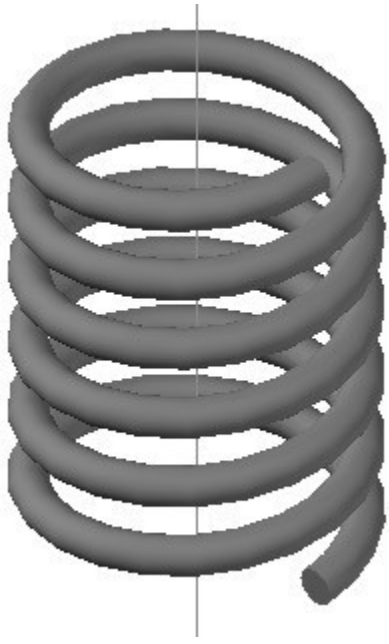


Figure Step 7B Spring – SE
Isometric View

Step 8

Draw a solid model of the lock washer on layer: Solid 4 using the dimensioned drawing as a reference. Draw a centre line in the model. (Figure Step 8A, 8B, and 8C)

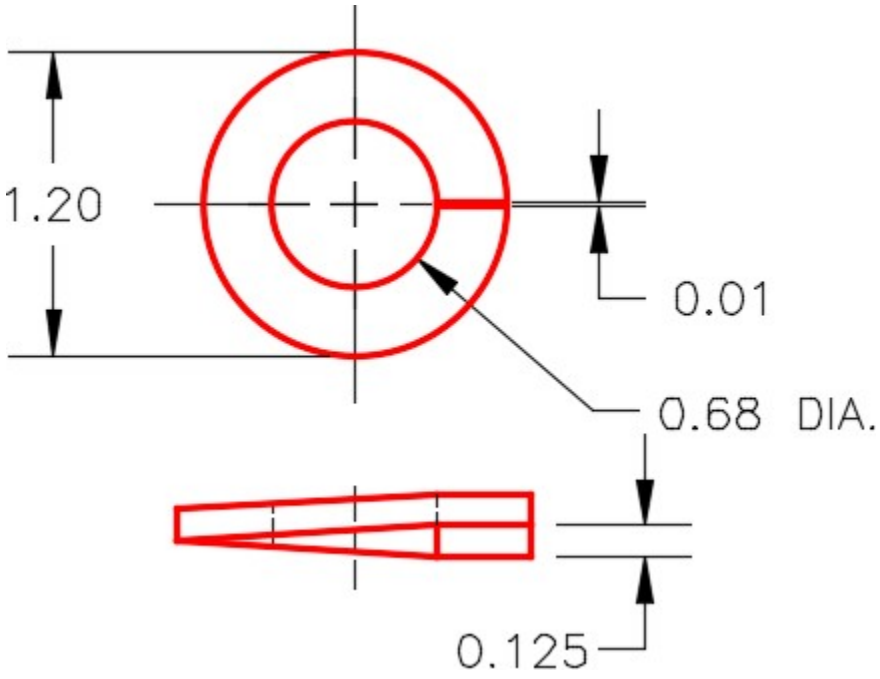


Figure Step 8A
Dimensioned Multiview Drawing of Lock Washer

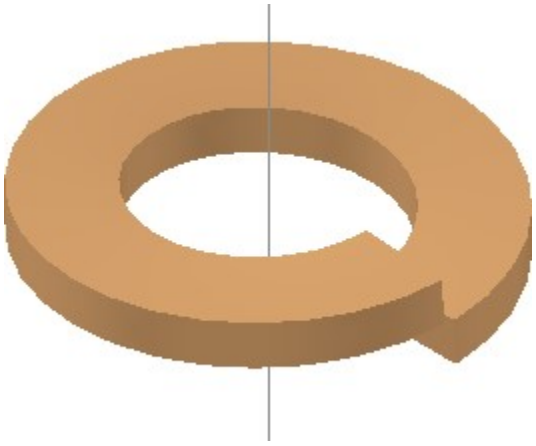


Figure Step 8B
Lock Washer – SE Isometric View

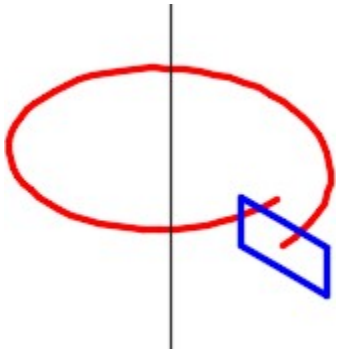


Figure Step 8C
Helix Hints Turn Height – 0.130
Turns – 0.99

Step 9

Using the centerlines as guide, copy the solid models to create an assembly solid model. After you assemble all the parts, by snapping to the centerlines. Change the current view to Right, the current UCS to Right and with Ortho enabled, move the parts ,by eye, to locate them vertically. (Figure Step 9A and 9B)

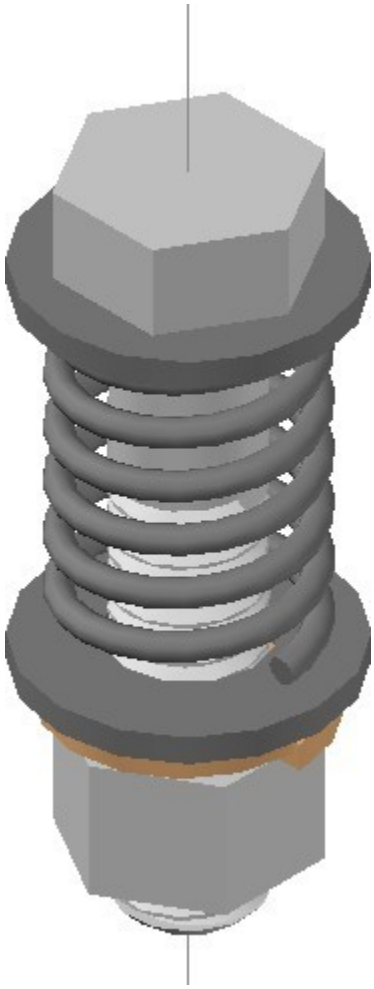


Figure Step 9A
Assembly –
SE Isometric View

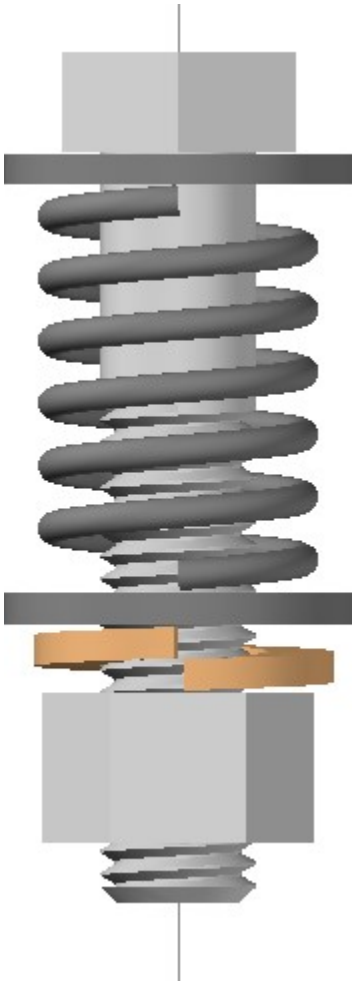


Figure Step 9B Assembly – Right Side View

Step 10

Save and close the drawing.

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3D Wireframe…….. Module 10
3D Polylines……. Module 15
3DARRAY…… Module 9
3DFACE…… Module 12
3DMESH…… Module 14
3DORBIT…….. Module 1
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Absolute X0Y0Z0…….. Module 3
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UCS……. Module 3
UCSICON……. Module 1
UCSMAN…….. Module 4
UNION…… Module 17
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User Coordinate System…….. Module 3-4
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Wireframe Modeling…… Module 1
World Coordinate System…… Module 3

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 5, 2021	Book published.	